Numerical Simulation of Turbulent Flow around an Airfoil with Blunt Trailing Edge

H. Sayadi*
Department of Engineering,
University of Shahrekord, Iran
E-mail: hamid.sayadi2009@gmail.com
*Corresponding author

A. R. Shateri
Department of Engineering,
University of Shahrekord, Iran
E-mail: shateri@eng.sku.ac.ir

Received: 13 May 2012, Revised: 9 September 2012, Accepted: 11 October 2012

Abstract: This paper is a computational study of the behaviour of aerodynamic characteristics of an airfoil with blunt trailing edge and investigates the effects caused by modifications made to the trailing edge, on aerodynamic performance. Blunt trailing edge airfoils are of interest in the engineering of large wind turbine blades because they allow for a strong structure with a high aerodynamic lift to structural weight ratio. Blunt trailing edge airfoils would not only provide a number of structural benefits, such as decreased structural volume and ease of fabrication and handling, but they have also been found to improve the lift characteristics of thick airfoils. The incorporation of blunt trailing edge airfoils would allow blade designers to more freely address the structural demands without having to sacrifice aerodynamic performance. These airfoils do have the disadvantage of generating high levels of drag as a result of the low-pressure steady or periodic flow in the near-wake of the blunt trailing edge. Also vortex shedding in these airfoils induces fluctuating loads and radiated noise. In the present investigation, we tested the effects of two cavities on the base drag and wake of an airfoil with blunt trailing edge. In two-dimensional subsonic flows, any method that increases the base pressure of the airfoil with blunt trailing edge consequently reduces the base drag. The base pressure increases subsequent to the introduction of the cavity to the trailing edge. Moreover the cavity causes the trapping and stabilizing of the vortex.

Keywords: Aerodynamic Cavity, Airfoil, Blunt Trailing Edge, Base Cavity, Vortex Shedding

Reference: Sayadi, H., and Shateri, A. R., “Numerical Simulation of Turbulent Flow Around an Airfoil with Blunt Trailing Edge”, Int. J of Advanced Design and Manufacturing Technology, Vol. 6/ No.1, 2013, pp. 61-67.

Biographical notes: H. Sayadi received his MSc in Mechanical Engineering from University of Shahrekord. **A. R. Shateri** received his PhD in Mechanical Engineering from Isfahan University of Technology (IUT) in 2004. He is an assistant professor in Mechanical Engineering Department in University of Shahrekord. His current research interest includes Gas dynamics, Grid generation and Nano fluid.

1 INTRODUCTION

In most cases airfoils are known as having rounded leading edge & sharp trailing edge geometries. However, nowadays blunt trailing edge airfoils also play a significant role in aerodynamic designs. These kinds of airfoils not only have many structural advantages, but are frequently used for the purpose of improving the aerodynamic features of thick airfoils [1], [2]. Among applications of such airfoils, large wind turbine blades, military planes, missiles and automobiles can be counted [3].

All in all, studies show that it is possible to increase the maximum ratio of lift to drag coefficient through using blunt trailing edge airfoils [4]. Besides, the studies focused on blunt trailing edge airfoils indicate that they possess better lift characteristics compared to sharp trailing edge airfoils. However, the point is that the aforementioned type airfoils produce lots of noise as well as producing drag. In addition, testing the vortex in this type of airfoils reveals that there is a severe degree of vortex shedding in their ending edge which in turn is directly related to the created drag force, the produced noise and also the fluctuating loads on the airfoils [5], [6].

Numerous methods have been investigated to lessen the strength of vortex shedding, produced noise by airfoils, fluctuating loads created on airfoils and the drag force. Among suggested solutions are implementing objects like cavity, splitter wedge, and splitter edge, in the end edge of the airfoil. In 2006 Solman [7] carried out extensive lab studies on the created vortex in the end edge of airfoils with blunt trailing edge. He could measure & report the turbulence degree and the velocity field in the vortex area of airfoils with blunt trailing edge.

In 2006, Lombardy et al. [8] investigated the effect of the existence of a flow blower in blunt trailing edge airfoils. The result showed that applying a flow blower with an appropriate rate, at the trailing edge of the airfoil can lead to a decrease in the drag force imposed on the airfoil. In 2008, Van dam et al. [9] conducting several experiments on airfoils with blunt trailing edges, concluded that although there may be some degree of lift force to be lost due to the change made in the primary bending of the airfoil, never the less, they are quite applicable; mainly because of their reduced weight and volume and also because of the lift force they create, compared to their structural weight.

In 2009, Abdullah et al. [10] designed a base cavity and placed it in the airfoil trailing edge. Their studies indicated that the presence of such a cavity leads to trapping the vortex formed in the trailing edge of the airfoil, reducing the produced noise, and also making a balance in the vortex shedding process.

In most numerical studies, an airfoil with sharp trailing edge is cut in order to make an airfoil with blunt trailing edge. This cut is applied vertically at a specified distance from the trailing edge. Accordingly, in order to produce the intended geometry in this study, the standard airfoil NACA 4412 (Fig. 1) having one meter card is employed and a cut at a distance of 0.3 meter is placed through it; thus, an airfoil with blunt trailing edge (Fig. 2) is prepared. In this figure "h" is the thickness of the trailing edge of the airfoil.

In this paper, initially, the aerodynamic characteristics and the flow around the blunt trailing edge airfoil will be investigated. Then, the improvement process of airfoils using the designed geometries will be dealt with. After that, the distinct aerodynamic characteristics of each airfoil along with their advantages will be examined separately. Finally an innovative design is sought. This design, called aerodynamic cavity (Figs. 3 and 4) leads to interesting and sometimes contradictory results which are described below.

Fig. 1 NACA 4412 airfoil

Fig. 2 Airfoil with blunt trailing edge

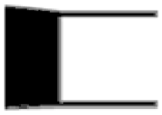

Fig. 3 Base cavity

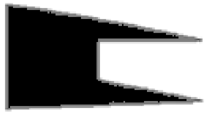

Fig. 4 Aerodynamic cavity

In each case, simulation was processed through Fluent software. In addition, the flow had a steady state condition and the flow regimen was turbulent. In order to achieve a high level of accuracy in the simulation process for each case, the Reynolds stress model (RSM) was employed. Finally, the air flow was considered to have fixed density and physical characteristics.

1.1. Boundary conditions

Figure 5 represents the computational space of the project. In this figure “c” stands for the airfoil cord. It is noted that selecting spaces differ according to the current research. In addition, as it is obvious from the figure and will be understood in the upcoming result section, the pressure and velocity fields reach a steady status after being adequately far from the airfoil which in turn implies the suitability of the selected space. Furthermore, the reason for the AE to be curved as depicted is the optimal results acquired due to this curvature.
Boundary conditions in each case are as follows:
AB, AE, and ED boundaries were considered as the flow inlet boundary condition. BD boundary condition also was considered as the pressure outlet. Due to the return flow in the trailing edge of the airfoil, using such a boundary condition (pressure outlet) is very useful.

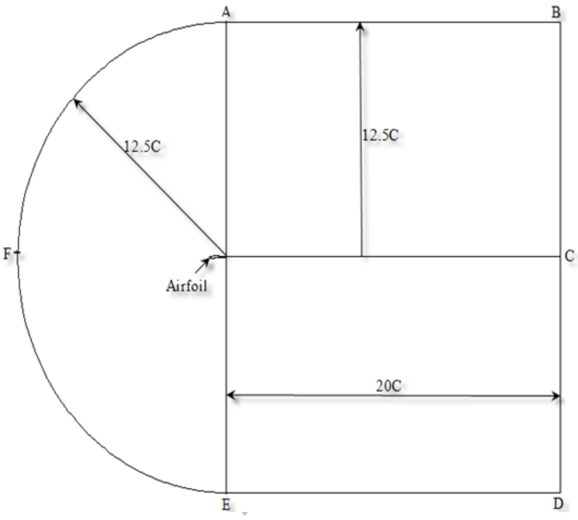

Fig. 5 Computational space considered around the airfoil

1.2. Solution conditions

In this paper input speed was 50 m/s, Reynolds number based on the input speed and the airfoil cord (c) was 3,400,000 and Mach number was 0.151. Applied flow in this project was standard air with the following features:
Density equals to 1.225 kg/m^3, special heat coefficient equals 1006.43 j/kg.k, heat conductivity 0.0242 w/m.k and flow viscosity equals 1.7894e-5 kg/m.s.

1.3. Mesh generating

Generating mesh for each geometry was performed using the Gambit software. In this paper structured meshes for the purpose of meshing airfoils were used. Meshing the surfaces was done with square elements. Furthermore, all used geometries in this project were meshed using Quad-map method. This process is very complex and time consuming. Figure 6 is a sample of the mesh used in this project. It has been minimized as much as possible due to the severe changes on the airfoil and its rear end.

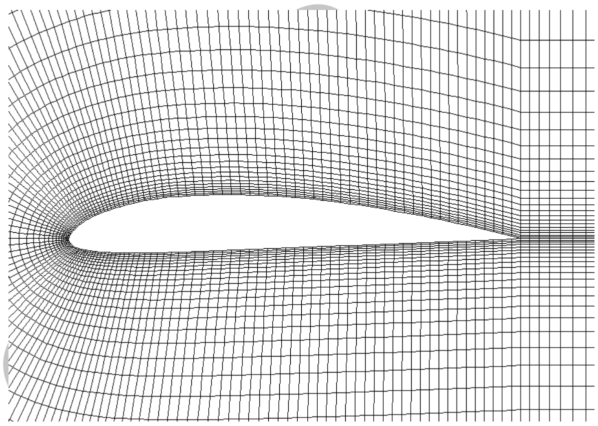

Fig. 6 A view of the used mesh in the paper

2 NUMERICAL RESULTS

2.1. Paper reliability

In order to examine the strength of the utilized mesh in solving the turbulent flow with boundary conditions and also the degree of accuracy of modelling the flow around the existing geometries of the project, the turbulent flow was simulated and then compared with experimental results gained by Abbott & Doenhoff [11].
In this experiment, Reynolds number was 2,000,000 and the input velocity was 29.215 m/s. Besides, the experiment was carried under unbounded flow conditions. Not all equations are necessary to obtain the pressure coefficient. In order to investigate the independence of the mesh and also to compare it with experimental results, the lift coefficient has been used. Table 1 involves the lift & drag coefficients for a 6 degree angle of attack.
As it is conspicuous from the table, after 13888 nodes, lift indexes attain independent from the number of nodes thus in order to enhance accuracy, 14568 nodes were used. To compare lift coefficient for different angles of attack, first the experimental diagram of the

lift coefficient based on the angle of attack was examined in xy-digitizer software and then the lift coefficient numeric indexes in some different angles of attack were obtained.

Figure 7 shows the obtained lift coefficient by RSM and also the experimental results' diagram. As it is obvious, the five equation model of RSM findings is very close to the experimental diagram. That is why the same model was used in subsequent simulations.

Table 1 Transitions selected for thermometry

Number of nodes	Lift coefficient	Drag coefficient
10035	1.247	0.0174
10927	1.233	0.0179
11893	1.118	0.0171
12465	1.122	0.0164
13888	1.115	0.0179
14568	1.114	0.0181

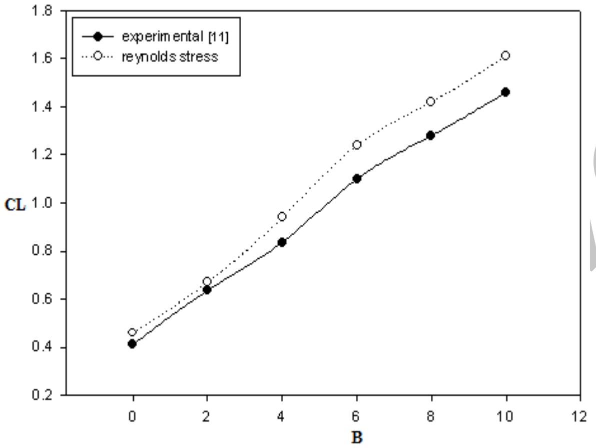

Fig. 7 Diagram of numerical & experimental results of the lift coefficient based on the angle of attack (B=angle of attack, CL=lift coefficient)

2.2. Airfoils with blunt trailing edge

First it should be mentioned that all contours used in the present paper were drown for a 6 degree angle of attack. In Fig. 8 the stream lines are obvious. The big vortex area formed at the rear end of the airfoil is also clear in the figure. In Fig. 9 the pressure contours have been represented in which the area having a high pressure around the stagnation point in the forward part of the airfoil is observable. Figures 10 and 11 represent the drag coefficient and also the ratio of the lift to drag coefficient.

As it is observed in this figure, while the attack angle increases, the drag coefficient also increases. However initially, the lift to drag coefficient, increases up to an angle of attack of 10 degrees; afterwards, it starts to decrease. It is noticed that from this section onward the blunt trailing edge airfoil diagrams are used as the basis for comparison with other diagrams.

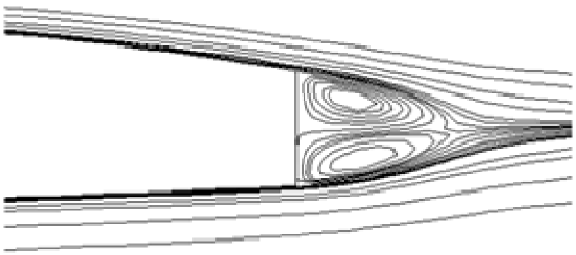

Fig. 8 Stream lines for airfoil with blunt trailing edge

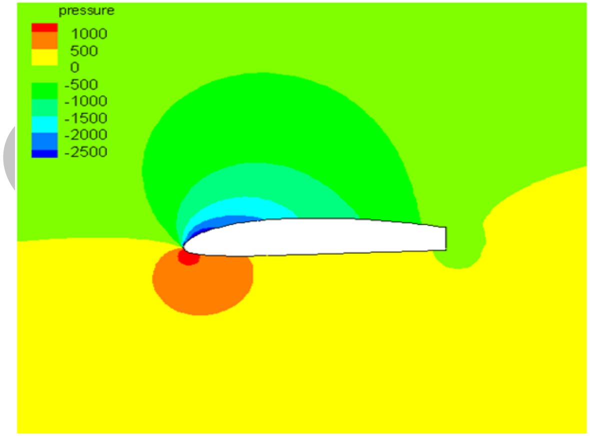

Fig. 9 Pressure contours for airfoil with blunt trailing edge

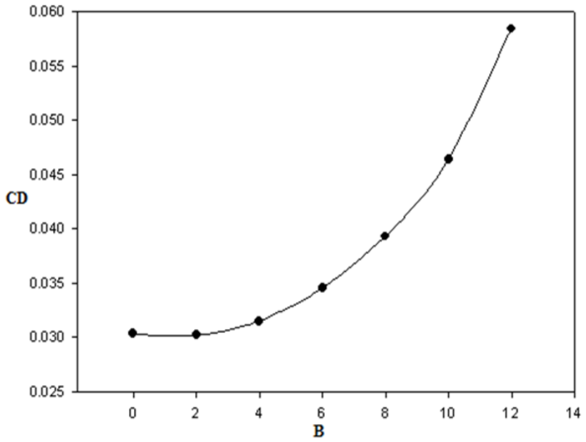

Fig. 10 Drag coefficient diagram based on the angle of attack for an airfoil with blunt trailing edge (CD= drag coefficient)

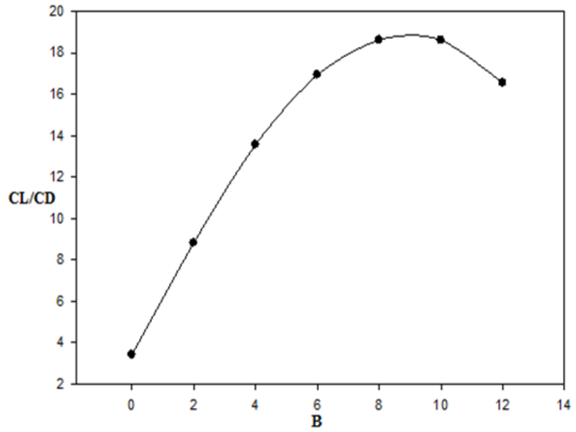

Fig. 11 Diagram of lift to drag coefficient based on the angle of attack for airfoil with blunt trailing edge

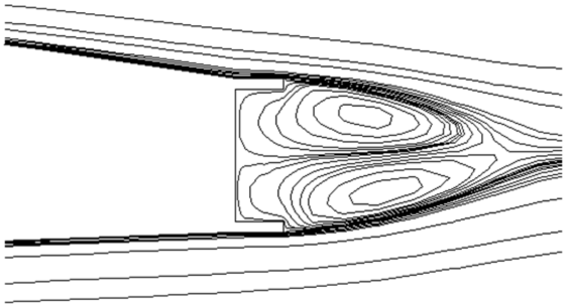

Fig. 12 Stream lines for modified airfoil with base cavity

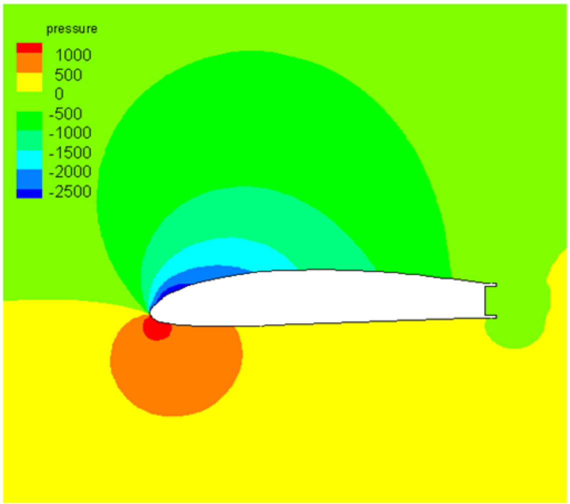

Fig. 13 Pressure contours for airfoil with base cavity

2.3. Airfoils with base cavity

The airfoil geometry is formed, connecting a cavity with 1/3 * h, to the airfoil trailing edge. In Fig. 12 the stream lines have been depicted for such geometry. As it is obvious from the figure, the cavity plays an important role in adjusting the vortex shedding process through trapping a portion of vortexes. It is also apparent from the figure that the existence of this cavity has caused the vortex area to be formed in a distance, far from the airfoil trailing edge.

In Fig. 13 pressure contours have been drawn. Figures 14 and 15 also represent the drag coefficient diagrams and the ratio of lift to drag coefficient for the aforementioned geometry and the basis stance. It is possible to decrease the vortex shedding process remarkably by placing the base cavity in the trailing edge of the airfoil with blunt trailing edge, maintaining its aerodynamic features.

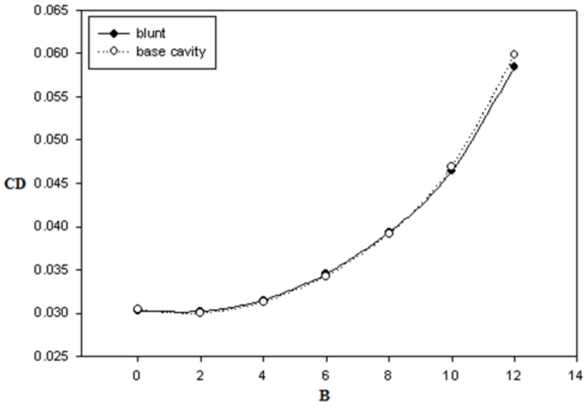

Fig. 14 Diagram of drag coefficient for the modified airfoil with base cavity and basis stance

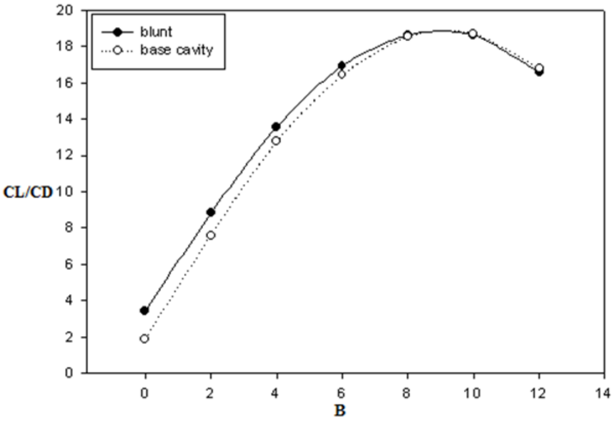

Fig. 15 Diagram of lift to drag coefficient for the modified airfoil with base cavity and basis stance

2.4. Airfoils with aerodynamic cavity

Having examined the results of different cavities and analyzed their stream lines, it is concluded that an aerodynamic design of the cavity's upper and lower edges produces more optimal drag and lift features compared to a flat design. Therefore, a cavity with aerodynamic edges was formed and referred to as an aerodynamic cavity. The existence of such cavity caused a remarkable reduction in the produced drag for lower angles of attack. Besides, installing this cavity in the trailing edge of the airfoil causes a suitable increase in the lift coefficient. Stream lines for this geometry is depicted in Figure 16.

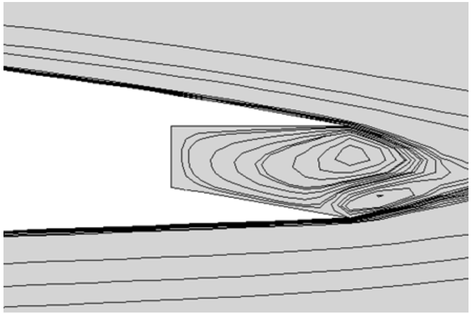

Fig. 16 Stream lines for modified airfoil with aerodynamic cavity

Figure 16 clearly shows the trapping process of a large portion of vortex, stabilization process of vortex zone, and also the formation of this zone in distances far from the airfoil trailing edge. In Fig. 17 pressure contours can be observed. In figure 18 the drag coefficient diagram for the geometry and the basis stance is depicted. It is clear from the diagram that drag coefficient for this geometry is less than the basis stance up to an attack angle of 7 degrees.

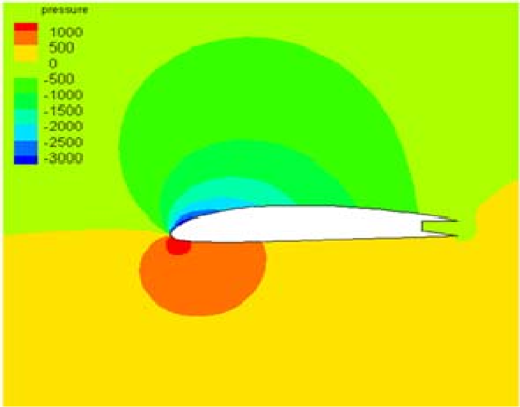

Fig. 17 Pressure contours for airfoil with aerodynamic cavity

In Fig. 19, the diagram of the ratio of the lift to drag coefficient is depicted. Apparently placing this cavity in the trailing edge of the airfoil will lead to a remarkable increase in the lift coefficient in addition to drag reduction. Therefore, implementing this kind of cavity in the airfoil will improve the aerodynamic characteristics of the airfoil and adjust the process of vortex shedding and its excess noise.

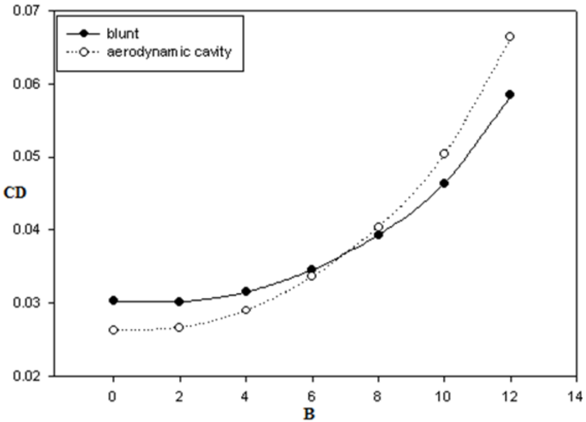

Fig. 18 Diagram of drag coefficient for the modified airfoil with aerodynamic cavity and basis stance

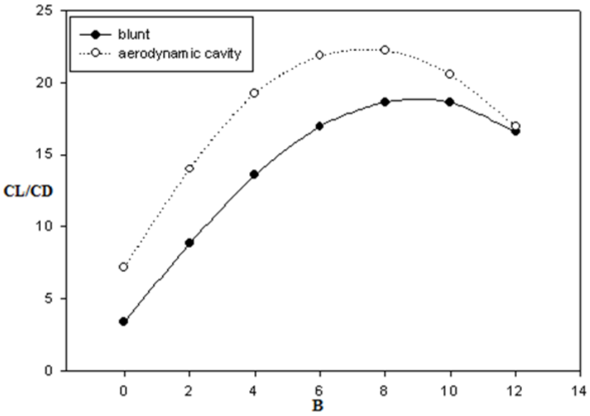

Fig. 19 Diagram of lift to drag coefficient for the modified airfoil with aerodynamic cavity and basis stance

3 CONCLUSION

Blunt trailing edge airfoils are abundantly used in the wind turbines due to their creation of high lift force in comparison with their structural weight & volume. However they produce a high drag force due to the remarkably low pressure vortex zone created behind them. Besides, due to the severe shed ejection of vortexes in their trailing edge, high noises and also huge alternative forces are produced on the blades.

It is possible not only to adjust the vortex throwing process through using a base cavity in the airfoil trailing edge, but also to trap vortexes inside the cavity while retaining suitable airfoil aerodynamic features. This will also lead to reducing excess noises and fluctuated loads produced on the airfoil. Finally, using an aerodynamic cavity in the trailing edge of the airfoil caused an improvement in the aerodynamic features of the airfoil and an increase in the ratio of the lift to drag coefficient.

REFERENCES

[1] Hoerner, S. F., “Base drag and thick trailing edges”, Journal of the Aeronautical Sciences, Vol. 17, No. 10, 1950, pp. 622-628.

[2] Hoerner, S. F., “Fluid-Dynamic Drag”, Hoerner Fluid Dynamics, Bricktown, NJ, 1965.

[3] Morel, T., “Effect of base cavities on the aerodynamic drag of an axisymmetric cylinder”, Aeronautical Quarterly. Vol. 30, part 2, 1979, pp. 400-412.

[4] Standish, K. J., and Van Dam, C. P., “Aerodynamic analysis of blunt trailing edge Airfoils”, Journal of Solar Energy Engineering, Vol. 125, 2003, pp. 479-487.

[5] Baker, J. P., Mayda, E. A., and van Dam, C. P., “Experimental analysis of thick blunt trailing-edge wind turbine airfoils”, Journal of Solar Energy Engineering, Vol. 128, No. 4, 2006, pp. 422–431.

[6] Baker, J. P., Van Dam, C. P., and Gilbert, B. L., “Flatback airfoil wind tunnel experiment”, Tech. Rep. SAND2008-2008, Sandia National Laboratories, April 2008.

[7] Souleman, E., “Anechoic Wind Tunnel Study”, Internship report, The University of Adelaide, 2006.

[8] Lombardi, G., Magazi, M., Caldiralo, L., Cannizzo, F., and Petrotta, G., “The aerodynamic development of the Ferrari Fxx”, 6th MIRA International Vehicle Aerodynamics Conference, Warwick (England), October 2006.

[9] Van Dam, C. P., Mayda, E. A., and Chao, D. D., “Computational Design and Analysis of Flatback Airfoil Wind Tunnel Experiment”, SAND2008-1782, Sandia National Laboratories, Albuquerque, NM, March 2008.

[10] Abdullah, M. Al-Garni, Ahmed Z. Al-Garni and Aravind, A., “Experimental Investigation of the effect of Base Cavities on the Base pressure and Wake of a Two-Dimensional Bluff Body”, 7th IASME/WSEAS conference on Fluid Mechanics and Aerodynamics (FMA’09) Moscow, Russia, August 20-22, 2009.

[11] Abbott, I. H., and .Von Doenhoff, A. E., “Theory of Wing Sections”, Dover, New York 1959.