AIRPLANE WAKE NEAR THE GROUND WITH NATURAL CONVECTION

Ana Paula Moni da Silva
e-mail: anamoni@unifei.edu.br
Luiz Antonio Alcântara Pereira
Institute of Mechanical Engineering, Federal University of Itajuba , CP 50, Itajubá, Minas Gerais, 37500-903, Brazil
e-mail: luizantp@unifei.edu.br

Miguel Hiroo Hirata
State University of Rio de Janeiro, FAT-UERJ, Resende, Rio de Janeiro, Brazil
e.mail: hirata@superonda.com.br

Abstract. *In this paper, the study of natural convection in the airplane wake near the ground is simulated numerically using a purely Lagrangian particle method. The interaction of temperature and vorticity requires the creation of vorticity from temperature; a model by changing the strength of vortices based on the direct interpretation of the vorticity equation is presented. The vorticity and heat are created from the ground plane and both are transported with the convection velocity due to the vorticity. The random walk method is used to simulate the diffusion of vorticity and heat. The present results are in a good agree with experimental results available in the literature and indicate that the creation of vorticity from heat is computed successfully.*

Keywords: *particle method, natural convection, heat-vortex interaction, aircraft wake behavior, Lagrangian description*

1. INTRODUCTION

An important problem in the aerodynamic design of wings flying near the ground is the control of the interaction between fluid flow and walls. Understand the simultaneous effects of the ground boundary-layer coupling and heat transfer is a key factor for the efficiency of the airport operations.

The creation of a pressure differential over the wing surfaces generates lift; the lowest pressure occurs over the upper wing surface and the highest pressure under the wing. This pressure differential triggers the rollup of the airflow aft of the wing resulting in swirling air masses trailing downstream of the wing-tips; after the rollup is completed, the wake consists of two counterrotating cylindrical vortices. 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 and economy of the Air Traffic Control. 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 $h \approx 30 \sim 70$ m range above ground level, when strong vorticity structures are interacting with the runaway ground”.

The interaction of vortical structures with ground plane has been studied by many authors in a variety of configurations. 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 as they interact with the nascent vortices near the ground - the ground vortex cloud (such as heat exchanger, chimneys and off-shore platforms (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. 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 (Alcântara Pereira and Hirata, 2005).

The vortex method has been applied to a variety of problems in fluid mechanics; eg. Chorin (1973), Leonard (1980), Sarpkaya (1989), Alcântara Pereira *et al.* (2004), Kamemoto (2004) and Stock (2007). With a purely Lagrangian a grid for the spatial discretization of the fluid region is not necessary and, therefore, special care to handle numerical instabilities associated to high Reynolds numbers flow is not needed. In addition, with the vortex method the attention is only directed to the regions of high activities, which are the regions containing vorticity; on the contrary, Eulerian schemes consider the entire fluid domain independent of the fact that there are sub-regions where less important, if any, flow activity can be found. Finally with the Lagrangian tracking of the vortices, one does not need to consider the far away boundary conditions. This is of importance in the wake region (which is not negligible in the flows of present interest) where turbulence activities are intense and unknown, a priori.

On the other hand, there are only a few examples of the simulations of vorticity and heat transport using a particle method.

Ghoniem & Sherman (1985) investigated one-dimensional heat diffusion using random walk scheme. In their investigation is presented a complete analysis of heat particles with different properties and the vorticity generation due to the heat transfers process.

Ghoniem *et al.* (1988) and Zhang & Ghoniem (1993) handle shear layers and a rise of a plume using a two-dimensional flow. The diffusion process was simulated using the core spreading method (Leonard, 1980) and the density difference was considered, although heat transfer was not.

Reulet *et al.* (1988) investigated the effect of an isolated vortex, produced by a flapping wing, convected by a turbulent stream over a heated wall. The main result was that the evolution of the thermal boundary layer was strongly coupled with the dynamic effects of the vortex.

Smith & Stansby (1989) and Stansby & Dixon (1983) studied the vorticity and the heat transport around a circular cylinder using the vortex in cell method incorporate with the random walk model. In their approach was introduced both vortex and temperature particles according to the similarity of equations of vorticity transport and energy. However, the vorticity generation due to heat and natural convection are not investigated.

Using an idealized flow, Romero-Méndez *et al.* (1998) analytically calculated the effects of isolated Rankine vortices over walls in different configurations; in all cases was observed an increase of the heat transfer and thus a significant insight on the fundamental process that occur during the interaction of vortices with walls and their implications with heat transfer.

Kamemoto & Miyasaka (1999) used the core spreading model to simulate the forced convection heat transfer around a circular cylinder at high Reynolds numbers. Discrete heat elements with thermal core were introduced in the thin thermal layer along the body surface. Although they made an approximation that the temperature in the thermal layer was constant along the normal direction, the time-averaged Nusselt number distribution showed reasonable agreement with that of experiment.

Alcântara Pereira and Hirata (2003) presented a vortex and heat element method for the analysis of unsteady heat transfer in a flow around a body. The time -averaged distribution of the local Nusselt number and the calculated values for aerodynamics loads around a non rotating circular cylinder showed good agreement with the data from the literature. The natural convection was not considered.

Martin and Zenit (2008) studied numerically the mechanisms by which heat is removed from a wall resulting from the interaction of a coherent vortical structure by using a finite volume technique. A simplified arrangement was chosen to solve the problem: the normal impingement of a two-dimensional vortex pair with a flat heated wall. Although it is clear that a forced fluid motion over a heated wall causes an increase of the heat transfer from the wall, the details of the process had not yet been investigated to date.

In this paper, the vortex method (Alcântara Pereira and Hirata, 2003) is employed to simulate numerically the mechanism of heat transfer resulting from the interaction of an aircraft wake vortex with a heated ground including an additional important effect: natural convection.

In order to handle fluid flow and heat transfer by the vortex method, the following model is used (Ogami, 2001): (i) the modeling of discretization of heat distribution into the heat particles; (ii) the modeling of the process in which the vortex is generated by the effect of a heat; (iii) the modeling of the diffusion process of heat and vortex. This new model is very suitable to simulate numerically complex flow fields, wich are of scientific importance as well as in engineering problems.

2. MATHEMATICAL FORMULATION

Figure 1 shows the problem domain with an aircraft modeled as a pair of vortex clouds. 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, U_a is the approaching velocity. The fluid domain 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. The initial fluid temperature is T_∞ and the temperature T_w is assumed constant around the ground plane surface.

For an incompressible fluid flow the dimensionless continuity equation is written as

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

where $\mathbf{u} \equiv (u, v)$ is the velocity vector.

If, in addition, the fluid is Newtonian with constant properties the momentum equation is represented by the Navier-Stokes equation (the Boussinesq Hypothesis is assumed) as

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\nabla p + \frac{1}{Re}\nabla^2 \mathbf{u} + \frac{Gr}{Re^2}\theta\, \mathbf{j}\text{, being } \theta = \left(\frac{T - T_\infty}{T_W - T_\infty}\right). \tag{2}$$

Here, p is the pressure field and Re stands for the Reynolds number defined as $Re = \frac{\Gamma}{\upsilon}$, where υ the kinematic viscosity of fluid. The Grashof number is defined as $Gr = \frac{g\beta(T_w - T_\infty)b^3}{\upsilon^2}$, where β of thermal expansion coefficient of the fluid; the dimensionless time is $t\Gamma/b^2$.

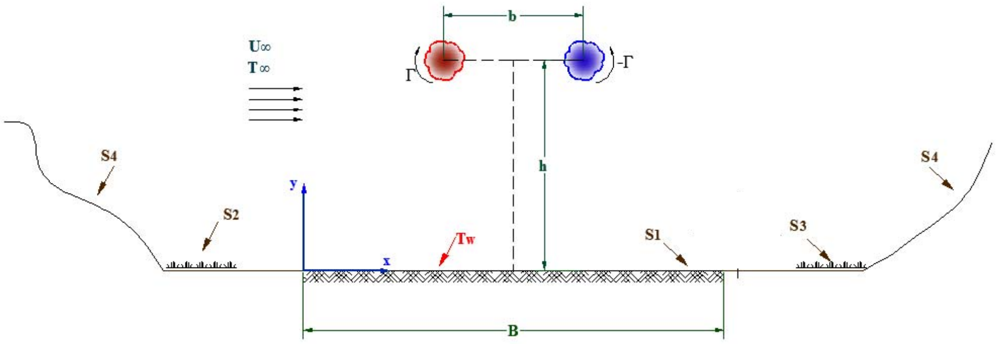


Figure 1: Definitions.

The energy conservation is given by

$$\frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla)T = \frac{1}{RePr}\nabla^2 T\,, \tag{3}$$

where T is the temperature field and $Pr=\nu/\alpha$ is the Prandtl number, being α the thermal diffusivity.

The following boundary conditions apply

$$u_n = v = 0\text{, impermeability on } y = 0, \tag{4}$$

$$u_\tau = u = 0\text{, no-slip on } y = 0, \tag{5}$$

$$|\mathbf{u}| \to \frac{U_\infty b}{\Gamma} \text{ at } S_4\,, \tag{6}$$

$$T=\theta=1 \text{ on } y=0, \tag{7}$$

$$T=\theta=0 \text{ at } S_4. \tag{8}$$

3. NUMERICAL METHOD

Taking the curl of the Navier-Stokes equation and with some algebraic manipulations one gets the vorticity equation which presents no pressure term. In two-dimensions this equation reads

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \frac{1}{Re} \nabla^2 \omega + \frac{Gr}{Re^2} \frac{\partial}{\partial x} \left(\frac{T - T_\infty}{T_W - T_\infty} \right), \tag{9}$$

where $\omega(\mathbf{x}, t) = \nabla \times \mathbf{u}(\mathbf{x}, t)$ represents the only non-zero component of the vorticity field. The left hand side of the above equation carries all the information needed for the convection of vorticity while the first term of the right hand side governs the diffusion.

The second term of the right hand side means that the vorticity must be created from heat due to the effect of buoyancy. Ogami (2001) presented two models for creating vortices from temperature particles. First, he modeled the vorticity equation as it is, and it is regarded as a natural extension of the method of Ghoniem and Sherman (1985). Second, a vortex pair (one positive and one negative) was generated from one temperature particle.

Our new scheme involving natural convection and interaction of temperature and vorticity is modeled by creating vorticity from temperature; according to the Ogami (2001), the strength of a vortex is increased at each time step Δt by the amount of

$$\Delta\Gamma = -Ri \, \frac{\Delta T}{2\sigma_T} \Delta t \, \Delta A \,, \tag{10}$$

where ΔA is the region occupied by vortex and $Ri = \dfrac{Gr}{Re^2}$ is the Richardson number.

A direct interpretation of Eq.(9) is given using this scheme; it is considered to be an extension of the scheme adopted by Ghoniem and Sherman (1985) to two-dimension. In the numerical approach used in this paper there are regions where both temperature particles and vortices exist and the present model is very suitable for it.

Following Chorin (1973) we use the viscous splitting algorithm, which, for the same time step of the numerical simulation, says that

Convection of vorticity is governed by

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = 0 \,, \tag{11}$$

Diffusion of vorticity is governed by

$$\frac{\partial \omega}{\partial t} = \frac{1}{Re} \nabla^2 \omega \,. \tag{12}$$

Note that Eq. (3) gives the law that the temperature distribution, T, moves both with the convection velocity. Vortex elements and discrete heat elements distributed in the flow field are followed during numerical simulation according to the Adams-Bashforth scheme (Ferziger, 1981). It is clear that the energy equation, Eq. (3), has the similar form to the vorticity transport equation, Eq. (9). This suggests that the energy equation can be solved in an analogous way using the random walk method to the motion of the heat elements to account for diffusion (Chorin, 1973).

For the convection of the discrete vortices (and heat particles) of the cloud, Eq. (11) is written in its Lagrangian form as

$$\frac{dx^{(i)}}{dt} = u^{(i)}(x, y, t) \,, \tag{13}$$

$$\frac{dy^{(i)}}{dt} = v^{(i)}(x, y, t)\,. \tag{14}$$

The u $^{(i)}$ and v $^{(i)}$ components of the velocity induced at the location of the vortex (i) and heat particle (i) can be written as

$$u^{(i)} = U_{\infty} b/\Gamma + ub^{(i)} + uv^{(i)}\,, \tag{15}$$

$$v^{(i)} = 0 + vb^{(i)} + vv^{(i)}\,, \tag{16}$$

where, **ui** $^{(i)} \equiv [1, 0]$ is the velocity vector of uniform flow,

ub $^{(i)} \equiv [ub^{(i)}, vb^{(i)}]$ is the velocity vector induced by the ground at the location of particle (i),

uv $^{(i)} \equiv [uv^{(i)}, vv^{(i)}]$ is the velocity vector induced at the particle (i) due to the Lamb vortex cloud.

The **ui** $^{(i)}$ calculations present no problems. The ground contributes with **ub**(**x**, t), which is obtained by using the Boundary Element Method (Katz and Plotkin, 1991). The two components can be written as

$$ub^{(i)} = \sum_{k=1}^{NP} \psi_k \, uc_k^{(i)} \tag{17}$$

$$vb^{(i)} = \sum_{k=1}^{NP} \psi_k \, vc_k^{(i)} \tag{18}$$

where NP is the total number of flat source panels representing ground plane surface. It is assumed that the source strength per length is constant such that ψ_k = const and $uc_k^{(i)}$ and $vc_k^{(i)}$ are the components of the velocity induced at particle (i) by a unit strength flat source panel located at k.

The heat transport from the ground to the fluid nearby the flat surface is determined by the temperature gradient at the surface. The surface heat flux is determined by Fourier's Law

$$\dot{q} = -\lambda \frac{dT}{dn}\,, \tag{19}$$

where n denotes the normal direction to the flat surface and λ is the thermal conductivity of fluid. The heat quantity transferred from the surface (j-th panel with length ΔS_j) to the k-th nascent heat element is given by

$$\Delta Q_j = \alpha \Delta t \frac{\left(T_w - T_j\right)}{\varepsilon} \Delta S_j\,, \tag{20}$$

in which $\alpha = \upsilon/Pr$ and ε is the displacement normal to the straight-line panel.

The temperature distribution $T(z)$ results from the contribution of all the heat particles in the field, so

$$T(z) = \sum_j \frac{\Delta Q_j}{\pi \sigma_T^2} \exp\left[-\frac{\left(z - z_j\right)^2}{\sigma_T^2}\right], \tag{21}$$

where σ_T is the core radius of the heat particles.

4. RESULTS AND DISCUSSIONS

The strength of the free vortices, starting at the wing tips, is governed by the weight, speed, and shape of the wing of the generating aircraft. To have an insight over the numerical results we first considered preliminary simulations. This allows us to analyze its consistency and define some numerical parameters. In the present numerical simulation each cloud composed by 500 free vortices with a total intensity equal to the isolated vortices. The ground plane surface is represented by NP=200 flat source panels with constant density. The simulation was performed up to 800 time steps with magnitude Δt=0.05. During each time step the new particles are shedding into the cloud through a displacement $\varepsilon=\sigma$=0.001 normal to the straight-line elements (panels). In all simulations the Reynolds number is kept a high value of 75,000. The Prandtl number is assumed as Pr=1.0 and the difference $T_w - T_\infty$ is assumed as 40.

In order to investigate the influence of natural convection on the heat transfer is adopted Ri=0.25. Figure 2 shows the influence of Richardson number on vortex trajectory histories.

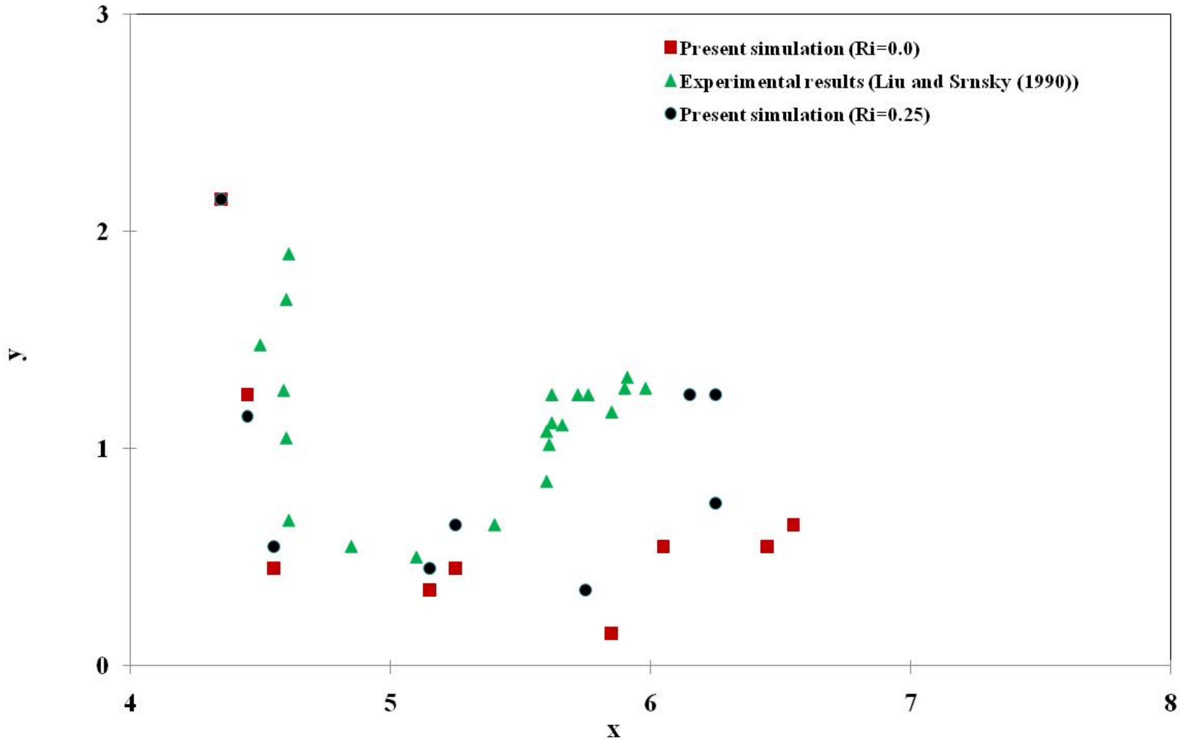
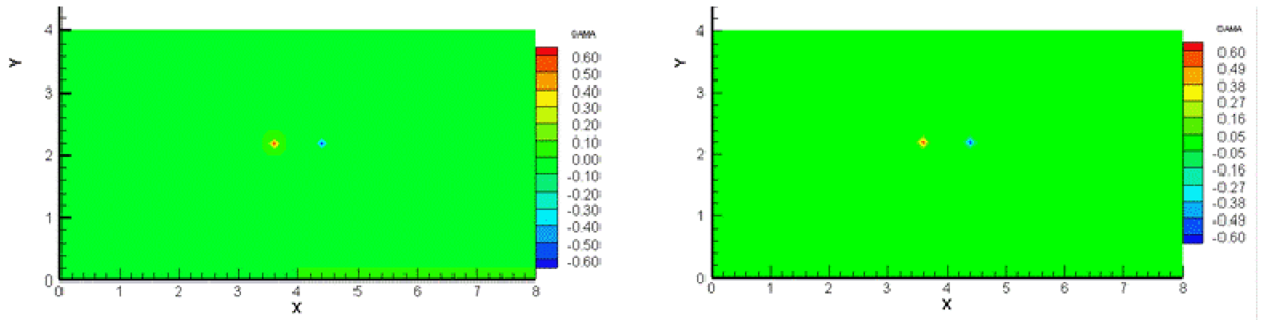


Figure 2. Comparison of measured and computed vortex trajectories
(Re=75,000 and Pr=1.0).

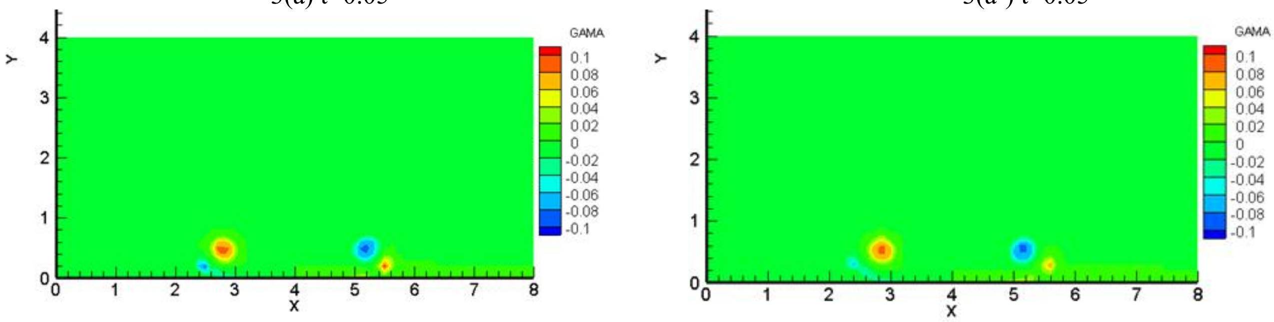
The numerical result using Ri=0.25 is in good agreement to the experimental values of Liu and Srnsky (1990); one can clearly observe that the computed trajectory vortical motion with natural convection does try to follow the experimental results, even for long time simulation. The case for which both vortical motion and natural convection contribute to the heat transfer is of great interest for practical applications. The pictures from Fig. 3(a) to Fig. 3(d) shows the vorticity distribution with forced convection (Ri=0), whereas in pictures from Fig. 3(a*) to Fig. 3(d*) the results refer to the simulation with natural convection (Ri=0.25).

In the absence of crosswinds, the vortical structures interact with the heated wall, and at t=15 is possible to identify the formation of secondary structures released from the ground. At t=30, see Fig. 3(c) and Fig. 3(c*), the vorticity contours indicate the formation of others secondary structures; one can clearly identify from Fig. 3(c*) the effect of heat transfer over the vortical structures. It is found that the heat from the ground plane is transported down the wake mostly by mean heat flux and that wake structure and heat transfer both significantly affect one another.

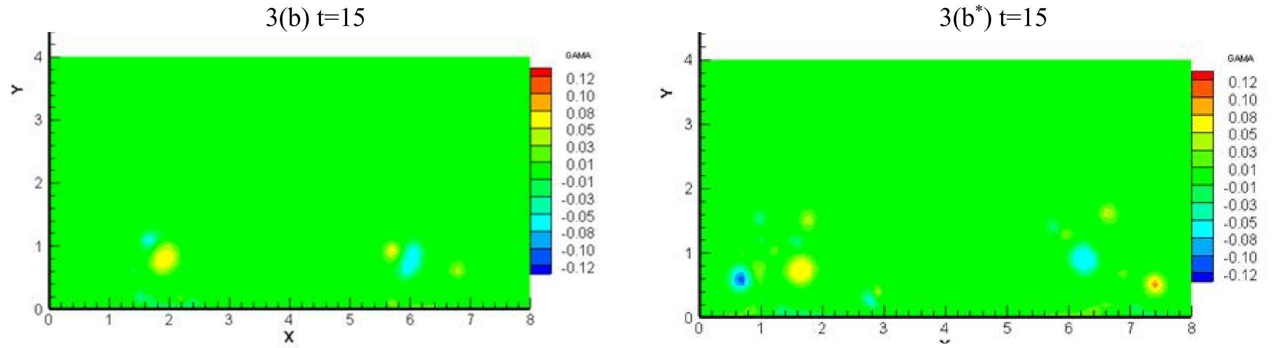
Figure 4 shows the evolution of temperature field for the same flow shown in Fig. 3. When the simulations starts, the temperature of the fluid is T=283 K and the temperature of ground plane is T=323 K. At t=0.05 the heat from the ground plane has begun to diffuse into the fluid. When the primary vortex structures moves closer and interacts with the ground plane (t=30), heat is strongly convected away from the wall by the motion of fluid. Buoyancy effect have a little influence on the initial time steps of simulation ($0 \leq t \leq 15$) because the ground plane inhibits the buoyant torques that result from the motion within a stratified fluid flow away from the ground.



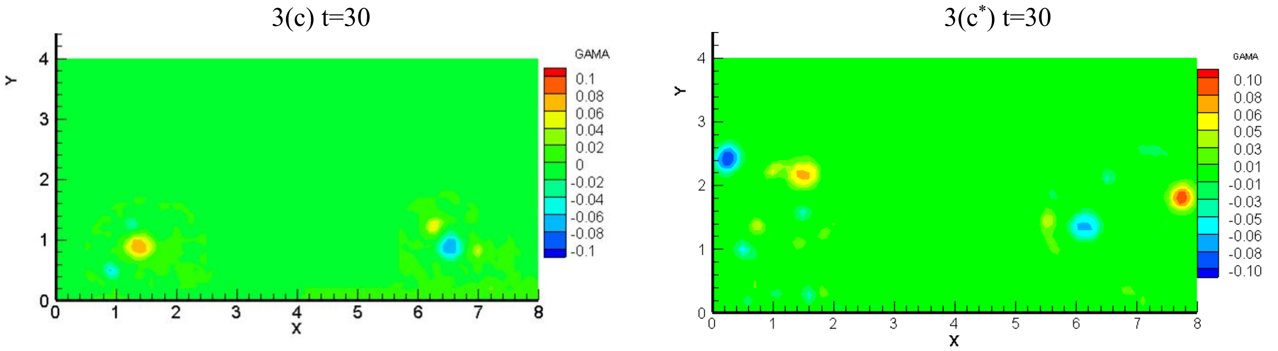
3(a) t=0.05 3(a*) t=0.05



3(b) t=15 3(b*) t=15



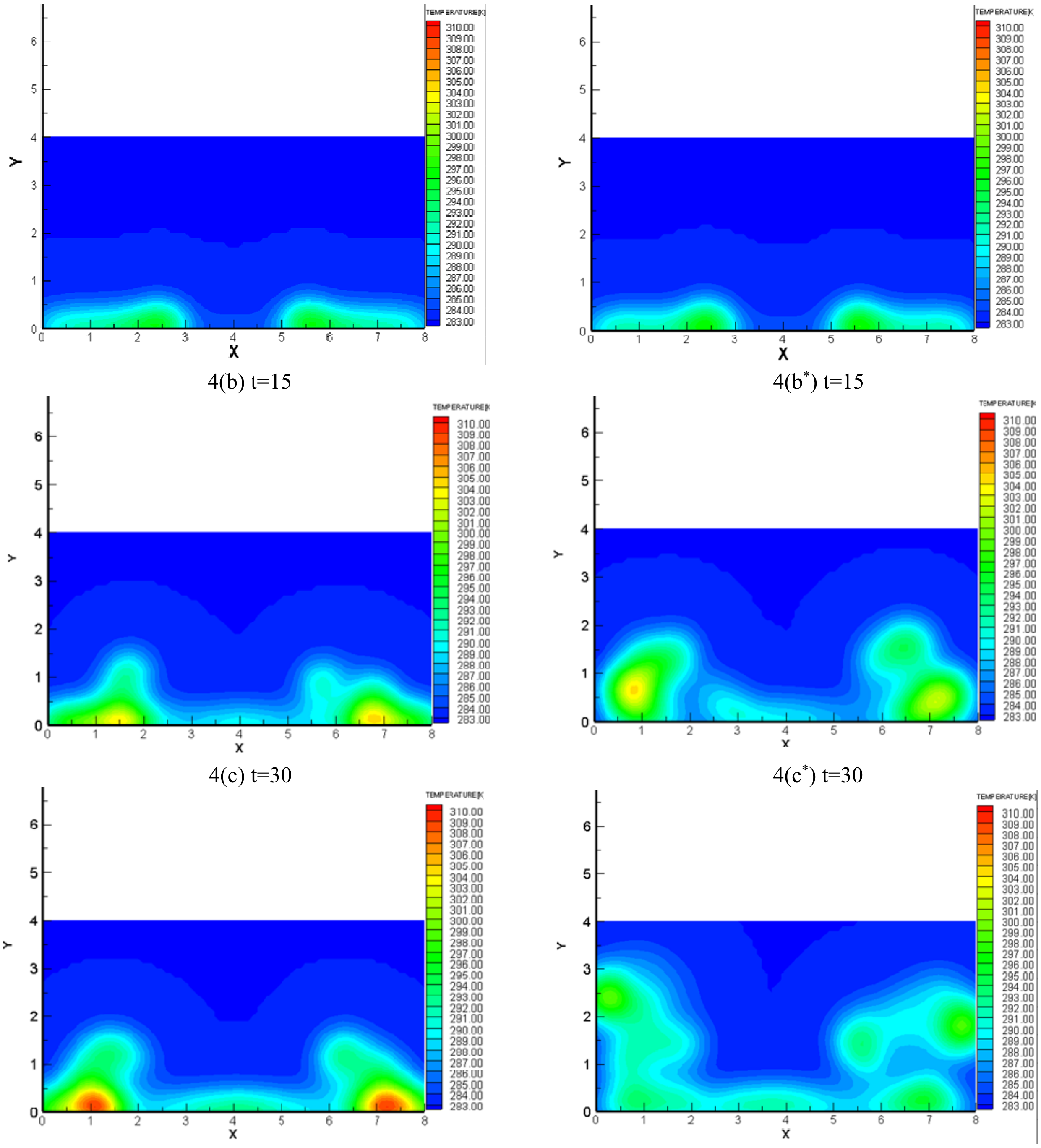
3(c) t=30 3(c*) t=30



3(d) t=40
computed with Ri=0

3(d*) t=40
computed with Ri=0.25

Figure 3. Vorticity distribution (GAMA) along the interaction of an aircraft wake with a heated wall (Re=75,000 and Pr=1.0).



4(b) t=15

4(b*) t=15

4(c) t=30

4(c*) t=30

4(d) t=40
computed with Ri=0

4(d*) t=40
computed with Ri=0.25

Figure 4. Evolution of the temperature field (Re=75,000 and Pr=1.0).

As can be seen in Fig. 4(c) and Fig. 4(c*), the buoyancy effect was shown to influence the heat transfer by affecting the circulation of the wake vortices.

Because the distributed vorticity and heat of the mainstream flow has been replaced in the numerical model by two clouds, each other with Z particles, the CPU time for particle-particle interaction turns expensive; the operation count of our algorithm is proportional to the square of Z. As NP increases Z also tends to increase, and the computational efforts becomes expensive. This is a major source of difficulties, and it can only be handled through the utilization of faster schemes for the induced velocity calculations, such as the multipole technique (Greengard and Rokhlin, 1987) and/or parallel computers to run long simulations (Takeda *et al.,* 1999). A typical numerical simulation requires 115 h of CPU time in a PENTIUM IV/400 Mhz.

Finally, the results are promising and encourage performing additional tests in order to explore the phenomena in more details.

5. CONCLUSIONS

Using a particle method, this study shows the effect of natural convection on wake vortices behavior near the heated wall. The final time of simulation is representative of terminal flight operations.

The main objective of the work with the implementation of a vortex and heat particles method for the analysis of unsteady and natural-convective heat transfer in a flow aircraft wake vortex behavior near the ground has been achieved. As can be seen in Fig. 2, 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. The present methodology, therefore, is able to provide good estimates for trajectory of the center of the main vortex, evolution of the vorticity field, evolution of the temperature field, and is able to predict the flow correctly in a physical sense.

As a future work, the turbulence modeling will be carried out (Alcântara Pereira *et al.*, 2003). A new method to simulate diffusion processes (Rossi, 2006) and to calculate Nusselt number as a function of the position over the ground plane for different time instants will be carried.

6. ACKNOWLEDGMENTS

This project was support by FAPEMIG (Research Foundation of the State of Minas Gerais) Proc. TEC APQ-01070-10.

7. REFERENCES

Alcântara Pereira, L.A. and Hirata, M.H., 2005, "A Particle Method for Aircraft Wake behavior Analysis near the Ground Using Turbulence Modeling", Proceedings of the 18th International Congress of Mechanical Engineering, November 6-11, Ouro Preto, MG, Brazil, Paper COBEM2005-0105.

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

Alcântara Pereira, L.A. and Hirata, M.H. 2003, "Heat Transfer in the Wake Behind a Body Using A Particle Method", 17th International Congress of Mechanical Engineering, Proceedings of COBEM 2003, November 10-14, São Paulo, SP, Brazil.

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.

Ghoniem, A.F., Heidarinejad, G. and Krishan, A., 1988, "Numerical Simulation of a Thermally Stratified Shear Layer Using the Vortex Element Method ", J. Comp. Physics., Vol. 79, pp. 135-166.

Ghoniem, A.F. and Sherman, F.S., 1985, "Grid-Free Simulation of Diffusion Using Randon Walk Methods", J. Comp. Physics, Vol. 61, pp. 1-37.

Greengard, L. and Rokhlin, V., 1987, "A Fast Algorithm for Particle Simulations", Journal of Computational Physics, Vol. 73, pp. 325-348.

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. J. Braz. Soc. Mech. Sci & Eng., vol. 26, n. 4, Rio de Janeiro, Oct. / Dec..

Kamemoto, K. and Miyasaka, T., 1999, "Development of a Vortex and Heat Elements Method and Its Application to Analysis of Unstead Heat Transfer around a Circular Cylinder in a Uniform Flow", Proc. of 1st Int. Conf. on Vortex Metods, Kobe, Nov. 4-5, pp. 191-203.

Katz, J. and Plotkin, A., 1991, "Low Speed Aerodynamics: From Wing Theory to Panel Methods". McGraw Hill, Inc.

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

Leonard, A., 1980. Vortex Methods for Flow Simulation. J. Comp. Phys., vol. 37, pp. 289-335.

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.

Martin, R. and Zenit, R., 2008, "Heat Transfer Resulting from the Interaction of a Vortex Pair with a Heated Wall", Journal of Heat Transfer, Vol. 130, pp.1-7.

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.

Reulet, P., Marchand, M. and Millian, P., 1988, "Experimental Characterization of the Convectice Vortex-Wall Interactions", Rev. Gen. Therm., 37 (8), pp. 661-668.

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, SP, Brazil, Paper COBEM2003-0887.

Romero-Méndez, R., Sen, M. Yang, K.T. and McClain, R.L., 1998, "Enhancement of Heat Transfer in an Inviscid-Flow Thermal Boundary Layer due to a Rankine Vortex", Int. J. Heat Mass Transfer, 41, pp. 3829-3840.

Rossi, L.F., 2006, "A Comparative Study of Lagrangian Methods Using Axisymmetric and Deforming Blobs", SIAM J. Sci. Comput., Vol. 27, No. 4, pp. 1168-1180.

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

Smith, P.A. and Stansby, P.K., 1989, "An Efficient Surface Algorithm for Randon-Particle Simulation of Vorticity and Heat Transport", J. Comp. Physics, Vol. 81, pp. 349-371.

Stansby, P.K. and Dixon, A.G., 1983, "Simulation of Flows around Cylinders by a Lagrangian Vortex Scheme", Applied Ocean Research, Vol. 5, No. 3, pp. 167-178

Stock, M. J., 2007. Summary of Vortex Methods Literature (A lifting document rife with opnion). April, 18: © 2002-2007 Mark J. Stock.

Takeda, K., Tutty, O.R. and Nicole, D.A., 1999, "Parallel Discrete Vortex Method on Commodity Supercomputers; an Investigation into Bluff Body Far Wake Behaviour", ESAIM: Proceedings, Third International Workshop on Vortex Flows and Related Numerical Methods, Vol. 7, pp. 418-428.

Zhang, X. and Ghoniem, A.F., 1993, "A Computational Model for the Rise and Dispersion of Wind-Blown, Buoyancy-Driven Plumes-I", Neutrally Stratified Atmosphere, Atmospheric Environ., Vol. 29A, pp. 2295-2311.

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.

RESPONSIBILITY NOTICE