DYNAMIC ANALYSIS OF A 10M DIAMETER WIND TURBINE ROTOR UNDER MAXIMUM WIND LOAD OF MALAYSIA

SOFIAN BIN MOHD

UNIVERSITI TEKNOLOGI MALAYSIA

PSZ 19:16 (Pind. 1/97)

UNIVERSITI TEKNOLOGI MALAYSIA

BORANG PENGESAHAN STATUS TESIS[υ]

JUDUL : DYNAMIC ANALYSIS OF A 10M DIAMETER WIND TURBINE ROTOR UNDER MAXIMUM WIND LOAD OF MALAYSIA

SESI PENGAJIAN : 2005/2006

Saya **SOFIAN BIN MOHD**

(HURUF BESAR)

mengaku membenarkan tesis (~~PSM~~/Sarjana/~~Doktor Falsafah~~)* ini disimpan di Perpustakaan Universiti Teknologi Malaysia dengan syarat-syarat kegunaan seperti berikut :

1. Tesis adalah hak milik Universiti Teknologi Malaysia.
2. Perpustakaan Universiti Teknologi Malaysia dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran di antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

☐	SULIT	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
☐	TERHAD	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/ badan di mana penyelidikan dijalankan)
✓	TIDAK TERHAD	

(TANDATANGAN PENULIS)

Alamat Tetap:

I-22, KG. AYER BEMBAN,
81020 KULAI,
JOHOR DARUL TAKZIM

Tarikh : **8 MEI 2006**

(TANDATANGAN PENYELIA)

Nama Penyelia:

PROF. IR. DR. ABAS ABDUL WAHAB

Tarikh : **8 MEI 2006**

CATATAN : * Potong yang tidak berkenaan.

** Jika Tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/ organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai SULIT atau TERHAD.

υ Tesis dimaksudkan sebagai tesis bagi Ijazah Doktor Falsafah dan Sarjana secara penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

UTM(PS)-1/02

School of Graduate Studies

Universiti Teknologi Malaysia

VALIDATION OF E-THESIS PREPARATION

Title of the thesis : Dynamic Analysis of a 10m Diameter Wind Turbine Rotor under Maximum Wind Load of Malaysia

Degree: Master of Engineering (Mechanical-Pure)

Faculty: Mechanical Engineering

Year: 2006

I SOFIAN BIN MOHD

(CAPITAL LETTER)

declare and verify that the copy of e-thesis submitted is in accordance to the Electronic Thesis and Dissertation's Manual, School of Graduate Studies, UTM

(Signature of the student)

Permanent address:

I-22, Kg. Ayer Bemban,

81020 Kulai,

Johor Darul Takzim

(Signature of supervisor as a witness)

Name of Supervisor: Prof. Ir. Dr. Abas Abdul Wahab

Faculty: Mechanical Engineering

Note: This form must be submitted to SPS together with the CD.

"I hereby declare that I have read this thesis and in my opinion this thesis is sufficient in terms of scope and quality for the award of the degree of Master of Engineering (Mechanical)".

Signature : ____________________

Supervisor : Prof. Ir. Dr. Abas Abdul Wahab

Date : 8 May 2006

DYNAMIC ANALYSIS OF A 10M DIAMETER WIND TURBINE ROTOR UNDER MAXIMUM WIND LOAD OF MALAYSIA

SOFIAN BIN MOHD

A dissertation submitted in partial fulfillment of the
requirements for the award of the degree of
Master of Engineering (Mechanical-Pure)

Faculty of Mechanical Engineering
Universiti Teknologi Malaysia

MAY 2006

"I hereby declare that this dissertation is the result of my own research except those cited in references."

Signature : ____________________

Name of Author : SOFIAN BIN MOHD

Date : 8 MAY 2006

To my beloved family,
The lover in you who brings my dreams comes true.

To my wife, Ruhiana Idayu Abd Hamid and daughter, Nurkhaleeda
who have brought a new level of love, patience
and understanding into our lives.

ACKNOWLEDGEMENT

"In the name of Allah that the most Gracious, the most Merciful"

Foremost, my greatest gratitude goes to ALLAH SWT for giving me the energy, strength and spirit to make it possible to complete this thesis.

In particular, I wish to express my deepest appreciation to my supervisor, Prof. Ir. Dr. Abas Abdul Wahab for encouragement, guidance and valuable critics. I am also very thankful to Mr. Nik Ahmad Ridhuan and Mr. Hafiz for their guidance and advices. Without their continued support and interest, this thesis would not have been the same as presented here.

Finally, I owed an immense debt of gratitude to my families for their love to me. To someone who always stays beside me in worse and better time, thanks for your advice, guidance and support.

THANK YOU

ABSTRACT

This project describes the application of finite element analysis (FEA) in studying the strength of the design of the wind turbine rotor under the wind speed of 36 m/s. The forces and pressure produced by the mentioned speed was initially estimated using Computational Fluid Dynamic (CFD). With the forces and pressure set as a boundary loads, the stress analysis was performed using Finite Element Method (FEM). The important criteria such as the displacement and factor of safety were considered in order to produce the optimized model of the wind turbine rotor. The optimized model was defined as the model with low maximum displacement and the minimum factor of safety of 1.5. As an option for cost effective design, the studies were also performed on the wind turbine model under the 15 m/s wind speed load. The model with the thinner AE2 blade (3mm thick) was found to be sufficient for the average wind speed of 15 m/s.

ABSTRAK

Projek ini membincangkan tentang aplikasi kaedah analisa unsur terhingga untuk menganalisa kekuatan dan keteguhan rekabentuk bagi struktur kincir angin yang dikenakan beban angin dengan kelajuan 36 m/s. Daya-daya dan tekanan yang terhasil dari kelajuan angin 36 m/s ini terlebih dahulu dianggarkan dengan menggunakan kaedah dinamik bendalir berkomputer. Daya dan tekanan yang diperolehi ditetapkan sebagai beban sempadan untuk analisa tegasan yang dilakukan dengan menggunakan kaedah unsur terhingga. Daripada keputusan analisa tegasan, anjakan dan faktor keselamatan adalah kriteria penting yang perlu diambilkira dalam menghasilkan model kincir angin yang optimum. Model optimum adalah merujuk kepada model yang berupaya menghasilkan anjakan maksima yang rendah dan faktor keselamatan sekurang-kurangnya 1.5. Sebagai alternatif untuk menghasilkan rekabentuk yang berkos efektif, analisa juga dijalankan ke atas model yang dikenakan beban oleh angin yang berkelajuan lebih rendah iaitu 15 m/s. Berdasarkan keputusan analisis, model dengan bilah AE2 yang lebih nipis (3mm tebal) berupaya menampung beban angin selaju 15 meter sesaat.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

v	-	Wind speed (m/s)
λ	-	Tip speed ratio
P_m	-	Mechanical shaft power (kW)
C_p	-	Power coefficient
a	-	Angle of attack (^{o})
b	-	Setting angle (^{o})
w	-	Resultant velocity (m/s)
E	-	Kinetic energy per unit volume (J/m^3)
ρ	-	Air density (kg/m^3)
P	-	Power in wind (kW)
A	-	Rotor frontal area (m^2)
v_{ave}	-	Average wind speed (m/s)
$\sigma_{1,2,3}$	-	Principal stresses (N/m^2)
σ_y	-	Yield stress (N/m^2)
Re	-	Reynolds number
l	-	Characteristic length (m)
l_{cr}	-	Critical characteristic length (m)
μ	-	Air viscosity (kg/ms)
v_{max}	-	Maximum wind speed (m/s)
r	-	Radius of the blade (m)
u	-	Rotating speed (rpm)

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

The wind energy conversion systems (WECS) have increasingly been developed over the last 10 years. The main reason of having wind as the source of energy is due to its capability of offering the energy without negative environmental impact. Wind energy has long been recognized as a potential source of free, clean and inexhaustible energy.

In order to realize the use of wind energy as the main energy source, there are still a lot of problems that need to be solved such as the wind turbine design, site and wind resources. The major issue recognized as the main barrier to the use of wind energy was the high cost to develop the whole system of wind energy conversion. The cost reduction need to be done in order to make the wind power cost-competitive with the other power source especially for the area which experiences low wind speed.

In Malaysia the wind energy is still new and not yet being applied for any practical use. In 1988 the research group from Universiti Teknologi Malaysia has made first move to investigate the Malaysian wind resource. From the wind data provided by the Malaysian Meteorological Department, it was found that there is a potential to use wind energy for electrical power generation especially in the East Coast of Peninsular Malaysia.

In this project, the static and flow analysis (under maximum wind load of Malaysia) will be conducted on two types of blades which form a complete set of wind turbine rotor (3 pieces each). The focus of this analysis is on both hub and rotor blade sections since they are the major components of the wind rotating system. And the positioning of the wind rotor is only set as perpendicular to the wind load. The main blade is named as AE2 blade which is manufactured through the bending of the flat aluminium plate into 8 bending lines to form its thin aerofoil section. The blade has wider area at its root as compared to its tip. The blade is then twisted lengthwise through a twist angle of 5 degrees. The above mentioned features enable the blade to absorb more energy from low speed wind thus giving the better efficiency to Low Wind Speed Wind Turbine. The smaller blade is called as Starter blade that acts as a starter to initiate the rotation of the whole rotor blade system. The starter blade is fabricated by bending the flat aluminium plate into 2 bending lines. The diameters of the main and starter blades are 10m and 6m respectively which have been scaled up from the previous prototype in order to increase the power produced by the wind turbine system.

The stress analysis is performed on both blades under static and dynamic loads. Static and dynamic loads are obtained from the Computational Fluid Dynamics (CFD) simulation. The wind turbine rotor model will be optimized in order to ensure that it will satisfy the stress analysis. The optimization will be covering certain aspects such as the blade thickness, the addition of stiffener and the extra supports. This particular project also will be considering the analysis at different wind speed as a comparison study in order to permit an option for cost reduction and easy fabrication.

1.1 Objective

The objective of this project is to analyze the strength of the wind turbine rotor under the wind speed of 36 m/s. In this study, it is important to come out with the optimized design of the wind turbine rotor that could be operated safely under the specified wind load.

1.2 Scopes

The project scopes are as follow:

- Study on the wind turbine rotor including the analysis required
- Design the hub for the wind turbine system
- Perform the solid modelling of the wing turbine rotor using SolidWork 2005
- Perform the flow analysis on the wind turbine rotor using FLUENT
- Perform the stress analysis on the wind turbine rotor using COSMOSWork
- Modification and improvement of the model to produce an optimized model

CHAPTER 2

LITERATURE REVIEW

An extensive literature search in the related area was conducted. It has been done to get some idea for the project. The main sources for the literature search are books and technical papers. One of the papers that closely related to this project is written by N.M. El Chazly.

As engineering investigation revealed that many of the structural failures of wind turbines occur in the blade root section, several possible solutions have been introduced in order to deal with this type of failure. One of the most promising solutions is to do a 3D analytical modeling to compute critical parameters of the rotor blades such as the deflection, stresses, and eigenvalues. As proposed by N.M El Chazly [1], this analytical modeling can be done using a bending triangular plate finite element.

In particular, lift and drag forces are set in a steady wind conditions and they are analyzed as normal and tangential forces on the blade sections at certain angle of attack. According to his work, these forces are applied as boundary loads to the computer program in order to perform both static and dynamic analysis of the rotor blades for a symmetrical aerofoil NACA 0015 series.

Constant chord, tapered blades and twisted ones have been analyzed at various wind speeds. The computer program can be validated by applying it to a standard cantilever box beam using the beam theory. The result of this study indicates that the maximum stress does happen at the root of the blades for all the configurations in the span wise direction. For that, tapered blades are used to diminish the stresses as well as to save the material weight. And with the twisting of the rotor blade, the stiffness can be increased and the relevant stresses can be reduced.

2.1 Working Principle of the Studied Wind Turbine

The designed wind turbine is the horizontal-axis type of wind machine. It has been develop since 1998. The use of this wind turbine is for electricity generation. The target power output is 1 kW with 6 m diameter (AE2 blade) of the frontal area. This wind turbine is working under the average wind speed of 2.5-6 m/s with the operating rotating speed around 90-200 rpm. The main components of the wind turbine system are AE2 blade, Starter blade, hub, shaft, front tail, rear tail and tower.

The function of the rear tail is to direct the wind turbine so that the coming wind is perpendicular to the frontal area of the wind turbine rotor. The role of the Starter blade is to accelerate the starting motion. Once the wind turbine is rotating, the AE2 blade which acts as the power blade in increasing the rotational speeds of the wind turbine up to the limit of 200 rpm under the wind speed of 6 m/s. When the average wind speed exceeds 6 m/s, the front tail will turns the wind turbine system so that the frontal area is not exposed to the extra load. The speed control system will make sure that the wind turbine rotor is working within the limit in order to prevent structures failure.

The airfoil section of AE2 blade design is taken from the shape of the camber line of the airfoil HK 8556 as shown in Figure 2.1.

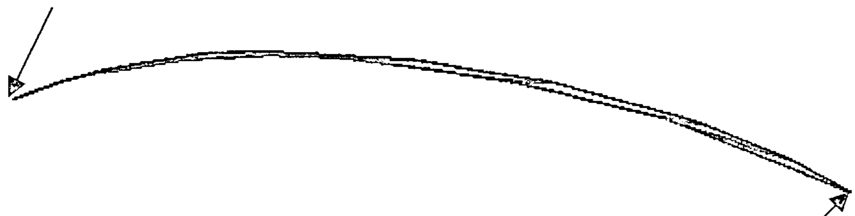


Figure 2.1: The top view of AE2 blade shows its airfoil section

2.2 The Use of FLUENT for the Flow Analysis around Wind Turbine

There are three approaches that can be used to analyze the flow around and downstream of the wind turbine. First is field testing, which provides accurate results but is too complex and expensive; Second is analytical and semi-empirical models, which adopt simplifying assumptions and are thus not universally reliable; and third is CFD, which offers the best alternative to direct measurements [2]. One of the most widely used CFD code is FLUENT.

At the University of Cagliari, FLUENT has been used to analyze an aerodynamic problem which is difficult to be tackled experimentally. The 41 m diameter of the wind turbine rotor rotating at a constant speed of 27 rpm was simulated using CFD and simplified analytical model. The purpose of the study is to investigate the disturbed flow field including the wake in the axial direction.

The computational domain used is in the shape of diffuser with 600 m length as shown in Figure 2.2. The diameter of the domain inlet and outlet are 5 m and 10 m respectively. In the plane of the rotor, the domain diameter is 5 times than that of the rotor. The model used to simulate the flow is Multiple Reference Frame (MRF). The flow is assumed as incompressible and steady-state with a uniform wind speed at the entrance of the domain. The "one-equation Spalart-Allmaras" model with standard wall functions was chosen for turbulence closure. GAMBIT was used to build a multi-block hexahedral mesh of approximately 1.5 million elements.

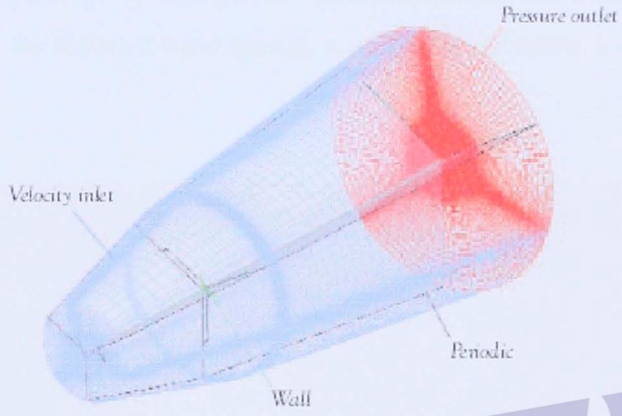


Figure 2.2: Computational domain and boundary conditions [2]

The classical blade element momentum (BEM) method was adopted for the design of the turbine rotor, using the specifications for the three-bladed horizontal axis Nordtank 41/500 turbine [3] and NACA 63-4xx profile [4]. The result obtained using FLUENT was then compared with the BEM result. In order to compare the results from both methods, the overall performance of the turbine was computed including an assessment of the mechanical power generated on the shaft axis and the corresponding power coefficient.

From Figure 2.3, it shows that there was a good agreement between the CFD results and those obtained using the BEM method. The total pressure and axial velocity contours predicted by FLUENT were used to show the wake development downstream of the turbine and to identify the transition from the near wake to the far wake region. Figure 2.4 and Figure 2.5 show the contours of the total pressure and axial velocity.

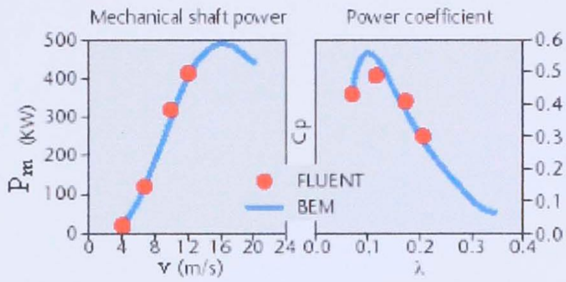


Figure 2.3: Mechanical power and power coefficient determined using the BEM method and CFD, for different wind speeds, v, and tip speed ratios, λ [2]

Figure 2.4: Contours of total pressure illustrate the wake [2]

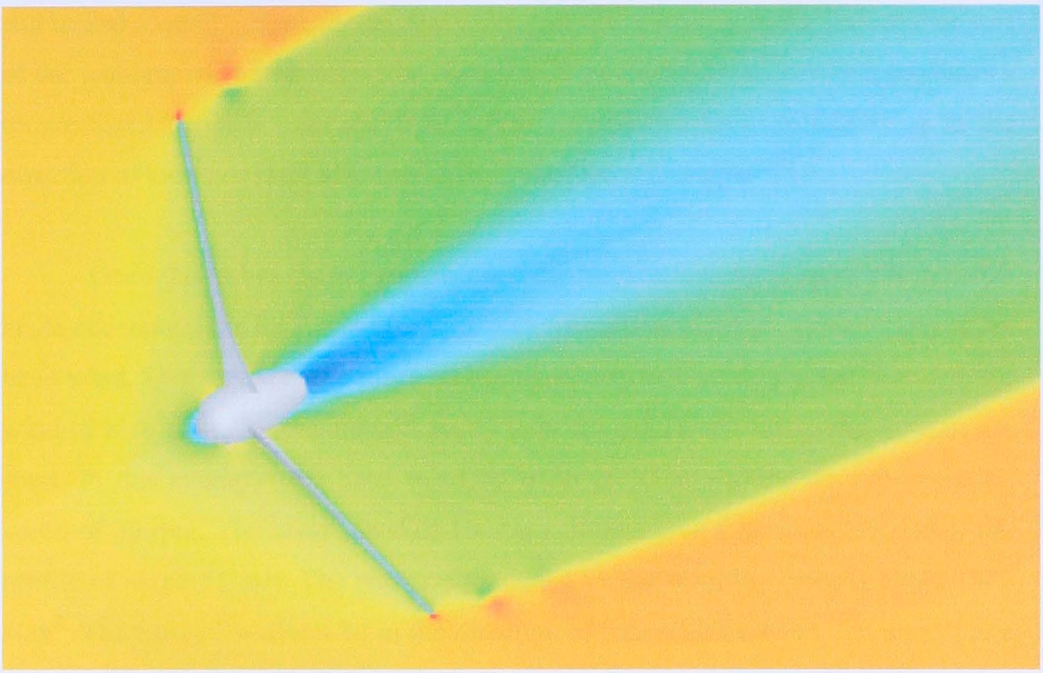


Figure 2.5: Axial velocity, showing the reduction of the de-energized core in the far wake region [2]

2.3 The Aerodynamics of Wind Turbine Blades

The direction of the air flow around the blade is depending on the rotation of the blade [5]. Thus, we need to consider both stationary and rotating conditions in order to describe the aerodynamics of the wind turbine blade. Figure 2.6 shows the force diagrams of the cross section near the blade tip operating in a wind speed, v of 10m/s. The example as discussed here is using the cross section of a Bonus 450 kW Mk III blade as a model.

When the rotor is stationary, as shown in Figure 2.6(A), the wind has a direction towards the blade which is the area swept by the rotor during the rotation of the blades. The wind speed of 10 m/s will produce a wind pressure of 80 N/m^2 of the blade surface. The wind pressure is roughly in the same direction as the wind load and is also roughly perpendicular to the flat side of the blade profile. The smaller part of the wind pressure blowing in the direction of the blades rotation which produces a torque that attempt to start the wind turbine, while the larger part blowing in the direction of the rotor shaft attempts to bend the blades and tower [5].

Once the turbine is in operation and the rotor is turning (Figure 2.6 (B)), the blade encounters the head wind from its own forward movement. The strength of the head wind, U at any specific place on the blade depends partly on just how fast the wind turbine blade is rotating and partly how far out on the blade one is from the shaft. In this example, the head wind, U is about 50 m/s at the normal operating speed of 30 rpm. The wind speed of 10 m/s will thus give a resulting wind over the profile of about 51 m/s. The pressure produced by the resulting force is about 1500 N/m^2. The force, F will not be in the direction of the resulting wind, but almost at a right angle to the resulting wind.

Figure 2.6(C) shows that the resulting force could be resolved into two components, which are in the direction of rotation and in perpendicular to the direction of rotation. In both cases, we may notice that the forces on the blade become very large during rotation. This is due to the resulting wind speed of 51 m/s strikes a blade during operation, which is many times larger than the resulting wind speed when the blade is at rest.

Another important observation is that the force on the blade is almost at a right angle to the resulting wind striking the blade due to its airfoil profile. This force is known as the lift and also produces a small resistance or drag. The direction of this lift is of great importance to give the power to the wind turbine in order to increase its rotational speed.

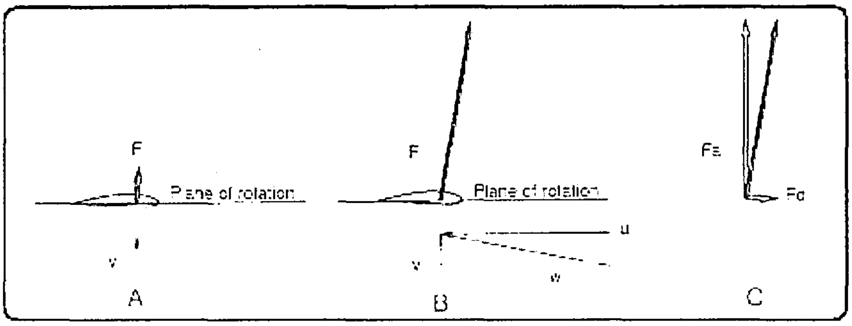


Figure 2.6: Airflow around a blade profile, near the wing tip [5]

2.4 The Change of Forces along the Blade

The magnitude of the forces and their direction change accordingly with their distance to the tip [5]. When the blade is in stationary, the force at root becomes slightly larger than the force at the tip as the blade is wider at the root. For the twisted blade, the force will be directed in the direction of rotation at the root. This is because the blade is more twisted at the root compared to the tip.

In comparison with the blade tip, the blade root section produces less aerodynamic forces during operational; however more of these forces are aligned in the correct direction, that is, in the direction of the rotation. The change of the magnitude and the direction of these forces from the tip in toward the root, determine the form and shape of the blade.

The head wind is not so strong at the blade root, therefore the pressure is likewise not so high and the blade must be made wider so that the forces should be larger enough. The resulting wind has a greater angle in relation to the plane of rotation at the root, so the blade must likewise have a greater angle of twist at the root.

Further out along the blade, the profile must be made thinner in order to produce acceptable aerodynamic properties, and therefore the shape of the profile at any given point on the blade is a compromise between the desire for strength (the thick wide profile) and the desire for good aerodynamic properties (the thin profile) with the need to avoid high aerodynamic stresses (the narrow profile).

2.5 Effect of Changes in Wind Speed

In order to understand the behavior of blade at different wind speeds, it is necessary to understand a little bit about how lift and drag change with a different angle of attack. Again, we will use the Bonus 450 kW Mk III blade as a model as an example. In Figure 2.7 below, the angle of attack is denoted as "a", whereas the setting angle is denoted as "b". The setting angle has a fixed value at any given place on the blade, but the angle of attack will grow as the wind speed increases.

The aerodynamic properties of the profile will change when the angle of attack "a" changes [5]. Figure 2.8 illustrates the changes of lift and drag with the increment of the angle of attack.

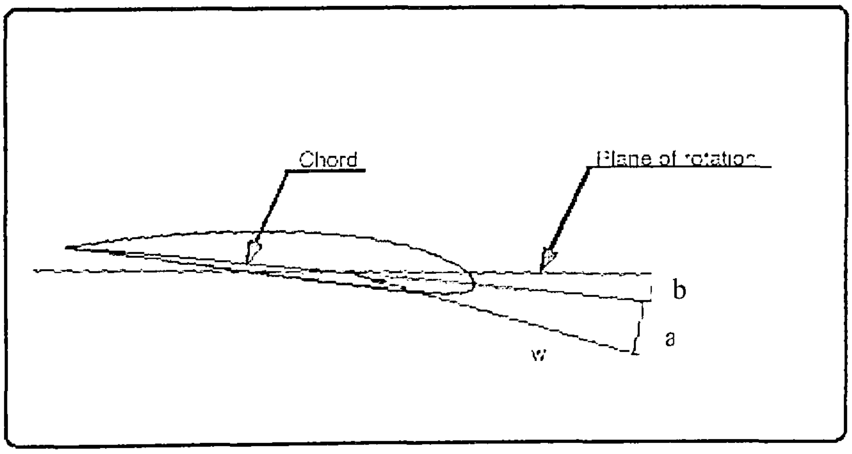


Figure 2.7: The angles of the profile [5]

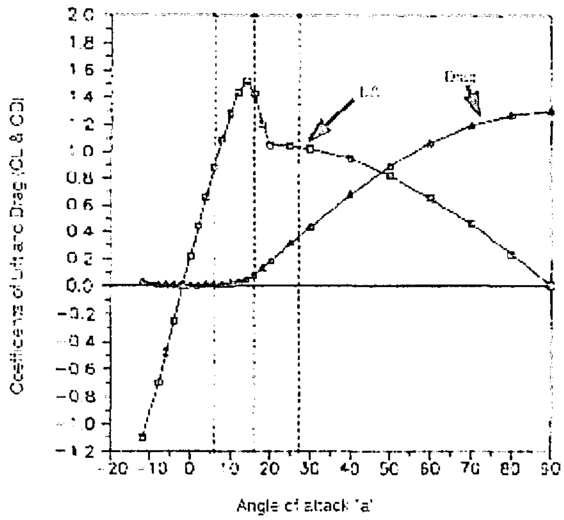


Figure 2.8: Relationship between lift and drag coefficients and the angle of attack [5]

There is a sudden change of both lift and drag when the angle of attack exceeds 15-20 degrees. The point of the sudden change is called *stall.* When this stalling point is reached, the lift will falls and the drag increases. To further study of the effect of the change of wind speed to the angle of attack, we can refer to Figure 2.9. The diagram shows the situations at three different wind speeds: 5 m/s, 15 m/s and 25 m/s at the blade tip. Considering the wind turbine has a constant rotational speed controlled by the grid connected generator which led to the constant head wind speed U, the increment of the wind speed v will cause a great change of the angle of attack. As discussed earlier, this change is of great importance for determining the strength of the aerodynamic forces.

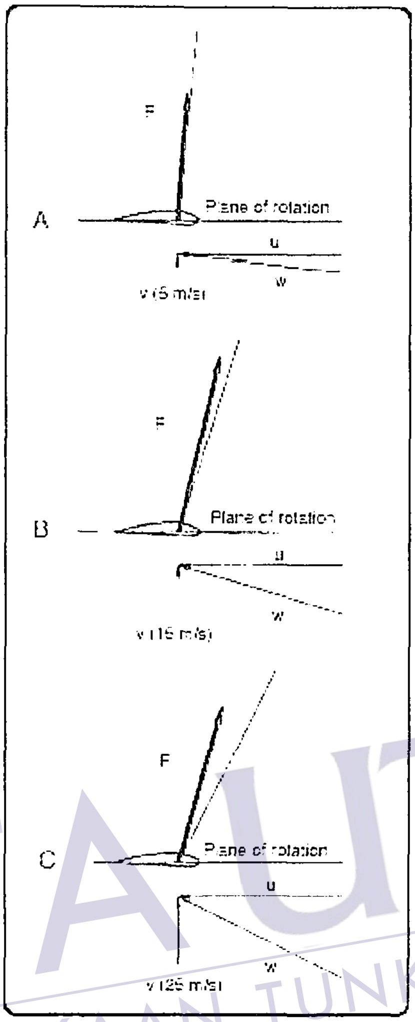


Figure 2.9: Conditions at three different wind speeds [5]

2.6 Works Done on the Wind Energy in Malaysia

Wind speed data provided by the Meteorological Department of Malaysia shows that the wind energy on the west coast of Peninsular Malaysia is low where the average wind speed recorded was only about 2 m/s [6]. Meanwhile, the east coast which is facing the South China Sea experiences the strong wind throughout the year.

A project to collect the data of wind speed at Pulau Layang-layang (an island located in South China Sea) has indicated that the average wind speed at that island is more than 6 m/s, which is suitable for electricity generation. The study was carried out by Tenaga Nasional R&D Berhad.

The information from Meteorological Department of Malaysia also shows that maximum wind speed of Malaysia was about 36 m/s which is predicted to reoccur once in 50 years.

2.7 Wind Energy

The energy associated with the wind is in the form of kinetic energy [7]. The kinetic energy in the air (speeding air) is proportional to the square of its velocity. The kinetic energy (E) per unit volume of moving air is expressed as

$$E = \frac{1}{2}\rho v^2$$

Where v is the average linear wind velocity and ρ is the density of the air. This energy in the wind is partially transformed to pressure against an object when that object is approached and the air slows down. By adding up the pressure acting over the entire object will gives the total force on that object.

Power is force times velocity. Since wind forces are proportional to the square of the velocity, wind power is proportional to cubic of wind speed. Doubled up the wind speed will increase the wind power by the factor of eight. The power (Watt) that wind turbine blades could extract from the wind is given by the following expression which is used in Commonwealth Regional Energy Resources Information System (1984).

$$Power = \frac{1}{2} \rho A (v_{ave(annual)})^3$$

V_{ave} = average wind velocity in a year
ρ = air density
A = rotor frontal area

2.8 Overspeed Control

Blades are design to withstand a certain centrifugal force and a certain wind load. The wind loads tend to bend the blades, whereas the centrifugal force tends to pull a blade from the rotor hub. It is essential to understand the various methods of rotors speed control. A control is needed to prevent over stressing the WECS system in extreme wind speed.

One of the methods is to design a wind turbine strong enough to withstand the highest possible wind. But this would involve an expensive cost especially for the installation. Another option is to design a good control system for more fragile unit. The examples of the method for controlling a wind turbine are [8]: 1) tilting the wind wheel out of excessive winds, and 2) changing the blade angles to lower their loads.

2.9 Von Mises Criterion

The von Mises Criterion, also known as the maximum distortion energy criterion, octahedral shear stress theory, or Maxwell-Huber-Hencky-von Mises theory, is often used to estimate the yield of ductile materials. It was introduced by Richard von Mises in 1913. The von Mises criterion states that the failure occurs when the energy of distortion reaches the same energy for yield/failure in uniaxial tension [9]. It could be expressed as

$$\frac{1}{2}\left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2\right] \leq \sigma_y^2$$

σ_1, σ_2, and σ_3 are the principal stresses. In the case of plane stress, σ_3 is zero and the above equation reduces to

$$\sigma_1^2 - \sigma_1\sigma_2 + \sigma_2^2 \leq \sigma_y^2$$

Figure 2.10 below illustrates a principal stress ellipse represented by the simplified equation. Also shown on the figure is the maximum shear stress (Tresca) criterion (dashed line). This theory is more conservative than the von Mises criterion since it lies inside the von Mises ellipse. In addition to bounding the principal stresses to prevent ductile failure, the von Mises criterion also gives a reasonable estimation of fatigue failure, especially for the cases of repeated tensile and tensile-shear loading. That could be the best reason to explain why most of the finite element analysis results are presented as von Mises stress.

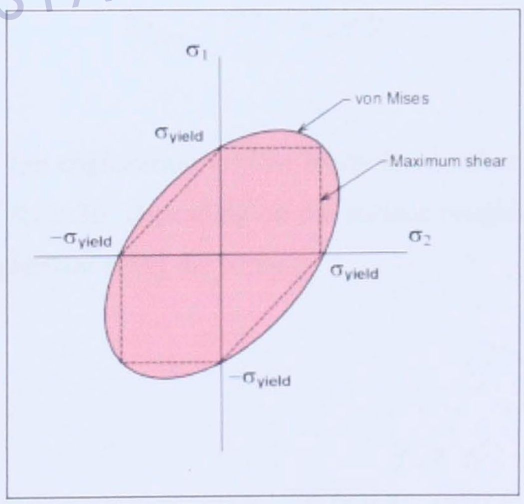


Figure 2.10: Plot of Von Mises and Tresca Criteria [9]

2.10 Laminar and Turbulent Flows

Most flows encountered in engineering practice are turbulent. Laminar flow is encountered when highly viscous fluids such as oils flow in small pipes or narrow passages [10]. Laminar flow is characterized by smooth streamlines and highly ordered motion whereas the turbulent flow characterized by random and rapid fluctuations of swirling regions of fluid, called eddies, throughout the flow.

In laminar flow, momentum and energy are transferred across streamlines by molecular diffusion. Meanwhile, in turbulent flow, the swirling eddies transport mass, momentum, and energy to other regions of flow much more rapidly than molecular diffusion. Even when the average flow is steady, the eddy motion in turbulent flow causes significant fluctuations in the values of velocity, temperature, pressure, and even density.

For the flow over a smooth flat plate, transient from laminar to turbulent begins at about $\mathrm{Re} \cong 1x10^5$ [10]. However it does not become fully turbulent before the Reynolds number reaches higher values, typically around $3x10^6$. In engineering analysis, a generally accepted value for the critical Reynolds number is

$$\mathrm{Re}_{Lcr} = \frac{\rho v l_{cr}}{\mu} = 5x10^5$$

The actual value of the engineering critical Reynolds number for a flat plate may vary from about 10^5 to $3x10^6$ depending on the surface roughness, turbulence level and the variation of pressure along the surface.

2.11 Finite Element Method

Finite element method is defined as a computerized method for predicting how a real world object will react to forces, heat, vibration, etc [11]. In a simple word, it is used to predict what will happen when the product is used under the specified loading and constrain. FEM is accepted as the standard analysis method due to its generality and suitability for computer implementation

With the advances in computer technology and CAD systems, the complex problems can be modelled with relative ease. Several alternative configurations can be tried out on a computer before the first prototype is built. All of this suggests that we need to keep pace with these developments by understanding the basic theory, modelling techniques and computational aspects of the finite element method.

The finite element method works by breaking a real object down into a large number of elements, such as little cubes. The process of breaking the model into small pieces is called meshing. The behaviour of each little element, which is regular in shape, is readily predicted by set mathematical equations. Then the computer adds up all the individual behaviours to predict the behaviour of the actual object.

The response at any point in an element is interpolated from the response at the element nodes. Each node is fully described by a number of parameters depending on the analysis type and the element used. For example, the temperature of a node fully describes its response in thermal analysis. For structural analyses, the response of a node is described, in general, by three translations and three rotations. These are called degrees of freedom (DOFs). Analysis using FEM is called Finite Element Analysis (FEA).

Nowadays there are a lot of finite element softwares available in the market. Some of them are easy to use and user friendly. One of the popular finite element software uses is COSMOSWorks. COSMOSWorks is a design analysis system fully integrated with SolidWorks. COSMOSWorks provides one screen solution for stress, frequency, buckling, thermal, and optimization analyses. Powered by fast solvers, COSMOSWorks enables us to solve large problems quickly using personal computer. Besides it is capable in analyzing the assembled components which gives the exact result for each component in that assembly. Figure 2.11 shows the example of the FEM simulation using COSMOSWorks.

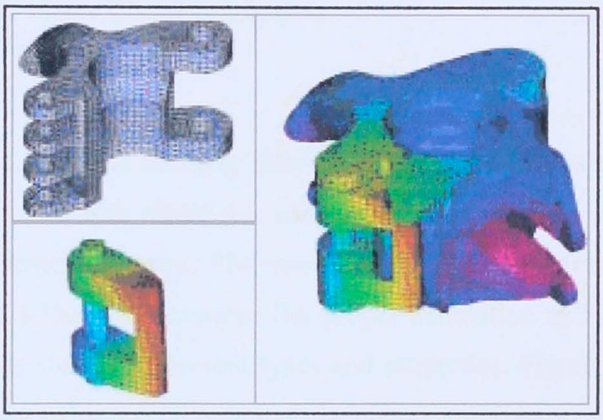


Figure 2.11: Finite element simulation using COSMOSWork

COSMOSWorks formulates the equations governing the behavior of each element taking into consideration its connectivity to other elements. These equations relate the response to known material properties, restraints, and loads. Next, the program organizes the equations into a large set of simultaneous algebraic equations and solves for the unknowns. In stress analysis, for example, the solver finds the displacements at each node and then the program calculates strains and finally stresses. COSMOSWork offers several types of analyses which are static, frequency, buckling, thermal, drop test and fatigue analysis.

Static (or stress) analysis calculates displacements, reaction forces, strains, stresses, and factor of safety distribution. Material fails at locations where stresses exceed a certain level. Factor of safety calculations are based on a failure criterion. COSMOSWorks offers four failure criteria and the most popular one is von Mises criteria which is applicable for ductile material. Static studies can help to avoid failure due to high stresses. A factor of safety less than unity indicates material failure. Large factors of safety in a contiguous region indicate low stresses and that we could probably remove some material from that region.

2.11.1 Conceptual in Finite Element Analysis

An element is the basic building block of finite element analysis. There are several basic types of elements where it is used depends on the type of object to be modelled for finite element analysis. The most common types are truss, beam, 2-D solid, plate/shell and 3-D solid elements. The proper simulation of a real structure depends on the proper choice of element types and properties. Figure 2.12 shows a several types of elements that can be used in finite element processor.

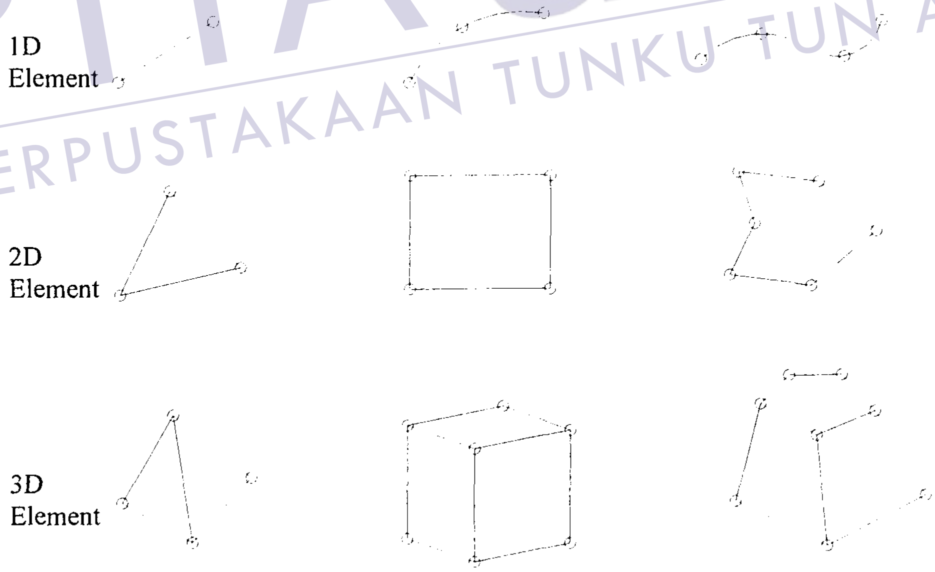
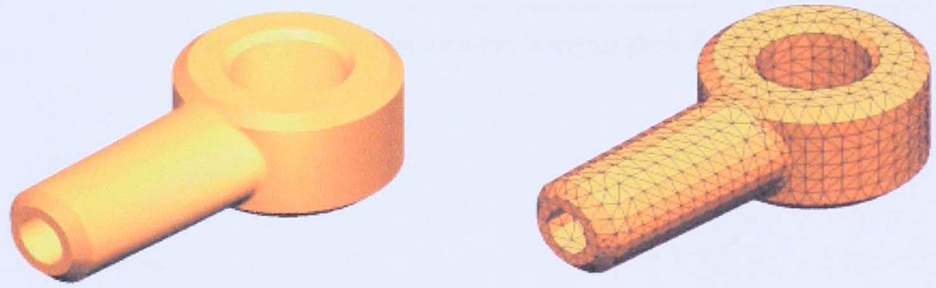


Figure 2.12: Types of element for meshing

Three dimensional solid elements are becoming the dominant form in mechanical engineering because most mechanical parts have a certain bulk and can convert solid models from modern CAD solid modeling systems such as Pro/ENGINEER and AutoCAD from Computer Vision into solid brick elements automatically. Also, solid elements can handle stress, dynamic, heat transfer, fluid flow, electrostatic and magnetic analysis, whereas other element types are more limited in application [12].

Two dimensional solid elements are essentially obsolete. Their main function in the past was to approximate three dimensional models with two dimensional representations because computers were much slower and good modeling tools were not available in the past. But nowadays with the advent of new technologies, all this matters have been solved.

Meshing is the second important process after defining the material properties and element properties for the structure. In this process, the geometry model is mesh into many nodes and elements. It is essential to define the meshing size or in other words to do a right choice of the number of nodes. This is because some of the critical point or area in the geometry model needs to have a small meshing size in order to give an accurate model. The smaller the meshing size the more accurate the results of the analysis. Figure 2.13 shows the example of the meshed model.



(a) CAD model of a part

(b) Model subdivided into small pieces (elements)

Figure 2.13: The model before and after meshed

In general, the finite element analysis consists of 6 major steps which have been shown in Figure 2.14 [13]. This figure shows the flow chart of steps in finite element analysis. All this step is very important and must been followed in order to solve for the finite element problem.

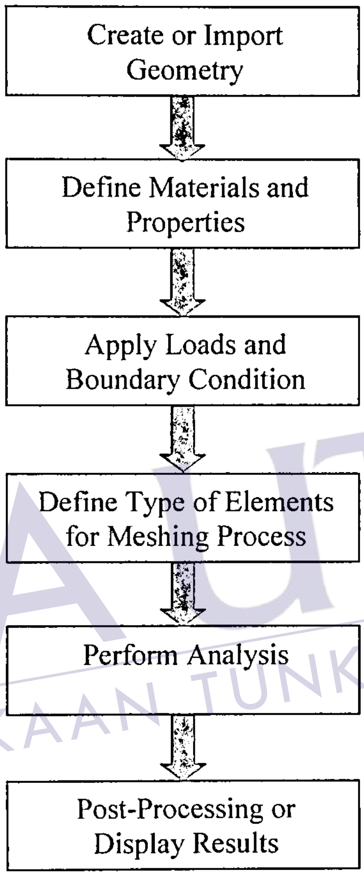


Figure 2.14: Finite element analysis procedure [13]

2.12 Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomenon by means of computer based simulation. The example of the problem that could be analysed using CFD is the aerodynamics of aircraft and vehicle (lift and drag).

From the 1960s onward the aerospace industry has integrated CFD technique into the design, R&D and manufacture of aircraft and jet engines [14]. Nowadays, the CFD has been widely applied to predict drag force, under- bonnet air flows and in car environment by motor vehicle manufacturer and also used to design of internal combustion engines, combustion chambers of gas turbines and furnaces.

There are the advantages of using the CFD compared to experimental testing. By using the CFD, ones can produce extremely large volume of results at virtually no added expenses and it is very cheap to perform parametric studies and also for instance to optimise the equipment performance. In contrast, conducting the experimental testing will involve a variable cost in terms of facility hire and/or man-hour costs and time consumption for data collection and model development.

FLUENT is a computational fluid dynamics (CFD) code widely used for flow-modeling applications. It is capable to handle subsonic or supersonic flows, steady or transient flows, laminar or turbulent flows, Newtonian or non-Newtonian flows, single or multiphase flows, chemical reactions including combustion, flow through porous media, heat transfer, and flow-induced vibration. GAMBIT is pre-processing code integrated with the FLUENT. Pre-processing involves building the model or importing model from a CAD package, applying a finite-volume-based mesh and entering data. The detail of other elements of CFD code is discussed in the following section.

REFERENCES

1. N.M.El Chazly, "Static and Dynamic Analysis of Wind Turbine Blades Using the Finite Element Method", Renewable Energy, Vol. 3, pp.705-724, Elsevier Science Ltd., 1996.
2. "The Economy of Large Scale Wind Turbines". http://www.fluent.com/about/news/newsletters/05v14i2/a10. htm
3. Hansen, M.O.L, "Aerodynamics of Wind Turbines: Rotors, Loads and Structure", James & James, London, 2000.
4. Abbott, J.H., Von Dohenhoff, A.E., "Theory of Wing Sections", Dover Publications Inc., New York, 1959.
5. "The Wind Turbine Components and Operation". http://www.windmission.dk/workshop/BonusTurbine. pdf
6. W.T Chong, (1997), "The Design And Testing of a Wind Turbine for Malaysian Wind Condition", Master thesis, University of Technology Malaysia.
7. P.C.Nicholas, "Fundamentals of Wind Energy", Ann Arbor Science Publisher, Michigan, 1978.
8. V.Daniel Hunt, "Windpower: A Handbook on Wind Energy Conversion Systems", Van Nostrand Reinhold Company,United State of America, 1981.
9. http://en.wikipedia.org./wiki/von_mises_stress
10. Yunus A. Cengel, John M. Cimbala, "Fluid Mechanics: Fundamentals and Applications", McGraw Hill Companies Inc., New York, 2006.
11. Tirupathi R.Chandrupatla, "Introduction to Finite Elements in Engineering". Second Edition, Prentice Hall Inc., 1997.
12. Kenneth H.Huebner, Earl A.Thorton, Ted G.Byrom, "The Finite Element Method For Engineers", Third Edition,

13. "MSC.Nastran for Windows: Quick Start Guide", MSC.Software Corporation, United States of America, 2000.
14. H.K.Versteeg, W.Malalasekera, "An Introduction to Computational Fluid Dynamics", Prentice Hall, England, 1995.