SIMULATION OF THE AIRPLANE WAKE USING THE VORTEX METHOD WITH TURBULENCE MODELING

José Eugênio Rios RICCI
Departamento de Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
luizantp@iem.efei.br

Luiz Antonio ALCÂNTARA PEREIRA
Departamento de Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
ricci@iem.efei.br

Miguel Hiroo HIRATA
Departamento de Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
hirata@iem.efei.br

Waldir de OLIVEIRA
Departamento de Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
waldir@iem.efei.br

Abstract. *This paper presents the results of the numerical simulation of an aircraft wake during the landing and take off operations. A pair of (Lamb) free vortex cloud is used to represent the free vortices at the wing tips. The time evolution of these clouds are followed in a typical Lagrangian fashion and is influenced by the present of the airport ground. The classical Vortex Method is properly modified to take into account the sub grid-scale phenomena; a Second-Order Velocity Function Model is adapted to the Lagrangian scheme to simulate the micro structures of the flow. 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. v*ortex methods, airplane wake, landing and take off operations, turbulence modeling, lagrangian description.*

1. Introduction

The ever-increasing airport traffic and aircraft size results in critical operations at almost all the important airports around the world. 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.

However the reduction of elapsed time between subsequent operations affects directly the safety of a landing (or take off) due to the aircraft wake that remains over the ground and that takes some time to dissipate or to be removed by lateral winds. 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 & 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”, and Zheng & Ash (1996), “The free vortices, leaving the wing tips, have an intensity proportional to the aircraft size and develops for considerable distances”.

The analysis of the aircraft wake, near the ground, is the main concern of this paper. To this end, incompressible inviscid fluid flow model, set up in a plane perpendicular to the airport runaway, have been used in many previous work. 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 & 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 well as the development of secondary structures (Dee & Nicholas, 1968; Barker & Crow, 1977; Liu & Srnsky 1990).

Using the viscous model, Zheng & 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 all (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.

In the present work an entirely different approach is used 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 fashion (the Vortex Methods, e.g. references Chorin, 1973; Lewis, 1991; Kamemoto, 1994) as they interact with the nascent vortices near the ground – the ground vortex cloud (Hirata et all, 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 this 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 all, 2002).

2. Formulation of the Physical Problem

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.

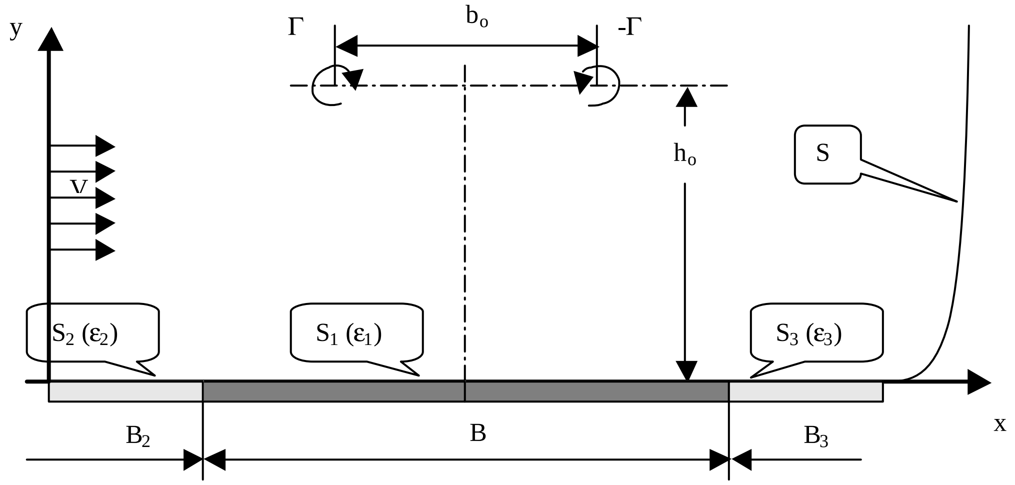


Figure 1. Definitons.

The runway breath is indicated by B and the initial positions of the center of the vortex clouds used to simulate the free vortex are defined by ($\pm$ 0.5b_o , h_o). V defines the lateral wind.

Let the fluid velocity be written as $\mathbf{u}$ ° $\{u_i\}$, i = 1,2. In order to separate the wave number scales an appropriately chosen low-pass filter, characterized by a function $\overline{G}$, is used. The filtered field is defined, for any quantity, as

$$\overline{f}(x,t) = \int_{\forall} f(x - y, t)\overline{G}(y)dy$$

According to Smagorinsky (1963)

$$\overline{u}(x,t) = u(x,t) * H_{\Delta x}$$

where $H_{\Delta x}$ is the filter and (∗) denotes convolution product. The turbulence scales must be split in the large scales and in the sub-grid scales

$$u = \overline{u} + u' \tag{1}$$

where $\overline{\mathbf{u}}$ is the filtered field or large scale field and $\mathbf{u}'$ is the fluctuation field or the sub-grid scales, which are smaller than Δx.

As the newtonian fluid flow is supposed to be incompressible the governing equations for the filtered field are (Lesieur, 1990)

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

$$\frac{\partial \overline{u_i}}{\partial t}+\frac{\partial}{\partial x_i}\left(\overline{u_i}\ \overline{u_j}\right)=-\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\upsilon\left(\frac{\partial \overline{u_i}}{\partial x_j}+\frac{\partial \overline{u_j}}{\partial x_i}\right)+T_{ij}\right],\quad i=j=1,2 \tag{3.a}$$

where the generalised sub-grid scale tensor is defined as

$$T_{ij}=\overline{u_i u_j}-\overline{u_i}\ \overline{u_j}$$

It is worth observing that Eq. (3) resembles the Reynolds equations for the mean flow, but the sub-grid scale tensor T_{ij} is different, as will be seeing below.

The following boundary conditions apply

$$u_n = u_2 = 0,\ \text{Impenetrability} \tag{4}$$

on y = 0

$$u_\tau = u_1 = 0,\ \text{no-slip} \tag{5}$$

$$|u| \rightarrow V\ \text{at}\ S_4 \tag{6}$$

and the relations below will be of interest are respectively

$$Re=\frac{\rho VB}{\mu}\ \text{(the Reynolds number)}\qquad Re_V=\frac{\rho\Gamma}{\mu}\ \text{(the vortex Reynolds)}\qquad Re=\frac{\rho Vx}{\mu}\ \text{(the running Reynolds number)}$$

3. Solution Procedure

As mentioned, the solution to the above-formulated problem is sought using a Lagrangian scheme. For that, a cloud of discrete Lamb vortices represents the vorticity in the domain of interest.

3.1. Free vortices from the wing tips

From the wing tips free vortices are emanating with intensity $\pm\Gamma$. In this paper 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 all, 2002).

The velocity induced by a Lamb vortex is (Mustto et all, 1998)

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

where σ_o is the radius of the vortex core, defined by

$$\sigma_o = 4.48364\sqrt{\frac{\Delta t}{Re}} \tag{7}$$

if turbulence is not accounted for.

3.2. The vortex method (Alcântara Pereira et all, 2002)

Now it is proposed to simulate numerically the large structures, represented by $\overline{u}$, and to use appropriate models to represent the small-scale effects. With the use of the eddy-viscosity assumption (Boussinesq’s hypothesis) to model the sub-grid scale tensor, the large structures are governed by

$$\frac{\partial \overline{u}_i}{\partial t}+\overline{u}_j\frac{\partial \overline{u}_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[(\upsilon+\upsilon_t).S_{ij}\right] \tag{3b}$$

where ν is the molecular viscosity and ν_t is the eddy-viscosity of the fluid. The deformation tensor for the filtered field is defined by

$$\overline{S}_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right)$$

From Eq. (2) and Eq. (3) one can write the non-dimensional vorticity equation in two dimensions as

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \frac{1 + \upsilon_t^*}{\mathrm{Re}} \nabla^2 \omega \tag{8}$$

where ω is the only component of the vorticity vector and

$$\nu_t^* = \frac{\nu_t}{\nu}$$

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

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

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

In order to satisfy impenetrability boundary condition, see Eq. (4), images clouds are provided in the lower half space.

In every time step of the numerical simulation, new vortices are placed near the ground surface; the strength of these vortices are such that the non slip boundary condition, see Eq. (5), is satisfied in a number of control points.

The Lagrangian solution to these equations is written using the second order Adams-Bashforth scheme as (Alcântara Pereira et all, 2002)

$$x_i(t + \Delta t) = x_i(t) + [1.5\, u_i(t) - 0.5\, u_i(t - \Delta t)]\Delta t + \xi_i(t) \text{ , } \quad i = 1,2$$

where ξ_i is the random displacement and the fluid velocity is written as the sum of the incident flow, ui_i , the body contribution, uc_i , and the vortex-vortex interaction, uv_i (Hirata et all, 2002).

$$u_i = ui_i + uc_i + uv_i$$

3.3. Turbulence modeling

In order to take into account the local activity of turbulence (Lesieur & Métais, 1996) considered that the small scales may not be too far from isotropy and proposed to use the local kinetic energy spectrum $E(k_c)$ to define the eddy viscosity as

$$\upsilon_t(\mathbf{x}, t) = \frac{2}{3} c_k^{-\frac{3}{2}} \left[\frac{E(k_c)}{k_c}\right]^{\frac{1}{2}}$$

where k_c is the cut-off wave number.

Using a relation proposed by Batchelor (1963) the local kinetic energy spectrum at k_c is calculated with a local Second Order Velocity Structure Function (Lesieur & Métais, 1996)

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

According to this formulation the velocities $\mathbf{u}(\mathbf{x} + \mathbf{r})$ are calculated over the surface of a sphere of radius Δ. In this paper this formulation is adapted for 2D problems and to take advantage of the Lagrangian scheme (Alcântara Pereira et all, 2002). Therefore, for each vortex of the cloud, one has

$$\overline{F_2} = \frac{1}{N_V}\sum_{i=1}^{N_V} \left\| u(x) - u(x + r_i) \right\|_i^2 \left(\frac{\sigma_o}{r_i}\right)^{2/3} \tag{9}$$

$$\left\| \mathbf{u}(\mathbf{x}) - \mathbf{u}(\mathbf{x} + \mathbf{r}_i) \right\|_i^2 = [u(\mathbf{x}) - u(\mathbf{x} + \mathbf{r}_i)]^2 + [v(\mathbf{x}) - v(\mathbf{x} + \mathbf{r}_i)]^2$$

where NV is the number of discrete vortices of the cloud found in the region defined by the distances (σ_0-ε) and (σ_0+ε) from the center of the reference vortex k. The velocities $u(\mathbf{x}+ \mathbf{r}_i)$ and $v(\mathbf{x}+ \mathbf{r}_i)$ are evaluated at the center of these NV vortices. A correction $(\sigma_o / r_i)^{2/3}$ 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 $r_2 = 2.0\sigma_0$.

Rendering for a Kolmogorov spectrum, it results on the eddy-viscosity as a function of $\overline{F_2}$

$$\upsilon_t(x,\Delta) = 0.104\, C_k^{-\frac{3}{2}} \Delta \sqrt{\overline{F_2}(x,\Delta,t)} \tag{10}$$

where $C_k = 1.4$ is the Kolmogorov constant.

Finally, for each time step of the simulation, the core radius, see Eq. (7), of each vortex of the cloud is updated according to

$$\sigma_o = 4.48364 \sqrt{\frac{\Delta t\left(1 + \upsilon_t^*\right)}{Re}} \tag{11}$$

The great computational advantage of this formulation over the Smagorinsky model, vis a vis the vortex method, is that in above formulation the notion of velocity fluctuations (differences of velocity) is used instead of the rate of deformation (derivatives).

4. Results and Discussion

In a previous paper Hirata et all (2002) used two isolated vortices with intensity $\pm\Gamma$ to simulate the free vortices from the 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 divergenge.

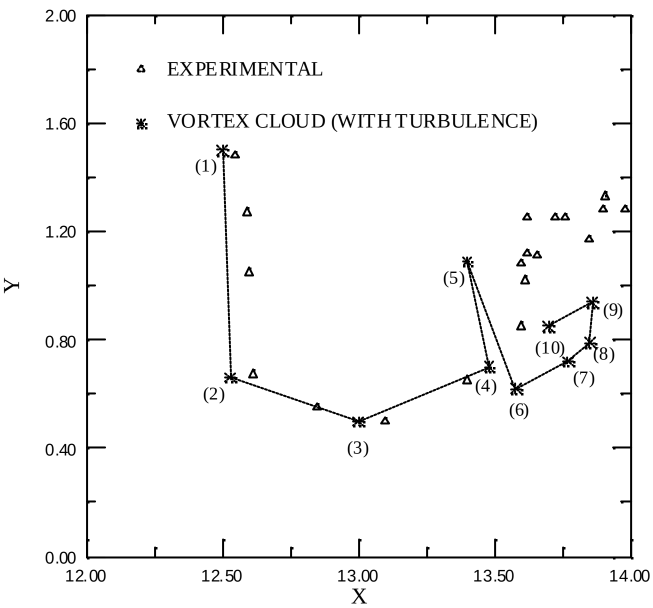
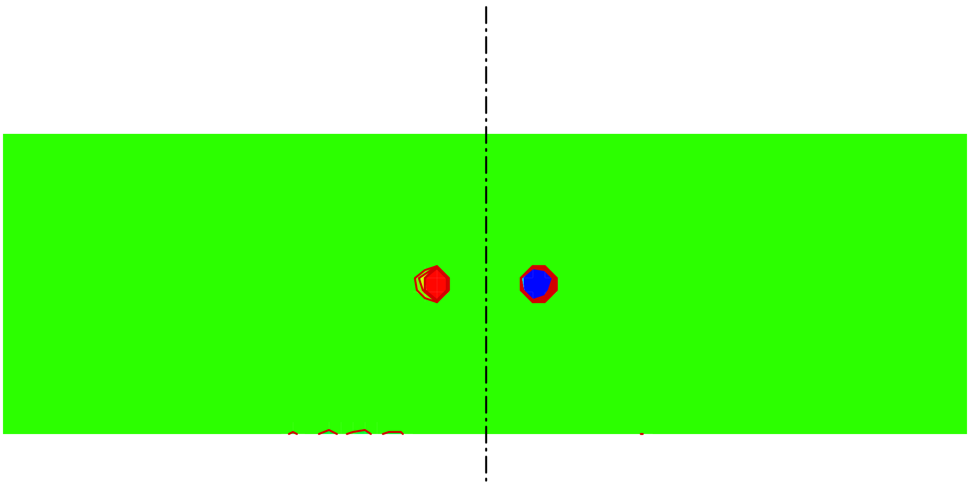


Figure 2. Trajectory of the free vorticity from wing tips

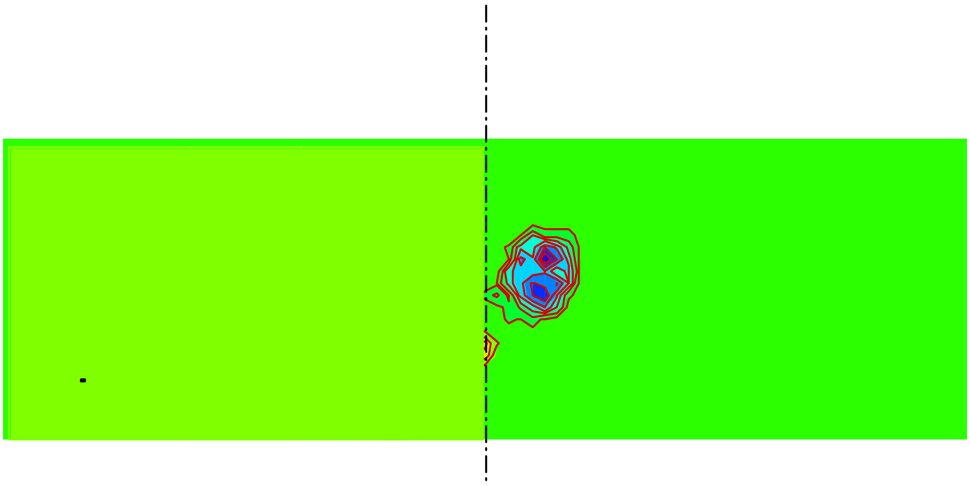
At that time the isolated vortices were substituted by a pair of vortex cloud, each cloud composed by 100 free vortices with a total intensity equal to the isolated vortices. The general picture of the flow were encouraging showing a better spatial distribution of the vorticity due to the fact that the vortex cloud structures are “less rigid” than the isolated vortex structures, thus having a better behavior when interacting with the new vortex structures- secondary vortex structures- that show-up near the ground.

In the present paper the same approach of Hirata et all (2002) was used. However to the vortex method algorithm, the turbulence modeling were added. The simulation with a pair of isolated vortices was repeated with broader ground strips at both side of the runaway; the results did not show a measurable improvement from the previous one. Even with the inclusion of the turbulence modeling there were no significant improvement. However, when using vortex cloud as primary structures and taken into account the local activity of turbulence one could observe a significant improvement of the numerical results as shown in Fig. 2. In this figure 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.

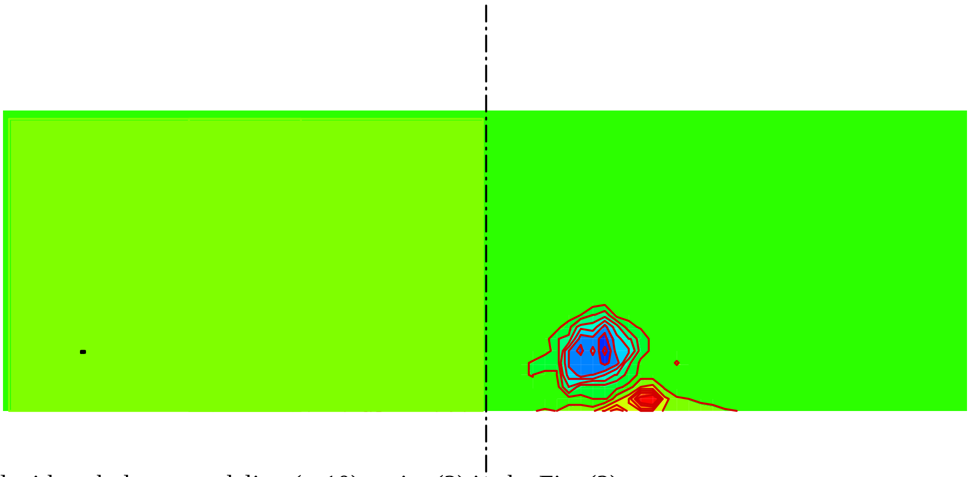
The sequence of pictures of Fig. (3) adds important information and shows details of the time evolution of the vortex structures. The primary structures follow, as they are released, the same trajectory as the ones predicted by previous results (which were obtained using the potential flow theory, other numerical simulations and the Vortex Method, see Hirata et all (2002)), 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 flow model. Previous numerical simulation, however, show an intense interaction with the ground boundary layer resulting in an upward motion of the primary vortex structures; soon after that point the simulation breaks down since flow separation occurs. Hirata et all (2002) using a Lagrangian description were able to simulate the flow beyond the separation point and the results from their simulation show great details of the primary vortex structures as well as the creation of new (secondary) vortex structures. In Fig. 3, one could also observe an interesting phenomenon that is identified in the calculated trajectory starting from point (4). The trajectory takes a steep upward direction until point (5), from which it follow a steep descent to point (6); from there on the trajectory more or less try to follows the experimental results. The sequence presented in the Fig 3 shows, for each point



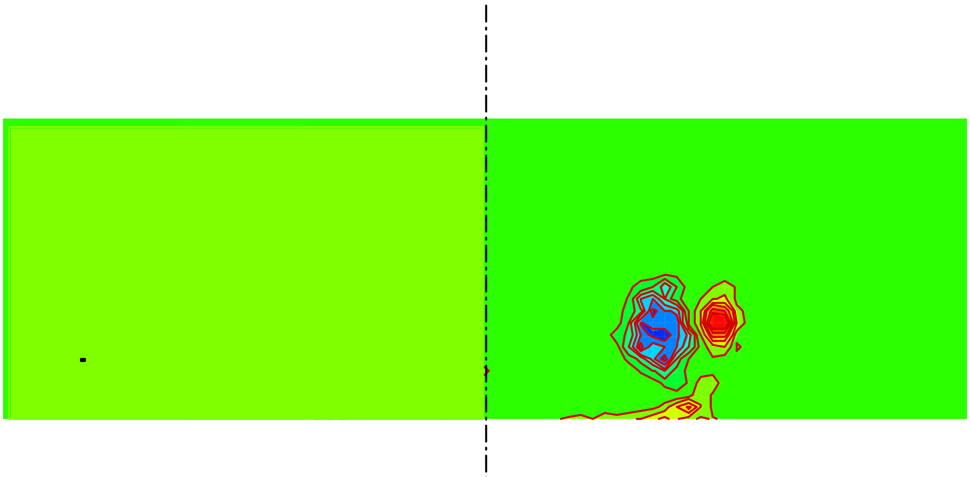
(a) vortex cloud with turbulence modeling (t=0)



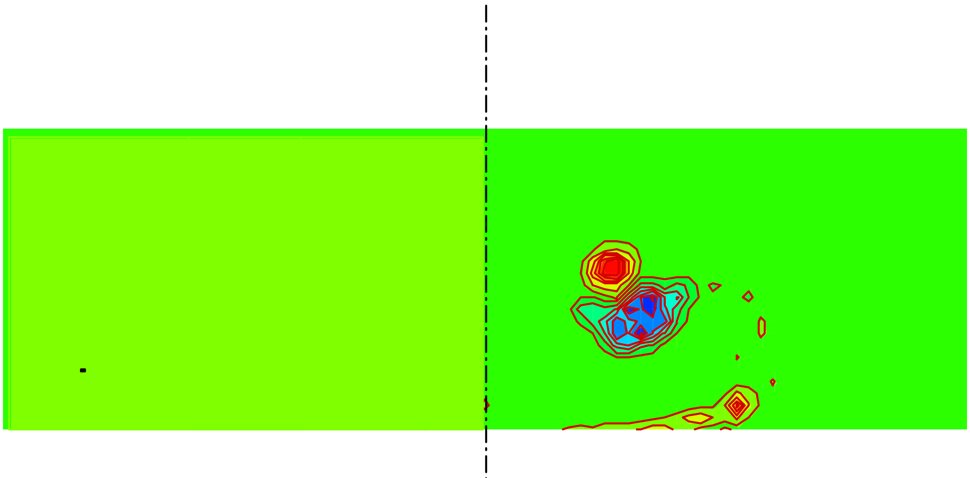
(b) vortex cloud with turbulence modeling (t=5): point (1) in the Fig. (2)



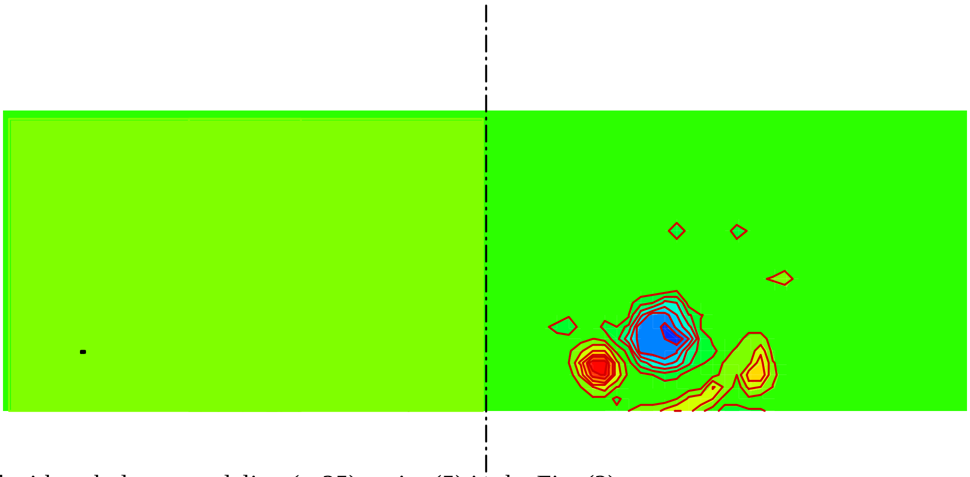
(c) vortex cloud with turbulence modeling (t=10): point (2) in the Fig. (2)



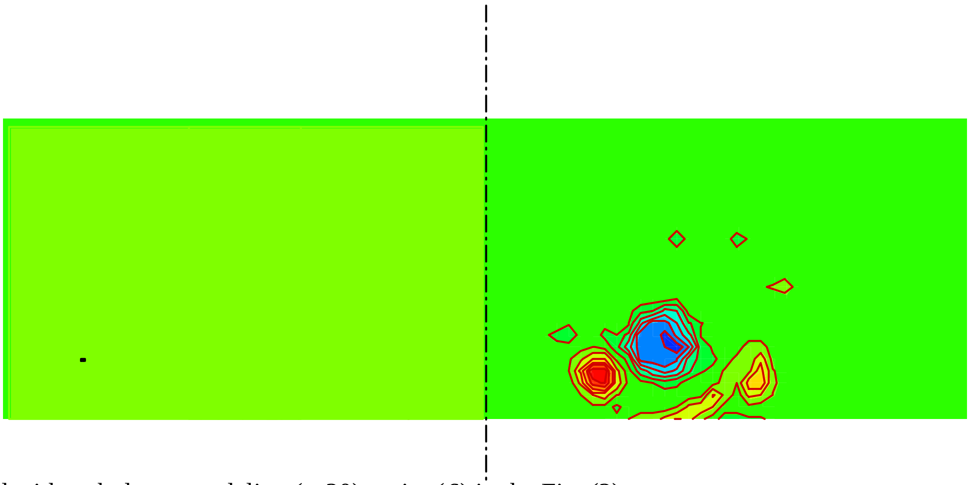
(d) vortex cloud with turbulence modeling (t=15): point (3) in the Fig. (2)



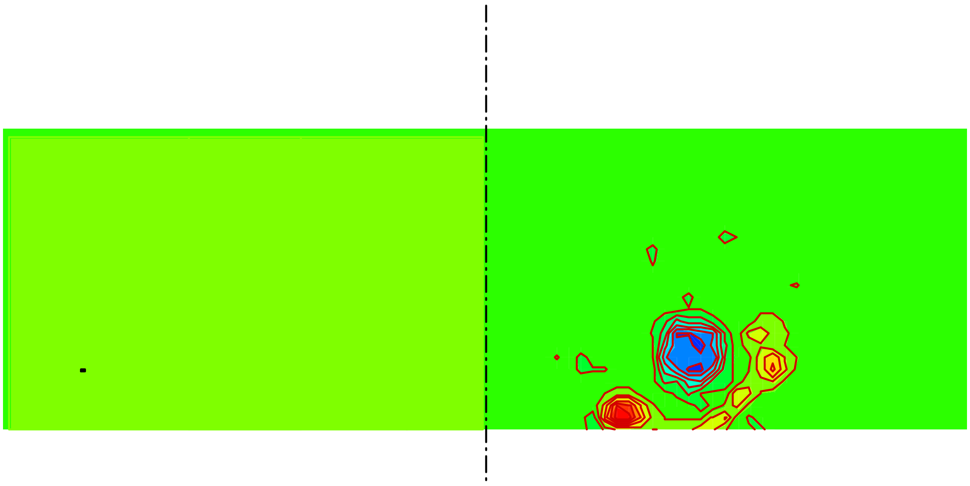
(e) vortex cloud with turbulence modeling (t=20): point (4) in the Fig. (2)



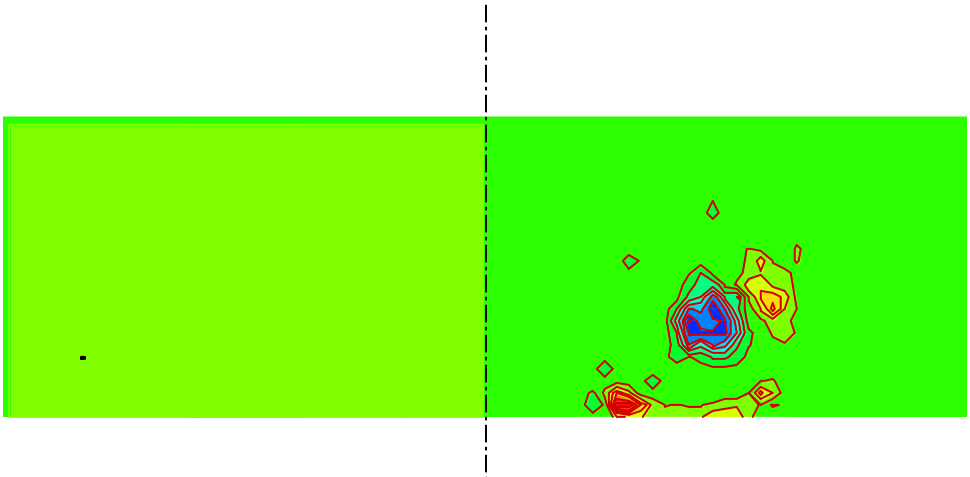
(f) vortex cloud with turbulence modeling (t=25): point (5) in the Fig. (2)



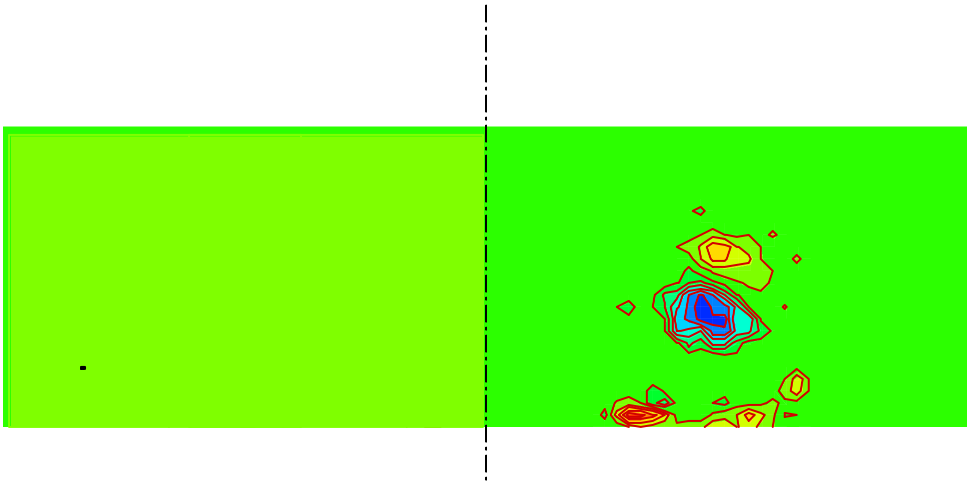
(g) vortex cloud with turbulence modeling (t=30): point (6) in the Fig. (2)



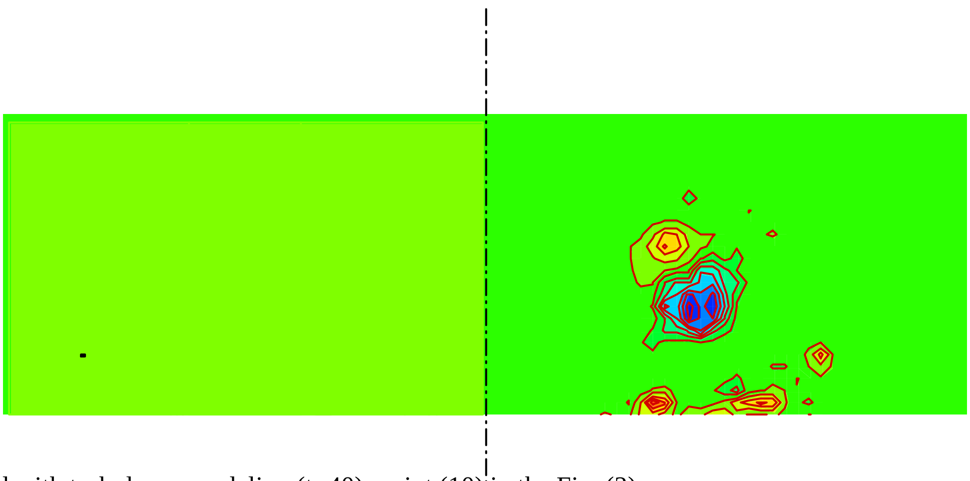
(h) vortex cloud with turbulence modeling (t=32.5): point (7) in the Fig. (2)



(i) vortex cloud with turbulence modeling (t=35): point (8) in the Fig. (2)



(j) vortex cloud with turbulence modeling (t=37.5): point (9) in the Fig. (2)



(k) vortex cloud with turbulence modeling (t=40): point (10) in the Fig. (2)

Figure 3. Vorticity distribution along the flow simulation.

of the calculated trajectory, the actual vorticity distribution. It is ease to see that at point (2) the free vortex cloud starts to strongly interact with the ground vortex cloud giving rise to a secondary vortex structure. At point (3) this interaction is intensified by the free vortex cloud that is close to the ground. At point (4) the secondary vortex structure departs from the ground and a new vortex structure appears. It seems reasonable to assume that the sudden separation (from the ground) of the secondary vortex structure free the vorticity to go up at point (5). One can also observe from the numerical results that the secondary structure, after being released from the ground, starts a circular motion around the primary structures. Also, new vortex structures show up near the ground during the simulation.

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 the experimental points, see Fig. 2; the numerical simulation enables one to follow this trajectory as shown in the figure.

As a final observation, 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.

5. Acknowledgement

The FAPEMIG (TEC-00025/2002 Process) and the CNPq (300126/92–1 Process) supported the work described in this paper.

6. 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", AGARDograph204.

Hirata, M.H., Ricci, J.E.R. and Oliveira, W., 2002, "Comportamento da Esteira de Aeronaves nas Operações de Pouso e Decolagem", Proc. 9^{th} Brazilian Congress of Thermal Engineering and Sciences, Caxambu, MG, Brasil (In Portuguese).

Kamemoto, K., 1994, "Development of the Vortex Methods for Grid-Free Lagrangian Direct Numerical Simulation", Proc. Third JSME-KSME, Sendai, Japan.

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

Lesieur, M., 1990, "Turbulence in Fluids", Kluwer, Dordrecht.

Lesieur, M. and Métais, M., 1996, "Spectral Large-Eddy Simulations of Isotropic and Stably-Stratified Turbulence", J. Fluid Mech., Vol. 239, pp. 157-194.

Lewis, R.I., 1991, "Vortex Element Methods for Fluid Dynamic Analysis of Engineering Systems", Cambridge University Press, Cambridge.

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

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", AIAA Paper 98-2409, Proceedings of the 16^{th} AIAA Applied Aerodynamics Conference, Albuquerque, NM, USA, June.

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", AIAA Journal, Vol. 34, No.3, pp. 580-588.