Proceedings of COBEM 2005
18th International Congress of Mechanical Engineering
November 6-11, 2005, Ouro Preto, MG
Copyright © 2005 by ABCM

NUMERICAL SIMULATION OF AIRFOIL-VORTEX CLOUD INTERACTION IN GROUND EFFECT

Leandro Silva de Oliveira
Instituto de Engenharia Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
leandrosiol@yahoo.com.br

Luiz Antonio Alcântara Pereira
Instituto de Engenharia Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
luizantp@unifei.edu.br

Miguel Hiroo Hirata
FAT/UERJ, Campus Regional de Resende, Estrada Resende-Riachuelo, Resende, Rio de Janeiro, Brasil
hirata@fat.uerj.br

Abstract. *The discrete vortex method is applied to simulate the unsteady, two-dimensional, incompressible flow that occurs during an airfoil-vortex cloud interaction in the vicinity of a ground plane. In the numerical simulation a cloud of Lamb vortices is used in order to simulate the vorticity generated in the surface of airfoil and that generated in the ground surface. A new cloud of Lamb vortices is also generated next to the airfoil in presence of a ground plane. The dynamics of the wakes is computed using the convection-diffusion splitting algorithm, where the convection process is carried out with a Lagrangian second-order Adams-Bashforth time-marching scheme, and the diffusion process is simulated using the random walk scheme. The aerodynamics loads are computed using the integral equation derived from the pressure Poisson equation.*

Keywords*: vortex methods, aerodynamics loads, airfoil-vortex cloud interaction, ground effect*

1. Introduction

In many cases of engineering practices, objects often appear in the form of groups (or multiple bodies), e.g. groups of buildings, chimneys, stacks, chemical reaction-towers, supports of off-shore platform and blades in turbomachines.

The main goal of the research line, where this paper is inserted, consists of the analysis of the flow in a turbomachines. This kind of flow is characterized by the interaction of the vortices generated by a rotor blade with the vortices generated by another blade of the same rotor. The viscous wake thus created interacts strongly with the casing surface. It is also frequently found the situation where the rotor blades are immersed in a viscous wake – from now on referred to as a shear flow - generated by an upstream rotor (stator).

A cascade of identical airfoils represents a simple model, suitable for the analysis of the main characteristics of this flow. Many researchers using the potential flow theory have exploited this model. Recently, Alcântara Pereira *et al.* (2004) extended the analysis including the viscous effects. In their work they used the Vortex Methods including an additional feature, the turbulence modeling, see Alcântara Pereira *et al.* (2002).

The above mentioned model considers the interaction of the flow generated by neighboring blades but does not take into account the presence of the casing surface; neither the oncoming vortex wake generated by an upstream stator our rotor. To analyze the influence of these aspects another simple model can be derived. This time one considers a single airfoil, near a ground plane, immersed in an upstream shear flow; this model is referred to as "the airfoil-vortex interaction in ground effect" – AVIG – and can be viewed as a combination of three interacting flows: airfoil-vortex interaction (AVI), airfoil-ground interaction (AGI), and vortex-ground interaction (VGI). A large number of papers on the unsteady, incompressible, two-dimensional AVI flow have been published. Within the context of the (two-dimensional) parallel AVI that occurs around helicopter rotors, known as blade-vortex interaction (BVI), Panaras (1987), Poling *et al.* (1989) and Lee and Smith (1991), among others, have devised numerical models based on the inviscid discrete vortex method coupled with linearized potential flow theory. More elaborate numerical models have also been employed, such as those based on Euler (Srinivasan and McCroskey, 1993) and Navier-Stokes (Rai, 1987) mesh-based methods. Detailed experimental investigations on the aerodynamics of parallel BVI have been performed by Seath *et al.* (1989), Straus *et al.* (1990), Chen and Chang (1997). See the review articles of McCune and Tavares (1993) and Mook and Dong (1994) for additional references on unsteady, incompressible flows over airfoils and the numerical simulation of wakes and BVI.

Chacaltana *et al.* (1995) analyze the flow around a thin airfoil immersed in a shear flow, in presence of a ground plane. The authors use the potential flow theory and taking into account the fact that the airfoil is thin were able to derive a simple algorithm. In this paper, the shear flow was simulated by a single moving free vortex.

Fonseca *et al.* (1997, 2003) in a series of two papers applied a numerical, inviscid, vortex method to simulate the unsteady, two-dimensional and incompressible flow that occurs during a parallel blade-vortex interaction in ground effect. A panel method was used to discretize the airfoil bound vorticity, where each panel has a linear and piecewisecontinuos distribution of vorticity. The impermeability condition was enforced on the airfoil contour, but the no-slip condition is not. The Kutta condition was imposed through the continuity of the pressure field at the airfoil trailing edge, which, combined with the condition that the circulation in the whole flow must be conserved, provides a model for the vorticity generation at the trailing edge. Thus the viscous wake was modeled by potential vortices shed into the flow at the trailing edge and the oncoming shear flow was modeled by a single potential vortex that interacts with the airfoil and its wake.

Ricci *et al.* (2001) presented a new methodology that utilizes the Vortex Method for the analysis of the vorticity generated in the surface of the airfoil with that generated on the ground plane. Lamb vortices are generated along the airfoil surface and ground plane to ensure that the no-slip condition is satisfied. Images clouds are provided in the lower half ground to ensure that the impermeability condition is satisfied. With the images clouds the computation becomes expensive. This is a major source of difficulties, and it can only be handled through the utilization of method of distributed singularities, the Panel Method.

In the present paper, the Vortex Method is employed to simulate the airfoil-vortex cloud interaction in ground effect, see Fig. 1. The no-slip condition is satisfied using Lamb vortices to simulate the vorticity generated in the airfoil surface and that generated in the ground surface. The impermeability condition is imposed through the application of a source panel method. The main feature of the paper is the oncoming shear flow that has two important characteristics. The first is that the shear flow is continuously generated is a plane perpendicular to the main flow, a feature not found in any of the previous paper, and the second characteristic is the possibility of having a time variation of the vorticity carried out by this shearing flow, see Silva de Oliveira *et al.* (2004). It is believed that with these two features the shear flow due to the upstream rotor could be simulated in a more realistic way; after all, the rotor always moves in relation to the upstream rotor or stator.

To simulate the oncoming shear flow, a cloud of free Lamb vortices is generated next to the body. For this one can imagine a row of vortex generating points that are aligned in a previously fixed position. This row of vortices moves in a direction perpendicular to the main flow at a pre-set velocity V, see Fig. 1.

As mentioned, the model is analyzed using the Vortex Method which is a meshless numerical method or a particle method. In this method, the vorticity in the fluid region is numerically simulated using a cloud of discrete vortices with a viscous core (Lamb vortex). To simulate the vorticity at the solid surfaces, nascent vortices are generated there at each time step of the simulation. In order to take care of the convection and the diffusion of the vorticity one makes use of the convection-diffusion splitting algorithm; accordingly the convection of the vortices in the cloud is carried out independently of the diffusion for each time step of the simulation. The convection process is carried out with the Adams-Bashforth time-marching scheme and the diffusion process is simulated using the random walk method. This is in essence the foundation of the Vortex Methods (e.g. references Chorin, 1973; Sarpkaya, 1989; Sethian, 1991; Lewis, 1999; Ogami, 2001; Alcântara Pereira *et al*., 2002 and Kamemoto, 2004). Please note that with the lagrangian formulation a grid for the spatial discretization of the fluid region is not necessary. Thus, special care to handle numerical instabilities associated to high Reynolds numbers is not needed. Also, the attention is only focused on the regions of high activities, which are the regions containing vorticity; on the contrary, Eulerian schemes consider the entire domain independent of the fact that there are sub-regions where less important, if any, flow activity can be found. With the Lagrangian tracking of the vortices, one need not take into account the far away boundary conditions. This is of important in the wake regions (which is not negligible in the flows of present interest) where turbulence activities are intense and unknown, a priori.

The present Vortex Method has been used to simulate the macro scale phenomena, therefore the smaller scale ones are taken into account through the use of a second order velocity function (Alcântara Pereira et *al*., 2002). In this approach, the effect of small scale is not considered.

2. Formulation of the physical problem

Consider the incompressible flow of a newtonian fluid around an airfoil in an unbounded two-dimensional region in ground effect. Figure 1 shows the incident flow, defined by free stream speed U and the domain Ω with boundary $S = S_1 \cup S_2 \cup S_3$, S_1 being the body surface, S_2 being the ground plane and S_3 the far away boundary. A cloud of free vortex is also generated next to the body at row which moves in the vertical direction with constant speed V; all the discrete vortices have the same strength $\Delta\Gamma$ (positive or negative).

The viscous and incompressible flow is governed by the continuity and the Navier-Stokes equations, which can be written in the form

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u} \, . \tag{2}$$

In the equations above $\mathbf{u}$ is the velocity vector field and p is the pressure. As can be seen the equations are non-dimensionalized in terms of U and b (a reference length). The Reynolds number is defined by

$$\mathrm{Re} = \frac{bU}{\upsilon} \tag{2a}$$

where υ is the fluid kinematics viscosity coefficient; the dimensionless time is b / U .

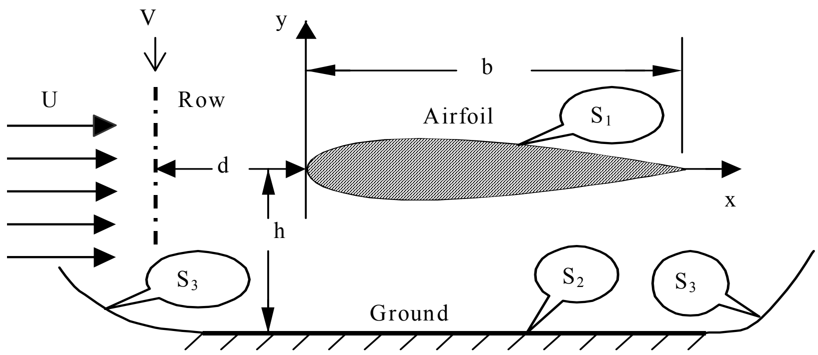


Figure 1. Definitions.

The impermeability and no-slip conditions on the body and ground plane surfaces are written as

$$u_n = \mathbf{u} \cdot \mathbf{e}_n = 0 \tag{3a}$$

$$u_\tau = \mathbf{u} \cdot \mathbf{e}_\tau = 0 \tag{3b}$$

$\mathbf{e}_n$ and $\mathbf{e}_\tau$ being, respectively, the unit normal and tangential vectors. One assumes that, far away, the perturbation caused by the body and ground plane fades as

$$|\mathbf{u}| \rightarrow 1 \text{ at } S_3 . \tag{3c}$$

The dynamics of the fluid motion, governed by the boundary-value problem (1), (2) and (3), can be alternatively studied by taking the curl of Eq. (2), obtaining the well-known 2-D vorticity transport equation

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \frac{1}{\mathrm{Re}} \nabla^2 \omega \tag{4}$$

in which ω is the only non-zero component of the vorticity vector.

3. Numerical simulation

According to the convection-diffusion splitting algorithm (Chorin, 1973) it is assumed that in the same time increment the convection and the diffusion of the vorticity can be independently handled and are governed by

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = 0 \tag{5}$$

$$\frac{\partial \omega}{\partial t} = \frac{1}{\mathrm{Re}} \nabla^2 \omega . \tag{6}$$

Convection is governed by Eq. (5) and the velocity field is given by

$$u - iv = 1 + \frac{1}{2\pi} \sum_{n=1}^{2M} \sigma(S_n) \int_{\Delta S_n} \frac{d}{dz} \ln(z - \zeta) d\zeta + \frac{i}{2\pi} \sum_{k=1}^{N} \frac{\Delta \Gamma_k}{z - z_k} . \tag{7}$$

Here, u and v are the x and y components of the velocity vector $\mathbf{u}$ and $i = \sqrt{-1}$. The first term in the right hand sides is the contribution of the incident flow; the summation of $2M$ integral terms comes from the sources panels with constant density distributed on the airfoil and ground surfaces. The second summation is associated to the velocity induced by the cloud of N free vortices; it represents the vortex-vortex interactions.

The process of vorticity generation is carried out from Eq. (3b), so as to satisfy the no-slip condition. According to the discussion above the panels method guaranties that the impermeability condition is satisfied in each straight-line element, or panel, at pivotal point. At each instant of the time M new vortices are created a small distance ε of the body and ground plane surfaces, whose strengths are determined from Eq. (3b) applied at M points right below the newly created vortices, along the radial direction. This procedure yields an algebraic system of M equations and M unknowns (the strengths of the vortices).

In order to remove the singularity in the second summation of Eq. (7) Lamb vortices are used, whose mathematical expression for the induced velocity of the kth vortex with strength $\Delta\Gamma_k$ in the circumferential direction u_{θ_k} , is (Mustto *et al.*, 1998)

$$u_{\theta_k} = \frac{\Delta\Gamma_k}{2\pi r}\left\{1 - \exp\left[-5.02572\left(-\frac{r}{\sigma_0}\right)^2\right]\right\} \tag{8}$$

where σ_0 is core radius of the Lamb vortex.

In this particular equation r is the radial distance between the vortex center and the point in the flow field where the induced velocity is calculated.

Each vortex particle distributed in the flow field is followed during numerical simulation according to the Adams-Bashforth second-order formula (Ferziger, 1981)

$$z(t+\Delta t) = z(t) + [1.5u(t) - 0.5u(t-\Delta t)]\Delta t + \xi \tag{9}$$

in which z is the position of a particle, Δt is the time increment and ξ is the random walk displacement. According to Lewis (1991), the random walk displacement is given by

$$\xi = \sqrt{4\beta\Delta t \ln\left(\frac{1}{P}\right)}\left[\cos(2\pi Q) + i\sin(2\pi Q)\right] \tag{10}$$

where $\beta = Re^{-1}$; P and Q are random numbers between 0.0 and 1.0.

The pressure calculation starts with the Bernoulli function, defined by Uhlman (1992) as

$$Y = p + \frac{u^2}{2},\ u = |\mathbf{u}| . \tag{11}$$

Kamemoto (1993) used the same function and starting from the Navier-Stokes equations was able to write a Poisson equation for the pressure. This equation was solved using a finite difference scheme. Here the same Poisson equation was derived and its solution was obtained through the following integral formulation (Shintani and Akamatsu, 1994)

$$H\overline{Y_i} - \int_{S_1} \overline{Y}\nabla G_i \cdot \mathbf{e}_n dS = \iint_{\Omega} \nabla G_i \cdot (\mathbf{u}\times\omega) d\Omega - \frac{1}{Re}\int_{S_1} (\nabla G_i \times \omega)\cdot \mathbf{e}_n dS \tag{12}$$

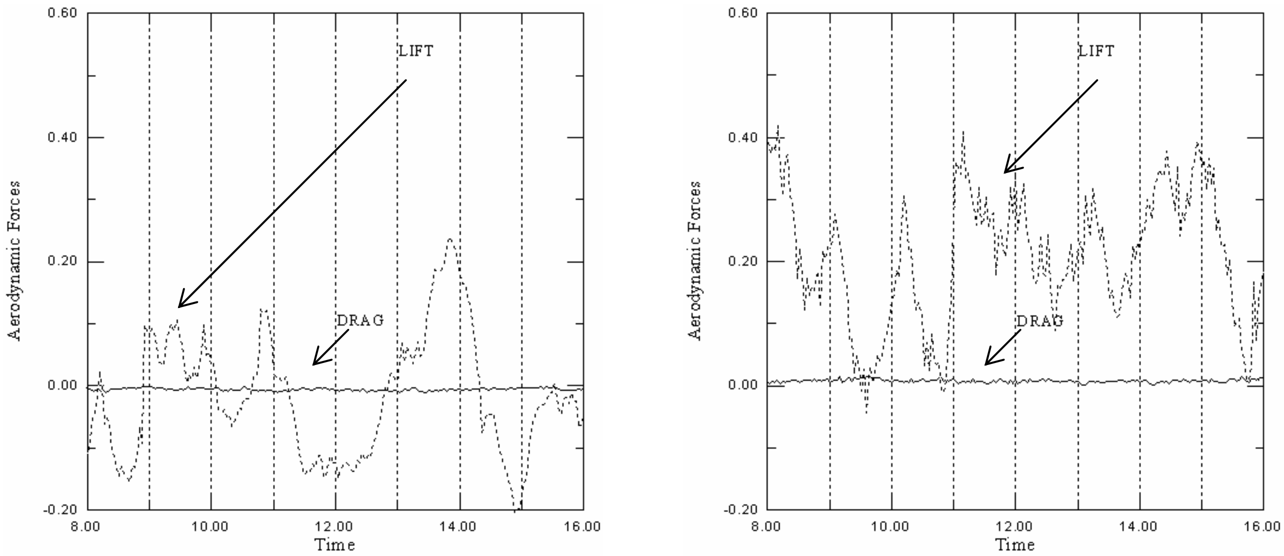
where H is 1.0 inside the flow (at domain Ω) and is 0.5 on the boundaries S_1 and S_2. $G_i = (1/2\pi)\log R^{-1}$ is the fundamental solution of Laplace equation, R being the distance from ith vortex element to the field point.

It is worth to observe that this formulation is specially suited for a Lagrangian scheme because it utilizes the velocity and vorticity field defined at the position of the vortices in the cloud. Therefore it does not require any additional calculation at mesh points. Numerically, Eq. (12) is solved by mean of a set of simultaneous equations for pressure Y_i.

4. Calculations results and discussions

The numerical simulations were restricted to the simple situation of a NACA 0012 profile at zero angle of attack relative to the freestream. In the calculations, each boundary S_1 and S_2 of Fig. 1 was represented by fifty (M=50) source panels, and the time step size and Reynolds number were taken as $\Delta t=0.04$ and $Re=10^6$ respectively. In each time step the nascent vortices were placed into the cloud through a displacement $\varepsilon=\sigma_0=0.0009b$ normal to the panels. The aerodynamics forces computations starts at t=8.

Using the solution technique described in the previous section, four representative cases will be analyzed. First, we remove the shear flow and simulate a NACA 0012 airfoil immersed in a uniform flow with height of the airfoil to the ground $h/b=\infty$. Figure 2(a) shows that the lift coefficient oscillates around zero. The differences encountered in the comparison of the numerical result with the expected result are attributed mainly the inherent three-dimensionality of the real flow for such a value of the Reynolds number, which is not modeled in the present simulation.



(a) computed without ground effect (h/b=∞)

(b) computed with ground effect (h/b=0.3)

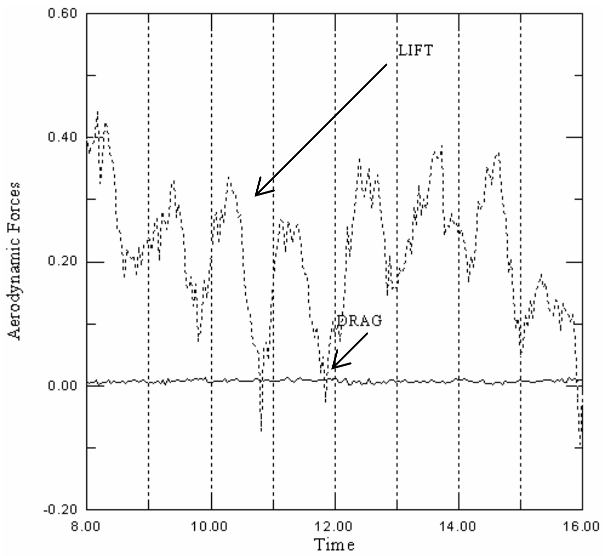
Figure 2. NACA 0012 airfoil: time variation of the lift coefficient for $Re=10^6$.

Second, we remove the shear flow and simulate a NACA 0012 airfoil immersed in a uniform flow to take into account the effect of the presence of the ground plane. The result from potential flow theory shows a decrease in the normalised lift coefficient as h/b approaches its small value. When the viscous effects are included in the present simulation, the suction effect caused by the ground plane (from potential flow theory) decreases and as a consequence the lift coefficient increase. The suction effect becomes weak because the leading edge stagnation point position changes and increases the lift coefficient. As shown in Fig 2(b) the lift coefficient is positive when the ground effect is simulated by vortex cloud modeling for h/b=0.3. News simulations are necessary to investigate the influence of ground height and angle of attack on the displacement of stagnation point.

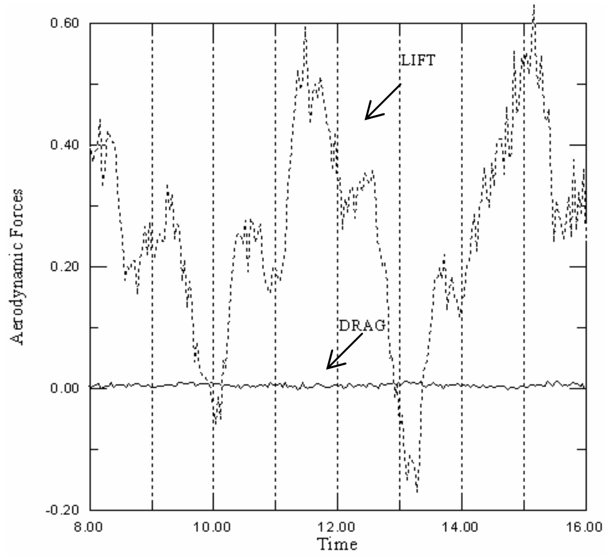
The interaction of a NACA 0012 airfoil with a shear flow was simulated by a second cloud of free vortices (generated at a row of moving points) that passes over the airfoil. Each discrete Lamb vortex generated at row of moving points has dimensionless strength $\Delta\Gamma=+0.001$. The row with length 0.6b and V=0.065U was located at distance d=3b, see Fig 1. The distance between two points of the row was equal to 0.09b. To avoid the influence of numerical transients, we start the simulation with the airfoil in an uniform flow without shear flow and we insert the shear flow only after the numerical simulation has been established (t=8).

Figure 3(a) shows the time development of the aerodynamics forces for an airfoil at zero incidence but immersed in a shear flow (simulated with a cloud of positive vortices, $\Delta\Gamma=+0.001$). We selected five dimensionless time positions in Fig. 3(a) to analyze the airfoil-vortex cloud interaction in ground effect. At t=8 the shear flow is inserted in an uniform stream. The cloud of free vortices generated at row of moving points follows a straight line path until t=10 when interacts with ground. From Fig. 2(b) at t=10 we can clearly observe that the lift coefficient increases; see in Fig. 3(a). When the front of vortex cloud (generated at row of moving points) is close to the airfoil leading edge (from t=11 on, approximately), the interaction with the body in ground effect with the vortices generated (boundary layer) is more active. At t=12 the interaction with the airfoil wake is strong and finally at t=13 the front of vortex cloud (generated at row of moving points) can be already considered as a part of the airfoil wake. After this, one has a full interaction of the three vortex cloud and the simulation goes on. This picture becomes clearer observing the Fig. 4(a), which shows the three vortex cloud as they interact. For a longer numerical new features can be observed as the strong vortex structures detaches from the body neighborhood from time to time. We are still analyzing such a complex phenomena and do not have yet a clear picture.

Figure 3(b) shows the aerodynamics forces evolution for the flow around the same airfoil immersed in a shear flow, this time simulated with a cloud of negative vortices (ΔΓ=-0.001). Figure 4(b) shows the instantaneous wake for the time t=16.



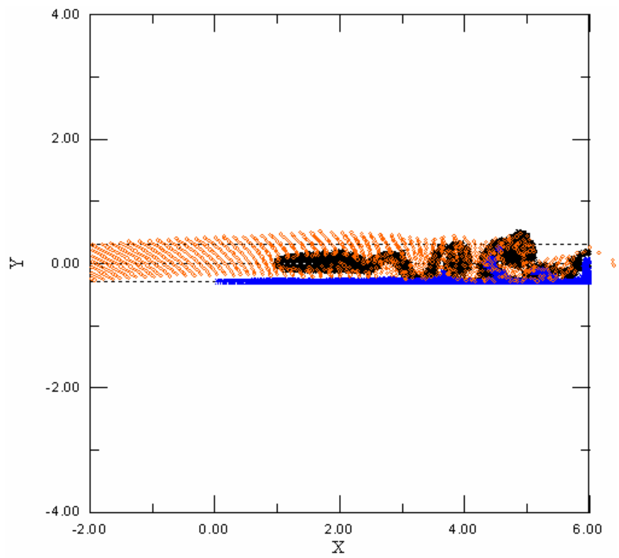
(a) computed with a cloud of positive vortices (ΔΓ=+0.001)



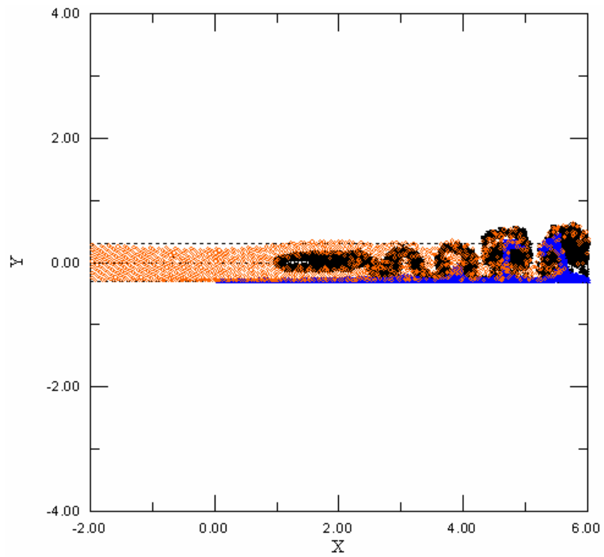
(b) computed with a cloud of negative vortices (ΔΓ=-0.001)

Figure 3. NACA 0012 airfoil in ground effect (h/b=0.3): time variation of the lift coefficient for Re=10^6.

It is also obvious that a denser vortex cloud is necessary to simulate the shearing flow; this will be an important point to take into consideration in future work.



(a) computed with a cloud of positive vortices (ΔΓ=+0.001)



(b) computed with a cloud of negative vortices (ΔΓ=-0.001)

Figure 4. NACA 0012 airfoil in ground effect (h/b=0.3): position of the wake vortices at t=16 for Re=10^6.

The comparison of the time evolution of the lift coefficient for a shear free oncoming flow, and the lift coefficient for the shearing flows shows some interesting features. While the front end of the shearing cloud is passing by the airfoil in presence of a ground plane (which can be considered as a transient regime in some sense) the effect on the lift due the interaction (see Fig. 3(a) and Fig. 3(b) between t=11 and t=12) decreases with a cloud of positive vortices and increases with a cloud of negative vortices. This can probably be explained by considering the change of leading edge stagnation point position. Once the front end reaches the wake, interactions which dramatically changes the lift behavior can be observed; with the positive shearing flow the lift is almost positive, while it an always a positive value for the negative shearing flow.

5. Conclusions

The main objective of the work with the implementation and initial test of a numerical vortex cloud model used to predict the aerodynamics of a two-dimensional airfoil-row wakes interaction in presence of a ground plane, has been achieved. The model employs a panel method with constant-strength source distribution to calculate the airfoil and ground plane contributions to the flow. Also a row which moves in the vertical direction with constant speed is employed to generate a third cloud of free vortices that interacts with the airfoil and ground wakes. The convective motion of each vortex is calculated using a time-marching scheme. The diffusion process of the vorticity is simulated using the random walk scheme. The flow over a NACA 0012 airfoil is considered to evaluate the integrated aerodynamics loads. The friction drag of the airfoil is not computed.

The distance between the airfoil and ground plane, the airfoil and the row, the row velocity, the distance between two vortices at row and the value of the vortex strength generated at row are extremely important in determining the degree of interaction. The influence of these parameters will be carried out.

The sub-grid turbulence modeling is of significant importance for the numerical simulation. The results of this analysis, taking into account the sub-grid turbulence modeling, are being generated and will be presented in due time, elsewhere.

From the present study, it is confirmed that the methodology used in the present study is convenient for investigation of unsteady characteristics of a two-dimensional airfoil-row wakes interaction in ground effect.

6. Acknowledgements

The authors would like to acknowledge FAPEMIG (Proc. TEC-748/04) and CNPq for the financial support during the time of this project.

7. References

Alcântara Pereira, L.A., Hirata, M.H. and Manzanares Filho, N. 2004, “Wake and Aerodynamics Loads in Multiple Bodies - Application to Turbomachinery Blade Rows”, J. Wind Eng. Ind Aerodyn., 92, pp. 477-491.

Alcântara Pereira, L.A., Ricci, J.E.R., Hirata, M.H. and Silveira-Neto, A., 2002, “Simulation of Vortex-Shedding Flow about a Circular Cylinder with Turbulence Modeling”, Intern’l Society of CFD, Vol. 11, No. 3, October, pp. 315-322.

Chacaltana, J.T.A., Bodstein, G.C. R. and Hirata, M.H., 1995, “Analytical Sutdy of the Time-Dependent 2D Inteaction of a Thin Airfoil and a Vortex in the Presence of a Ground Plane”, XIII Brazilian Congress of Mechanical Engineering - XIII COBEM - Belo Horizonte, MG, Brazil.

Chen, J.M. and Chang, D.M., 1997, “Unsteady Pressure Measuriments for Parallel Vortex-Airfoil Interaction at Low Speed”, Journal of Aircraft, Vol. 34, No. 3, pp. 330-336.

Chorin, A.J., 1973, “Numerical Study of Slightly Viscous Flow”, Journal of Fluid Mechanics, Vol. 57, pp. 785-796.

Ferziger, J.H. , 1981, “Numerical Methods for Engineering Application”, John Wiley & Sons, Inc.

Fonseca, G.F., Bodstein, G.C.R. and Hirata, M.H., 2003, “Numerical Simulation of Inviscid Incompressible Two-Dimensional Airfoil-Vortex Interaction in Ground Effect”, Journal of Aircraft, Vol. 40, No. 4, July-August, pp. 653-661.

Fonseca, G.F., Bodstein, G.C.R. and Hirata, M.H., 1997, “A Numerical Inviscid Vortex Model Applied to Parallel Blade-Vortex Interaction”, Journal of the Brazilian Society of Mechanical Sciences, Vol. 19, No. 3, pp. 341-356.

Kamemoto, K., 2004, “On Contribution of Advanced Vortex Element Methods Toward Virtual Reality of Unsteady Vortical Flows in the New Generation of CFD”, Proceedings of the 10th Brazilian Congress of Thermal Sciences and Engineering-ENCIT 2004, Rio de Janeiro, Brazil, Nov. 29-Dec. 03, Invited Lecture-CIT04-IL04.

Kamemoto, K., 1993, “Procedure to Estimate Unstead Pressure Distribution for Vortex Method” (In Japanese), Trans. Jpn. Soc. Mech. Eng., Vol. 59, No. 568 B, pp. 3708-3713.

Lee, D.J. and Smith, C.A., 1991, “Effect of Vortex Core Distortion on Blade-Vortex Interaction”, A.I.A.A. Journal, Vol. 29, No. 9, pp. 1355-1362.

Lewis, R.I., 1999, “Vortex Element Methods, The Most Natural Approach to Flow Simulation - A Review of Methodology with Applications”, Proceedings of 1st Int. Conference on Vortex Methods, Kobe, Nov. 4-5, pp. 1-15.

Lewis, R. I., 1991, “Vortex Element Method for Fluid Dynamic Analysis of Engineering Systems”, Cambridge Univ. Press, Cambridge, England, U.K..

McCune, J.E. and Tavares, T.S., 1993, “Perspective: Unsteady Wing Theory - The Karman/Sears Legacy”, Journal of Fluids Engineering, Vol. 115, Dec., pp. 548-560.

Mook, D.T. and Dong, B., 1994, "Perspective: Numerical Simulations of Wakes and Blade-Vortex Interaction", Journal of Fluids Engineering, Vol. 116, March, pp. 5-21.

Mustto, A.A., Hirata, M.H. and Bodstein, G.C.R., 1998, "Discrete Vortex Method Simulation of the Flow Around a Circular Cylinder with and without Rotation", A.I.A.A. Paper 98-2409, Proceedings of the 16th A.I.A.A. Applied Aerodynamics Conference, Albuquerque, NM, USA, June.

Ogami, Y., 2001, "Simulation of Heat-Fluid Motion by the Vortex Method", J.S.M.E. International Journal, Series B, Vol. 44, No. 4, pp. 513-519.

Panaras, A.G., 1987, "Numerical Modeling of the Vortex/Airfoil Interaction", A.I.A.A. Journal, Vol. 25, No. 1, pp. 5-11.

Poling, D.R., Dadone, L. and Telionis, D.P., 1989, "Blade-Vortex Interaction", A.I.A.A. Journal, Vol. 27, No. 6, pp. 694-699.

Rai, M.M., 1987, "Navier-Stokes Simulations of Blade-Vortex Interaction Using High-Order Accurate Upwind Schemes", A.I.A.A. Paper 87-0543, November.

Ricci, J.E.R., Alcântara Pereira, L.A., Hirata, M.H. and Manzanares Filho, N., 2001, "Influence of a Ground Plane in the Aerodynamic Loads on an Airfoil" (In Portuguese), Proceedings of COBEM 2001, Fluid Mechanics, Vol. 8, pp. 130-138. Uberlândia, MG.

Sarpkaya, T., 1989, "Computational Methods with Vortices - The 1988 Freeman Scholar Lecture", Journal of Fluids Engineering, Vol. 111, pp. 5-52.

Seath, D.D., Kim, J.M. and Wilson, D.R., 1989, "Investigation of Parallel Blade-Vortex Interaction at Low Speed", Journal of Aircraft, Vol. 26, No. 4, pp. 328-333.

Sethian, J.I., 1991, "A Brief Overview of Vortex Method, Vortex Methods and Vortex Motion", SIAM. Philadelphia, pp. 1-32.

Shintani, M. and Akamatsu, T, 1994, "Investigation of Two Dimensional Discrete Vortex Method with Viscous Diffusion Model", Computational Fluid Dynamics Journal, Vol. 3, No. 2, pp. 237-254.

Silva de Oliveira, L., Alcântara Pereira, L.A., Hirata, M.H. and Manzanares Filho, N., 2004, "Numerical Modelling of the Airfoil-Vortex Cloud Interaction", Proceedings of the 10th Brazilian Congress of Thermal Sciences and Engineering - ENCIT 2004, Rio de Janeiro, Brazil, Nov. 29 - Dec. 03, Paper CIT04-0253.

Srinivasan, G.R. and McCroskey, W.J., 1993, "Euler Calculations of Unsteady Interaction of Advancing Rotor with a Line Vortex", A.I.A.A. Journal, Vol. 31, No. 9, pp. 1659-1666.

Straus, J., Renzoni, P. and Mayle, R.E., 1990, "Airfoil Pressure Measurement During a Blade Vortex Interaction and a Comparison with Theory", A.I.A.A. Journal, Vol. 28, No. 2, pp. 222-228.

Uhlman, J.S., 1992, "An Integral Equation Formulation of the Equation of an Incompressible Fluid", Naval Undersea Warfare Center, T.R. 10-086.

8. Responsibility notice

The authors are the only responsible for the printed material included in this paper.