Design Optimisation of Building mounted Wind Acceleration Aerofoil Wing powering Wind Turbines using CFD, an integrated Architectural-aerodynamic Approach.

Dr. Abdel Rahman Elbakheit
King Saud University
E-mail: a_rahman41@hotmail.com, Albakheit@ksu.edu.sa

Abstract

This paper represents some work aimed at providing a systematic approach in building design towards a successful integration of wind harnessing renewable energy technologies. The approach adopts the use of building forms and profiles to assist local wind patterns of the site in question to trigger a continuous stream of air that can be used to power a turbine of a suitable shape and size.

In this sense it takes into consideration the use of building masses and adjacencies to create such flows, as well as the general outline of the urban settings. This enables the Architectural design process of form finding and justification in terms of functionality and economic suitability a further step to achieve sustainability.

In this process computational fluid dynamics CFD provides the corner stone in evaluating, testing and optimising an envisaged design solution. The capability of the CFD code to predict the patterns of air movements with magnitude and direction with potential pressures on surfaces is the most valued privilege.

The study tackled the issue of wind separation in buildings and how to simulate it in CFD, as well as an investigation into the parameters of wing shapes utilizing wind separation in buildings to accelerate wind velocities for capturing by wind turbines.

The study concluded that CFD is a good tool for modelling, designing and optimizing of aerofoil shapes for wind energy utilization. The main parameters affecting the shape of wing profile for wind harnessing purposes would be: 1- the distance of the wing from the building surface.

2- The angle of attack of the wing.

This process could magnify natural wind up to 30% increase in low velocities (i.e., below 5m/s) and up to 150% in high velocities (i.e., above 5m/s).

1. Wind energy availability

Globally, wind energy has well-established patterns of flow as to the geographical locations and timing of the year. Although this would be a major stimulant for any particular location, yet still any site would be unique in terms of its potential to capture wind energy. This is particularly evident in urban terrains compared to rural areas. As the presence of building clusters, tall or short, trees and other obstructions would be detrimental for wind directions and velocities. However, unlike solar energy wind energy can be available all the time, if only relevant measures are made to reduce the effect of obstructions and be assisted with carefully sculpted designs.

The wind prevailing pattern of a particular site can be obtained in the form of wind-rose diagram as shown in figure 1, from relevant metrological registration office, showing information about the distribution of wind speeds as well as the frequency of varying wind directions along the year.

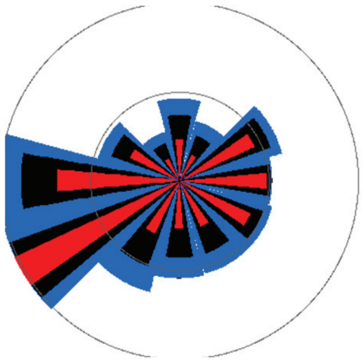

Figure 1: Wind rose diagram shows the wind distribution and likely frequency plotted as 12 sectors according to the European wind atlas1.

2. Building design Optimisation for potential wind energy collection

2.1. Background

Considerable effort and research have been conducted to establish wind flows over typical building cross sections through simulation and field-testing. It was found that at the top corner of a rectangular2 building facing wind direction or slopped3 roof sides subjected to wind; an acceleration effect of the wind would take place figure 2.

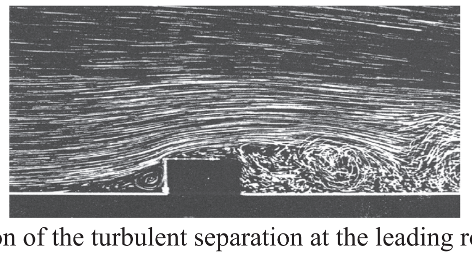

Figure2 2: Visualisation of the turbulent separation at the leading roof edge of a flat roof.

The total picture of the flow around a building subjected to wind stream, (i.e., figure 2) reveals that the flow would be separated at the top of the building with areas of large pressure in front of the building and for some distance over it. Areas of re-circulation occur at the top and back of the building as well as in the foot of the building from the front side.

The flow passing a tall building separates in many ways. Figure 3 shows separating streamlines in a vertical cross sectional plane in the mean flow direction being separated in different zones around the building. At the front, the flow divides upwards and to the sides. The flow around the building separates at each sharp edge and generates re-circulation.

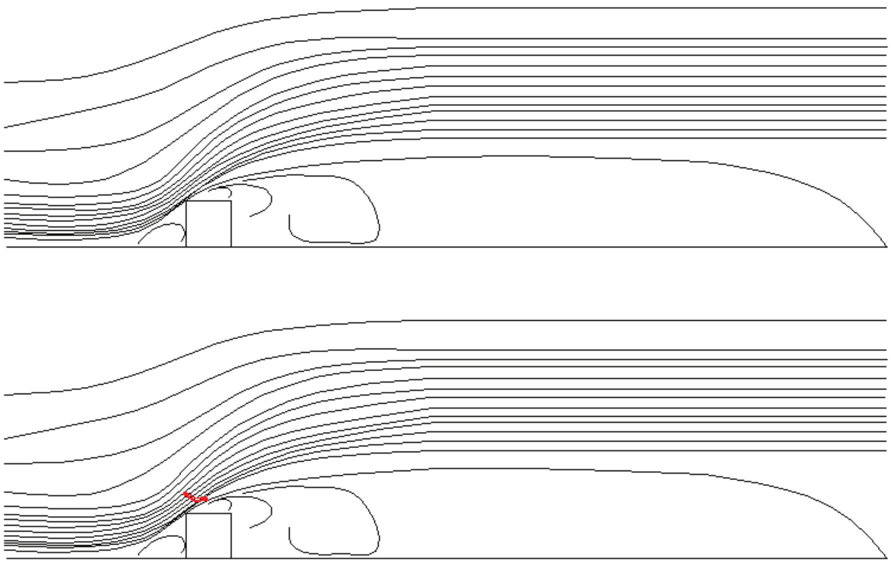

Figure 3: Flow schematic sketch, with areas of re-circulation in front, top and back of the building (left to right).

This phenomenon has been studied by many researchers trying to determine where exactly the limits of these circulations takes place compared to the geometrical parameters of the building and due to Reynolds stresses. (Schlichting, H. 19794, Hunt and Smith5 1969, etc6,7,8,9,10,11,12,13,..)

Of all the above-mentioned scenarios of wind separation along the edges of tall buildings only the acceleration effect at the very front of the building is of major importance in this study. Therefore buildings, which lie in the wake of tall buildings, would not take advantage of the acceleration effect due to the fact that they are under re-circulation flows (turbulence) resulting from other buildings, Figure 4. Unless they are tall enough to fall outside this effect, then they can be incorporated with this application within this study. That is simply because turbulent flow would create problems and lowers the efficiency of wind harnessing technologies (i.e., wind turbines).

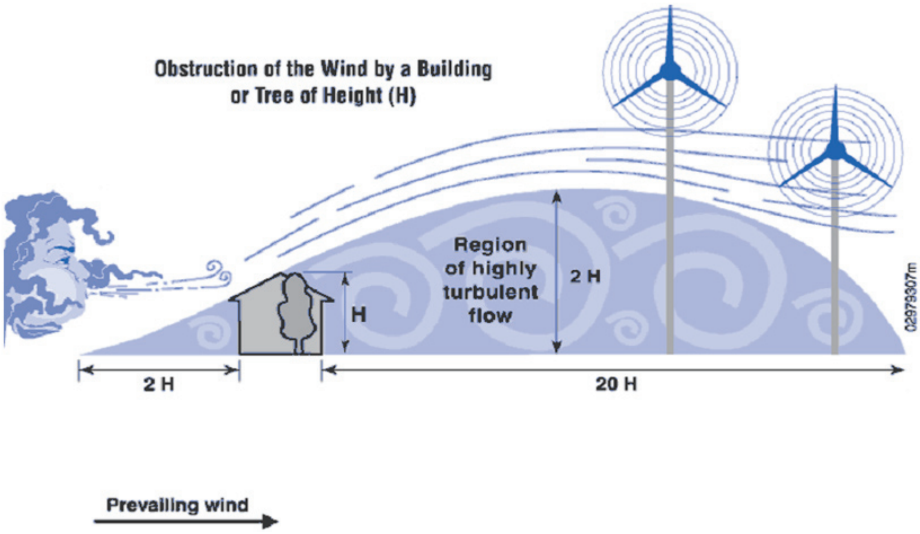

Figure14 4: Projected areas were re-circulation (turbulences) takes place.

In order to utilise this acceleration to power a building mounted wind turbine, the flow can be further enhanced through the provision of a wing shape profile at the start of the corner on the top of a rectangular building or at the top/start of the ridge of slopping roof.

The main parameters that influence the performance of such a shape would be:

1. The distance the wing is positioned away from the roof. (D) in Figure 5
2. The opening angle of the wing (angle of attack; angle (A) in Figure 5.

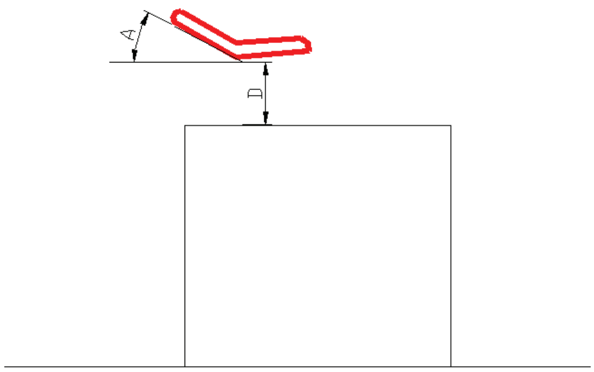

Figure 5: Main parameters affecting the performance of a wing shape on top of a building.

The effect of placing such a shape in the accelerated free stream would result in further increase of flow in the region where it is placed and on the top of it, hence providing an ideal location to place wind-harnessing technologies. The wing shape through the above stated parameters of its distance from the roof and the angle of opening would pose acceleration to air mass flow rate by concentrating large amount of air through a restriction. This shape should be of a suitable material and properly anchored to the building to withstand the likely high structural loads such as uplifting or drag forces. Figure 6 shows the likely shape of flow after installing the wing shape on top of the building.

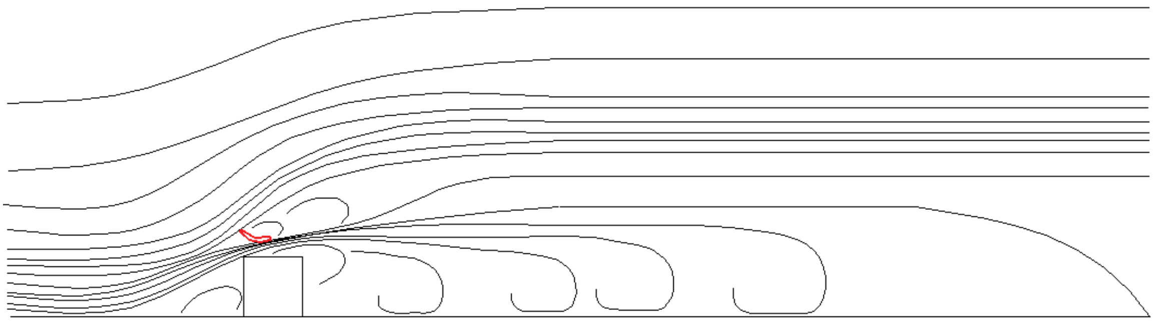

Figure 6: Flow separation after installing a wing shape on top of a building.

Figures 7 and 8 below show a 2D slice of a 3D CFD simulation (i.e., 3D CFDsimulations3 generally more accurate and reliable in depicting such flows around buildings).

When comparing figure 7 to figure 2 above we can see that the 3D CFD simulation provided a fairly close matching flow to that measured or theoretically established flows shown on figures 3 and 6.

Figure 8, below shows the acceleration effect after placing a wing shape to figure 7 flow, with the yellow velocity vectors of higher velocity magnitude (i.e., 25m/s) compared to figure 7, where the velocity was in the region of 17m/s in a free air stream on top of a rectangular building.

2.2. Underline simulation strategies

Since the 1980s, the flow of wind in urban terrain and around buildings has been intensively studied using CFD. According to Oliveira and Younis as a consequence of this; the blockage effect imposed by the building on an approaching wind flow is very important in 2D simulations, requiring a domain higher than 10H and an upstream developing length longer than 15H (i.e., were H is the height of the obstacle)

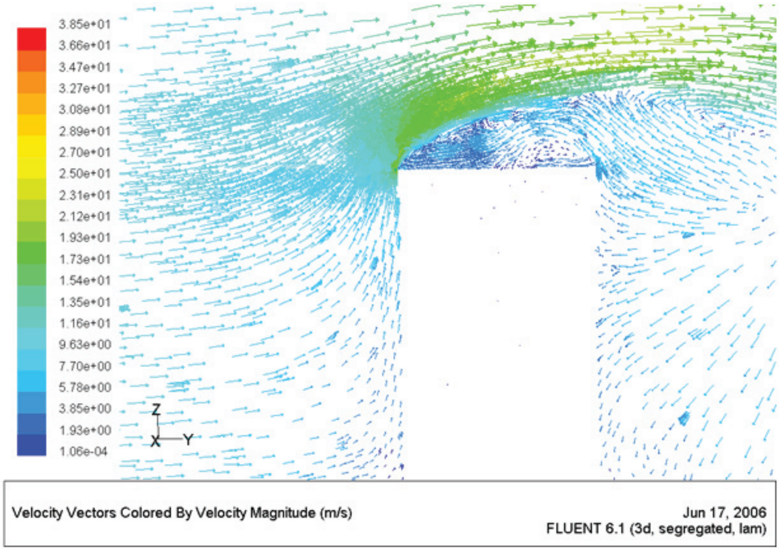

Figure 7: Velocity vectors depicting flow separation on top of a rectangular building.

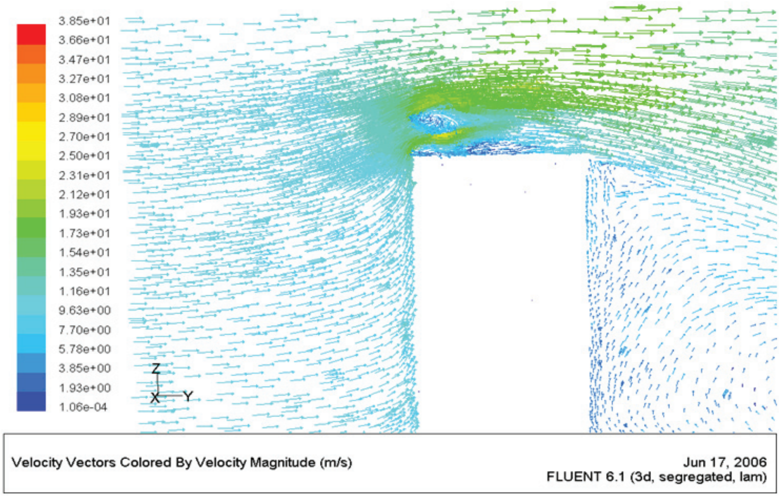

Figure 8: Velocity vectors after installing a wing shape on the same building in figure 7.

3.Analysis of wind turbine integration into Architectural Design.

3.1. Optimisation of wing shape position and Angle of attack.

3.2. Introduction

In an attempt to optimise building design for wind renewable energy a design of a single story detached house is developed figure 9. It has ground and first floors with a height of 7m on the low side of the pitch to 8m on the high end. The house was meant to integrate wind turbines and several other renewable energy technologies, (i.e., Photovoltaic). It is assumed that the site of the house is free from obstructions to wind flow. In practical terms if there are obstructions it would be solved by adding more height to the building as shown in figure 4 above.

3.3. Wind turbine integration

Within urban areas higher grounds or top of buildings would theoretically be more suited for uninterrupted and more persistent wind flow compared to lower level areas. In addition to this the flow would be most likely free from turbulence effects. Due to these reasons, the top of the roof was suggested as the suitable place where wind turbines will be integrated. The roof shape of the house is a mono-pitch slopping to one side (i.e., wind direction) with the lower side on top of a curved curtain wall façade integrating PV modules, figure 9. This shape would be of little resistance to air flow and is intended to maximize air mass flow rate at the opposite edge of the roof where the turbines are placed. A wing like profile is attached to the top end of the roof, and the turbines are placed underneath it. The approach taken here is to use vertical axis wind turbines which are compact, produce no noise and are also less risky, compared to horizontal axis wind turbines which are more subject to failure of turbine blades and produce much more noise at high wind velocity. Vertical axis wind turbines when aligned horizontally as in figure 9 and figure 10 can also be installed on top of the terraced or detached houses.

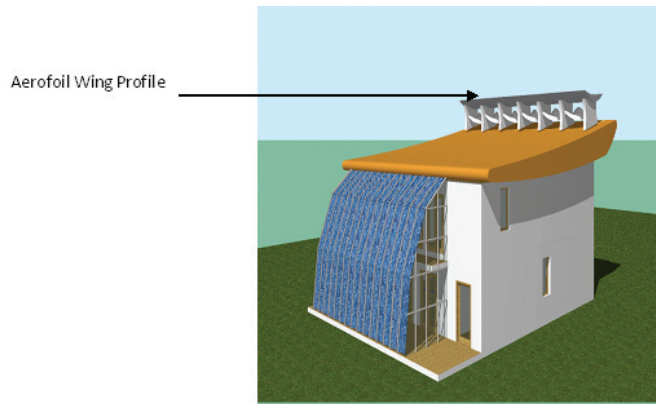

Figure 9: Typical wind turbine integrated in a prototype-detached house, Wing profile on high end of the roof.

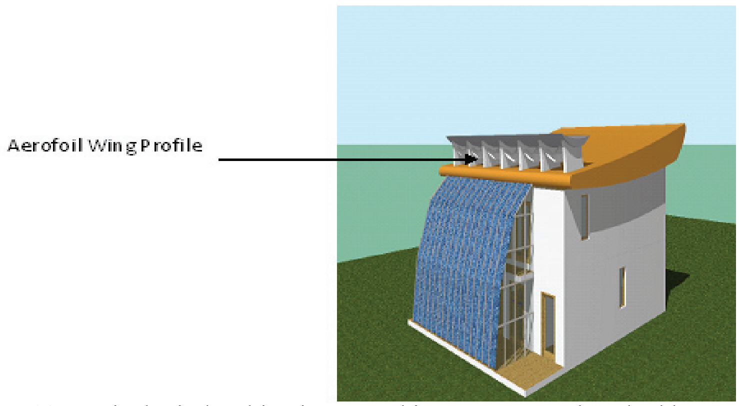

Figure 10: Typical wind turbine integrated in a prototype-detached house, Wing profile on low end of the roof.

3.3.1. Design optimisation for the height of the turbine slot 15

In order to investigate the effect of varying the proximity of the wing profile in relation to the roof and finding the optimum height that produce maximum wind speed for turbine operation. A number of wing proximities to the roof are studied. These range from 25cm to105cm.

The study15 is carried out using computational fluid dynamics CFD. A 2-D unstructured grid was chosen to simulate the wing on top of the house, with the above-mentioned range of distances from the rooftop, while keeping the building dimensions intact.

3.3.2. Effect of Domain size

As the main point of this investigation is to determine the potentially optimum distance from the roof top where the wing profile can be located, it is essential to employ a domain which can provide a plausible description to the flow of air in response to the change of the wing distance from the roof surface, Figure 11.

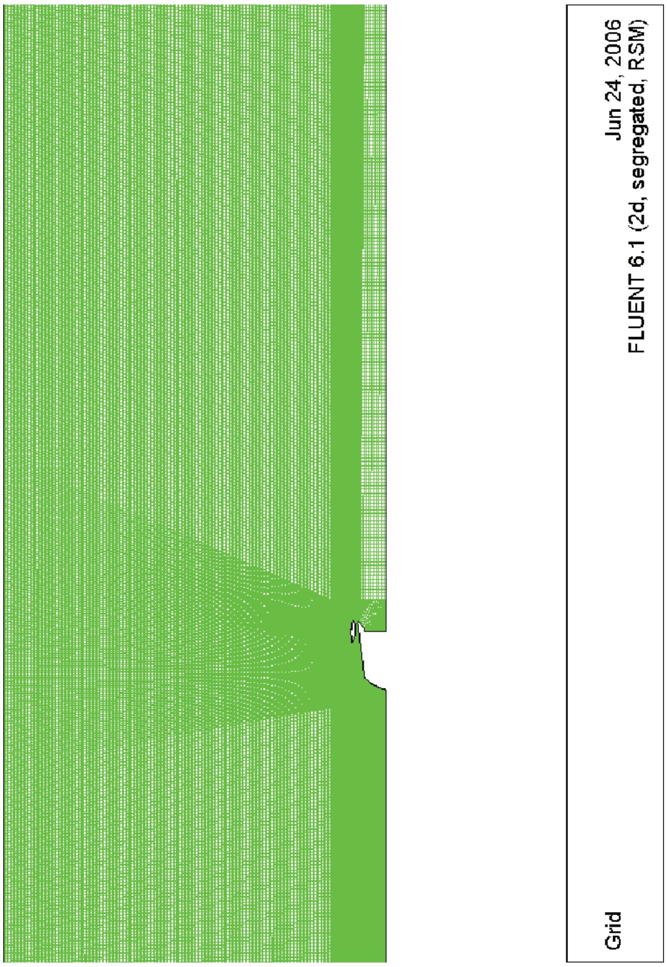

Figure 11: 2D mesh for the case of 85cm distance between wing and rooftop.

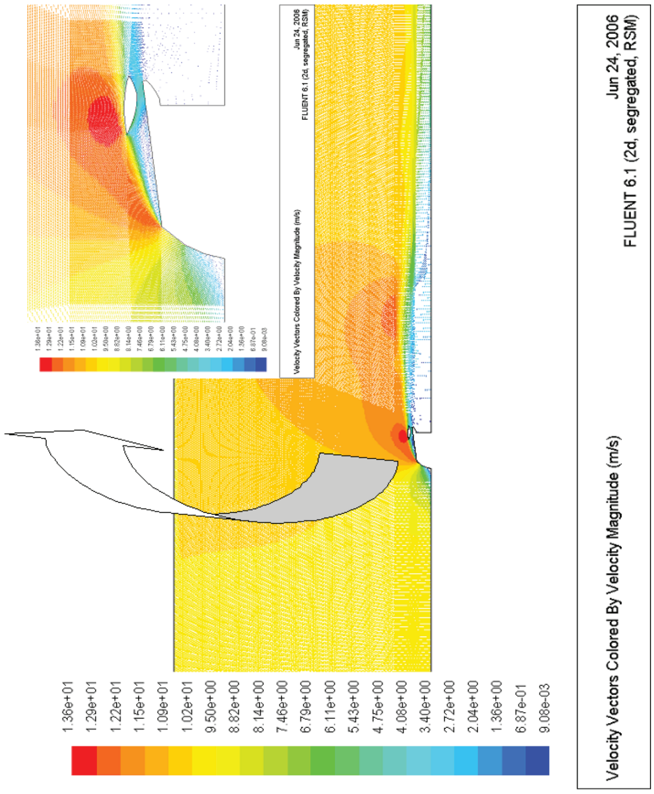

Figure 12: Resulting velocity vectors for the case of 85cm distance between wing and rooftop. (Large grid under Reynolds model of Turbulence).

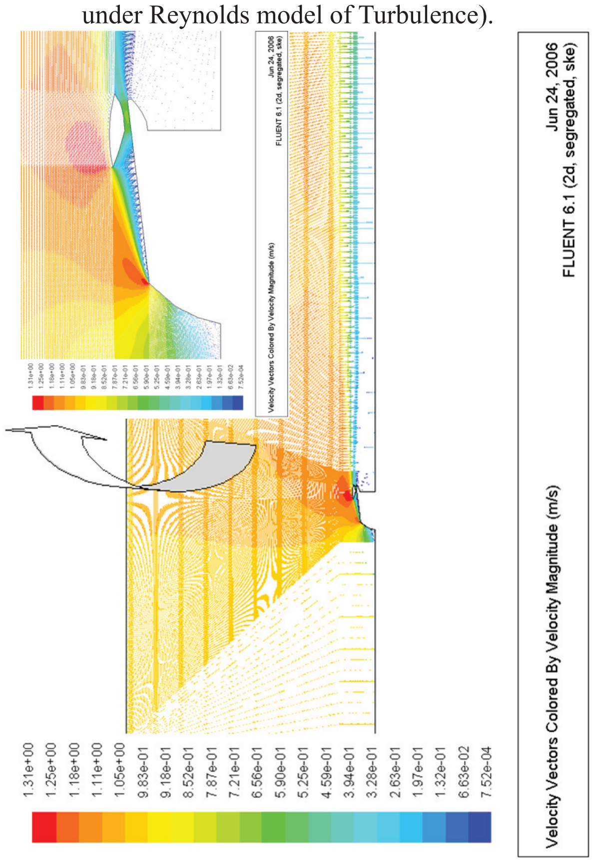

Figure 13: Resulting velocity vectors for the case of 85cm distance between wing and rooftop. (Large grid under K-ε of Turbulence).

3.3.3. The effect of different models of turbulence

The flow around the building and its wing is totally turbulent as the Reynolds number for this problem is 6x106 based on the characteristic length D, taken as the length of the wing (i.e., 6m long).

A comparison of the flow obtained by Reynolds stresses and the industry standard k-ε models is made for the grid, figures 12 and 13. The later model (i.e., k-ε), under estimate the area of recirculation at the lower part of the roof, while the former model, projects the recirculation in areas where the later model couldn't.

This result also provide evidence that the configuration of the building and roof form accelerate air at the eave and after it, much more than the anticipated position between the wing and the roof top. Therefore it might be better to relocate the wing profile on the lower part of the roof to boost the acceleration effect further in the region below the wing.

The convergence rate for these models are for k-ε model will converge at around 400 iterations and more, While Reynolds Stress model can converge up to 750 or more.

3.3.4. The effect of wing positions on top of building roof

This particular point is a result of examining the nature of the flow resulting from choosing the design configuration in question, presented by the prescribed model of turbulence (i.e., Reynolds Stress as explained in point 3.3.3 above). The flow seems to separate better at the eave and the acceleration effect is experienced there and beyond. So at this point, we can quantify how good is it to shift the wing from the back of the roof to the front figures 9 to 10 above. To do this, the same wing has been located at a distance of 85cm from the roof in two scenarios, one at the front and the other at the rear of the roof. The results obtained for the rear wing is shown in figure 12 above while the results of front wing are presented in figure 14 below. The two cases were identical in every aspect, as well as both of them are run under Reynolds stress model. So the only difference is the shift in the wing position from the top edge of the roof to the lower edge of it. The two cases were run under an assumed inlet speed of 10m/s from the left hand side of the domain.

The front wing position was (i.e., figure 14) the highest resulting velocities under the wing. i.e., 23.53 m/s. while the rear wing position gave a maximum speed of 4.75m/s only. That is almost ¼- times of that of the front position.

Also an important point here is that the flow has mainly accelerated under the wing in the whole domain as shown in figure 14. Whilst in figure 12 the flow did affected the domain more than it did for the under wing area.

3.3.5 The effect of coplanar wind Directions

A comparison is made between the cases where coplanar wind directions can be expected. For a 2-D case this normally means two lateral directions opposing each other. (i.e., windward becomes leeward and vice versa). In actual terms this can be learned from a wind rose diagram, therefore it would be anticipated that the main direction would be the one studied with the direction opposite as just a check to see how the design perform in a different scenario, which is normally the second best. Sometimes according to the site characteristics it may have a wind pattern of high velocity from a particular direction at a certain time of the year and its reverse (i.e., 180º) on a different one. So in this particular situation a 2-D simulation for the design shape would be a great advantage.

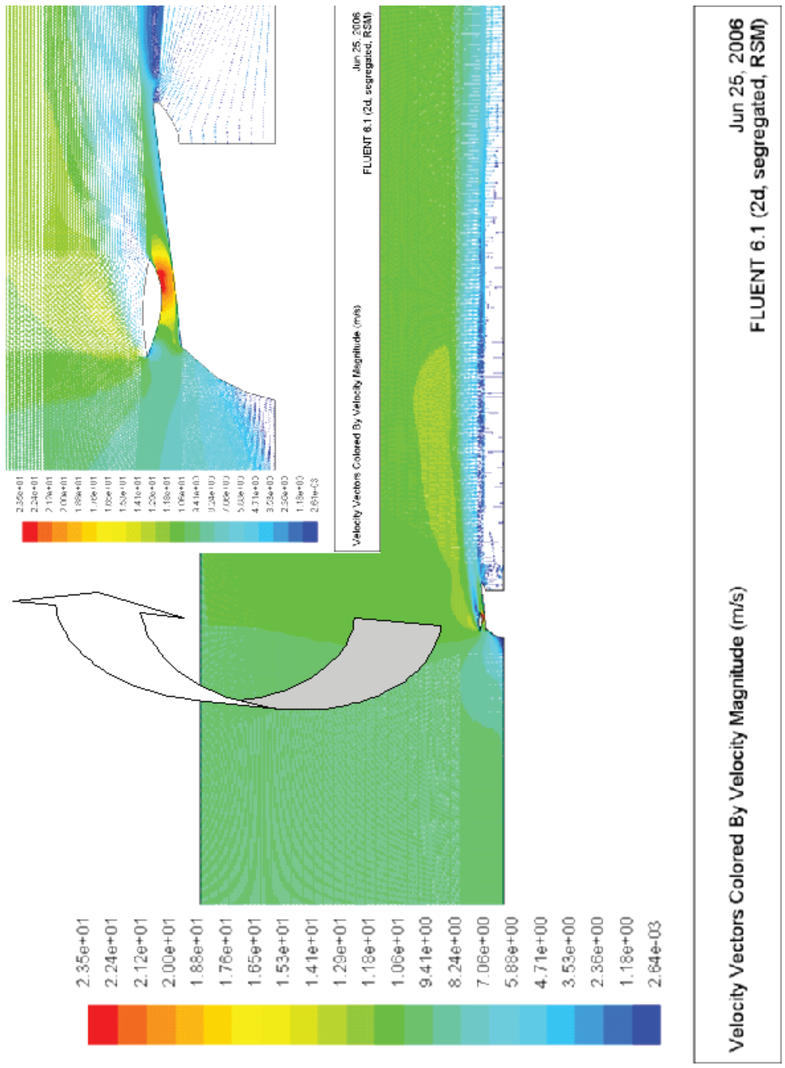

Figure 14: Velocity vectors for the wing shape positioned above the eave in the front of the house.

Here we can take the grid cases where the wing is placed at either the lower side or the upper side of the roof. Discussed earlier at 3.3.4, and having the wind subjected from the left side of the domain as the front wind and the opposite side as the back wind. Again the resulting velocity underneath the wing will be the major factor to determine the most suitable design configuration.

Table 1: Effect of the wind direction at the front edge of the building.

Direction of the wind	Resulting velocity below the wing.
Front wind	23.53
Back wind	3.5

Table 2: Effect of the wind direction at the rear edge of the building.

Direction of the wind	Resulting velocity below the wing.
Front wind	4.75
Back wind	14.3

From tables1 and 2 above, it is clear that the scenario where the wind coming from the front side and the wing on the lower position would generate the most of acceleration (i.e., 23.53m/s) within the studied scenarios. While the scenario where the wind coming from the back side would be more favourable for the upper wing position producing 14.3m/s.

However, based on the flow patterns generated by the two different positions of the wings, it seems that the lower position is more favourable. But the upper position can be better performing if the wind came from the back side. In this situation the choice of the location of the wing would be based on the average percentage of yearly wind direction coming from the most frequent wind side, as depicted by the wind rose diagram.

4. Summary of optimising the wing proximity to the roof of the house

This point summarizes the resulting optimisation for the proximity of the wing to the roof, the wing is located at a lower position on the roof as explained in point 3.3.5 above. Figure 15 below, sums the resulting velocities from the different proximities of the wing after applying a range of 10 consecutive velocities through the inlet of the domain. The resulting velocities tend to increase with the increase of the distance from the roof up to width of 85cm, and reduce after that. Maximum velocities resulting are within a region of 25m/s.When plotting the velocity profile along the different wing proximities to the roof as seen in figure 16, below. We find that most of the plotted resulting velocities do increase with the increase of gap distance from the roof, but up to a distance of 85cm and decreases gradually after that. Therefore the distance of 85cm would be the optimum proximity of the wing to generate maximum wind speeds.

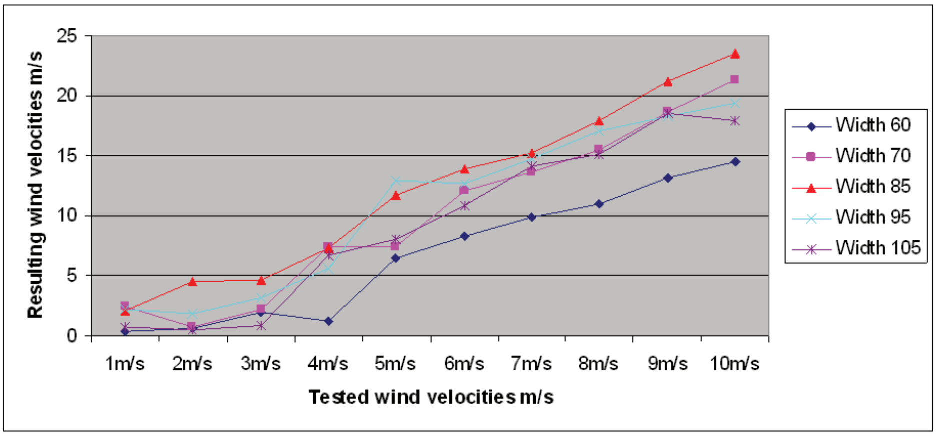

Figure 15: Summary of different proximities of the wing to the roof and its effect of velocity. generated.

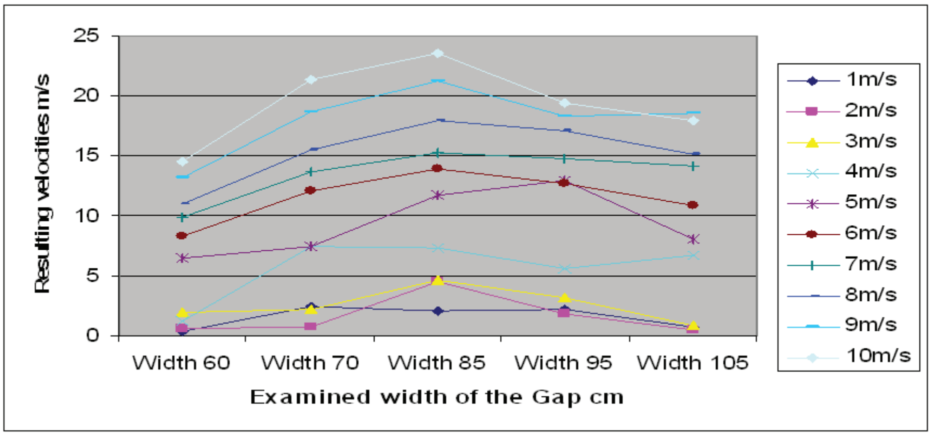

Figure 16: Comparison of the resulting velocities due to examined widths at various tested velocities.

5. Summary of optimisation of wing front shape

The second parameter which determines the magnitude of wind speed at the locality of wind turbine is the wind profile (i.e., geometry). The wing shape need to be designed to maximise the wind speed, by allowing as much wind to be captured and also reduce pressure drop as well as turbulence flow. After establishing the optimum distance the wing need to be placed away from the roof, a further enhancement to the wing shape to enable it to capture more wind would be theoretically possible. This could be by increasing the opening angle of the wing figure 17 below.

However, it is fairly safer to use more monolithic construction materials (i.e., concrete, etc), which are less susceptible to high uplift forces. But that obviously necessitate a proper structural anchoring of the wing. Figure 17 shows the suggested increase in the opening angle by an increment of 2º from the base angle 20 on wards to 30º and 50º.

By this way the wing takes a shape of an aerofoil that enhances the air flow.

By increasing the front attack angle of the wing, more wind is captured by the wing, this increases the mass flow rate through the space beneath the wing. It is assumed that optimum front attack angle would be reached where the mass flow rate is at maximum and the aerofoil shape of the wing is optimised.

5.1. Effect of increasing the angle of attack

As proposed in point 3.1 and 6 above that more opening for the angle of attack would produce more capturing of wind (i.e., air mass flow rate) and therefore more energy. This point however introduces the effect of a large grid depicting a wider and more precise vision of the flow around the building and the wing. By doing so, more realistic results and therefore conclusions can be arrived at. Nevertheless, with the large grid the use of Reynolds stress module of turbulence would slow down the convergence considerably. Hence, the use of the module (k-ε) would be recommended without appreciable loss of accuracy in predicting the flow patterns. Figure 18 below, depicts the effect of investigating the opening angle of the wing by using large grid cases under (k-ε) model. The figure shows the resulting velocities obtained after applying 10-consecutive air velocities through the inlet starting from 1m/s upwards. The profile of the graphs for all angles predicts an increase in the resulting velocities with the increase of angle of the front of the wing from 30º up to 50º. However, the result of angle 60º was below that of 50º. This suggests that angle 50º would be the optimum choice for this wing. Figure 19 below shows that all the tested velocity range (i.e., from 1-10m/s) there is a peak value of resulting velocity at angle 50º. This confirms the fact that this is the optimum angle for this wing.

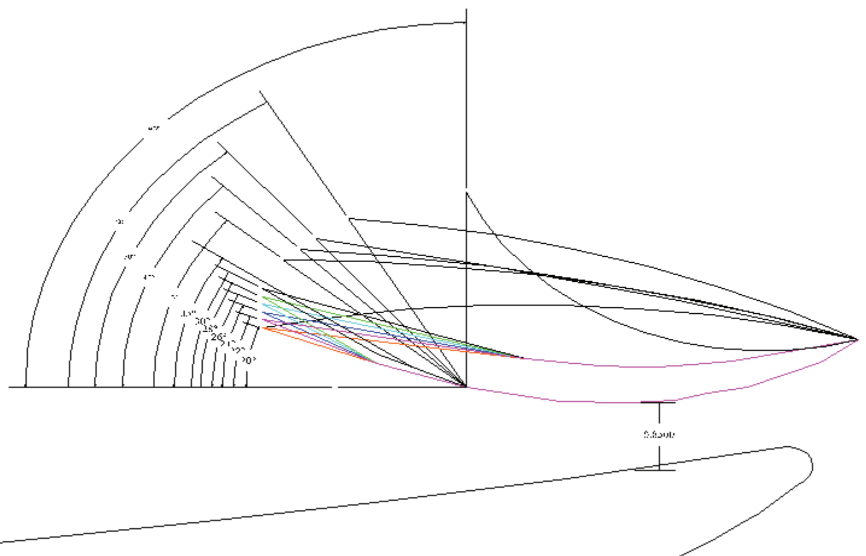

Figure 17: A schematic showing the different opening angles of wind energy capturing wings.

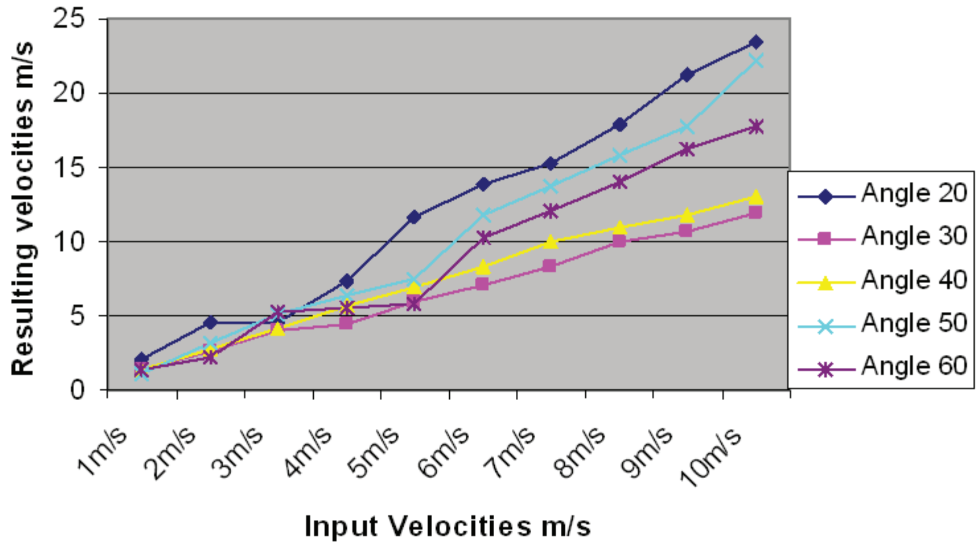

Figure 18: Variation of resulting wind velocities with wing angles at different input wind speed, In Large grid cases.

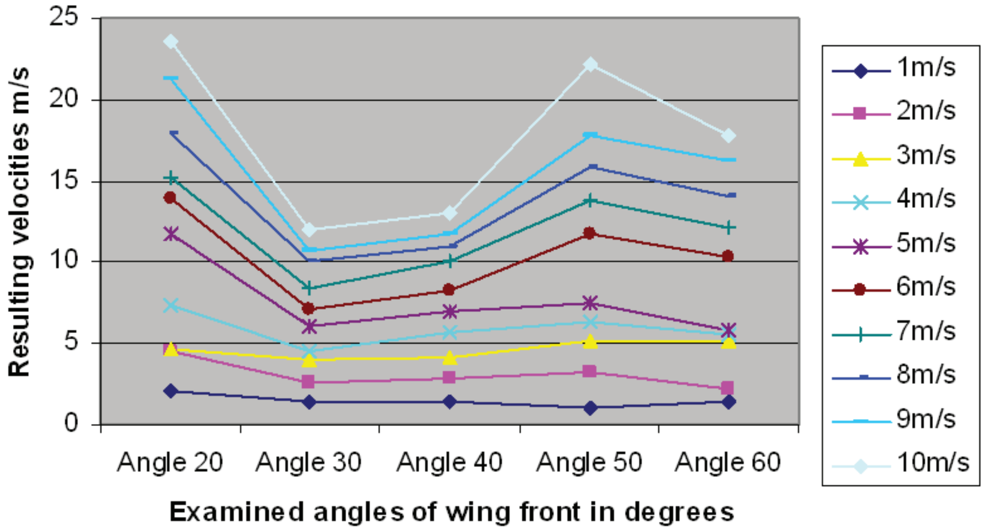

Figure 19: Comparison of the resulting velocities due to examined widths at various tested velocities and angles of attack, in large grid cases.

However, these figures (i.e., 18 and 19) also reveal that angle 20 º, which is the initial angle of the wing front, has the highest resulting velocity in all the range of tested velocities. Therefore, all the resulting velocities from the other angles expect those of angle 50º would be lower than it. This suggests that the large grid has exposed the real flow more clearly and that there would be a possibility for wind to be deflected away from the turbine areas with the increase of angle of attack, but not at 50º. The predicted resulting flow vectors around the wing after applying the angle of attack of 20-60º are shown in figures 20 to 24 below.

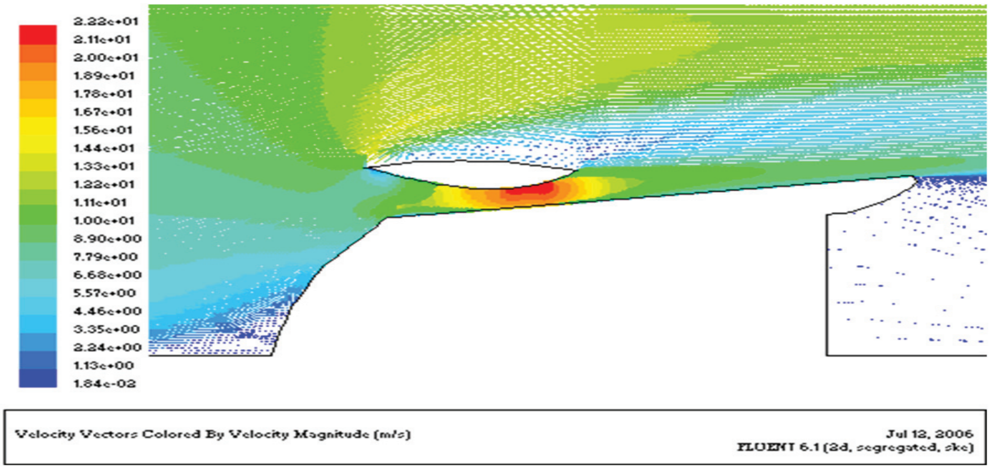

Figure 20: Velocity vectors for wing front angle of attack 20º, in a large grid case.

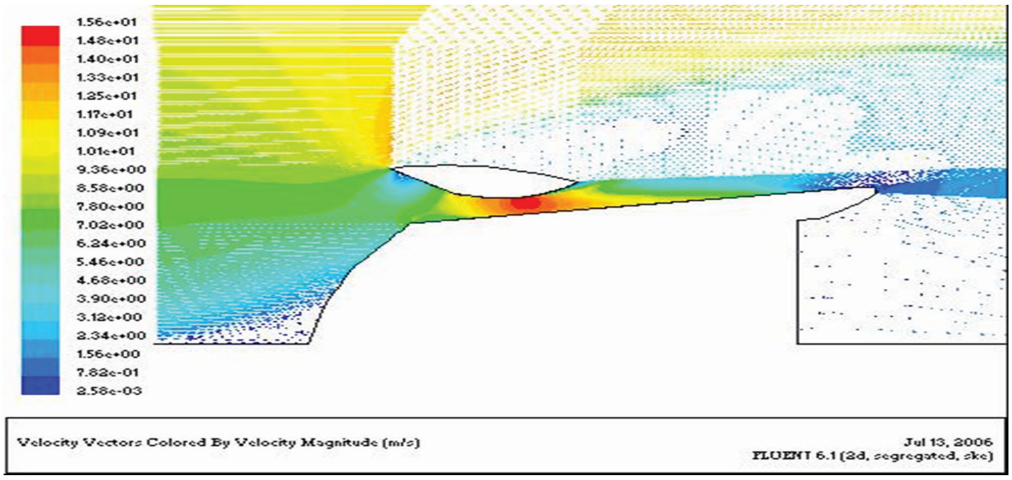

Figure 21: Velocity vectors for wing front angle of attack 30º, in a large grid case.

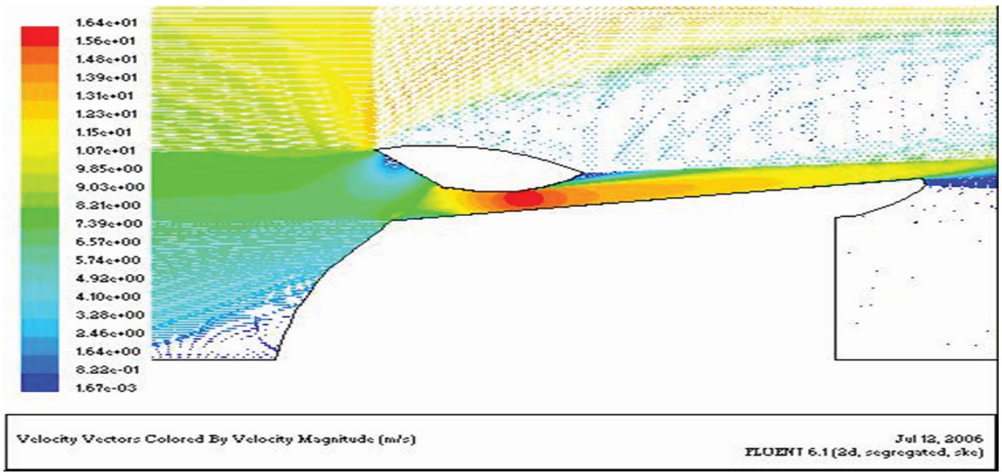

Figure 22: Velocity vectors for wing front angle of attack 40º, in a large grid case.

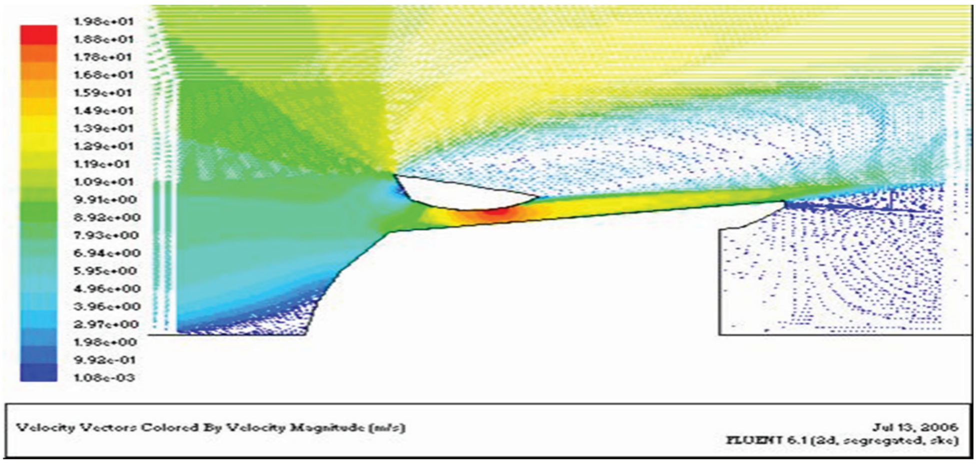

Figure 23: Velocity vectors for wing front angle of attack 50º, in a large grid case.

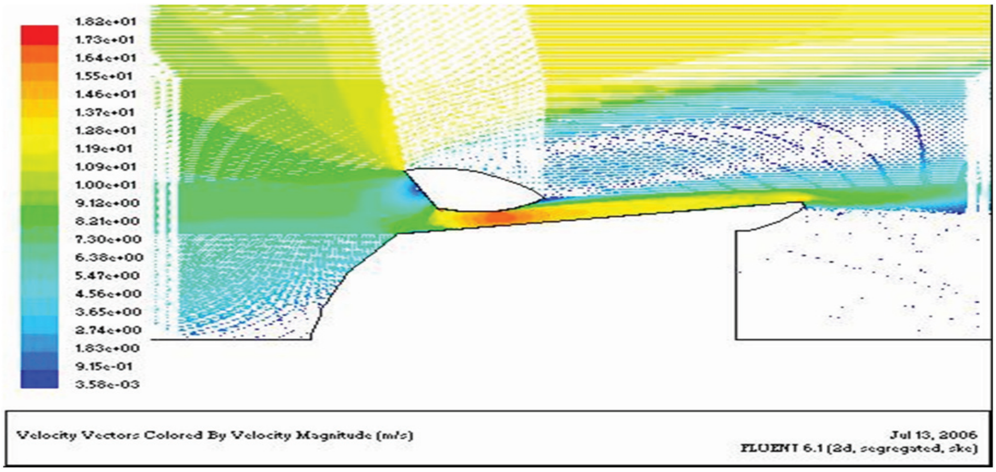

Figure 24: Velocity vectors for wing front angle of attack 60º, in a large grid case.

These figures (i.e., 20-24) show that the air would have a stagnant point appearing just after the wing tip point (i.e., the dark blue point on left side of the wing). From wing of angle 30º onwards. This point tends to move closer into the gap between the wing and the roof as the angle of attack increases. The increase of the of the angle of attack lead also to a larger wind separation from the tip of the wing upwards and at the back of the wing, causing wind to deflect upwards instead of the area between the wing and the roof. Therefore reducing wind speed resulting, this confirms the fact that the large domain would be essential to investigate the angle of attack of the wing. Despite the fact that there is an increase of velocity with the increase of the angle of attack, the nature of flow made this increase falls behind what was in the initial case of angle 20º. Hence the wing angle of attack can affect the speed generated by the wing according to the shape it is in, either positively or negatively.

6. Power estimation

The power in the wind can be expressed as:

$$P_w = 0.5\rho A \nu^3 \quad \text{...} \quad (1)$$

Where, Pw = power in the wind (W), ρ = air density (kg/m^3), ν = wind velocity (m/s), A =swept area of rotor (m^2). The power in the wind increases with the cube of the wind speed, which explains why wind turbines tend to be located at relatively windy sites and trend towards increased tower height to raise the turbine higher avoiding turbulence flow.

6.1 Effect of wing front angle and wing Proximity on the power output

The power output from a wind turbine located under the wing depends basically on the resulting wind velocities. The power output is given for a unit area (W/m^2), and the resulting power at each speed and wing angle examined are obtained as presented on figures 25 and 26, which show the general trend of the power generation with the increase of wing angle. With the increase of angle of attack from 20-50º there is a resulting increase of wind speed and therefore the power production. However increasing the angle to 60º would reduce the resulting power; therefore the angle of 50º would be the optimum angle for this system. Yet there is a large unsteadiness predicted on the percentage of increase of power at all angles figure 26.

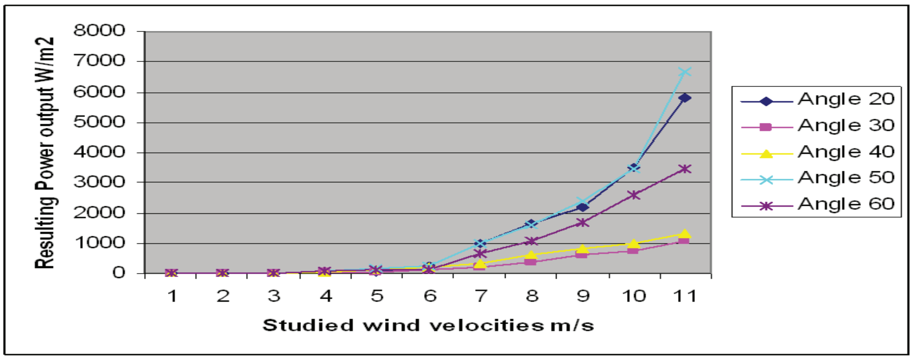

Figure 25: Resulting power output at different wind velocities and angles of wing.

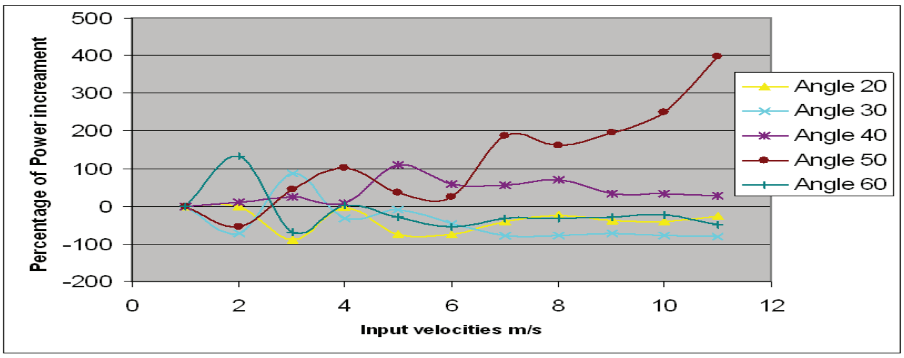

Figure 26: Percentage of power increment as the angle of wing increases.

7. Conclusions:

- Computational fluid dynamics CFD can provide detailed descriptions and guidance on the physical designing and placing of wind harnessing technologies within the built environment.
- An increase in the upstream wind velocity in the region of 30%- 150% is theoretically possible with building design and wind catching shapes and forms considered in this analysis.
- The analysis has proved that introducing aerofoil shapes such as wings suitable at the top and corners of buildings have the potential to provide wind velocity boosting, which can be harnessed using suitable wind turbines.
- The use of aerofoil shapes into buildings can be either during the initial design stages or retrofitted to existing buildings. In either case they can be more cost effective for small (i.e., less than 10kw) integration compared to large-scale machines (i.e., >100kw). That is due to the reduction of capital cost, maintenance cost and there are more prospects of wind capturing within small sites and high-rise buildings.
- The possibility of better operational management, as in the case with multi-small scale systems compared to single large machines (i.e. possibility of small scale turbines backup each other to produce the required output.
- The process of designing aerofoil shapes necessitates some feedback assessing the structural loading and implications as to the effect of induced pressure and drag forces on buildings.
- The main parameters affecting the shape of a wing profile for wind harnessing purposes would be: 1. The distance of the wing from the building surface.

  2. The angle of attack of the wing.
- The study shows that the angle of attack of 50º would provide the optimum wind speed for the design of wing shape under question.
- For the studied wing shape, the optimum proximity of the wing to the roof surface was found to be 85cm. This distance provided the best resulting velocities for the air passing in between the wing and the roof surface.
- The study showed also that the position of the wing on the top of the roof of the house either in the front or rear can affect the resulting performance of the wing according to the direction of the incident wind.

References

1 Danish wind industry Association, www.windpower.org, April 2006.

2 Mertens, S., Henk Krus, R.P.O.M. Van RooiJ, wind description for roof location of wind turbines, A design guidelines of the required height of a wind turbine on a horizontal roof of a mid-to high rise building, proc global wind power, Paris, France, 2-5 April 2002.

3 P.J. Olivera and B.A. Younis, On the Prediction of turbulent flows around full-scale buildings, Journael of wind engineering and industrial Aerodynamics 86 (2000) 203-220.

4 Schlichting, Herman, 'Boundary Layer Theory'McGraw-Hill, Seventh Edition, 2000, Newyork.

5 Hunt, J. C. R., and Smith, G. P., "A Theory of Wakes behind Buildings and Some Provisional Experimental Results," CERL Note RD/L/N 31/69, 1969.

6 R.P. Hoxey, P.J. Richards; flow pattern and pressure field around a full-scale building, J. wind Eng. Ind. Aerodyn. 50 (1993) 203-212.

7 Hunt, J. C. R., "A Theory f o r t h e Laminar Wake of a Two-Dimensional Body in, a Boundary Layer," Journal of Fluid Mechanics, Vol. 49, Part 1, 1971, pp. 159-178.

8 R.P.Hoxey, P.Moran. full scale wind pressure study of the geometric parameters that influence wind loads on low rise buildings, J. wind eng. Ind. Aerodyn. 13. (1993) 277-288.

9 On the prediction of turbulent flows around full-scale buildings

Journal of Wind Engineering and Industrial Aerodynamics, Volume 86, Issues 2-3, June 2000, Pages 203-220 Paulo J. Oliveira, Bassam A. Younis

10 Hunt, J. C. R., "Wakes behind Buildings, I 1 A paper presented to The Atmospheric Environment Committee of the Aeronautical Research Council for their meeting on October 23, 1974.

11 Counihan, J., Hunt, J. C. R., and Jackson, P. S., "Wakes behind Two-Dimensional Surface Obstacles in Turbulent ,Boundary Layers," Journal of Fluid Mechanics, Vol. 64, part 3, 1974, pp. 529-563.

14 AWEA slide Show, www.awea.org/pubs/documents/swslides/toc.htm, June 2006.

15 Elbakheit A.R. 'Enhanced Architectural Integration of Photovoltaics and Wind Turbines into Building Design'. PhD Thesis 2003-2007, Nottingham University, UK.

طريقة مثلى لتصميم المباني المتلائمة مع توربينات تسريع الرياح باستخدام برنامج (CFD): منهجية لتكامل العمارة وطاقة الرياح المستدامة

د. عبد الرحمن البخيت

كلية العمارة والتخطيط

جامعة الملك سعود

البريد الالكتروني: a__rahman41@hotmail.com. Albakheit@ksu.edu.sa

ملخص:

تهدف هذه الورقة لإيجاد طريقة ممنّهجة للتصاميم المعمارية من أجل الاستخدام الأمثل لتكنولوجيا و طاقة الرياح في المباني. تتبنى الطريقة المتبعة استخدام الرياح السائدة في الموقع المعنى وكتل وهيئات المباني بالإضافة إلى أشكال من الأجنحة الهوائية لتسريع الرياح لكي يتسنى استخدامها لتشغيل مراوح التوليد الكهربائي ذات الأشكال والأحجام المناسبة . بهذا الاتجاه فإنها تتعامل مع كتل المباني و علاقاتها مع محيطها لتفعيل التيارات الهوائية و لذلك فإنها تعطى للتصميم المعماري في محاولات اختيار و إيجاد هيئات و أشكال المباني و جدلية مناسبتها لوظيفتها المطلوبة و الجدوى الاقتصادية للتصميم (ومن ثم المشروع) بعدا آخر لتحقيق الاستدامة.

(حوسبة حركة الموائع) حجر الزاوية في اختيار CFD في هذه الطريقة المتبعة ، يأخذ برنامج الحاسوب المسمى

وتقييم و تحسين الأداء للتصميم المعماري تحت الدراسة. مقدرة هذا البرنامج في رصد و توقع حركة الهواء و اتجاهها سرعة و ضغطا ...الخ هي الميزة الكبرى للبرنامج.

تطرقت الدراسة لظاهرة انفصال كتل الهواء نتيجة لاعتراضها بواسطة كتل المباني و ما ينتج عنه من تسريع للهواء مما يجعله صالحا للاستخدام في توليد الطاقة الكهربائية بواسطة المراوح التوربينية وأشكال من الأجنحة المناسب وإمكانية محاكاتها بالبرنامج المذكور .

تخلص الدراسة بان البرنامج المذكور بمثابة أداة فعالة لتصميم و تقييم أشكال الأجنحة لاستخدامها في توظيف طاقة الرياح في المباني. كما تخلص إلى إن خواص هذه الأجنحة ممثلة في:

١. زاوية مواجهتها للرياح.

٢ . بعد هذه الأجنحة من سطوح المباني.

هي أهم العوامل المؤثرة على سرعة الرياح الناتجة من تصميم الأجنحة.

بهذه الطريقة يمكن زيادة سرعة الرياح بواقع ٣٠٪ للرياح الطبيعية المنخفضة السرعة (١-٥ م/ث) والى ١٥٠٪ للرياح الطبيعية المرتفعة السرعة (من ٥م/ث فأكثر).