AIR FLOW PROFILE EVALUATION AROUND MOVING VEHICLE

DEENESH KUMAR A/L NALATHAMBI

A project report submitted in partial
fulfillment of the requirement for the award of the
Degree of Master of Mechanical Engineering

Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia

JANUARY 2014

ABSTRACT

Highway lighting at night does increase the electricity costing because despite using solar energy to generate the electric for the lighting, mostly still uses direct current to work. The purpose of this study is to evaluate the air flow dispersed around moving vehicles and the wind turbine selection in harnessing electricity from the air flow dispersed. Solidworks and ANSYS software were used in modeling the vehicles and to analyze the flow characteristic, respectively. Analyses were done by the selection of 3 different vehicles which are a sedan car, a MPV and a bus by considering 3 different velocities i.e. 80km/h, 100km/h and 120km/h at 2 different environments which are the open road highway and road tunnel. Simulation computed the velocity field of the flow around the vehicles and also the turbine. The maximum wind speed dispersed obtained were used to calculate the wind power and mechanical power of the turbine analytically. It proves that the increment of vehicle's speed increases the wind speed dispersed thus increasing the power generated by the selected wind turbine, Savonius Vertical Axis Wind Turbine with Two Semicircular blades.

ABSTRAK

Pencahayaan lampu jalan pada waktu malam akan meningkatkan kos elektrik walaupun menggunakan kuasa solar kerana kuasa solar menggunakan arus terus. Tujuan kajian ini adalah untuk menyiasat penilaian Profil Aliran Udara sepanjang kenderaan bergeak dan turbin udara pilihan dalam usaha menjana elektrik daripada angin yang tersebar. Perisian Solidworks dan ANSYS digunakan untuk menjana model kenderaan dan menganalisa profil angin bergerak. Analisa telah dijalankan dengan pilihan 3 jenis kenderaan iaitu kereta sedan, MPV dan bus dengan mempertimbangkan 3 halaju berbeza iaitu 80km/h , 100km/h dan 120km/h pada 2 lokasi berbeza iaitu jalan raya terbuka dan terowong jalan. Simulasi telah menyelesaikan medan halaju sepanjang kenderaan bergerak dan turbin udara. Nilai udara tersesar maksima telah digunakan untuk mengira kuasa angin dan kuasa mekanikal turbin udara secara pengiraan analitikal dan dibuktikan bahawa peningkatan halaju kenderaan akan meningkatkan halaju angin tersesar dan pada masa yang sama meningkatkan kuasa yang dijana oleh turbin pilihan, Savonius Vertical Axis Wind Turbine with Two Semicircular blades.

CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Research Background

This paper aims to set out a description of the air flow profile evaluation around moving vehicle. This approach is adopted so that the result of the research will clarify the basic air flow mechanisms that exist around vehicles and future steps to be taken in harnessing electricity from the air flow mechanism.

The wind energy harnessing can be considered in moderate wind speed. Other than that, high wind speed is also found in highway called as crosswinds. The highest mean daily wind speed in Malaysia (recorded at Mersing, Johor) is 3.8m/s. In addition to that, movement of land vehicle on highway will induce high speed air motion around it. The energy from crosswinds and the moving air can be converted to electrical energy for road & billboard lighting. Besides that, increasing use of gasoline-powered vehicles significantly contributes to environmental pollution, noise and depletion of crude oil reserves.

In today's life the demand on electricity is much higher than that of its production the main objective of our project is to produce electricity by using the force of air created by the moving vehicle in highways. In highways the vehicle suffers a lot to travel in night time because of lightning problem. This problem can be overcome by using the Vertical Axis Highway Windmill (VAHW). This is a new unique method of power generation. The force will rotate the vertical turbine blade and this blade is coupled with the generator and this generator will produce electricity [1].

This project involves the simulation of the car model with three different cases which are a sedan car, a MPV car and a bus. Computer simulation software (Ansys software) will be used to simulate and investigate the effects of wind turbine against aerodynamics performance of the vehicle. In addition, the study also covers the process of designing an optimum wind turbine system for harnessing wind energy at the highway sides and road tunnels. At the end of the study, a result that consist the dispersed wind speed (at highway side and road tunnel) will be obtained and the optimum design of wind turbine system for harnessing wind energy by the vehicles will be proposed. The amount of electricity produced is expected to support the lighting around the highway and road tunnels – in order to save the electricity cost from road side lightings.

1.2 Problem statement

Highway lighting at night does increase the electricity costing because despite using solar energy to generate the electric for the lighting, mostly still uses direct current to work. The other option left is to harness wind energy generated by the air deflection from the moving vehicles by placing wind turbine along the highway and tunnels sideways. The appropriate data regarding air flow profile on the moving larger, medium and smaller vehicles will be obtained from this study and can be used for further studies.

1.3 Objective

a) To determine air profile of airflow around moving vehicle on open road
b) To determine air profile of airflow around moving vehicle in a road tunnel
c) To design a wind turbine to be placed on the open road and tunnel sideways
d) To determine the amount of electricity be generated by wind turbine

1.4 Scope of the project

The study will emphasize on large vehicles (bus), medium vehicles (MPV) and small vehicle (sedan car) and determining profile of airflow around moving vehicle on open road (highways) and in tunnel based on Malaysia's standard climate and wind situation. Following are the scope or limit of the study:

a) Development of 3D model of the selected vehicles by using Solidworks software
b) Simulation of an air flow (using ANSYS software) around single moving vehicles (on open roads and in road tunnel)
c) Selection of the optimum wind turbine design to be placed on the open road and in tunnel sideways
d) Determine the amount of electricity be generated by wind turbine

1.5 Project Overview

The project activities consists of Chapter1 (Introduction), Chapter 2 (Literature Review), Chapter 3 (Methodology), Chapter 4 (Result), Chapter 5 (Discussion) and Chapter 6 (Summary and Conclusion). The project was given on the early of March 2013.

1.6 Project schedule

(In Gantt chart) on Appendix A

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Vehicles on the road or going through a tunnel will experience air flow that passes through the body of the vehicles. Type of air flow might be laminar or turbulent and considered to be quite dangerous for smaller vehicles users such as motorists. An idea was generated based on the sustainability energy in re-using the air flow that passes the vehicles because the speed of the air flow (based on the normal vehicles speed in highways which are 80km/h, 100km/h and 120km/h) can be used for generating electric via wind turbine attached at the road sides and tunnel. Besides that, it was found that many researches has been made regarding the use of wind as an alternative of nuclear or any other dangerous electric generating method especially at countries that has massive air flow and wind cross. A wind farm is a locally-clustered group of wind turbines at the same location used to produce electric power with the goal of zero carbon footprints. A large wind farm may consist of several hundred individual wind turbines, and cover an extended area of hundreds of square kilometers. There are many advantages to this commercial structure. In most countries, areas with commercially viable wind speeds are restricted to certain regions. This creates the necessity of concentrating as many wind turbines as possible in these regions to take advantage of the geographically-limited rich wind resources. The spatial concentration of several wind turbines also offers considerable advantages to the maintenance aspects of the operation since it is more cost-effective to maintain a large number of wind turbines in close proximity to each other [2]. This study aims to set out a description of the air flow profile evaluation around moving vehicle.The reason why we want to study this is relates to the safety of smaller vehicle (e.g. motorcycle users) when a bigger one (buses) passing through or overtake them. This approach is adopted so that the result of the research will clarify the basic air flow mechanisms that exist around vehicles and future steps to be taken in the design part to lessen the disturbance due to the air flow mechanism. The tools used in this study are SolidWorks software (for developing the 3D model of selected vehicles) and ANSYS software (for CFD simulation) while considering the aerodynamics theories. Computational Fluid Dynamics (CFD) methods are safe and economic for experimental methods and play an increasingly important role in simulating the air flow profile because a real experiment which is dangerous can be avoided. Besides that, the air flow profile, the effects and instability of the vehicle due to the air flow can be detected by using ANSYS.

2.2 Installing Wind Turbine at the Road Ways and Tunnel

Based on the air flow profile, an analysis will be done whether displaced air can able to rotate the air wind. The utilization of wind to generate power provides an alternative and renewable energy source compared to current fossil fuels based power generation. The world's fossil fuel energy is finite and is depleting at a faster rate. Moreover, the fossil fuel is directly related to air pollution, land and water degradation. Despite significant progresses have been made in power generation using large scale wind turbines recently, domestic scale wind turbines especially vertical scale wind turbines have been received less attention which have immense potentials for standalone power generation. Snaebjornsson et. al reported that increased concern for environment has led to the search for more environment friendly sources of energy.

Research about installing wind turbine along highways has been done so that it can generate electricity from moving vehicles [4]. In this paper, the author reported that as the automobiles moves from highways/expressways, there is a creation of pressure column on both the sides of the road. This pressure column is created due to imbalance of high pressure/low pressure energy band created by the automobiles. Due to this pressure band wind flow and create pressure thrust. This wind Pressure thrust depends upon the:-

a) The intensity of the traffic.
b) The size of the automobile.
c) The speed of the automobiles.

This Pressure thrust of wind energy can be converted into mechanical energy with the help of small turbines placing them just nearby these highways sides and center. Figure 2.1 shows an example of installing wind turbines at highways.

Figure 2.1 Example of installing wind turbine at highways

2.3 Air Profile around Moving Vehicles

When a vehicle moves in a certain velocity along an open road or through a tunnel, it will displace air around it causing an air flow profile creation along the vehicle. The air profile is always evaluated for the risk factor of causing accidents and also the potential of generating electricity. Several wind- and weather-related accidents of road vehicles occur every year, especially at exposed locations where topographical features magnify the wind effects as they researched about the parameters influencing wind-related accidents of road vehicles. The most notable ones involve high-sided vehicles. The theory presented is applied to a multitude of scenarios to explore the interrelation between the various basic variables and how they affect the probability of accident given in terms of the so-called accident index [3].The analysis demonstrates that wind-related accidents are the consequence of a combination of several basic variables as represented by the accident index.

The knowledge gained from the experimental or numerical approach is mainly on a single ground vehicle/scale-down model vehicle either running on road or in a testing facility (e.g. wind tunnel). On the other hand, the automobile population is ever increasing in most urban cities [5]. It leads to the growing traffic density with a shortened interval distance (i.e., vehicle spacing, safety distance etc.) between the consecutive running vehicles on the traffic lanes, and the application of modern technologies such as intelligent transport systems will further reduce the traffic/vehicle spacing [6]. It is clearly known that the moving ground vehicle may alter the incoming flow structures for the following vehicles. For over a century, the aerodynamic performance of road vehicles has mainly been assessed on the basis of their drag coefficient C_d value, sometimes in conjunction with their lift coefficient C_l value, which was obtained through steady-state wind tunnel measurements. However, these mean quantities do not adequately reflect vehicle performance with respect to unsteady aspects that are commonly encountered in real-life situations. There are also an increasing number of computational studies that had been carried out on the unsteady aerodynamic effects on road vehicles. Unlike the steady-state solution that can rely on the Reynolds Averaged Navier–Stokes (RANS) method, the application computational fluid dynamics in solving unsteady aerodynamics problems required the use of Large Eddy Simulation (LES) technique [7]. In this case study, ANSYS will replace the usage of LES. In a research states that in fundamental terms, the nature of the flow field is that the flow fields that exist are the primary cause of the aerodynamic forces on the train and its components which result in a whole range of aerodynamic issues. A simulated driver and vehicle are subject to two time-dependent crosswinds, one representative of a windy day and the second an extreme crosswind gust. Initially a quasi-static response is considered and then 5 additional sources of aerodynamic unsteadiness, based on experimental results, are added to the model [2]. From the simulated vehicle and driver, the responses are used to produce results based on lateral deviation, driver steering inputs and also to create a ‘subjective’ handling rating. These results show that the largest effects are due to the relatively low frequency, time-dependent wind inputs. The additional sources of simulated unsteadiness have much smaller effect on the overall system and would be experienced as increased wind noise and reduced refinement rather than a worsening of the vehicle's handling [8].

The two-way coupling of full-scale vehicle motion and the surrounding turbulence motion research has been realized on a developed unsteady aerodynamic simulator for a road vehicle. The simulation code is implemented on a massively parallel processor to meet the demands of the large-scale, long-term aerodynamic simulations of full-scale road vehicles. As a typical application of the coupled analysis, an unsteady aerodynamics simulation of a simplified heavy-duty truck in windy conditions is demonstrated, and the effects of the unsteady aerodynamics on the truck's motion are investigated. The obtained results are compared with the results of a conventional quasi-steady analysis, and certain differences in the vehicle path and the yaw angle are identified. The effects of the transitional aerodynamics on variables related to driver's perception are significant. These results clearly indicate the importance of estimating the unsteady aerodynamic forces in a vehicle motion analysis [9].

The design of the vehicle also determines the air flow characteristics. Inter-vehicle spacing on highways is considered and an analysis of spacing is presented, deduced from data from an instrumented highway. Vehicle drag reductions arising from close spacing are discussed and drag and lift data from wind-tunnel tests on two co-linear Ahmed bodies. For greater spacing, the drag of the rear body fell below the value of the isolated case, up to the maximum spacing considered. The lift coefficient of the rear body was also found to be very sensitive to spacing. It was concluded that the effect of the strong vortex system arising from the slant back was the cause of the drag and lift changes of the rear vehicle [10]. Figure 2.2 shows vehicles that are moving a highway in a certain speed which can produce enough wind speed distribution to rotate the wind turbine.

Figure 2.2 Vehicles on Road [11]

2.2.1 Reasons the Vehicles Chosen

Based on a statistic [12], the statistic of vehicles as at December 31, 2011, there were 21,401,269 vehicles registered with the Malaysian Road Transport Department (RTD) (Malay: Jabatan Pengangkutan Jalan Malaysia - JPJ). These include motorcycles, motorcars, buses, taxis, rental vehicles, goods vehicles, excavators, and other vehicles for which registration is required. Figure 2.3 will show the statistics of vehicles based on each states in Malaysia.

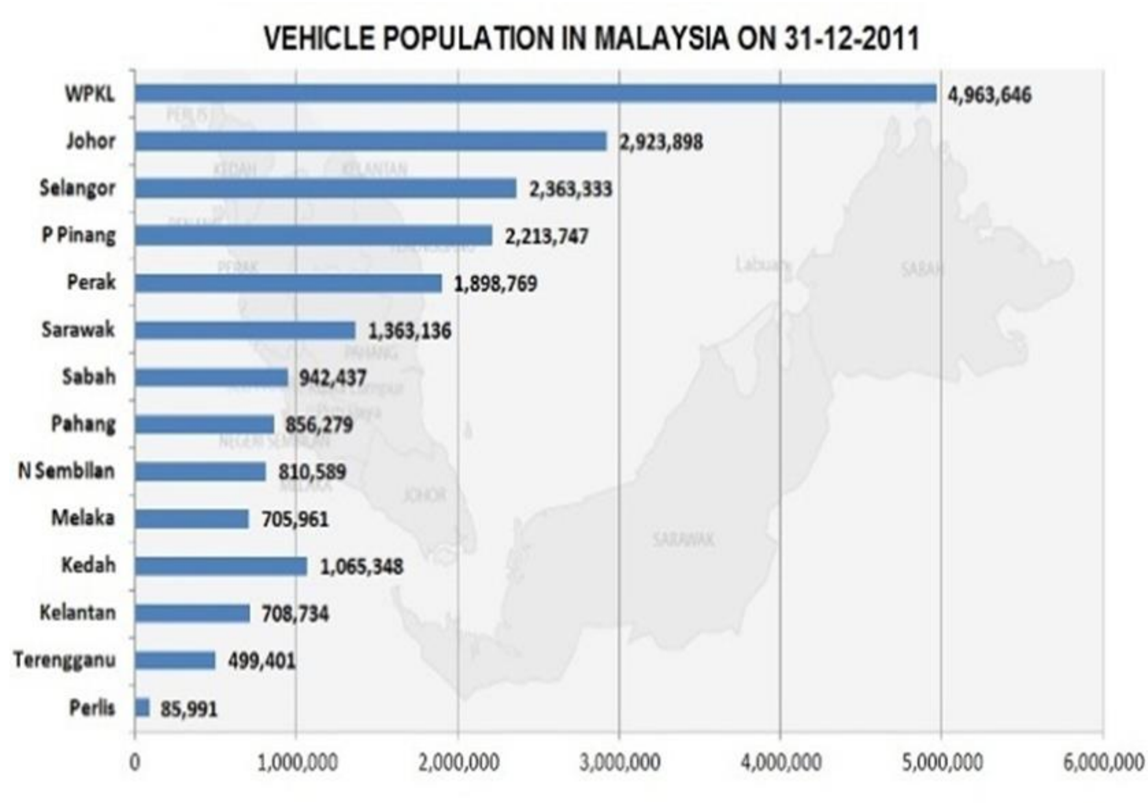

Figure 2.3 Statistics of vehicles based on each states in Malaysia [12]

Due to the statistic of vehicles in Malaysia, it can be said that there will be a lot of potential air flow displacement in the roads and also tunnel. This indicates that a good air flow profiling simulation can be done based on Malaysia's statistic of vehicles. High sided vehicles (bus), semi high sided vehicles (MPV) and normal height vehicle (sedan car) are chosen due to velocity that can reach up to 120km/h and they are the common vehicles on the roads.

2.2.2 Malaysian Climate and Normal Wind Velocity

Geographically, Malaysia is almost as diverse as its culture. 11 states and 2 federal territories (Kuala Lumpur and Putrajaya) form Peninsular Malaysia which is separated by the South China Sea from East Malaysia which includes the 2 states (Sabah and Sarawak on the island of Borneo) and a third federal territory, the island of Labuan. This shows that Malaysia will experience an Equatorial climate [13]. The highest daily wind speed in Malaysia (recorded at Mersing, Johor) is 3.8m/s [14].

2.2.3 Reynolds Number

The Reynolds number (Re) is the single most important non-dimensional number in fluid dynamics and is recommended to be calculated before beginning any new CFD modeling project. The Reynolds Number as formula shown in Figure 2.4 is defined as the dimensionless ratio of the inertial forces to viscous forces and quantifies their relevance for the prescribed flow condition:

$$Re = \frac{\rho U_\infty L}{\mu}$$

Figure 2.4 Formula to obtain Reynolds number (Re)

Where U_∞ and c are the characteristic velocity and length scale of the problem, ρ is the fluid density and μ is the dynamic viscosity. The use of the Reynolds number frequently arises when performing a dimensional analysis and is known as Reynolds principle of similarity. The Re also allows characterizing whether a flow is laminar or turbulent. Laminar flow is characterized by lower Re and high diffusion over convection. Turbulent flow on the other hand is characterized by higher Re where inertial forces dominate considerably, resulting in largely chaotic flow. The flow may also undergo a transitioning phase whereby the flow exhibits neither completely laminar nor completely turbulent characteristics. Inertial forces are proportional to the square of velocity, while viscous forces vary linearly with velocity. Therefore as the Re $\rightarrow 0$ it is reasonable to neglect convective terms (e.g. creeping flow). Similarly, as the Re $\rightarrow \infty$ the viscous terms of the momentum equation can be neglected and the problem can be characterized purely by the generation of inertial forces (e.g. high supersonic and hypersonic flows) [15]. Experiments were done in order to help in defining the influences of Reynolds number, after-body length, angle of wind incidence and approaching flow turbulence on the aerodynamics of rectangular prisms. The influence of the edge treatment of the rectangular prisms, i.e. square edges versus small or large chamfer, was also studied at high Reynolds numbers [16].

2.3 Integration of Solidworks and ANSYS

SolidWorks is a low-cost competitor to CAD programs such as PRO/ENGINEER, Unigraphics and Autocad. Solidworks has intuitive 3D design, built in intelligence and very low cost in owning. Assisting in this model developing, Solidworks creates 2D and 3D drawings faster and is has been so accurate in developing models [17]. ANSYS is a general purpose software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers.

ANSYS is one of CAE software. ANSYS, which enables to simulate tests or working conditions as shown in Figure 2.5, enables to test in virtual environment before manufacturing prototypes of products.

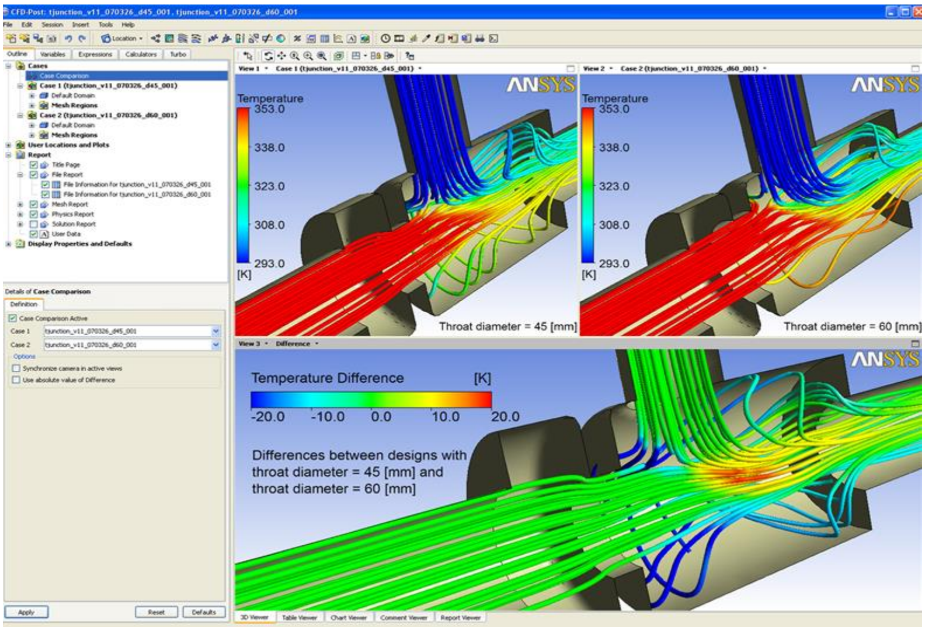

Figure 2.5 ANSYS result for flow simulation

Furthermore, ANSYS software with its modular structure is used for determining and improving weak points, computing life and foreseeing probable problems are possible by 3D simulations in virtual environment. Figure 2.6 below gives an opportunity for taking only needed features. ANSYS can work integrated with other used engineering software on desktop by adding CAD and FEA connection modules [18].

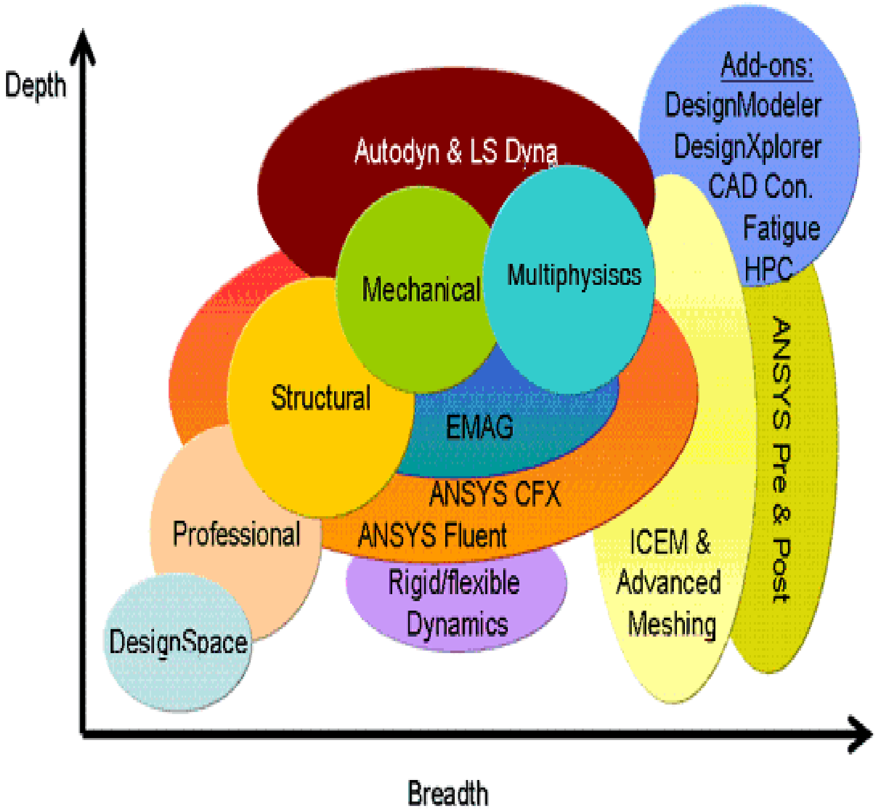

Figure 2.6 The integration of ANSYS with most engineering software

Cost-effective Computational Fluid Dynamics (CFD) methods with sufficient accuracy are complementary to experimental methods and play an increasingly important role in simulating maneuvers conditions. Examples are conditions that cannot be simulated in a wind tunnel or are too dangerous to be performed in flight tests. Stability and control characteristics can be obtained for such conditions using CFD. Prior to the usage of the computed stability and control characteristics the CFD methods should be well validated and evaluated against state-of-the-art wind tunnel and/or flight test data. The proper selection of the level of geometrical detail to be used in a CFD simulation has a large impact on the cost effectiveness. Including more geometrical detail gives rise to a more complex and thus more expensive grid generation task. Incorporating more geometrical detail will also result in grids with more cells. Since the computing time and hence the computational cost depend directly on the number of grid cells, incorporating more geometrical detail will result in more costly simulations [19]. Based on the necessity and to lessen the cost of production, CFD software such as ANSYS will be used to do the analysis part as it is cheap and affordable to be used.

2.4 Wind Turbine Types

Two major types of wind turbines exist based on their blade shape and operation. The first type is the horizontal axis wind turbine (HAWT). This type of wind turbine is the most common and can often be seen littered across the landscape in areas of relatively level terrain with predictable year round wind conditions. HAWTs sit atop a large tower and have a set of blades that rotate about an axis parallel to the flow direction. These wind turbines have been the main subject of wind turbine research for decades, mainly because they share common operation and dynamics with rotary aircraft. All the wind turbine types mentioned can be seen in Figure 2.7. The HAWT is the most popular and widely used type of wind turbine. Companies such as Vestas, Siemens, and GE develop and deploy HAWTs around the world, making them the largest and most successful wind turbine manufacturers. All major wind farms around the world employ many HAWTs working as a team to aide in the generation of electricity for small towns and large cities. The blades of a HAWT work to extract energy from the wind by generating lift, resulting in a net torque about the axis of rotation. As torque is produced through the slow turning of the wind turbine blades and the gearbox speeds up this rotation for the production of electricity through the use of a generator.

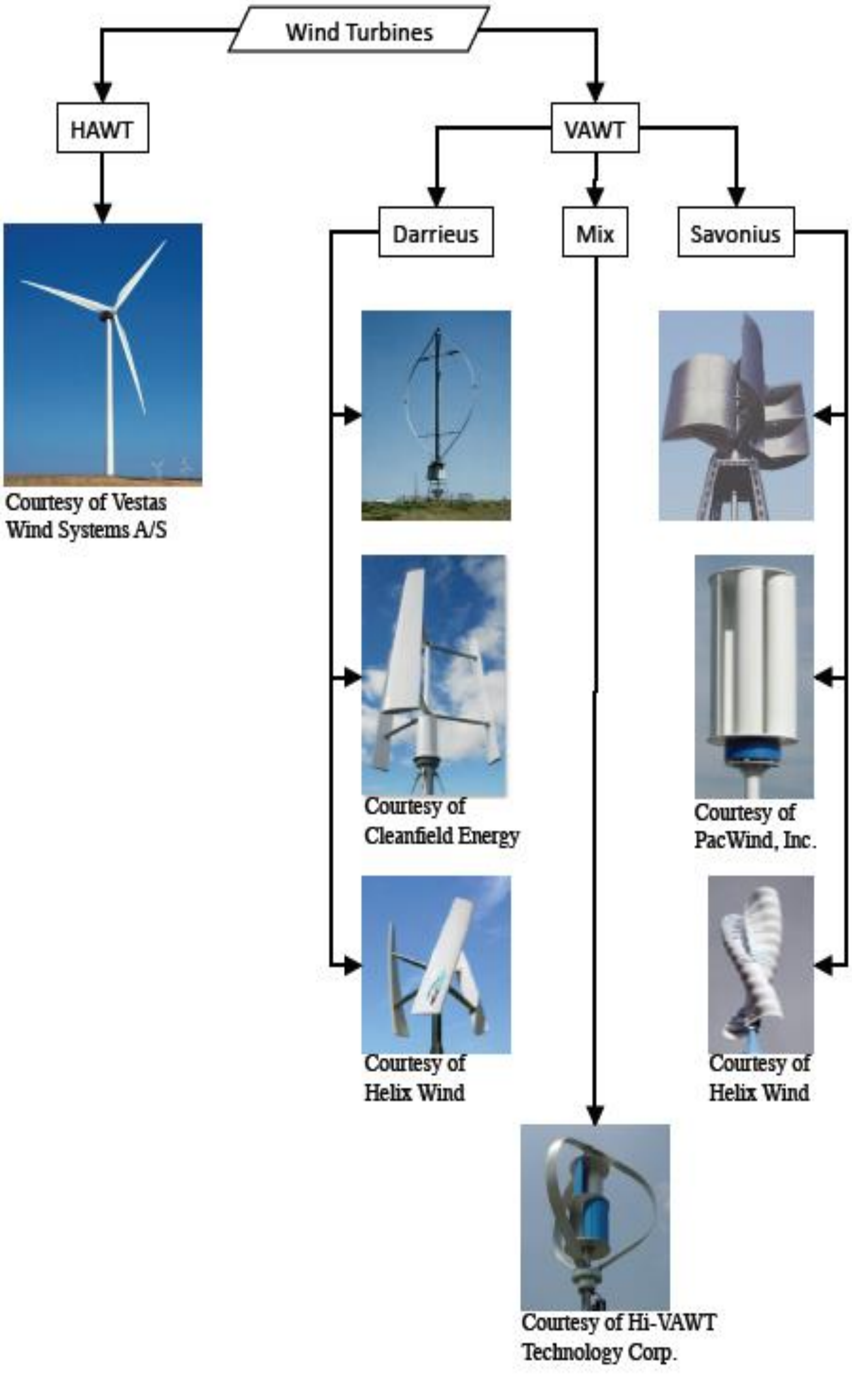

Figure 2.7 Types of wind turbine

To accomplish this task efficiently, especially for large HAWTs, active pitch controllers are used to ensure that each blade is adjusted to maintain an optimal angle of attack for maximum power extraction for a given wind speed. A yaw controller is also used to actively yaw the blades into the wind. However, these active control systems are complex and require more moving parts and effort to install than a

VAWT assembly where the only moving part is the rotor, and the majority of components are located at the base of the turbine [1].

The design and manufacturing of a HAWT blade is complex as the blade is tapered and twisted with varying cross-sections in order to achieve optimum aerodynamic performance. The change in the cross-section and twist of the blade from the root to the tip is due to the variation of the relative velocity component. Because the tip of the blade spins much faster than the root, the twist of the blade is shallow and the cross-section is thin. It can be seen due to the high relative velocity at the tip, the resultant force F acting on the blade section is extremely high, but only a small portion of that force Fd is driving the rotation. However, because of the blades design, the thin blade profile with a low angle of twist at the tip produces roughly the same amount of torque as the root of the blade due to its large moment arm. Because of the slower rotation speed at the root, the blade profile is much thicker with a higher angle of twist than at the tip. This design means a larger portion of the resultant force is in the direction of rotation. However, due to the lower moment arm at the root, the torque distribution over the entire blade is fairly uniform. A HAWT blade assembly and HAWT blade root and tip airfoil sections can be seen in Figure 2.8.

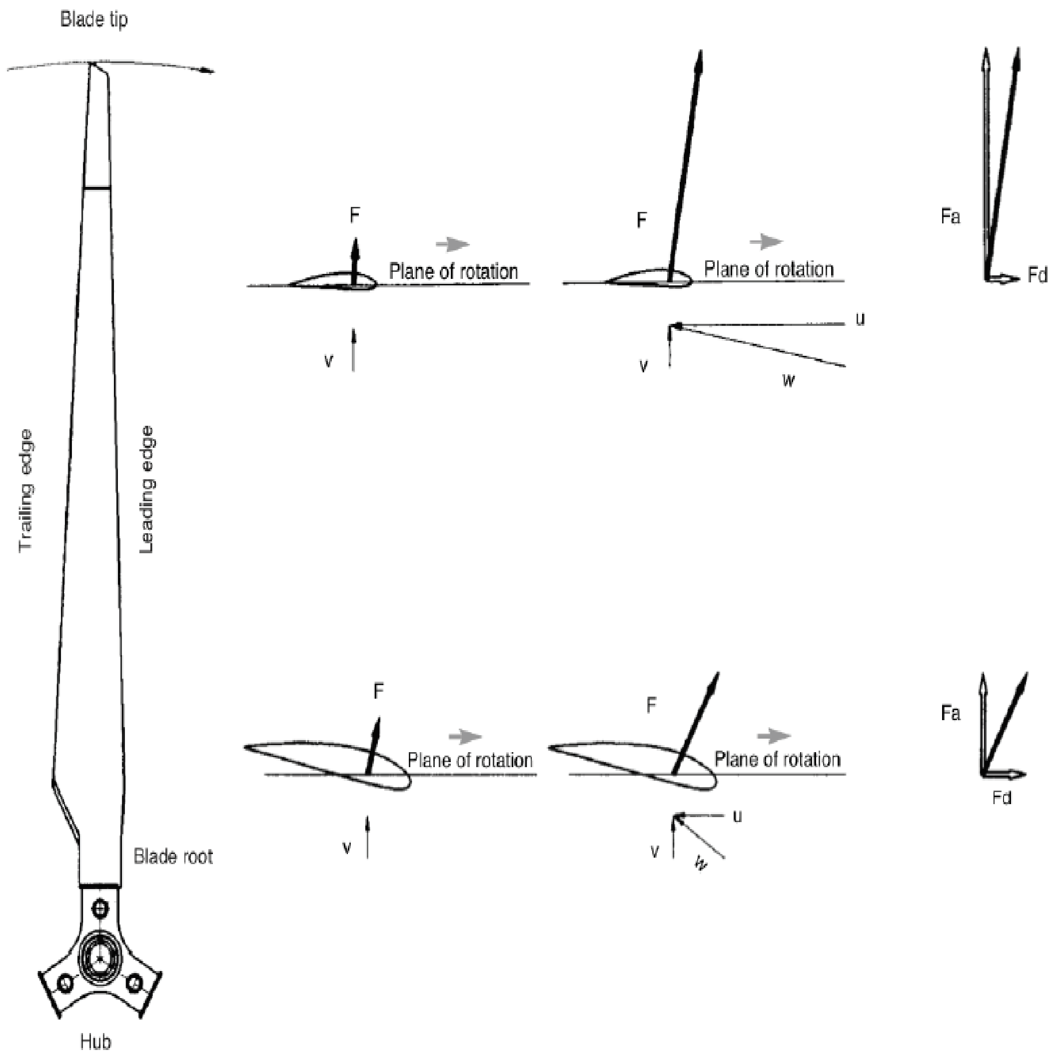

Figure 2.8 HAWT blade root and tip airfoil sections

The second major type of wind turbine is the vertical axis wind turbine (VAWT). This type of wind turbine rotates about an axis that is perpendicular to the oncoming flow; hence, it can take wind from any direction. VAWTs consist of two major types, the Darrieus rotor and Savonius rotor. The Darrieus wind turbine is a VAWT that rotates around a central axis due to the lift produced by the rotating airfoils, whereas a Savonius rotor rotates due to the drag created by its blades. There is also a new type of VAWT emerging in the wind power industry which is a mixture between the Darrieus and Savonius designs. Recently, VAWTs have been gaining popularity due to interest in personal green energy solutions. VAWTs target individual homes, farms, or small residential areas as a way of providing local and personal wind energy. This reduces the target individual's dependence on external energy resources and opens up a whole new market in alternative energy technology. Because VAWTs are small, quiet, easy to install, can take wind from any direction, and operate efficiently in turbulent wind conditions, a new area in wind turbine research has opened up to meet the demands of individuals willing to take control and invest in small wind energy technology [20].

The device itself is relatively simple. With the major moving component being the rotor, the more complex parts like the gearbox and generator are located at the base of the wind turbine. This makes installing a VAWT a painless undertaking and can be accomplished quickly. Manufacturing a VAWT is much simpler than a HAWT due to the constant cross section blades. Because of the VAWTs simple manufacturing process and installation, they are perfectly suited for residential applications. The VAWT rotor, comprised of a number of constant cross-section blades, is designed to achieve good aerodynamic qualities at various angles of attack. Unlike the HAWT where the blades exert a constant torque about the shaft as they rotate, a VAWT rotates perpendicular to the flow, causing the blades to produce an oscillation in the torque about the axis of rotation. This is due to the fact that the local angle of attack for each blade is a function of its azimuthal location. Because each blade has a different angle of attack at any point in time, the average torque is typically sought as the objective function. Even though the HAWT blades must be designed with varying cross-sections and twist, they only have to operate at a single angle of attack throughout an entire rotation. However, VAWT blades are designed such that they exhibit good aerodynamic performance throughout an entire rotation at the various angles of attack they experience leading to high time averaged torque. The blades of a Darrieus VAWT (D-VAWT) accomplish this through the generation of lift, while the Savonius type VAWTs (S-VAWT) produce torque through drag. A snapshot of the velocity and force components for a Darrieus type VAWT blade can be seen in Figure 2.9.

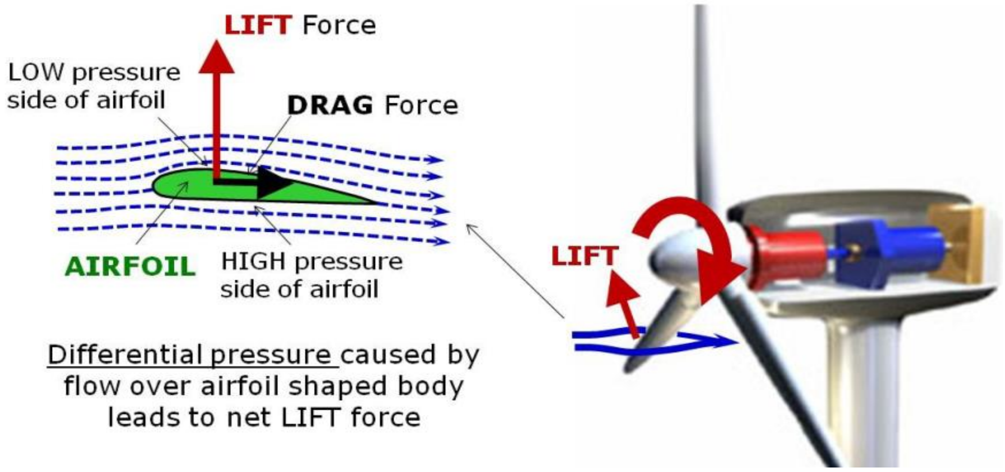

Figure 2.9 Velocity and force components for a Darrieus type VAWT.

As the blade rotates, the local angle of attack for that blade changes due to the variation of the relative velocity. The induced velocity and the rotating velocity of the blade govern the orientation and magnitude of the relative velocity. This in turn changes the lift L and the drag D forces acting on the blade. As the lift and drag change their magnitude and orientation, the resultant force FR changes. The resultant force can be decomposed into both a normal component FN and a tangential component FT . It is this tangential force component that drives the rotation of the wind turbine and produces the torque necessary to generate electricity. Due to the mechanism driving the rotation of a D-VAWT, it is possible that the blades can travel faster than the speed of the free stream velocity, or in other words, the tip speed ratio at which these wind turbines operate is often higher, leading to higher efficiency [20].

A S-VAWT generates electricity through drag rather than lift like the D-VAWT. A cross-section of a Savonius wind turbine can be seen in Figure 2.10**.** As the wind hits the concave portion of the blade (the bucket), it becomes trapped and pushes the blade around, advancing the next bucket into position. This continues as long as the wind is blowing and can overcome the friction of the shaft about which the blades rotate. A Savonius rotor typically rotates with a velocity equivalent to the speed of the freestream velocity. Because of its lower rotation speed, Savonius rotors are associated with lower e_ciencies and are not capable of providing adequate electricity, but rather serve as a device used to reduce the overall dependence on other energy resources. However, due to the Savonius wind turbines simplicity, it is extremely easy to construct; some have even been built using large plastic blue polydrums with the capability of providing up to 10% of a household's electricity [4].

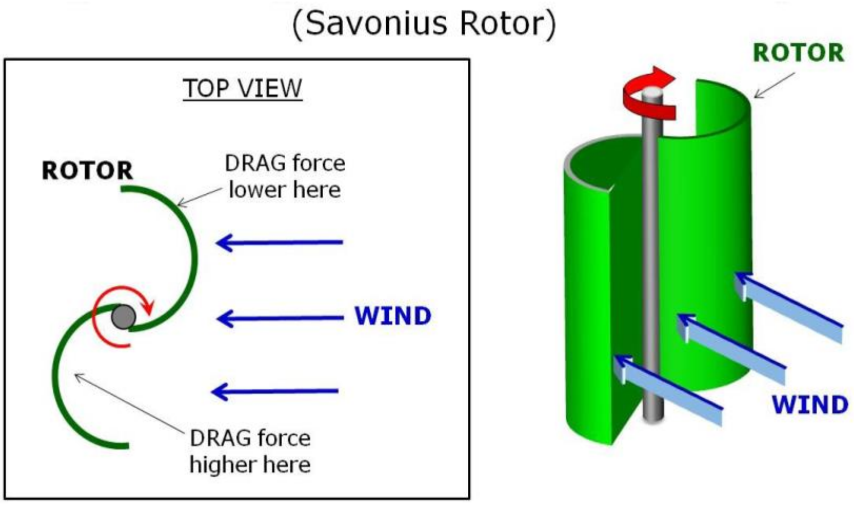

Figure 2.10 Velocity and force components for a Savonius type VAWT**.**

2.5 Computational Modeling

The majority of wind turbine research is focused on accurately predicting efficiency. Various computational models exist, each with their own strengths and weaknesses that attempt to accurately predict the performance of a wind turbine. Being able to numerically predict wind turbine performance replacing the classic experimental technique, the major benefit is computational studies is more economical than costly experiments.

A survey of aerodynamic models used for the prediction of VAWT performance was conducted [21]**.** The first major method used for predicting VAWT performance is the momentum model. The momentum model assumes the rotor is an actuator disk across which the forces acting on the blades of the wind turbine are equal to the change in momentum of the air. In the multiple stream tube models, several stream tubes were used that stretched across the blade rotor [22]. In each stream tube the induced velocity was assumed to be constant. Various other momentum models have been introduced using curvilinear streams rather than rectilinear flow approaching the wind turbine and improving the prediction of local blade Reynolds number when determining the aerodynamic forces acting on the blade [23]. While these methods are relatively simple to understand and implement, they tend to break down for high tip speed ratios and high solidity where viscous effects become dominant.

Vortex models are the second major method used for predicting VAWT performance and operate under the assumption of incompressible, potential flow. The blades are replaced by a moving bound vortex. As the blades spin, the strength of the bound vortex varies as blade forces change given by the Kutta-Joukowski theorem. This can be attributed to the change in relative wind and angle of attack the blades experience as they rotate. In order to conserve total circulation, a vortex sheet is shed into the wake. At any point in the flow, the induced velocity due to a vortex sheet can be found using the Biot-Savart law. With the free stream velocity and induced velocity known, the relative velocity can be determined for each of the blades. Once the relative velocity for the blades is known at all azimuthal locations, the lift and drag of the airfoil can be determined using empirical airfoil coefficient data tables. While the model operates under the assumption of potential flow, the use of experimental data allows for unsteady, viscous affects to be considered [24] .

The last method used for predicting VAWT performance is through the use of CFD where the entirety of the flow is calculated by numerically solving the Navier-Stokes equations. The Navier-Stokes equations are a set of non-linear, coupled, partial differential equations for which an exact solution still does not exist. For the simulation of a wind turbine, the equations include the continuity equation and momentum equations. Turbulence is considered using the Reynolds-averaged Navier-Stokes equations where the introduction of a new term representing the turbulent stress gives rise to a number of turbulence models. Because of its flexibility, CFD has been gaining popularity for analyzing the complex. CFD has shown no problems predicting the performance of turbines.

REFERENCES

[1] M. S.V Saravanan, "Design and Fabrication of Vertical Axis Highway Windmill, " *ISSN (Print) : 2278-8948,* p. 31, 2013.

[2] C. Baker, "The Flow around high Speed Trains," *Journal of Wind Engineering and Industrial Aerodynamics,* pp. 277-298, 2010.

[3] J. Snæbjörnsson, C. Baker and R. Sigbjörnsson, "Probabilistic assessment of road vehicle safety in windy environments, " *Journal of Wind Engineering and Industrial Aerodynamics, Volume 95, Issues 9–11,* pp. 1445-1462, 2007.

[4] M. K. Sharma, "Assesment Of Wind Energy Potential From Highways, " *International Journal of Engineering Research & Technology (IJERT),* 2012.

[5] J. Huang, T. Chan and Y.Zhou, "Three Dimensional flow structure measurements behind a queue of studied model vehicles, " *International Journal of Heat and Fluid Fow,* pp. 647-657, 2009.

[6] E. Deets, "Wind driven generator for powered vehicles". US Patent 7135786 B1, 2006.

[7] S. Cheng, M.Tsubokur, T.Nakashima, T.Nouzama and Y.Okada, "A numerical analysis of transient flow past road vehciles subjected to pitching oscillation," *Journal of Wind Engineering and Industrial Aerodynamics,* pp. 511-522, 2011.

[8] J. Fuller, M. Best, N. Garret and M. Passmore, "The importance of unsteady aerodynamics to road vehicle dynamics, " *Journal of Wind Engineering and Industrial Aerodynamics,* pp. 1-10, 2013.

[9] T. Nakashima, M. Tsubokura, M. Vázquez, H. Owen and Y. Doi, "Coupled analysis of unsteady aerodynamics and vehicle motion of a road vehicle in windy conditions," *Computer Fluids,* 2012.

[10] S. Watkins and G. Vino, "The Effect of vehicle spacing on the aerodynamics of a representative car shape, " *International Journal of Heat and Fluid Flow,* pp. 647-657, 2009.

[11] B. Wang, S. Olson, P. Wolff and A. Wang, "Next big future," 24 January 2012. [Online]. Available: http://nextbigfuture.com/2012/01/car-platooning-successfully.html. [Accessed 27 May 2013].

[12] PROTO, "Motor Trader," 2012. [Online]. Available: http://www.motortrader.com.my/news/malaysia-s-vehicle-population/. [Accessed 27 May 2013].

[13] T. Malaysia, "Tourism Malaysia," 2013. [Online]. Available: http://www.tourism.gov.my/en/my/Web-Page/About-Malaysia. [Accessed 27 May 2013].

[14] M. M. Department, "Malaysian Meteorological Department," 2012. [Online]. Available: http://www.met.gov.my/index.php?option=com_content&task=view&id=30&Itemid=1090&lang=english. [Accessed 27 May 2013].

[15] L. S. Team, "Leading Engineering Application Providers," 20 March 2013. [Online]. Available: http://www.computationalfluiddynamics.com.au/reynolds-number-cfd/. [Accessed 27 May 2013].

[16] G. Larose and A. D'Auteuil, "Experiments on 2D rectangular prisms at high Reynolds numbers in a pressurised wind tunnel, " *Journal of Wind Engineering and Industrial Aerodynamics 96* , pp. 923-933, 2008.

[17] Solidworks, "http://www.solidworks.com," 2013. [Online]. Available: http://www.solidworks.com/sw/why-choose-solidworks.htm. [Accessed 2013].

[18] F. Engineering, "© 2003-2013 FIGES A.S.," 2013. [Online]. Available: http://www.figes.com.tr/english/ansys/ansys.php.

[19] O. Boelens, "CFD analysis of the flow around the X-31 aircraft at high angle of attack," *Aerospace Science and Technology,* pp. 38-51, 2012.

[20] M. M. A. Bhutta, N. Hayat, A. U. Farooq, Z. Ali, S. R. Jamil and Z. Hussain, "Vertical axis wind turbine – A review of various configurations and design techniques," *Renewable and Sustainable Energy Reviews,* p. 1926–1939, 2012.

[21] J.-O. Mo, A. Choudhry, M. Arjomandi, R. Kelso and Y.-H. Lee, "Effects of wind speed changes on wake instability of a wind turbine in a virtual wind tunnel using large eddy simulation, " *Journal of Wind Engineering and Industrial Aerodynamics* , pp. 38-56, 2006.

[22] D. T. a. A. F. M. Islam, "Aerodynamic models for darrieus-type straight-bladed vertical axis wind turbines, " *Renewable and Sustainable Energy Reviews,* pp. 1087-1109, 2008.

[23] I. Paraschivoiu, "Aerodynamic loads and performance of the darrieus rotor," *Journal of Energy.*

[24] H. B. a. T. B. S. McIntosh, "Aerodynamic modeling of swept bladed vertical axis wind turbines," *47th AIAA Aerospace Sciences Meeting Including The New Horizons Forum and Aerospace Exposition,* 2009.

[25] H. Larsen, "Summary of a vortex theory for the cyclogiro, " *Proceedings of the second US national conferences on wind engineering research.*

[26] M. Lant, "Best Practice Guidlines for Handling Automotive External Aerodynamics wit FLUENT," 2005.

[27] M. Z. I. A. Albani, "Preliminary Development Of Prototype Of Savonius Wind Turbine For Application In Low Wind Speed In Kuala Terengganu, Malaysia," *INTERNATIONAL JOURNAL OF SCIENTIFIC & TECHNOLOGY RESEARCH VOLUME 2, ISSUE 3,* 2013.

[28] S. Watkins, Vino and Gioacchino, "The effect of vehicle spacing on the aerodynamics of a representative car shape," *Journal of Wind Engineering and Industrial Aerodynamics,* pp. 1232-1239, 2008.

[29] W.Medjroubi, B.Stoevesandt, B.Carmo and J.Peinke, "High order numerical simulations of the flow around a heaving foil," *Computers & Fluid,* pp. 68-84, 2011.

[30] H. Kozmar, K. Butler and A. Kareem, "Transient cross wind aerodynamics loads on a generic vehcile due to bora gusts," *Journal of Wind Engineering and Industrial Aerodynamics,* pp. 73-84, 2012.

[31] I. ANSYS, "ANSYS," 2013. [Online]. Available: http://www.ansys.com/.

[32] F. Alama and S. Goldeb, "An Aerodynamic Study of a Micro Scale Vertical Axis Wind Turbine," *Procedia Engineering,* p. 568–572, 2013.

[33] D.-C. Tseng, "Device of wind electric power on transportation vehicles". US Patent 20030155464 A1, 2003.

[34] M. H. Ali, "Experimental Comparison Study for Savonius Wind Turbine of Two & Three Blades At Low Wind Speed," *International Journal of Modern Engineering Research (IJMER,* pp. 2978-2986, 20133.