Copyright © 2005 by ABCM

A PARTICLE METHOD FOR AIRCRAFT WAKE BEHAVIOR ANALYSIS NEAR THE GROUND USING TURBULENCE MODELING

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 purpose of this paper is to numerically simulate the aircraft wake during the landing and take off operations using a new mesh-free two-dimensional discrete vortex method associated to a method of distributed singularities, the Panel Method. A pair of (Lamb) free vortex cloud is used to represent the free vortices at the wing tips. Lamb vortices are also generated along the airport ground, whose strengths are determined to ensure that the no-slip condition is satisfied. The impermeability condition is imposed through the application of a source panel method. In order to keep the computational effort within a manageable range, the micro scale manifestations of the turbulence are taken into account using a second-order velocity structure function model which is adapted to the Lagrangian scheme. Numerical results showing the free vortex clouds trajectory and their interaction with the ground vortex cloud are presented, showing new vortex structures that result from this interaction.*

Keywords*: vortex methods, panels method, aircraft wake vortices, landing and take off operations, turbulence model*

1. Introduction

Due the increasing frequency of operations and the size of airplanes, the landing and take off operations are becoming critical in almost all congested airports. The airplane wake vortices have circulation levels that scale directly with the size and speed of the generating aircraft and persist as tightly wound rotating flows for significant distances (Zheng and Ash, 1996). The reduction of the elapsed time between landing and take off as well as almost simultaneous operations in parallel runaways are vital for the efficiency of the airport operations. Therefore, to avoid the flight of an aircraft in the wake of another one is the main concern in airport operation (Machol, 1993). To this observation one should mention Critchley and Foot (1991), “Accidents occur in subsequent operations, mainly in the 30 ~ 70 m range above ground level, when strong vorticity structures are interacting with the runaway ground”.

Because of the inherent difficulty in making measurements in the wakes of aircraft, computer codes have been developed and applied for analysis of behavior and predict vortex interaction. The analysis of the aircraft wake, near the ground, is the main concern of this paper.

The incompressible inviscid fluid flow model, set up in a plane perpendicular to the airport runaway, have been used in many previous works. According to this model, the trajectory of the two free vortices, initially located at the wing tips, separates as a result of the ground effect, but does not rebound (Lamb, 1932). Donaldson and Bilanin (1975) present a thoroughly literature survey up to 1975 and most of the results are based on the inviscid model.

Other phenomena are, however, observed due to the combined effect of the ground and the lateral winds. Viscous fluid flow models enable the simulation of the boundary layer, which develops on the ground surface and affects substantially the vorticity dynamic. In addition to the vortices rebound one can observe the deformation of the main structures as well as the development of secondary structures (Dee and Nicholas, 1968; Barker and Crow, 1977; Liu and Srnsky 1990).

Using the viscous model, Zheng and Ash (1996) present an analysis of the influence of the Reynolds number and the atmospheric effects on the wake development near the ground. A matched asymptotic expansion technique is used to initialize the vortex flow system, prior to the finite difference numerical simulations. The prediction of the vortex trajectories is in good agreement with the experimental results and the vorticity contours show clearly the secondary structures; the influence of the Reynolds number on the vortex rebound trajectories is presented. Doligalski *et al.* (1994) present an analysis of the interactions that occur between the primary vortical structures with the ground boundary layer; in their analysis the boundary layer equations are used which does not allow the flow simulation beyond the separation points.

Ricci *et al.* (2003) used an entirely different approach to analyze the wake interactions with the ground. Initially a pair of discrete vortex is used to simulate the free vortices from the wing tips; the time evolution of the vortices is followed in a Lagrangian description (the Vortex Method, e.g. references Chorin, 1973; Sarpakaya, 1989; Sethian, 1991; Lewis, 1999; Kamemoto, 2004) as they interact with the nascent vortices near the ground – the ground vortex cloud (Hirata *et al.*, 2002). As the pair of vortices separates and rebound, due to the ground effect, one can observe the change in the primary vortical structure as well as secondary structures that appear in the flow, near the ground. A pair of single discrete vortices, as opposed to a pair of vortex clouds, was initially utilized inasmuch as it allows one to easily follow their trajectories. However, they are too restrictive with respect to the deformation of the vorticity structures as can also be seen in their work. A pair of vortex cloud is then used instead. The vortex method is used to simulate the macro scale phenomena and the smaller scale ones are taken into account through the use of a second order velocity function (Alcântara Pereira *et al.*, 2002). Lamb vortices are generated along the 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 Panels Method.

In the present paper, the Vortex Method with turbulence modeling (Alcântara Pereira *et al.*, 2002) is employed to simulate the airplane wake. The impermeability condition is imposed through the application of a source panel method and the no-slip condition is satisfied using discrete Lamb vortices to simulate the vorticity generated in the ground plane surface.

2. Mathematical formulation

The problem to be considered is that of aircraft wake interactions with the ground during the landing and takes off operations. The free vortices, starting at the wing tips, are defined by $\Gamma = \pm W/(\rho b U_a)$, where W is the aircraft weight, b is the wingspan and U_a is the approaching velocity. The main quantities of the model used to simulate the phenomenon are illustrated in Fig. 1. The domain of interest is defined by boundary $S = S_1 \cup S_2 \cup S_3 \cup S_4$; S_1 being the airport runway, with roughness ε_1, S_2 and S_3 being the runway side ground, with roughness ε_2 and ε_3 and S_4 the far away boundary.

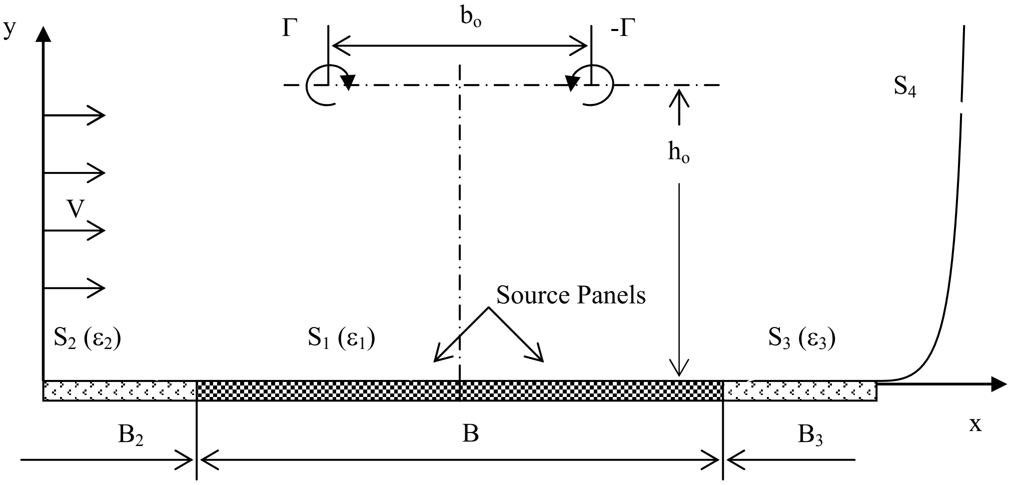


Figure 1. Schematic of wake vortex representation and associated coordinate system.

As the Newtonian fluid flow is supposed to be incompressible the governing equations are (Alcântara Pereira *et al.*, 2002)

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \text{ , i=1,2} \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[(\upsilon + \upsilon_t) \overline{S}_{ij} \right] \tag{2}$$

where the summation convention applies. The above governing was filtered ($u_i = \overline{u_i} + u_i'$, u_i' denotes the fluctuation field), ν is the molecular viscosity, ν_t is the eddy-viscosity of the fluid, ρ is the fluid density, $\overline{S}_{ij}$ is the deformation tensor of the filtered field and p is the pressure.

The large structures are governed by Eq. (2) and the eddy-viscosity assumption (Boussinesq's hypothesis) is used to model the sub-grid scale tensor $T_{ij} = \overline{u_i u_j} - \overline{u_i}\;\overline{u_j}$ (Smagorinsky, 1963).

The following boundary conditions apply

$u_n = u_2 = 0$, impermeability on y = 0 (3)

$u_\tau = u_1 = 0$, no-slip on y = 0 (4)

$|u| \to V$ at S4. (5)

In order to take into account the local activities of turbulence, Métais and Lesieur (1992) considered that the small scales may not be too far from isotropy and proposed to use the local kinetic-energy spectrum $E(k_c)$ at the cut-off wave number (k_c) to define the eddy-viscosity ν_t. Using a relation proposed by Batchelor (1967), the local spectrum at k_c is calculated with a local second-order velocity structure function $\overline{F_2}$ of the filtered field (Lesieur and Métais, 1996)

$$\overline{F_2}(x,\Delta,t) = \overline{\left\| \bar{u}(x,t) - \bar{u}(x+r,t) \right\|^2}_{\|r\|=\Delta} . \tag{6}$$

From the Kolmogorov spectrum the eddy-viscosity can be written as a function of $\overline{F_2}$

$$\upsilon_t(x,\Delta) = 0.104\, C_k^{-\frac{3}{2}} \Delta \sqrt{\overline{F_2}(x,\Delta,t)} \tag{7}$$

where C_k =1.4 is the Kolmogorov constant.

The great computational advantage of above formulation over Smagorinsky (1963) model is that in Eq. (6) the notion of velocity fluctuations (diferences of velocity) is used instead of the rate of deformation (derivatives). The velocities $u(x+r)$ are calculated over the surface of a sphere of radiusΔ. In this paper this formulation is adapted for 2D problems and to take advantage of the Lagrangian scheme (Alcântara Pereira *et al.*, 2002). Therefore, for each vortex of the cloud, one has

$$\overline{F_2} = \frac{1}{N_V} \sum_{i=1}^{N_V} \left\| u(x) - u(x+r_i) \right\|_i^2 \left(\frac{\sigma_o}{r_i} \right)^{2/3} . \tag{8}$$

In Eq. (8), Nv is the number of discrete vortices of the cloud found in the region defined by distances $r_1 = 0.1\sigma_0$ and $r_2 = 2.0\sigma_0$ from the centre of the reference vortex. A correction $(\sigma_o/\mathbf{r}_i)^{2/3}$ is necessary due to the fact that the Nv vortices are not located at equal distance from the center of the reference vortex.

In the numerical simulation, consider a point vortex of the cloud, which is located at point L. The value of the velocity structure function $\overline{F_2}$ which measures the turbulence manifestations is statistically sound only if the neighborhood of L is sufficiently populated with other point vortices. After some numerical experiments with the flow around a circular cylinder, it was assumed that this happens if Nv / A > 100, where Nv is the number of point vortices in the region, of area A , defined by two circumferences centered in L and with radius $r_1 = 0.1\sigma_0$ and $r_2 = 2.0\sigma_0$.

The relations below will be of interest are respectively (Ricci *et al.*, 2003)

$$Re = \frac{\rho V B}{\mu} \text{ (Reynolds number)} \qquad Re_V = \frac{\rho \Gamma}{\mu} \text{ (vortex Reynolds)} \qquad Re = \frac{\rho V x}{\mu} \text{ (running Reynolds number).}$$

It is also worth to observe that the turbulence is essentially a 3-D phenomenon and yet one is modeling it using a 2-D approach; obviously it is then assumed 2-D turbulence. With this procedure one are still left with important turbulence aspects and the final results are also improved. The use of 2-D turbulence may explain some numerical results that depart from the experimental values.

3. The discrete vortex method with turbulence model

From Eq. (1) and Eq. (2) one can write the non-dimensional vorticity equation in two dimensions as

$$\frac{\partial \omega}{\partial t} + u \cdot \nabla \omega = \frac{1 + \upsilon_t^*}{Re} \nabla^2 \omega \tag{9}$$

where ω is the only component of the vorticity vector and

$$\nu_t^* = \frac{\nu_t}{\nu} . \tag{10}$$

The vorticity equation carries information about the convection and the diffusion of vorticity. For the numerical simulation, the viscous splitting algorithm, first proposed by Chorin (1973), says that, in each time step, these process are governed by

$$\frac{\partial \omega}{\partial t} + \mathrm{u} \cdot \nabla \omega = 0 \text{, convection process} \tag{11}$$

$$\frac{\partial \omega}{\partial t} = \frac{1+\upsilon_t^*}{Re} \nabla^2 \omega \text{, diffusion process.} \tag{12}$$

Convection is governed by Eq. (11) and the velocity field is given by

$$u - iv = u_{crosswind} + \frac{1}{2\pi} \sum_{n=1}^{M} \sigma(S_n) \int_{\Delta S_n} \frac{d}{dz} \ln(z-\zeta) d\zeta + \frac{i}{2\pi} \sum_{k=1}^{N} \frac{\Delta\Gamma_k}{z - z_k} . \tag{13}$$

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 side is the contribution of the crosswind; the summation of M integral terms comes from the sources panels distributed on the ground surface. 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. (4), 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 ground plane surfaces, whose strengths are determined from Eq. (4) 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. (13) 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 given by (Mustto *et al.*, 1998)

$$u_{\theta_k} = \frac{\Gamma_k}{2\pi r} \left[1 - \exp\left(-5.02572 \frac{r^2}{\sigma_o^2} \right) \right] \tag{14}$$

where σ_o is the radius of the vortex core, which is updated according to

$$\sigma_o = 4.48364 \sqrt{\frac{\Delta t (1+\upsilon_t^*)}{Re}} . \tag{15}$$

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

$$z(t+\Delta t) = z(t) + [1.5u(t) - 0.5u(t-\Delta t)]\Delta t + \xi \tag{16}$$

in which z is the position of a vortex particle, Δt is the time increment and ξ is the diffusion displacement.

It is important to observe that the viscous diffusion of vorticity, governed by Eq. (12), was taken care of by using the random walk method, a molecular (laminar) diffusion process (Lewis, 1999). In our approach the variation of the core radius is only performed locally where the flow is turbulent, that means an additional (turbulent) diffusion process.

4. Discussion and results

Aiming at assessing the method capability to predict the main quantitative features of the flow we chose one value of the Reynolds for all cases, that is, Re=75000. The airport runway with roughness ε_1, see Fig. 1, was represented by M=120 straight-line source panels with constant density. In each time step, a new free vortex is generated at a distance $\varepsilon=\sigma_o=0.001$ on a straight-line, passing by the control point, normal to the panels; these new free vortices are added to the vortex cloud. All runs were performed with 800 time steps of magnitude $\Delta t=0.05$. The time increment was evaluated according to $\Delta t=2\pi k/M$, $0<k\leq 1$ (Mustto *et a.l, 1998*).

According to Ricci *et al.* (2003), from the wing tips free vortices are emanating with intensity $\pm\Gamma$; these vortices are represented either as a pair of isolated Lamb vortices or a pair of free vortex cloud (each with 100 discrete Lamb vortices). The vortex clouds are first generated using a random walk procedure, which start with all the vortices concentrated at a single point and ends when the outermost vortex reaches $0.1b_o$ (Hirata *et al.*, 2002).

We first present the results for the trajectory of the pair of vortex cloud. In a previous paper Hirata *et al.* (2002) used two isolated vortices with intensity $\pm\Gamma$ to simulate the free vortices from wing tips; the trajectory of the isolated vortices is in good agreement with experimental results for short time simulations; however, for long time simulations they showed some divergence. In their paper the simulation with a pair of isolated vortices was repeated with broader ground strips at both side of the runway; the results did not show a measurable improvement from the previous one. Even with the inclusion of the turbulence modeling there were no significant improvements. However, when using vortex cloud as primary structures and taken into account the local activity of turbulence one could observe a significant improvement of their numerical results.

A comparison of the computed trajectory in this paper using vortex cloud as primary structures – each cloud was composed by 100 free vortices with a total intensity equal to the two isolated vortices – and taken into account the local activity of turbulence to the experimental results of Liu and Srnsky (1990) and the numerical results of Ricci *et al.* (2003) are shown in Fig. 2. As one can see, the agreement between the two numerical results is very good; one can clearly observe that the computed trajectory of the primary vortex structure does try to follow the experimental results, even for long time simulation.

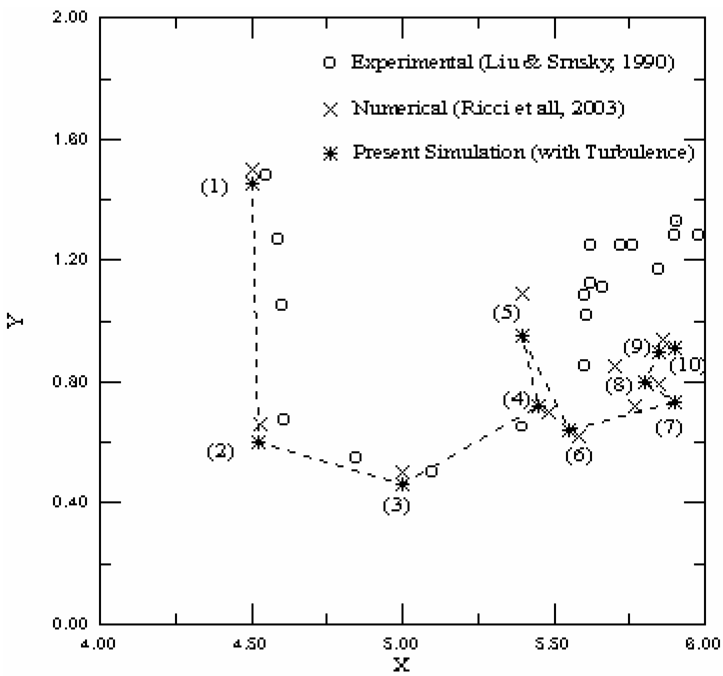
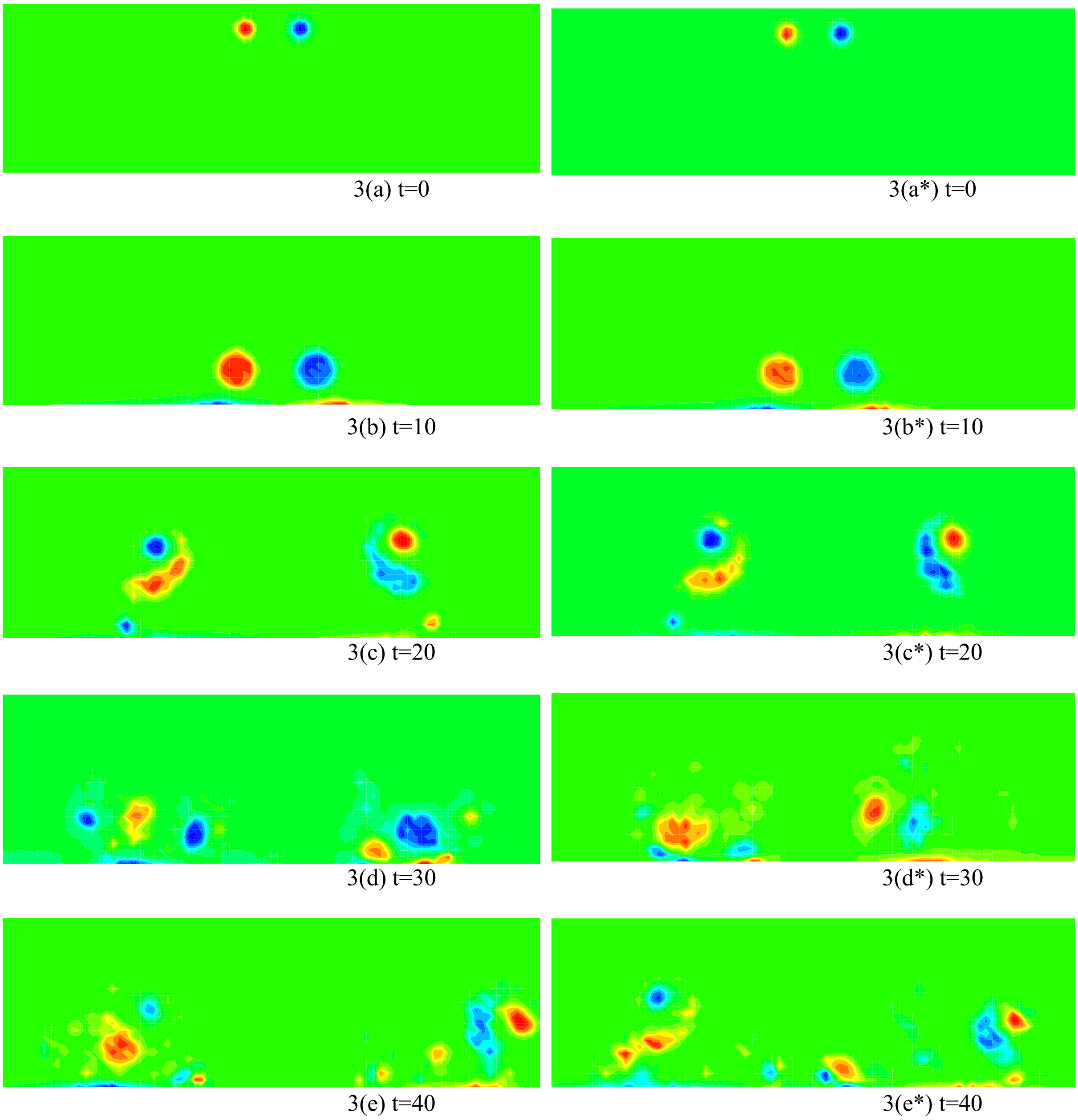


Figure 2. Comparison at Reynolds number Re=75000 of measured and computed vortex trajectories.

The sequence of pictures of Fig. 3 adds important information and shows details of the time evolution of the vortex structures.

The pictures from Fig. 3(a) to Fig. 3(e) shows the vorticity distribution without turbulence modeling, whereas in pictures from Fig. 3(a*) to Fig. 3(e*) the results refer to the simulation with turbulence modeling. The primary structures follow, as they are released, the same trajectory as the ones predicted by previous results – which were obtained using the potential flow theory, other numerical simulations and the Vortex Method – starting a downward motion until close to the ground, when they split moving toward $x\rightarrow\pm\infty$, this is the prediction of the potential model; see Ricci *et al.* (2003).

One observe in Fig. 3(b) and Fig. 3(b*) that the vorticity distribution behaviour is almost the same, both for the simulation with and without sub-grid turbulence modeling. However, when the pair of vortex cloud interacts with the ground, the turbulence model is more sensitive; compare the simulations illustrated from Fig. 3(c) to Fig. 3(e) with the simulations from Fig. 3(c*) to Fig. 3(e*). It is ease to see that at point (3) in Fig. 2 the free vortex cloud starts to strongly interact with the ground vortex cloud giving rise to a secondary vortex structure. Also, during strong ground coupling, it is possible that the vortex bending and stretching can produce important flow effects that will not be captured by the two-dimensional model. When compared to any type of inviscid approximation, the inclusion of ground coupling and the effects of turbulence are important features that can be gained within the present numerical simulation.



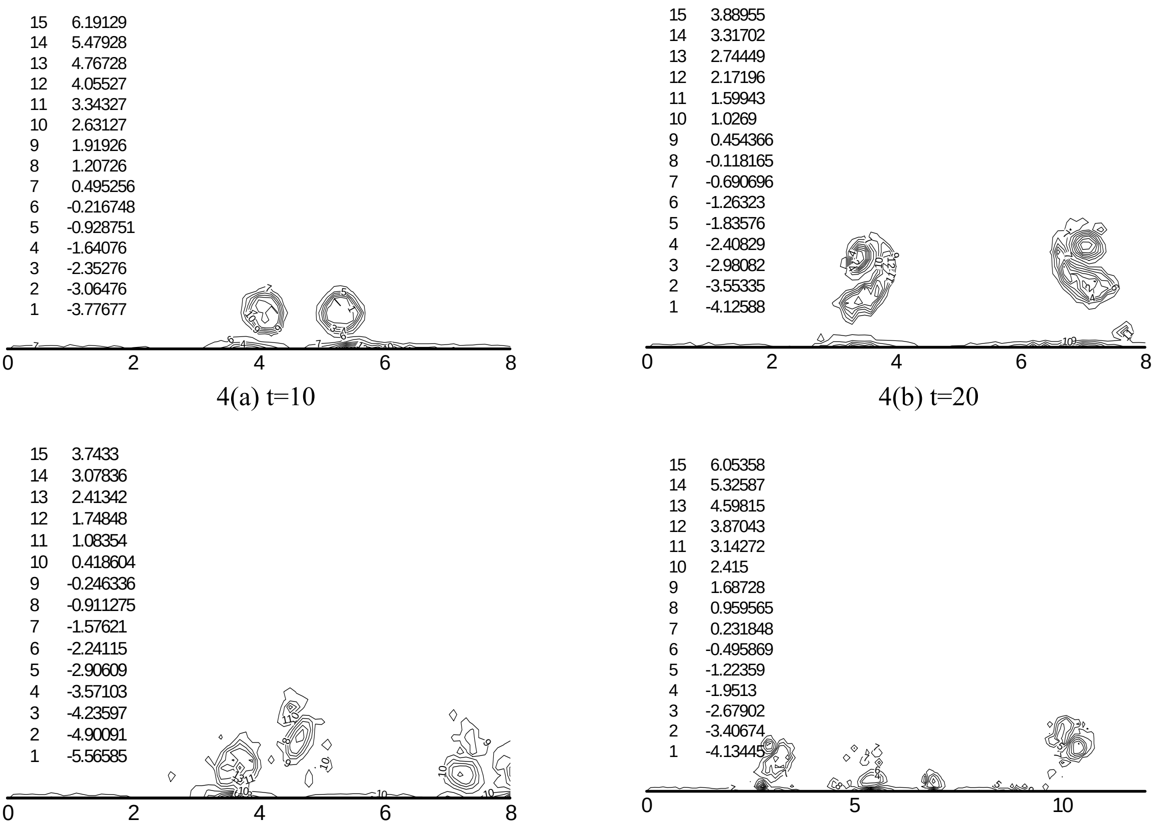
computed without turbulence model *computed with turbulence model

Figure 3. Vorticity distribution along the flow simulation at Re=75000.

From Fig. 3(c*), it is apparent that the vorticity distribution change radically between dimensionless times of 20 and 40, at Re=75000. Those effects can be interpreted if we examine the vorticity contour prots at t=30, as shown in Fig. 3(d) and Fig. 3(d*). There we see that at t=30, though the left secondary vortices with turbulence model are weaker than primary vortex, they influence its motion (as they revolve around the primary vortex).

One should observe, however, that, after the point where the secondary vortex structures are released from the ground, it is hard to identify the real trajectory of the primary structures in the sequence of experimental points, see Fig. 2; the numerical simulation enables one to follow this trajectory as shown in figure.

When crosswinds were considered for V=0.04, the pair of vortex cloud interacts no symmetrically with the ground, see Fig. 4(a). Figure 4(b) shows that the higher height achieved by primary structures at t=20 is equal to the initial (specified) pair of vortex cloud height. At t=30, see Fig. 4(c) the vorticity contours indicate that a secondaries vortices has been shed form the ground region, while others secondaries structures are beginning to form. At the same time, the secondaries structures reside directly above the primaries vortices and can strongly retard its lateral motion. Finally, at t=40, see Fig. 4(d), the right primary vortex is strongly interacting with a secondary vortex and is located at the runway side ground, with roughness ε_2 . In the present paper the simulation with broader ground strips at both sides of the runway is not computed.



4(a) t=10 4(b) t=20

4(c) t=30 4(b)t=40

Figure 4. Vorticity contour plot for a crosswind V=0.04 at Re=75000 computed with turbulence modeling.

5. Conclusions

A new mesh-free-two-dimensional discrete vortex method coupled with a source panel method is implemented to simulate of an aircraft wake during the landing and take off operations. A new implementation for the impermeability condition on the ground plane surface was tested with good results. The differences encountered in the comparison of the numerical results with the experimental results are attributed mainly to the inherent three-dimensionality of the real flow for such a value of the Reynolds number, which is not modeled in the simulation. To this observation one should mention Zheng and Ash (1996), “To use a two-dimensional model, it is important to identify some of the flow features that are neglected a priori. Specifically, the initial wake vortices include high-velocity axial jets (in the direction opposite to the line of flight, Zilliac *et al.*, 1993), and these axial jets that imply minimum pressure regions along the vortex axes probably burst immediately behind the aircraft, becoming low-speed axial wakes. There are rigorous theoretical links between the expansion of the vortex core regions and local axial pressure gradient, and those constraints are excluded automatically by a two-dimensional model. Also, during strong ground coupling, it is possible that vortex bending and stretching can produce important flow effects that will not captured by the two-dimensional model. When compared to any type of inviscid approximation, however, the inclusion of ground coupling and the ability to estimate decay rates are important features that can be gained within the two-dimensional approximations”.

New simulations will be carried out with more straight-line source panels and with broader ground strips at both side of the runway. Use of a larger number of panels distributed on the ground plane surface can also improve the results, but for this it is necessary a larger number of free vortices in the cloud and consequently a larger computational effort. It is worth to mention that the smaller scale analysis leading to the turbulence modeling is a necessary step for the study of the ground roughness; this is a subject under present investigation and to be presented elsewhere. Also, the buoyancy effects must be included as a future work. The atmospheric stratification is typically the only mechanism that can reverse the downward motion of an aircraft wake away from the ground. When the atmosphere is stably stratified, the ambient temperature increases with height. As a result, buoyant forces act on the pair of vortex cloud, reversing their descent.

From the present study, it is confirmed that the methodology used in the present study is convenient for investigations of unsteady characteristics of the flow.

6. Acknowledgements

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

7. References

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", Inter'l Society of CFD / Computational Fluid Dynamics Journal, Vol. 11, No. 3, pp. 315-322.

Batchelor, G.K. , 1967, "An Introduction to Fluid Dynamics", Cambridge University Press, Cambridge, UK.

Barker, S.J. and Crow, S.C., 1977, "The Motion of Two-Dimensional Vortex Pairs in a Ground Effect", JFM, Vol. 82, pp. 659-671.

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

Critchley, J. and Foot, P., 1991, "UK CAA Wake Vortex Database: Analysis of Incidents Reported Between 1982 and 1990", Civil Aviation Authority, CAA Paper 91.

Dee, F.S. and Nicholas , O.P., 1968, "Flight Measurement of Wing Tip Vortex Motion Near the Ground", CP 1605, British Aeronautical Research Council, London, England.

Doligalski, T.L., Smith, C.R. and Walker, J.D.A., 1994, "Vortex Interactions with Walls", Annual Review of Fluid Mechanics, Vol. 26, pp. 573-616.

Donaldson, C. D. and Bilanin, A., 1975, "Vortex Wakes of Conventional Aircraft", AGARDograph No.204, May.

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

Hirata, M.H., Ricci, J.E.R. and Oliveira, W., 2002, "Dynamic Behaviour of Aircraft Wake During Landing and Take Off Operations", Proc. 9th Brazilian Congress of Thermal Engineering and Sciences, Caxambu, MG, Brasil (In Portuguese).

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.

Lamb, H., 1932, "Hydrodynamics", Dover Publication, NY.

Lesieur, M. and Métais, O., 1996, "Spectral Large-Eddy Simulations of Isotropic and Stably-Stratified Turbulence", An. Rev. in Fluid Mech., Vol. 28, pp. 45-82.

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

Liu, H.T. and Srnsky, R.A., 1990, "Laboratory Investigation of Atmospheric Effects on Vortex Wakes", Flow Research, Inc., Report No. 497, Belleville, WA.

Métais, O. and Lesieur, M., 1992, "Spectral Large-Eddy Simulations of Isotropic and Stably-Stratified Turbulence", J. Fluid Mech., Vol. 239, pp. 157-194.

Machol, R., 1993, "Wake Vortices - A Primer", FAA Aviation Safety Journal, vol. 3, No. 1, pp. 16-19.

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.

Ricci, J.E.R., Alcântara Pereira, L.A., Hirata, M.H. and Oliveira, W., 2003, "Simulation of the Airplane Wake Using the Vortex Method with Turbulence Modeling", Proceedings of the 17th International Congress of Mechanical Engineering, November 10-14, São Paulo, Brazil, Paper COBEM2003-0887.

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

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

Smagorinsky, J., 1963, "General Circulation Experiments With the Primitive Equations", Mon. Weather Rev., vol. 91, pp. 99-164.

Zheng, Z.C. and Ash, R.L., 1996, "Study of Aircraft Wake Vortex Behavior Near the Ground", A.I.A.A. Journal, vol. 34, No.3, pp. 580-588.

Zilliac, G.C., Chow, J.S., Dacles-Mariani, J. and Bradshaw, P., 1993, "Turbulent Structures of a Wingtip Vortex in the Near Field", A.I.A.A. Paper 93-3011, July.

8. Responsibility notice

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