12th Spring School on Transition and Turbulence
September 21st-25th, 2020, Blumenau, SC, Brazil

EPTT- 2020-0070 - WING TIP VORTICES DYNAMICS WITH CROSS WIND EFFECT USING THE LARGE EDDY SIMULATION (LES) THEORY

Marcos André de Oliveira
Paulo Guimarães de Moraes
Luiz Antonio Alcântara Pereira
Mechanical Engineering Institute, Federal University of Itajubá (UNIFEI), Itajubá, Brazil
marcos.oliveira@uft.edu.br
pauloguimor@yahoo.com.br
luizatp@unifei.edu.br

Abstract.
In operational control of the airports it is required to address two important issues: The risk of meeting the vortex wake of an aircraft in the takeoff or landing operations of another aircraft and the interest in reducing the interval between these frequent subsequent operations. However, for studies of these phenomena such as the interaction of vortex wake with surfaces, using Lagrangian methods, it is necessary to implement solutions to accelerate the simulations. This paper describes a refined numerical method developed to significantly reduce the final processing time of simulations with expensive Lagrangian approaches. The numerical method is based on Lagrangian vortex method with Large-Eddy Simulation (LES) theory adapted to 2D. The image method is used to guarantee the condition of impenetrability on the airport runway. In this sense, parallel computing (OpenMP) and an algorithm with a box structure to accelerate the calculation of the local eddy viscosity, requisite for each Lamb vortex during each stage of time, are used in a Fortran code. Quantitative and qualitative results are presented at different Reynolds numbers, in dimensionless cross wind conditions. Preliminary results for a Boeing 757-200 (representative in aviation problems) are also presented. The preliminary results maintain good agreement with experimental results and other numerical simulations. It is possible to capture satisfactorily the trajectories of the centroid of the vortices structures that give off the wing tip, the temporal deformation of these structures, as well as the dynamics of the primary and secondary structures in the interaction with the runway.

Keywords: *LES Turbulence, Lagrangian methods, OpenMP-Fortran, Image method, Aircraft vortex wake*

1. INTRODUCTION

Increasing advances in the computational area have allowed simulations of complex phenomena associated with the movement of a viscous fluid around solid boundaries by numerical methods.

Due to the nonlinearity of the governing equations and their characteristics for numerical analysis, the problems of Computational Fluid Dynamics - CFD and coupled hydroelastics are very expensive from the point of view of computational time (Kuzmina *et al*., 2015). Increasingly, developers of scientific codes for fluid dynamics studies are seeking High Performance Computing (HPC) solutions to reduce simulation time and increase accuracy and precision of results. Parallel programming of type OpenMP is one of these solutions when the computational code allows the parallelism and respective increase of performance. More accuracy of the problem analysis has been possible due the possibility of using more precise numerical methods. That in turn would require a lot of processing time if the parallelism of the computational code wasn't used.

OpenMP is a model shared memory programming. Thus each execution unit is a thread. All threads have access to the same (shared) memory. And each thread also has its own memory independent of the others (private). According to Los Reis and Alcântara Pereira (2016), since all data is stored on a single machine, data exchange between the cores is faster than in distributed memory systems. The main features of the OpenMP standard are summarized as follows: it includes a library that provides the functionality and an Application Programming Interface (API) that specifies how to use it; is independent of language, compiler, operating system and hardware; is an open and continuously evolving standard; it is implemented through compilation directives (instructions that will be executed by the compiler); traditionally supported by C, C ++ and Fortran compilers; and has simpler programming than the standard distributed memory (example: Message Passing Interface - MPI).

Techniques of approach to solution of computational fluid dynamics problems may be classified as follows: Eulerian, Lagrangian and Lagrangian-Eulerian (hybrid) methods. The numerical method of simulation of the process of generation and convection (advection and diffusion) of vorticity using pseudorandom numbers generated by computer, without grid (Lagrangian approach), presented in the classic paper of Chorin (1973), allowed concentrating the computational effort in regions of interest by sampling velocity field values near a threshold. Said numerical method allows the simulation of viscous fluids - solution of the Navier-Stokes equations - in two dimensions and for high Reynolds numbers. The numerical method proposed by the author is presented as a tool to overcome difficulties encountered in Eulerian methods, especially in the finite difference method. The work did not apply to high numbers of Reynolds, of practical importance, and also required prohibitive computer work.

The Discrete Vortex Method – DVM is the best-known representative of this general class of elements/particles methods. The DVM (Lewis, 1999; Kamemoto, 1994) refers to mesh free Lagrangian particle methods and basically is based on the generation of a cloud of discrete vortices which are generated at the surface of the body and at subsequent instants of the simulation are convected to the fluid domain. The advection of these vortices is made by some advancement scheme, such as that of Euler, Adams-Bashforth or Runge-Kutta. The diffusion of these vortices can also be performed by various techniques.

Santiago and Bodstein (2006) implemented four diffusion simulation techniques: random walk method, redistribution of vorticity method, diffusion velocity method and modified core spreading method. Using the purely diffusive analytical solution for the Lamb vortex as reference, the four methods were compared, considering their accuracy and computational cost, aiming at the application to the Vortex Method. The authors concluded that the modified core spreading method is superior to the random advance method in the convective/diffusive problem of the Blasius boundary layer development simulation on a flat plate. This superiority is related to the accuracy and ease of implementation, but with a high computational cost. The solution to the high computational cost for the modified core spreading method, according to the authors, would be a vortices agglomeration algorithm. In the present paper the random walk method is used (Chorin, 1973).

An aircraft during landing or takeoff operations at airports around the world sheds large vortical structures, which can interfere with other aircraft. In general, the aircraft wing tip vortices persist for significant distances, present a circulation proportional to aircraft size and aircraft velocity (Zheng and Ash, 1996), interact intensely with the ground plane (Hirata *et al.,* 2002), and still are influenced by effects of cross wind conditions. For interests of safety, conservative separation distances between two aircrafts are adopted attempting to avoid such interference (Wakim *et al.,* 2017). However, for the operational optimization of the airports, it is indispensable to reduce the interval between two consecutive landing or takeoff operations (Wakim *et al.,* 2017; Hirata *et al.,* 2002).

In past work by Ricci *et al.* (2003), a pair of counter vortical structures, formed by elemental Lamb vortices, was utilized to effectively consider the deformation of such structures during their impact with the ground. They blended the Lagrangian vortex method with LES modeling (Alcântara Pereira, 2002), the latter to take into account the sub grid-scale phenomena. They also used the images method to automatically satisfy the impermeability boundary condition. The elemental Lamb vortices were only used for the desingularization of the velocity filtered field computation (Mustto *et al.*, 1998). The simulations were performed at Reynolds number of Re=7,650 and the primary vortical structures trajectories were successfully compared with experimental results. However, they did not consider effects of cross wind conditions and parallel computing.

In this work, the two-dimensional Lagrangian vortex method with LES modeling presented by Ricci *et al.* (2003) is improved aiming to reduce the final processing time of the simulations. The final processing time of a typical simulation is expensive because of the Biot-Savart's law, which has a computational cost proportional to $O(N^2)$. The aim here is triple: *(i)* Elemental Lamb vortices are used in all computations dispensing the potential model; *(ii)* The local eddy viscosity computation, necessary to each Lamb vortex during each time step, is accelerated through a structure of the boxes developed by Andrade *et al.* (2016); *(iii)* Through parallel computing (OpenMP) in Fortran, are computationally integrated the expensive approaches of the Lagrangian vortex method and image method for accuracy and precision of the results, including cross wind conditions. Lastly, quantitative and qualitative results are presented at different Reynolds numbers, namely, 7,650, 10,000, 100,000 and 1,000,000, in dimensionless cross wind conditions, namely, 0, 0.02 and 0.04. The chosen example was of a Boeing 757-200, which is representative in aviation problems.

2. THE PROBLEM STATEMENT

The computational code applied to the aircraft wake next to the ground was implemented considering the free vortices shedding from the wings. Since, according to Ricci *et al.* (2003), the free vortices that start at the wing tips of aircraft are defined through the relationship:

$$\Gamma_0 = \frac{\pm W}{\left(\rho b_0 U_a\right)} \tag{1}$$

where, Γ_0 is the circulation, W is the aircraft weight, ρ is the fluid density, b_0 is the wingspan and U_a is the approaching velocity.

Figure 1 represents the vortex shedding wake and the magnitudes of interest used to simulate the phenomenon. The domain of the problem is defined by boundary $S=S_1 \cup S_2 \cup S_3 \cup S_4$, being S_2 the airport runway, S_1 and S_3 the runway side ground and S_4 the far away boundary.

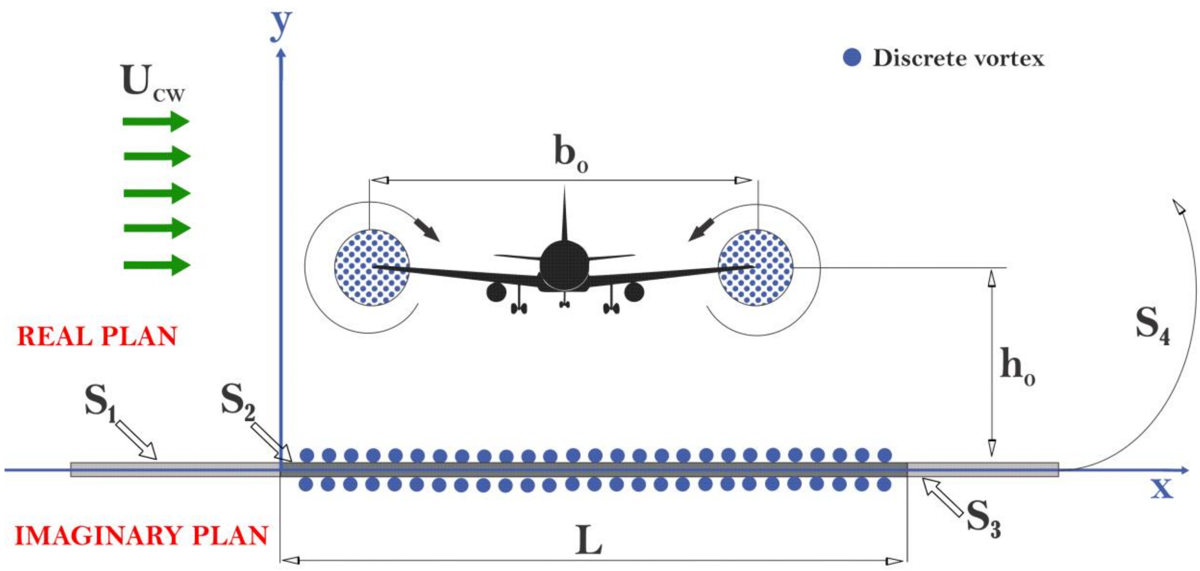

Figure 1. Schematic of the aircraft vortex wake and quantities of interest.

Large Eddy Simulation (LES) is used to separate the flow scales. The macro scales are where the phenomena of greatest interest are concentrated (Lesieur and Métais, 1996) and the analysis of these phenomena can be performed, in this paper, using a considerable number of discrete vortices. The microscales must be modeled.

Through the filtering process of the governing equations, the large scales $\left(\overline{f}\left(\boldsymbol{x},t\right)\right)$ and the sub-mesh scale $\left(f'\left(\boldsymbol{x},t\right)\right)$ are separated:

$$f\left(\boldsymbol{x},t\right)=\overline{f}\left(\boldsymbol{x},t\right)+f'\left(\boldsymbol{x},t\right) \tag{2}$$

The filtered part is given by:

$$\overline{f}\left(\boldsymbol{x},t\right)=\int_{\forall} f\left(\boldsymbol{x}-\boldsymbol{y},t\right)\overline{G}\left(y\right)dy \tag{3}$$

where the function $\overline{G}$ represents a low pass filter.

Considering the indicial form, there is then

$$u_i=\overline{u}_i+u_i' \tag{4}$$

where, $\overline{u}_i$ is the filtered field and u_i' is the fluctuation field.

Thus, the governing equations for the filtered field, considering the incompressible flow of Newtonian fluid (also in the indicial form):

$$\frac{\partial \overline{u}_j}{\partial x_j}=0 \tag{5}$$

$$\frac{\partial \overline{u}_i}{\partial \mathrm{t}}+\frac{\partial\left(\overline{u}_i\,\overline{u}_j\right)}{\partial \mathrm{x_j}}=-\frac{1}{\rho}\frac{\partial \overline{P}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\left(\nu+\nu_t\right)\left(\frac{\partial \overline{u}_i}{\partial x_j}+\frac{\partial \overline{u}_j}{\partial x_i}\right)\right] \tag{6}$$

where, ν is the kinematics viscosity coefficient, ν_t is the eddy viscosity coefficient and $\overline{P}$ is the pressure modified by the turbulent kinetic energy.

2.1. Boundary Conditions

Boundary conditions are given by the no-slip and impermeability conditions on solid surfaces:

$$u_{\tau f} = u_{\tau s} \quad \text{at} \quad S_1,\ S_2 \ \text{and} \ S_3 \tag{7}$$

$$u_{n f} = u_{ns} \quad \text{at} \quad S_1,\ S_2 \ \text{and} \ S_3 \tag{8}$$

where, u is the velocity vector, n represent the normal direction, τ is the tangential direction and f and s subscripts represent the fluid and solid boundary, respectively.

For regions of the domain, distant from the phenomena involved, the condition is:

$$|u| \to U_\infty \quad \text{at} \quad S_4 \tag{9}$$

3. NUMERICAL METHOD

3.1. Establishment of the dimensionless problem

The equations and parameters of interest are written in dimensionless form based on the following quantities: *(i)* Length scale, represented by the aircraft's wingspan (characteristic length), b_0; *(ii)* Velocity scale, represented by the representative velocity of the phenomenon (Ricci, 2003), $U = \dfrac{\Gamma_0}{b_0}$; Time scale that results from the relationship between the two previously defined scales, $t = \dfrac{b_0}{U} = \dfrac{b_0^{\,2}}{\Gamma_0}$.

The dimensionless parameter used in the simulation is the Reynolds number, which is defined differently when considering the vortex cloud and when considering the cross wind, respectively:

$$\mathrm{Re}_\Gamma = \frac{\Gamma_0}{\nu} \tag{10}$$

$$\mathrm{Re}_{CW} = \frac{U_{CW} b_0}{\nu} \tag{11}$$

where, Γ_0 is the vorticity intensity from the wing tip vortices, ν is the viscosity of the air, U_{CW} is the velocity of the crosswind, when considered.

Applying the rotational operator (∇×) in Eq. 6, the pressure term is eliminated and the two-dimensional vorticity transport equation can be used in its dimensionless form:

$$\frac{\partial \overline{\omega}}{\partial t} + \left(\overline{\boldsymbol{u}} \cdot \nabla\right)\overline{\omega} = \left(\frac{1}{Re} + \nu_t^*\right)\nabla^2 \overline{\omega} \tag{12}$$

It should be noted that ν_t^* is the dimensionless eddy viscosity coefficient.

Finally the other dimensionless quantities involved in the problem are established: b the wing span length and h is the height of the flight from the airport runway when are shed the wing tip vortices.

3.2. Initial conditions

The option was made to use two discrete vortices clouds instead of the option to use a counter-rotating pair of simple vortices to represent the vortices structures that shed themselves from the wing tips of aircraft in takeoff or landing operation. A quantity of 50 vortices was used in each particle cloud.

The free vortices cloud shedding from the wings have a dimensionless intensity $\Gamma^* = -1$ and $\Gamma^* = 1$, respectively (Fig. 1). The center of the two discrete vortices clouds are located initially at points $(-b/2, h)$ and $(b/2, h)$, considering the center line of the aircraft. Being that, $b/2 = 0.5$ and $h = 1.9$. This option allows comparison with other experimental and numerical works.

Initially the discrete vortices of each cloud are concentrated in a single point, from there a random advance is made to spread these vortices until the outermost vortex reaches the dimensionless radius of 0.1 (Ricci *et al*., 2003; Hirata *et al* ., 2002), that is, guaranteeing a Gaussian distribution.

3.3. Other parameters or quantities used in the simulation

The runway length of the airport was defined as being $L^* = 8b$, which is sufficient to establish the phenomenon of interference between the ground and the vortices shedding from the aircraft.

Dimensionless cross wind conditions are considered $U^*_{CW} = 0$, 0.02 and 0.04, and time increment is $\Delta t=0.05$.

From the initial computational measurements, the core of the discrete vortices was established at a constant value σ=0.001.

3.4. The Discrete Vortex Method integrated with the Image Method

The viscous splitting algorithm applied to the Eq. 12, proposed by Chorin (1973), allows the problems of advection and diffusion of vorticity to be separated and represented by the following equations, respectively:

$$\frac{D\bar{\omega}}{Dt} = \frac{\partial \bar{\omega}}{\partial t} + \left(\bar{\boldsymbol{u}} \cdot \nabla\right)\bar{\omega} = 0 \tag{13}$$

$$\frac{\partial \bar{\omega}}{\partial t} = \left(\frac{1}{Re} + \nu_t^*\right)\nabla^2 \bar{\omega} \tag{14}$$

The Discrete Vortex Method spatially discretize the vorticity field adopting a cloud of vortex elements, which are related to a vorticity distribution $(\varsigma_{\sigma i})$, the circulation strength (Γ_i), and the core size (σ_i). Hence, the discretized vorticity is listed by:

$$\boldsymbol{\omega}(\boldsymbol{x},t) \approx \boldsymbol{\omega}^h(\boldsymbol{x},t) = \sum_{i=1}^{Z} \Gamma_i(t)\varsigma_{\sigma i}\left(\boldsymbol{x} - \boldsymbol{x}_i(t)\right) \tag{15}$$

where, $\boldsymbol{x}_i$ is the position vector of each discrete vortex, Z is the number of vortices existent used to simulate the vorticity field. The numerical analysis is established over a series of small discrete time Δt for each of which a Lamb vortex element is shed from a determined number of points located on the ground.

The nominal core of the discrete vortices is given by the dimensional relationships:

$$\sigma(t) = \sqrt{4\nu\Delta t} \tag{16}$$

In this context the present work higher precision methods are used, however with a higher computational cost: the image method (Katz and Plotkin, 1991) for ground representation and the second-order Adams-Bashforth scheme for the advective motion of particles. Lamb vortices are generated along the ground plane to ensure that the no-slip condition is satisfied and images clouds are provided in the lower half ground to ensure that the impermeability condition is satisfied.

The second-order Adams-Bashforth scheme (advective term), with the addition of the diffusive term, is written as follows (Alcântara *et al*., 2002; Hirata *et al*., 2002):

$$\boldsymbol{x}(t+\Delta t) = \boldsymbol{x}(t) + \left[1.5\bar{\boldsymbol{u}}(t) - 0.5\bar{\boldsymbol{u}}(t-\Delta t)\right]\Delta t + \boldsymbol{\xi}(t) \tag{17}$$

where, $\bar{\boldsymbol{u}}$ is the fluid velocity that is given by the sum of the contributions of the incident flow, of the singularities that represent the body (ground) and the vortex-vortex interaction (Biot-Savart law); $\boldsymbol{\xi}(t)$ is the random displacement.

The random displacement (random advance vector), $\boldsymbol{\xi}(t) \equiv (x_d, y_d)$, is done with the inclusion of turbulence modeling (for more details see Bimbato *et al.* (2011)). The components x_d and y_d of the random advance vector are defined for each discrete vortex, respectively:

$$x_d = \sqrt{4\Delta t\left(\frac{1}{Re} + \nu_t\right)\ln\left(\frac{1}{P}\right)}\left[\cos(2\pi Q)\right] \tag{18}$$

$$y_d = \sqrt{4\Delta t\left(\frac{1}{Re} + \nu_t\right)\ln\left(\frac{1}{P}\right)}\left[\sin(2\pi Q)\right] \tag{19}$$

where, P and Q are random numbers in the range 0 to 1

3.5. 2D turbulence modeling according to Large Eddy Simulation theory

In the random walk method, the vortex elements are displaced similarly to what happens in the viscosity diffusion process (Eq.14). This step includes LES modeling which was originally adapted by Alcântara Pereira *et al.* (2002) to take into account turbulence manifestations into the two-dimensional viscous wake. Bimbato *et al.* (2011) validated the LES modeling used in this work and it is important to comment that the use of a turbulence modeling based on two-dimensional LES stabilizes the numerical solution.

The calculation of turbulent viscosity (Eq. 6) can be performed using the formulation proposed by Smagorinsky (1963) given by the equation:

$$\nu_t = \left(C_{SM}\, l\right)^2 \sqrt{2\overline{S_{ij}}\,\overline{S_{ij}}} \tag{20}$$

where, $C_{SM} = 0{,}18$ and $l = \sqrt{\Delta x\, \Delta y}$. The quantities Δx and Δy are, respectively, the width and height of the mesh.

Smagorinsky's model (1963) uses a mesh in its development, which is not compatible with the Discrete Vortex Method. Then the definition of Chollet and Lesieur (1981) is used, which observed that in regions of low turbulent activity, sub-mesh modeling is not necessary and that local manifestations of turbulence are dissipated where the sub-mesh scales are found – where the phenomena are approximately homogeneous and isotropic. As such, they define turbulent viscosity through the local kinetic energy spectrum, $E(K_C, t)$:

$$\nu_t(K_C, t) = \frac{2}{3} C_K^{-3/2} \sqrt{\frac{E(K_C, t)}{K_C}} \tag{21}$$

where, $C_k = 1{,}4$ is the Kolmogorov constant and K_C is the cut-off wave number.

The local kinetic energy spectrum is calculated using the Second Order Velocity Structure Function model, $\overline{F}_2$, (Métais and Lesieur, 1992**):**

$$\overline{F}_2(\mathbf{x}, \Delta, t) = \left\| \overline{\bar{\mathbf{u}}(\mathbf{x}, t) - \bar{\mathbf{u}}(\mathbf{x} + \mathbf{r}, t)} \right\|^2_{\|\mathbf{r}\| = \Delta} \tag{22}$$

The "average" operator is applied between the velocities $\bar{\boldsymbol{u}}(\boldsymbol{x} + \boldsymbol{r}, t)$, calculated on points on the surface of a sphere with the center at $\boldsymbol{x}$ and radius $\|\boldsymbol{r}\| = \Delta$, and the velocity $\bar{\boldsymbol{u}}\ (\boldsymbol{x}, t)$, calculated on the flow point defined by $\boldsymbol{x}$, where it is desired to determine the turbulent activity. Métais and Lesieur (1992) use the Second Order Velocity Structure Function to calculate the turbulent viscosity according to the equation:

$$\nu_t(\boldsymbol{x}, \Delta, t) = 0{,}105 C_K^{-3/2} \Delta \sqrt{\overline{F_2}(\boldsymbol{x}, \Delta, t)} \tag{23}$$

In the context of the Discrete Vortex Method, Alcântara Pereira *et al.* (2002) adapted the definition of the Second Order Velocity Structure Function to the Lagrangian scheme in two dimensions, such as:

$$\overline{F}_{2k} = \frac{1}{NVC}\sum_{j=1}^{NVC}\left\|\bar{\boldsymbol{u}}_k\left(\boldsymbol{x}_k,t\right)-\bar{\boldsymbol{u}}_j\left(\boldsymbol{x}_k+\boldsymbol{r}_{kj},t\right)\right\|^2\left(\frac{\sigma_{0k}}{r_{kj}}\right)^{2/3} \tag{24}$$

In Eq. (24), NVC are the Lamb discrete vortices found inside the circle crown from the reference center and σ_{0k} is the Lamb vortex core. It is required to make the correction $\left(\sigma_{0k}/r_{kj}\right)^{2/3}$ because the vortices of the crown are not equidistant from the vortex under analysis. So, $\overline{F}_{2k}$ is a local statistical average. This function represents, physically, the turbulent activities in the vicinity of the vortex located at k position.

This formulation is very appropriate to the Discrete Vortex Method, since the Second Order Velocity Structure Function uses the concept of velocity fluctuations (velocity differences) and not the concept of derivatives.

To reduce the computational cost, the turbulent viscosity at points in the fluid domain where vortices exist, is calculated with the inclusion of an adaptation made for the algorithm developed by Andrade *et al.* (2016). The idea is based on using a box structure to identify groups of discrete vortices confined to the crown of a vortex of interest that is found in a certain region of the fluid domain. This solution was originally designed by Andrade *et al.* (2016) for cylinders subjected to cross flow.

4. RESULTS

In the simulations performed by Ricci *et al.* (2003), using the two-dimensional Lagrangian vortex method and LES modeling, a pair of elementary vortices clouds was represented with the aid of the potential vortex theory (Hirata *et al.*, 2002) to bypass excessive computational time. The results presented showed good agreement with experimental data for a Re = 7,650 (Liu and Srnsky, 1990).

The present work does not require the use of potential vortex theory and simulations are performed for Reynolds numbers: 7,650; 10,000; 100,000 and 1,000,000. Also considering dimensionless cross wind conditions equal to 0, 0.02 and 0.04.

Figure 2 shows the results of the simulations for Re=7,650. In Fig. 2a it can be seen that the trajectory of the centroid of the right primary vortical structure, sheds from the starting point, follows the trend of the experimental data during the simulation. The primary structures make a downward movement from the shedding point until they interact with the boundary layer of the ground, which results in the movement changing to an upward condition up to a maximum height of around 1.3 (Fig. 2a and Fig. 3). In Fig. 2b, it is possible to verify the influence of the magnitude of the cross winds in the referred vortical structure, which in addition to being pushed proportionally with the increase in the intensity of the cross wind still reaches maximum ascending heights higher than in the condition without cross wind, for example, around 1.6 for a 0.04 cross wind.

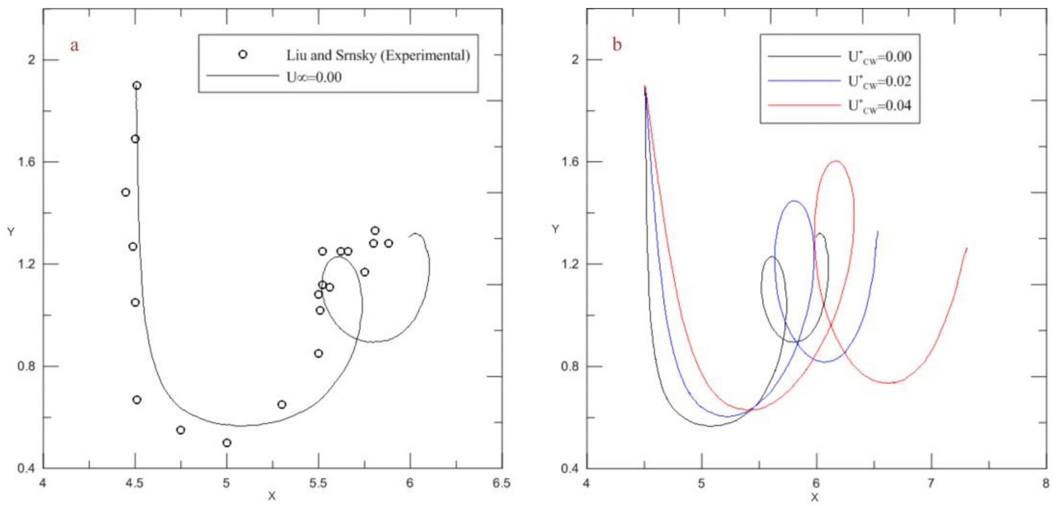

Figure 2. (a) Comparison of the present simulation with the experimental result of Liu and Srnsky (1990) for Re = 7,650 (without cross wind); (b) Influence of the magnitude of the cross winds on the trajectory of the vortical structure.

Through Fig. 3, it is possible to follow the development of the vorticity field of the primary structures (left and right), without the influence of cross wind, from the point of their respective shedding until their interaction with the boundary layer of the ground. From this interaction with the ground secondary structures are formed.

The representative trajectories of the right vortical structure, without the action of cross wind, were also generated for the other Reynolds numbers considered, according to Fig. 4a.

In order to consider aviation applications, simulations were also carried out for wing tip vortices of a Boeing 757-200. The following representative parameters were adopted problem (Proctor *et al.*, 1998): $\Gamma_0 = 306,9\ m^2/s$ and $b_0 = 29,80\ m$. The trajectory of the vortical structures for different cross wind conditions for a Boeing 757-200 is shown in Figure 4b.

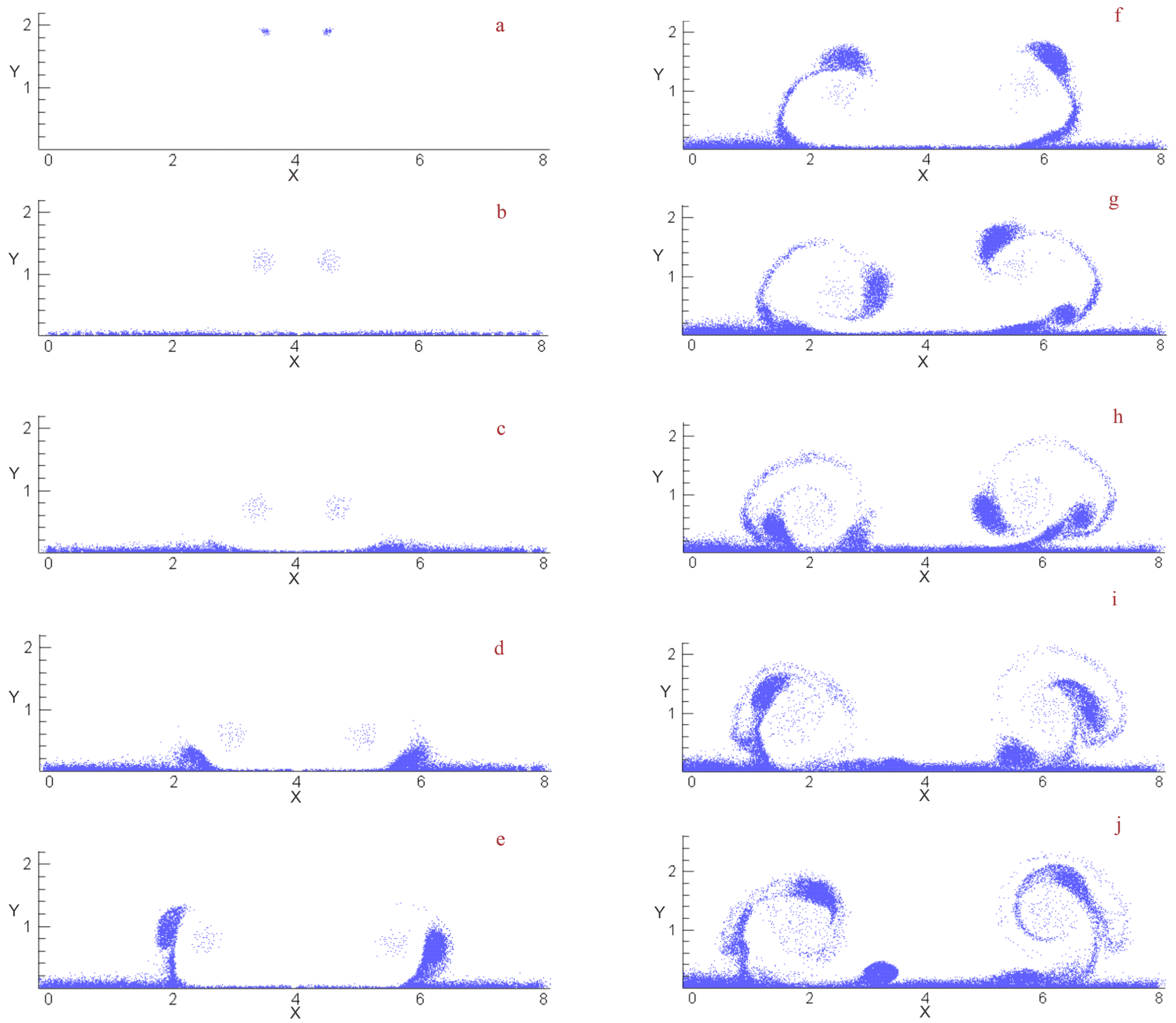

Figure 3. Evolution of the vorticity field without cross wind and interaction of the vortical structures of the aircraft wake with the ground to Re = 7,650. Simulation steps: (a) 0, (b) 100, (c) 200, (d) 300, (e) 400, (f) 500, (g) 600, (h) 700, (i) 800 and (j) 900.

Following the evolution of the movement of the vortical structures generated by the Boeing 757-200 (Fig. 5) under a cross wind equal to 0.04, it can be seen that in their descending paths these primary structures interact with the boundary layer of the ground immediately creating new structures (Fig. 5c and Fig. 5d). At this point, it can be seen that two new vortical structures are formed, close to the ground (Fig. 5d), for each of the primary structures: one (larger) starts a circular movement around the primary structure, but with rotation, in its own center, in the opposite direction to the rotation of the primary structure (Fig. 5e); the other (smaller) structure initially tends to move away from the primary structure.

In their ascending paths, the primary structures reach heights equivalent to that of the initial shedding point (Fig. 5i and Fig. 5j). In addition, new vortex structures appear close to the ground during the simulation.

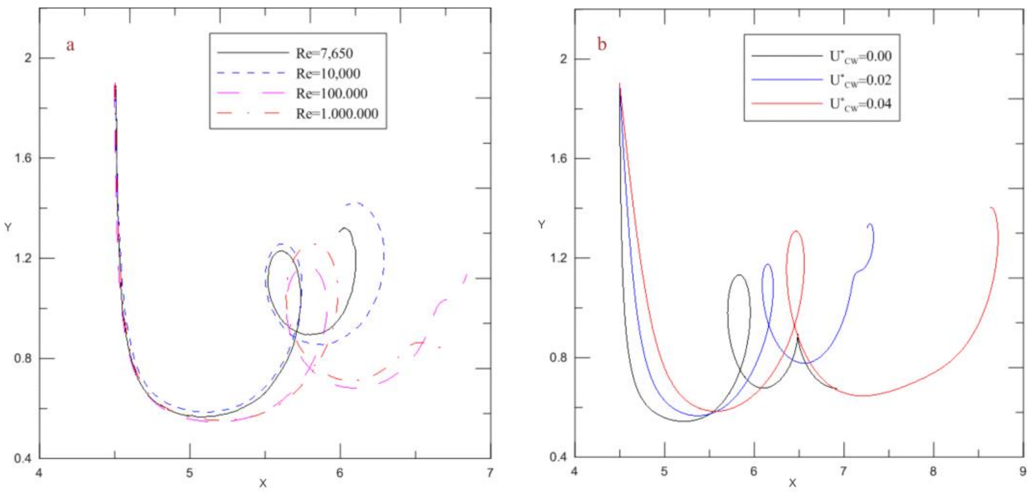

Figure 4. (a) Trajectory of vortical structure for different Reynolds numbers without cross wind; (b) Trajectory of the vortical structure for different cross wind conditions for Boeing 757-200.

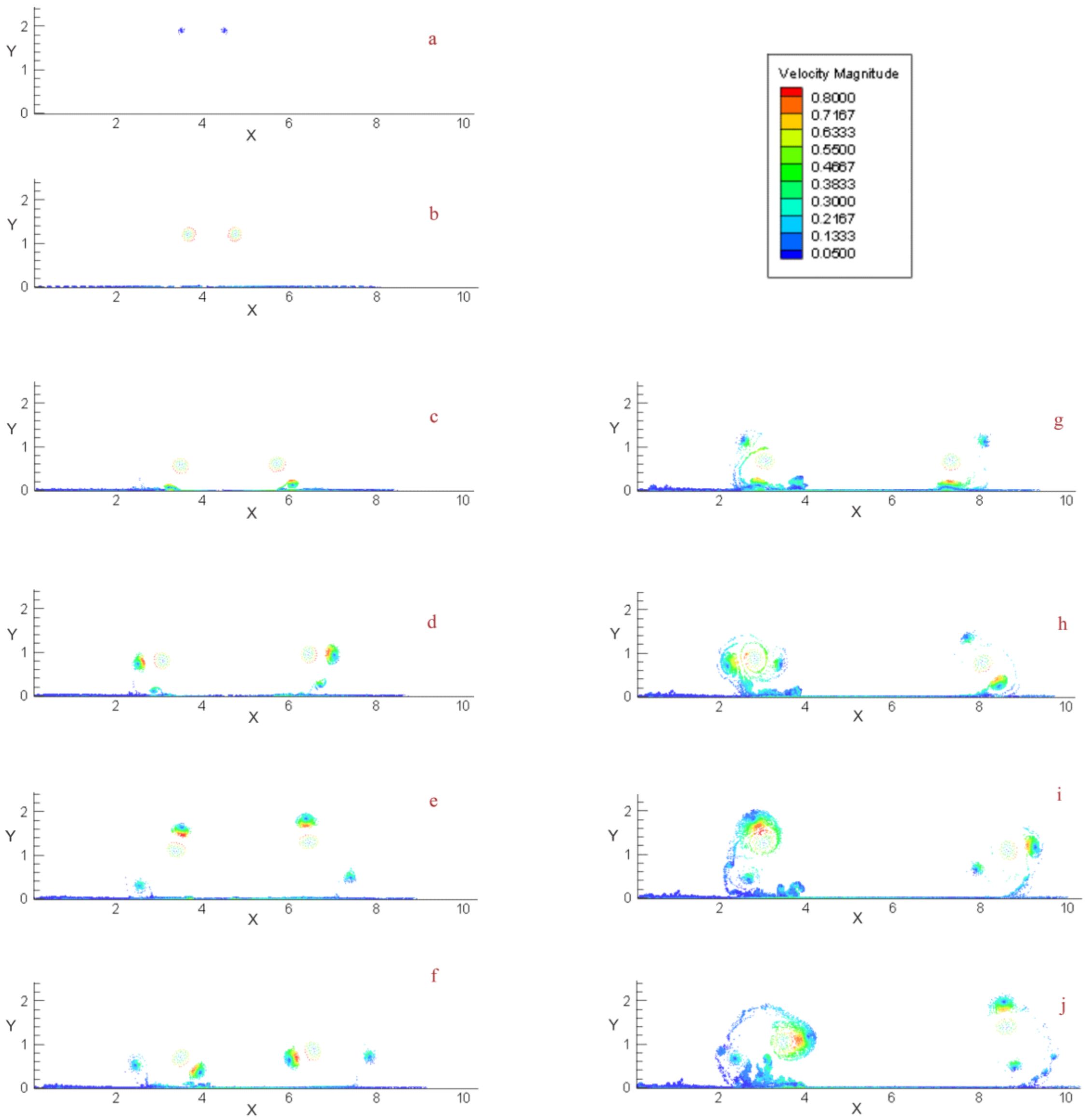

Figure 5. Evolution of the velocity field interaction of the vortical structures of the aircraft wake with the ground under a cross wind equal to 0.04 for the Boeing 757-200. Simulation steps: (a) 0, (b) 100, (c) 300, (d) 400, (e) 500, (f) 600, (g) 700, (h) 800, (i) 900 and (j) 1,000.

For the simulations were used i7 type computers (available at the computational laboratory of the Federal University of Itajubá) with 8 processing cores and with the following configurations: Intel(R) Core(TM) i7-2600 CPU @3.40GHz and 7.8 GB de RAM; Windows 10 system; Intel ® Parallel Studio Xe Cluster Edition Compiler.

Only the routines with the highest computational cost were parallelized, that is, routines related to the Biot-Savart's law (responsible for vortex-vortex interactions) whose computational cost is proportional to $O(N^2)$ and the routine responsible for solving the systems of linear equations. Fig. 6a shows the performance difference between the parallelized and the serial code, during the simulation time intervals.

The application of parallel processing in the developed computational code allowed a reduction of approximately 73% in the total processing time (Fig. 6b). The total time for a typical simulation with serial processing code is 215 minutes. However, for one typical simulation with parallel processing code it is only 57.3 minutes. This computational gain can be even greater with the use of better machines processing configurations (or clusters).

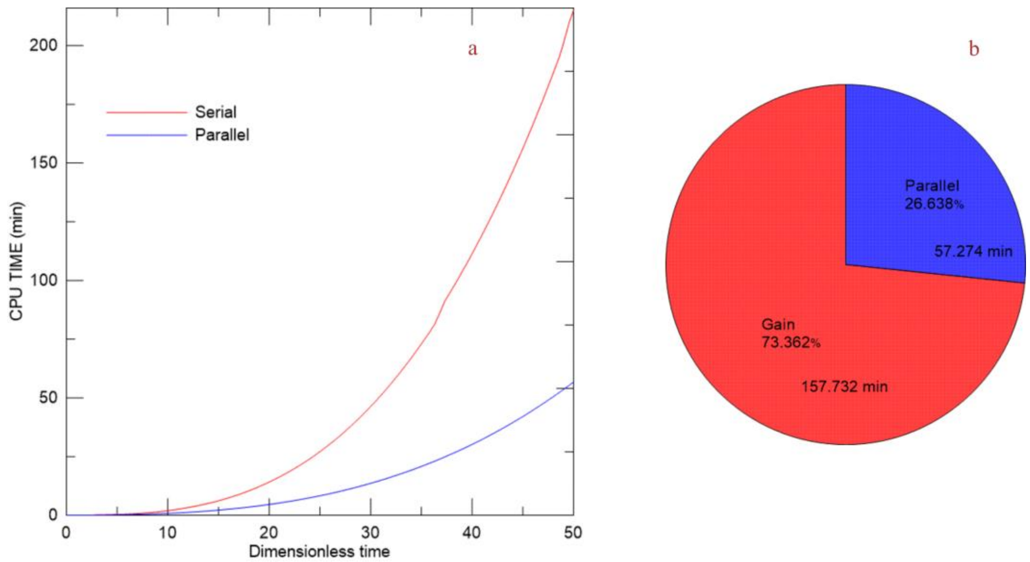

Figure 6. (a) Performance of parallel programming during a typical simulation; (b) Total computational time gain with OpenMP for a typical simulation.

5. CONCLUSIONS

The two-dimensional turbulent flow of aircraft wing tip vortices has been successfully simulated using the Discrete Vortex Method integrated with LES turbulence modeling.

From the simulations for the analysis of the interaction of aircraft vortices wake with ground and crosswinds, it was possible to obtain quantitative results of the vorticity fields. It also allowed qualitative analyzes of the deformations of the primary structures, as well as the secondary ones, the dynamics of the flow and the effects involved. And the parallel implementation OpenMP allowed the simulation of many cases in a timely manner, with results with greater accuracy and precision due to the methods used (Image method and second-order Adams-Bashforth scheme), which would be restrictive in an approach with serial programming. The final time of a typical numerical simulation was reduced around 73%.

In the next works, effects of roughness (Oliveira *et al*., 2020; Alcântara Pereira *et al*., 2020) on the ground and heat transfer by turbulent mixed convection from the heated ground to the fluid domain will be included. The inclusion of these effects has already been done in a computational code made at home and the turbulence modeling described in this paper, as well as the strategies adopted to reduce computational time, made this possible.

6. REFERENCES

Alcântara Pereira, L.A., 2002. *Simulação numérica do escoamento ao redor de perfis aerodinâmicos montados em grades lineares de turbomáquinas utilizando o Método de Vórtices Discretos com Modelagem Sub-Malha de Turbulência*. Ph.D. thesis, Federal University of Itajubá, Brazil.

Alcântara Pereira, L.A., de Oliveira, M.A., de Moraes, P.G. and Bimbato, A.M., 2020. “Numerical experiments of the flow around a bluff body with and without roughness model near a moving wall”. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*. Vol. 42, pp. 1-17.

Andrade, C.L., Alcântara Pereira, L.A. and Bimbato, A. M., 2016. “Boxes structure construction around the clusters of vortex elements to reduce the computational cost of a lagrangian vortex method with roughness model”. In *Proceedings of the 16th Brazilian Congress of Thermal Sciences and Engineering - ENCIT2016*. Vitória, Brazil.

Bimbato, A.M., Alcântara Pereira, L.A. and Hirata, M.H., 2011.“Study of the vortex shedding flow around a body near a moving ground”. *Journal of Wind Engineering & Industrial Aerodynamics,* Vol. 99, pp. 7-17.

Chollet, J.P., Lesieur, M., 1981. “Parameterization of Small Scales of Three-Dimensional Isotropic Turbulence Utilizing Spectral Closures”. *Journal of the Atmospheric Sciences*, Vol. 38, pp. 2747-2757.

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

Hirata, M.H., Ricci, J.E.R. and Oliveira, W., 2002. “Dynamic Behaviour of Aircraft Wake During Landing and Take Off Operations”. In *Proc. 9th Brazilian Congress of Thermal Engineering and Sciences – ENCIT2002*, Caxambu, Brazil.

Kamemoto, K., 1994. “Development of the Vortex Methods for Grid-Free Lagrangian Direct Numerical Simulation”. In *Proceedings of the Third JSME-KSME*, Sendai, Japan.

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

Kuzmina, K., Marchevsky, I., Moreva, V., 2015. “Parallel Implementation of Vortex Element Method on CPUs and GPUs”. *4th International Young Scientists Conference on Computational Science*, Vol. 66, pp. 73–82.

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.

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

Los Reis, J.H.F. and Alcântara Pereira, L.A., 2016. “Particle-Particle Interactions in Parallel Computations”. In *Proceedings of the 16th Brazilian Congress of Thermal Sciences and Engineering*, Vitória, Brazil.

Métais, O., Lesieur, M., 1992, “Spectral large-eddy simulation of isotropic and stably stratified turbulence”. *Journal of Fluid Mechanics*. Vol. 239, pp. 157-194.

Mustto, A.A., 1998. *Simulação Numérica do Escoamento em torno de um Cilindro Circular com e sem Rotação Utilizando o Método de Vórtices*. Master thesis, Federal University of Rio de Janeiro, Rio de Janeiro, Brazil.

Oliveira, M.A, Moraes, P.G., Andrade, C.L., Bimbato, A.M. and Alcântara Pereira, L.A., 2020. “Control and Suppression of Vortex Shedding from a Slightly Rough Circular Cylinder by a Discrete Vortex Method”. *Energies*. Vol 13, pp. 1-23.

Proctor, F.H, Hinton, D.A., Han, J., Schowalter, D.G. and Lin, Y.L., 1998. “Two Dimensional Wake Vortex Simulations in the Atmosphere: Preliminary Sensitivity Studies”. *AIAA Meeting Papers on Disc*., Paper 97-0056.

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”. In *Proceedings of the 17th International Congress of Mechanical* Engineering - COBEM 2003, São Paulo, Brazil.

Santiago, V.S., Bodstein, G.C.R., 2006. "Comparative Study of Four Different Methods of Viscous Diffusion for Application in the Vortex Method". In *Proceedings of the 11th Brazilian Congress of Thermal Sciences - ENCIT2006*, Curitiba, Brazil.

Smagorinsky, J., 1963. “General Circulation Experiments with the Primitive Equations”. *Monthly Weather Review*, Vol. 91, pp. 99-164.

Wakim, A., Jacquin, L., Brion, V. and Dolfi-Bouteyre, A., 2017. “Vortex Pair in ground vicinity: Optimal Perturbation and Optimal Control”. *23ème Congrès Français de Mécanique*.

Zheng, Z.C. and Ash, R.L., 1996. “Study of Aircraft Wake Vortex Behavior Near the Ground”. *A.I.A.A. Journal*, Vol. 34, N^{o}.3, pp. 580-588.

7. RESPONSIBILITY NOTICE

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