Journal of Visualization, Vol. 3, No. 3 (2000) 245-252

Numerical Flow Visualization of Stall Suppression of a Symmetrical Aerofoil by Leading-edge Moving Surface

Chew, Y. T.*, Lee, T. S.*, Li, W. P.*, Pan, L. S.* and Ng, Y. T.*

* Faculty of Engineering, National University of Singapore, 117576 Singapore.

Received 8 October 1999.
Revised 31 October 2000.

***Abstract*:** This paper applies the numerical simulation techniques based on the generalized conservation of circulation (GCC) method to investigate the effects of momentum injection by a leading-edge moving surface on flow past a two-dimensional aerofoil at a Reynolds number of 1000. The stream function and vorticity contours obtained together with the animated flow visualization show that the stall flow region is highly unsteady and consist mainly of large vortices being shed alternately. They are confined to a narrow region near the upper surface of aerofoil as C_u (the ratio of the speed of the moving surface to the free stream velocity) is raised. The proximity of vortices to the upper surface of aerofoil at high C_u is caused by the ability of free stream to negotiate around the leading edge since the leading-edge moving surface suppresses the growth of boundary layer by reducing the relative between the inviscid flow and the wall. As well-formed large scale vortices are associated with low pressure regions, their proximity to the aerofoil leads to increase in lift as speed ratio increases.

***Keywords*:** numerical simulation, moving surface boundary layer control, visualization.

1. Introduction

This paper applies the numerical simulation techniques based on the generalized conservation of circulation method (Pan et al., 1995) to investigate the effects of momentum injection by the leading edge moving surface on the stall flow of a two-dimensional symmetrical aerofoil.

It is well known that the performance of many fluid systems is limited by the onset of flow separation as it affects the drag and lift forces of a moving body in viscous fluid. Hence, the control of flow separation has become an important problem in fluid mechanics. A typical example is the control of stall on aircraft wings by leading-edge moving surface. A moving surface attempts to accomplish this in two ways: it prevents the initial growth of the boundary layer by minimizing relative motion between the surface and the freestream, and it injects momentum into the existing boundary layer. Regardless of the methods used, the common objective of controlling flow separation is to reduce the wake region and to design a high lift and high efficiency aerofoil for engineering applications.

Extensive literature accumulated over the years has been reviewed by several authors, such as Goldstein (1938), Lachmann (1961), Rosenhead (1966), Schlichting (1968), Chang (1970) and Modi (1997). In the early days, the methods of flow separation control were mainly focused on the utilzation of suction, blowing, vortex generation and turbulence promoters. The application of moving wall boundary layer control was demonstrated first by Favre (1938). Using an aerofoil with an upper surface formed by a belt moving over two rollers, he showed that it was able to delay separation until the angle of attack α reached 55 where the lift coefficient was as high as 3.5. After a lull of more than 20 years, Alvarez-Calderon and Arnold (1961) carried out tests on a rotating cylinder flap that was utilized in the design of a high-lift aerofoil for STOL-type aircraft. Brooks (1963) published his preliminary results of tests on a hydrofoil with a rotating cylinder at the leading or trailing edge. The results showed a small increase in lift for the leading-edge configuration but a substantial gain for the latter case. This motivated an improved design to increase the fin performance for torpedo control. Steel and Harding (1970) investigated the application of rotating cylinders to improve ship maneuverability. Tennant (1973) presented an interesting analysis of a two-dimensional, moving-wall diffuser with a step change in area. Tennant et al. (1976,1977,1978) reported the control of circulation for a symmetrical aerofoil with a rotating cylinder forming its trailing edge. For zero angle of attack, a lift coefficient C_L of 1.2 was attained with $C_u=U/U_\infty=3$, where U_∞ is the free stream velocity and U is the surface velocity of the rotating cylinder.

Recently, Modi et al. (1981) showed that a moving surface can be an effective means of boundary layer control to provide a significant increase in the maximum lift coefficient and stall angle. It appears that a rotating cylinder at the leading-edge of an aerofoil is likely to provide the maximum benefit. Modi et al. (1990, 1991) also showed that moving surface appears to be quite promising in reducing drag of bluff bodies. Munshi (1995) conducted the test on a two-dimensional Joukowsky aerofoil with a rotating cylinder at its leading edge. The results showed that the stall angle and lift coefficient increase with increasing rotation speed.

In the previous work of Chew et al. (1998), the leading edge consists of a rotating circular cylinder instead of the present moving surface of the leading edge profile of a Joukowsky aerofoil. This configuration was chosen in order to enable the verification of the numerical method by comparing with Munshi's (1995) experimental results of the same configuration. The numerical results were computed at the experimental Reynolds number of 1.43×10^5. They are in agreement with the experimental results for various speed ratio C_u and angles of attack. In the present paper, flow past a Joukowsky aerofoil with leading-edge moving surface is computed at a Reynolds number of 1000 in order to study the flow patterns and stall suppression at various angles of attack α and speed ratio C_u through numerical flow visualization.

2. Description of Problem and Numerical Method

For constant density, viscous, stall flow past a Joukowsky aerofoil, the flow is always unsteady due to the fluctuating wake behind the aerofoil. This is irrespective of whether the ambient velocity and surface conditions of the aerofoil are time independent. Thus the flow field must be computed as an unsteady problem and basic unsteady equations for vorticity ω and stream function ψ governing the separation flow are

$$\frac{\partial\omega}{\partial t} + \boldsymbol{V}\cdot\nabla\omega = -\frac{1}{\mathrm{Re}}\nabla^2\omega \tag{1}$$

$$\nabla^2\psi = -\omega \tag{2}$$

$$\boldsymbol{V} = \nabla\times\left(\psi\boldsymbol{e}_3\right) \tag{3}$$

In the above equations, all variables and operators are in corresponding dimensionless forms and ∇ is gradient operator and $\nabla^2 = \nabla\cdot\nabla$ in (x, y) co-ordinate system (see Fig.1). The reference velocity, length and time used to non-dimensionalize the variables are the ambient velocity U_∞, the length of the aerofoil C and (C/U_∞) respectively. Hence the Reynolds number is $\mathrm{Re}=U_\infty C/\nu$, where ν is kinetic viscosity. In Eq. (3), $\boldsymbol{e}_3$ is a unit vector perpendicular to flow plane.

It is obvious that the ordinary boundary conditions of velocity are inconvenient for solving Eqs. (1) and (2). This can be overcome by altering the form of boundary conditions into that in terms of the vorticity ω and stream function ψ. Suppose (ξ, η) is a local orthogonal coordinate system near the boundary and the boundary is denoted by ξ=0. It can be shown (Chew et al., 1998) that if the coordinate transformation is conformal, the boundary condition for vorticity on the surface can be transformed into the form

$$\omega\big|_B = -2\frac{\left|\nabla\xi\right|^2}{\Delta\xi^2}\psi\left(\Delta\xi,\eta\right) + 2\frac{\left|\nabla\xi\right|}{\Delta\xi}\boldsymbol{V}_B\cdot\boldsymbol{e}_\mathrm{s} \tag{4}$$

where $\boldsymbol{e}_s$ is unit tangent vector and $\boldsymbol{V}_B$ is surface velocity on the aerofoil.

Since the vorticity at infinity is zero this boundary condition can theoretically be used to solve Eqs. (1) and (2). However, numerical simulation requires that the outer-boundary of computational domain must be bounded. Thus, some closed curve far away from the aerofoil within flow field has to be defined as a computational boundary artificially. Clearly, it is impossible to give the condition of vorticity on the artificial outer-boundary before obtaining the solution of Eq. (1). In fact, this difficulty is not overcome completely in fluid mechanics. If the generalized conservation of circulation (GCC) method (Pan et al., 1995) is used, this difficulty can be avoided. The details of the present method were described in another paper (Chew et al., 1998).

3. Numerical Results and Discussion

The model of aerofoil to be simulated is as shown in Fig. 1. It is a fixed geometry aerofoil, but with part of its leading edge moving at the speed ratio C_U. The moving leading surface is within 7% of the chord length.

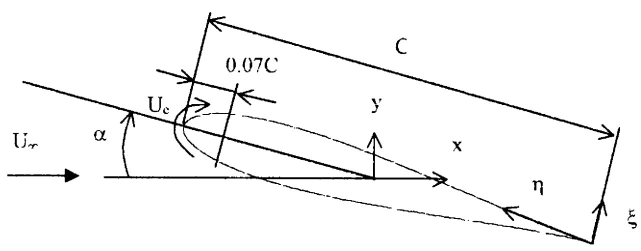


Fig. 1. Sketch of the aerofoil model with leading-edge moving surface.

To obtain a better understanding of the effects of speed ratio C_U on the suppression of stall at various angles of attack α, it is necessary to conduct numerical flow visualization through vorticity contour and streamline plots. As the solutions of Eqs. (1) and (2) are time dependent vorticity and stream function, their contour patterns are amenable to animation and much understanding of the physics of flow can be obtained through the study of such animated flow patterns. Although the study is extensive and covers a wide range of speed ratio and angle of attack, only selected results at C_U = 0, 2, 4 and α = 15°, 45° for Re=1000 will be presented here.

In Figs. 2 (a) and (b), the time sequence of vorticity evolution around the aerofoil and in the wake is shown for two cases of Cu = 0, 4 at α = 45° respectively. The time sequence presented is over approximately one period of the vortex shedding behind the aerofoil. Here the period for α = 45° is about 2.5. Comparing Figs. 2 (a) and (b), it can be seen that at α = 45°, irrespective of whether there is a leading-edge moving surface, the boundary layer on the upper side of the aerofoil has separated and the flow is similar to that past a bluff body. The vorticity is not confined to a thin region within the boundary layer and the usual concept of boundary layer on the upper surface is no longer valid. Thus the traditional belief of stall suppression through a delay in boundary layer separation at high angle of attack is questionable. The question to be answered is: if there is no suppression of boundary layer separation, how do we explain the increase in lift with increasing speed ration C_u ?

In order to answer the above question, we need to examine carefully Figs. 2 (a) and (b) again. It can be seen that for the case of C_u = 0, the near wake vortices are formed farther away from the upper aerofoil surface. For the case of C_u = 4, the leading-edge moving surface reduces the relative velocity between the inviscid flow and the wall and suppress the growth of boundary layer at its leading edge. This enables the free stream to negotiate around the leading edge even at high angles of attack, resulting in the formation of vortices close to the upper surface of the aerofoil. A well-formed vortex has a low pressure at its center and it is well known in bluff body aerodynamics that the shorter is the formation length of vortices measured from the wall, the lower is the near wake or base pressure. Thus, the leading-edge moving surface produces vortices confined to a narrow region on the upper surface of the aerofoil and generates large vortex-induced lift. Consequently, the width of the vortex wake is also reduced. These variations imply that the lift coefficient tends to increase and the drag coefficient decreases as the speed ratio C_u is raised.

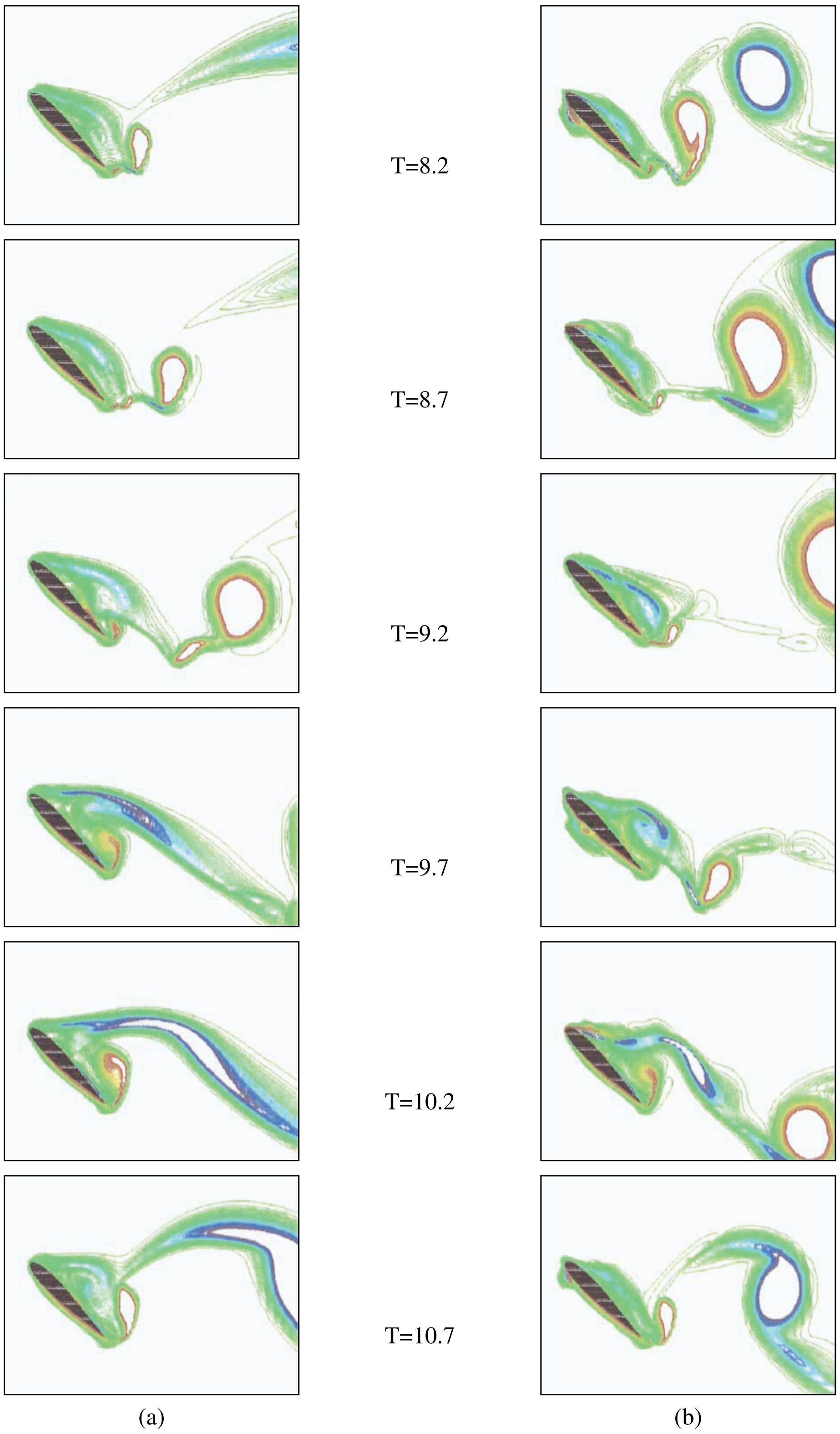


Fig. 2. Vorticity plots for aerofoil at angle of incidence of 45º: (a) $C_u = 0$, (b) $C_u = 4$.

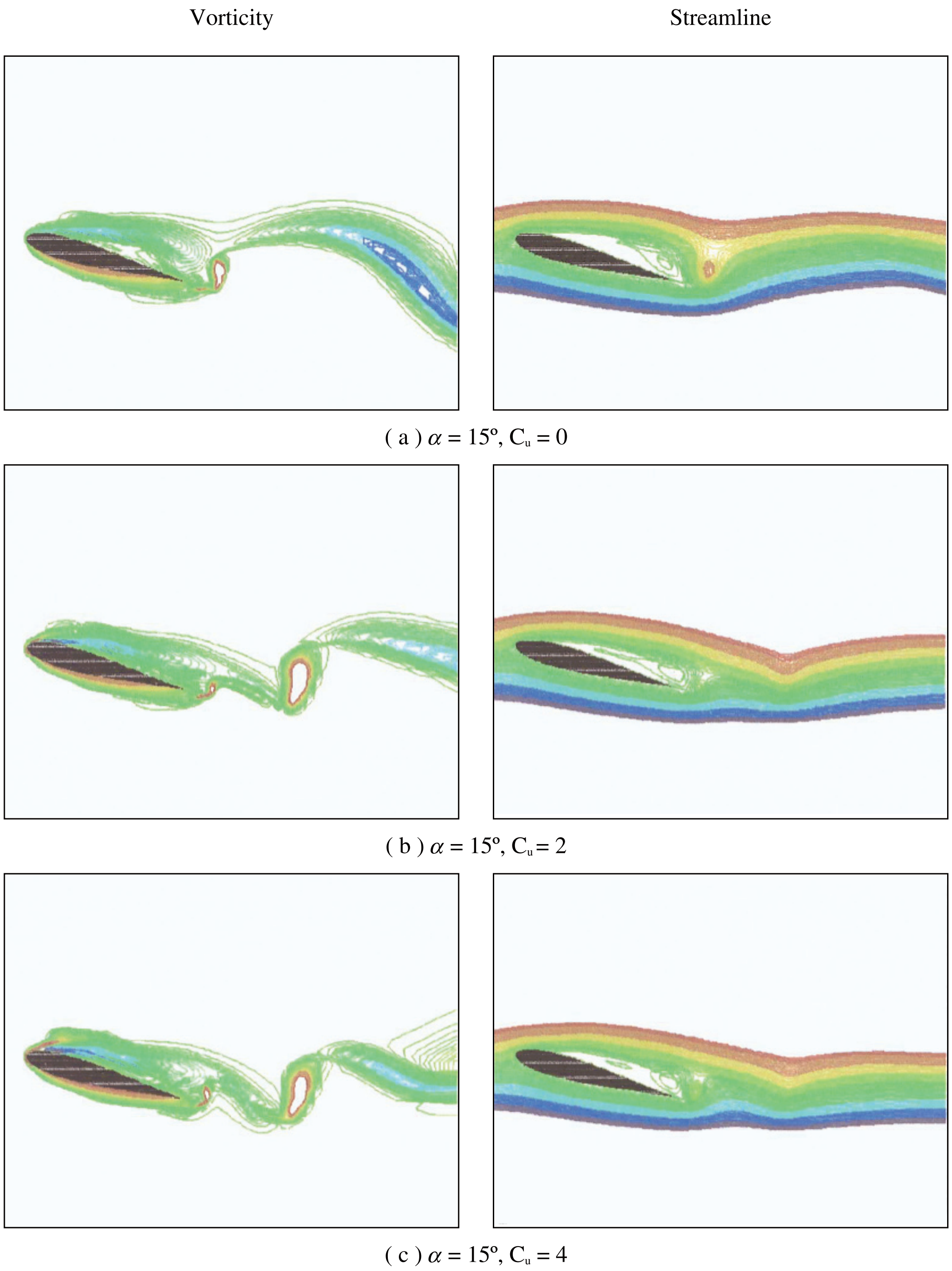


Fig. 3. Streamline and vorticity contours at Re = 1000 and C_u = 0, 2 and 4 for α = 15º.

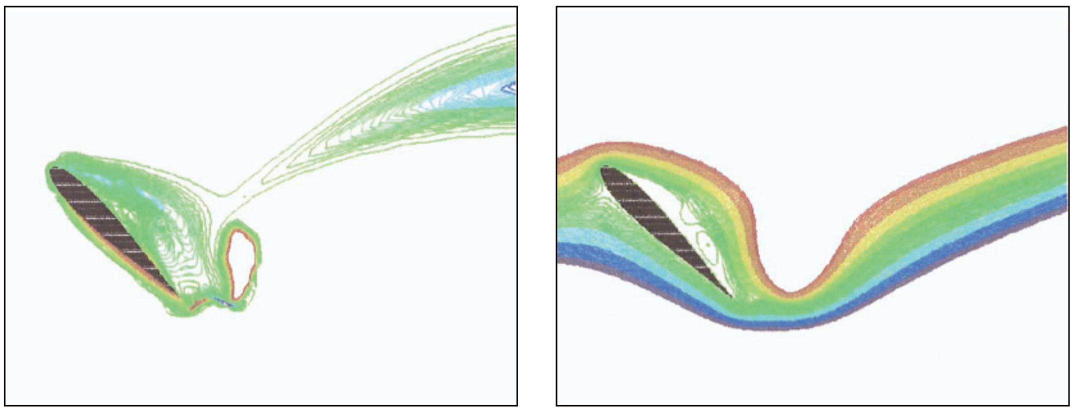
Figures 3 and 4 show the numerical streamline patterns and vorticity contours at Re=1000 and C_U=0, 2 and 4 for α = 15º and 45º respectively. They correspond to the instant of maximum lift coefficient. Comparison of results at different C_u may not be easy as they are at different phases of vortex shedding. However, they support the general observations made in Fig. 2. At α = 15º and C_u = 4 in Fig. 3(c), anti-clockwise vorticity can be observed near the leading edge on the upper surface due to leading-edge surface moving at velocity larger than that in inviscid flow. At α = 45º when C_U = 4 in Fig. 4 (c), anti-clockwise vorticity can also be observed near the leading edge on the lower surface cased by the moving surface. The bulging of this vorticity region and its location on the lower surface is dependent on the phase angle of vortex shedding as can be observed in Fig. 2 (b).

Kelvin-Helmholtz instability can also be observed in the strong shear layer at the trailing edge in Fig. 4 (b).

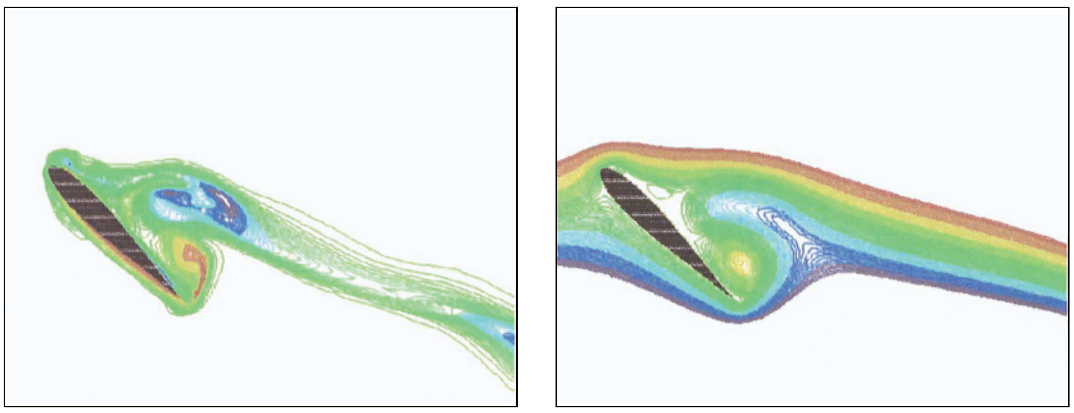
In a similar work done by Chew et al. (1998) which was carried out at a much higher Reynolds number of 1.43×10^5, small vortices tend to form on the upper surface of the aerofoil at $\alpha = 45^{\circ}$ when C_u is raised.

It is well known that stall is directly related to flow separation and the size of the wake region. The above results indicate that the stall flow region is highly unsteady and consist of large vortices being shed alternately . It is confined to a narrow region near the upper surface of aerofoil as C_u is raised.

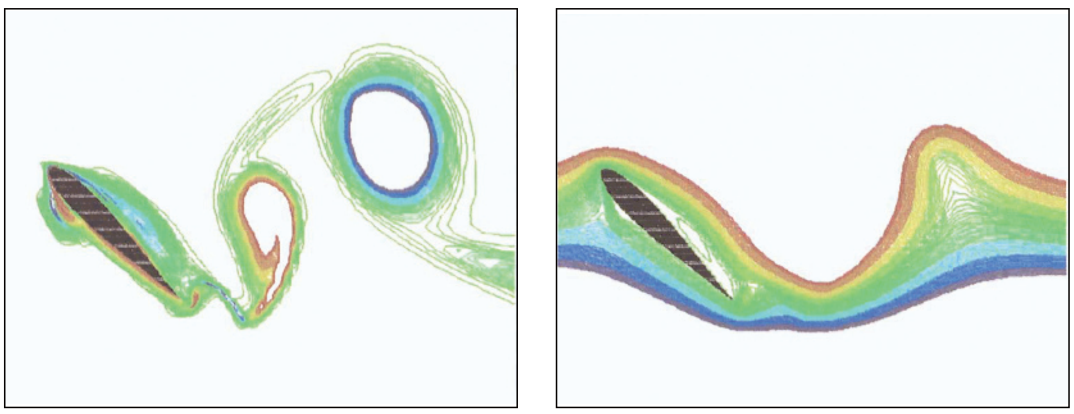
Vorticity Streamline



(a) $\alpha = 45^{\circ}$, $C_u = 0$



(b) $\alpha = 45^{\circ}$, $C_u = 2$



(c) $\alpha = 45^{\circ}$, $C_u = 4$

Fig. 4. Streamline and vorticity contours at Re = 1000 and C_u = 0, 2 and 4 for $\alpha = 45^{\circ}$.

4. Conclusions

Based on the GCC method used to simulate the separation flow of an aerofoil with leading-edge moving surface, it can be concluded that the application of such moving surface can effectively control the separation flow around a symmetrical aerofoil.

It has been shown from the numerical flow visualization results that the stall flow region is highly unsteady and consists mainly of large vortices being shed alternately. It is confined to a narrow region near the upper surface of aerofoil as C_u is raised.

The effective stall flow control by the means of leading-edge moving surface leads to an increase in lift and a reduction in drag.

References

Alvarez-Calderon, A. and Arnold, F. R., A Study of the Aerodynamic Characteristics of a High Lift Device Based on Rotating Cylinder Flap, Stanford University Technical Rept. RCF-1, 1961.

Brooks, J. D., Effect of a rotating cylinder at the leading and trailing edges of a hydrofoil, U.S. Naval Ordnance Test Station, Department of the Navy, NAVWEPS Rept. 8042, April 1963.

Chew,Y. T., Pan, L. S. and Lee, T. S. Numerical simulation of the effect of a moving wall on separation flow past a symmetrical aerofoil, J. Power & Energy, (1998), V212, Part A, 69-77.

Chang, P. K., Control of Flow Separation, (1970), 154-177, Hemisphere Publish Corporation, McGraw-Hill Book Company, Washington.

Favre, A., Contribution a L'etude Experimentale des Mouvement Hydrodynamiques a Deux Dimensions, (1938), Thesis University of Pais.

Goldstein, S., Modern Developments in Fluid Mechanics, (1938), Vols. I and II, Oxford University Press, Oxford, England.

Lachmann, G. V., Boundary-Layer and Flow Control, Its Principles and Application, Vol. I, (1961), pp. 1-2, Pergamon Press, New York.

Modi, V. J. Moving Surface Boundary Layer Control: A Review. J. Fluids and Structures, 11, (1997), 627-663.

Modi, V. J., Sun, J. L. C., Akutsu, T., Lake, P., McMillan, K., Swinton, P. G. and Mullins, D., Moving surface boundary layer control for aircraft operation at high incidence, Journal of Aircraft, 18, (1981), 963-968.

Modi, V. J., Ying, B. and Yokomizo, T., Boundary-layer control of bluff bodies through momentum injection, SAE Trans.:, Journal of Commercial vehicles, Sec. 2, 99, (1990),778-794.

Modi, V. J., Fernando, M. S. U. K. and Yokomizo, T., Moving surface boundary-layer control: Studies with bluff bodies and application, AIAA Journal, 29-9, (1991), 1400-1406.

Munshi,S. R., Aerodynamics and dynamics of bluff bodies in presence of the moving surface boundary-layer control, (1995), Ph.D Thesis, The University of British Columbia.

Pan, L. S., Chew, Y. T. and Lee, T. S., A proposed new vortex calculation method using generalized conservation of circulation model, Proceedings of 6th Asian Congress of Fluid Mechanics (Singapore), (1995), Vol. 1, 445-448, Published by Centre for Continuing Education, Nanyang Technological University.

Rosenhead, L., Laminar Boundary Layers, (1996), 111, 322-335, 547, Clarendon Press, Oxford.

Schlichting, H., Boundary Layer Theory, (1968), 362-390, McGraw-Hill Book Company, New York.

Steele, B. N. and Harding, M. H., The Application of Rotating Cylinder to Ship Maneuvering, National Physical Laboratory, Ship Division, U.K., Report No.148, December 1970.

Tennant, J. S., A subsonic diffuser with moving walls for boundary-layer control, AIAA Journal, 11-2, (1973), 240-242.

Tennant, J. S., Johnson, W. S. and Krothapalli, A., Rotating cylinder for circulation control on an aerofoil, Journal of Hydronautics, 10-3, (1976), 102-105.

Tennant, J. S., Johnson, W. S. and Keaton, D. D., On the calculation of boundary layer along rotating cylinder, Journal of Hydronautics, 11-2, April (1977), 61-63.

Tennant, J. S., Johnson, W. S. and Keaton, D. D., Boundary layer flow from fixed to moving surfaces including gap effects, Journal of Hydronautics, 12-2, (1978), 81-84.

Author Profile

Y. T. Chew: He has received his BEng (Hons.) and MEngSc degrees from the University of Western Australia, and his PhD degree from Cambridge University in 1976. He worked as a Visiting Assistant Professor with the University of Texas at Austin and Southern Methodist University at Dallas, USA briefly before joining the National University of Singapore where he is currently the Professor and Head of the Fluid Mechanics Division. He is also the President of the Institution of Engineers Singapore. His research interests include near-wall turbulence measurements, bluff body aerodynamics, bio-fluid dynamics and computational fluid dynamics.

T. S. Lee: Currently, Deputy Head - Fluid Mechanics Division and Associate Professor, Mechanical Engineering Department, National University of Singapore. He graduated from the University of New South Wales with BE (Hons.) and PhD. He is a senior member of the American Institute of Aeronautics and Astronautics; members of the American Society of Mechanical Engineers, Royal Institution of Naval Architects (UK), Institution of Engineers (Australia), Institution of Engineers (Singapore), Environmental Engineering Society of Singapore and Society of Automobile Engineering (USA). He is a Chartered Engineer (UK) and a Chartered Professional Engineer (Australia). His current research interest is in the areas of Computational Fluid Mechanics.

W. P. Li received his Bachelor's and Master's degree from Hua Zhong University of Science and Technology, Wuhan, China in 1982 and 1984 respectively. He graduated from Tianjin University with a PhD degree in 1988. His research interest comprises experimental and numerical methods in hydrodynamics.

L. S. Pan received his Bachelor and Master's degree from Hua Zhong University of Science and Technology, Wuhan China in 1982 and 1984. He graduated from the National University of Singapore with a PhD degree in 1977. His current research interests include LES and DNS computational methods and the use of Direct Simulation Monte Carlo (DSMC) methods on molecular dynamics.

Y. T. Ng: He has received his B.Eng (Hons.) from the National University of Singapore in 1998. He proceeded to graduate study and conducted research on bluff body flows. Using numerical simulation and different experimental flow visualisation techniques, he studied the flow structures in square cylinder flows, placed at different angles of incidence. His current research interest includes bluff body and high angle of attack aerodynamics and computational fluid dynamics.