26th ABCM International Congress of Mechanical Engineering
November 22-26, 2021. Florianópolis, SC, Brazil

# COB-2021-0239
# IMPLEMENTATION OF AN ACCELERATOR ALGORITHM TO REDUCE THE COMPUTATIONAL COSTS IN LAGRANGIAN SIMULATIONS

**Marília Fernandes Vidille**
School of Engineering, São Paulo State University (UNESP), Guaratinguetá, SP, Brazil
marilia.f.vidille@unesp.br

**Luiz Antonio Alcântara Pereira**
Mechanical Engineering Institute, Federal University of Itajubá (UNIFEI), Itajubá, MG, Brazil
luizantp@unifei.edu.br

**Alex Mendonça Bimbato**
School of Engineering, São Paulo State University (UNESP), Guaratinguetá, SP, Brazil
alex.bimbato@unesp.br

***Abstract.*** *The purely lagrangian discrete vortex method is a computation technique in which meshes are avoided. The method is used to represent a flow property as the vorticity field. In order to represent such property, the vorticity transport equation is obtained from the Navier-Stokes and continuity equations and the viscous splitting algorithm is used; according to it, one can solve the advection and diffusion equations separately but in the same time increment, just to make the vortex method implementation easier. The diffusion equation is solved by the classical random walk method; according to it, each discrete vortex used to represent the vorticity field is dislocated in a radial and circumferential direction to simulate vorticity scattering. The advection equation is solved using the material derivative in a classical lagrangian approach. One of the most problems when using lagrangian simulations concerns to the solution of advection equation, since it is necessary to compute the velocity field in the position occupied by each one of Z discrete vortex present in the computational domain. In a typical problem the velocity field is composed by three contributions: (i) the incident flow; (ii) the boundary surfaces; (iii) the vortex-vortex interaction. The vortex-vortex interaction is especially expensive since its computational cost is proportional to* $Z^2$ *when the Biot-Savart law is used. The aim of the present work is to implement a fast multipole method to reduce the computational costs associated to the Biot-Savart computations. The fast multipole method divides the computational domain using boxes which contains the discrete vortices; the goal of the algorithm is to promote more interactions between boxes than interactions between particles. One hopes that the computational costs can be proportional to Z log Z or even to Z. In order to simplify the numerical implementation of the fast multipole method the physical situation investigated is the vortices detached from airplane wings; there are no incident flow and solid boundaries in the problem. The pair of vortices is represented by Lamb discrete vortices and the velocity field is composed only by the vortex-vortex interaction which is computed using only the Biot-Savart law and using the fast multipole method to verify the CPU time reduction.*

***Keywords:*** *discrete vortex method, fast multipole method, CPU time reduction, lagrangian description*

## 1. INTRODUCTION

The growth in air traffic and the use of increasingly larger and more powerful aircraft make landing and takeoff operations critical. On the other hand, reducing the time interval between takeoff and landing operations is of paramount importance to improve the efficiency of the air traffic control system at the largest airports in the world.

The generation of vorticity from the wings of the airplane is a disturbance of such magnitude that it can changes the fluid velocity field abruptly, causing serious accidents to a second aircraft that will operate on the same runway several minutes after the first pass. Thus, preventing an aircraft from operating in the wake of another that precedes it constitutes the first requirement for air traffic control at airports (Machol, 1993).

According to Critchley and Foot (1991), the highest frequency of accidents occurs with subsequent operations in the range of 30 ~ 70m in height, when the wake vortices interact with the airport track. Besides that, Zheng and Ash (1996) found that the free vortices created from the wings have a circulation that develops in the same proportion as the size and speed of the aircraft that generates them and persist for significant distances. It is not difficult to observe this problem in large airports in which the landing and take-off operations of large aircrafts occurs frequently. As an example, the Airbus A340-600 has a wingspan of 63.70m, a length of 75.30m and a height of 17.80m.

Therefore, the analysis of the behavior of aircraft wake and its dissipation is a crucial tool for air traffic operators and serves as a motivation for the development of the present work, especially when the International Air Transport Association (IATA) predicts that the number of tickets issued in the world will double in the next two decades compared to the year 2017, totaling 8.2 billion passengers in 2037. The estimate is based on an average annual growth of 3.5% in the sector and was carried out before the pandemic of SARS-CoV-2. In Figure 1 is shown the number of flights that occurred in the last two decades.

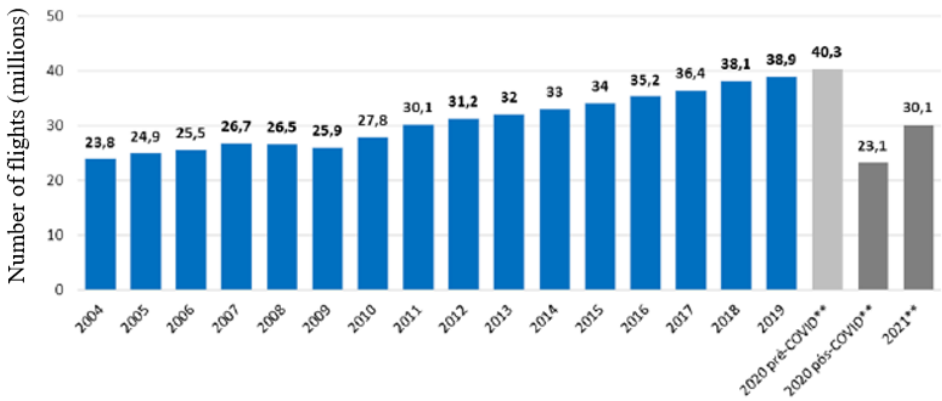



Figure 1. Number of flights operated by the global aviation industry between 2004 and 2021 (Statista, 2020).

The problem analyzed consists in the numerical simulation of the behavior of two vortex structures, which represent the free vortices generated at the aircraft's wings. These vorticity structures maintain a constant spacing as they travel vertically downward. In order to perform such simulations, a Lagrangian vortex method is employed to simulate numerically the vorticity field dynamics. The two-dimensional flow is investigated at a Reynolds number of $7.5\times10^4$.

The vortex method have been developed and applied for analysis of complex, unsteady and vortical flows in relation to problems in a wide range of industries, because they consist of simple algorithm based on physics of flow (Kamemoto, 2004). Vortex cloud modeling offers great potential for numerical analysis of important problems in fluid mechanics. A cloud of discrete vortices is used in order to simulate the vorticity and each individual free vortex of the cloud is followed during the numerical simulation in a typical Lagrangian scheme. This is in essence the foundations of the vortex method (Chorin, 1973; Sarpakaya, 1989; Lewis, 1999; Kamemoto, 2004; Alcântara Pereira *et al*., 2004; Stock 2007; Bimbato *et al*. 2018).

Vortex method offers a number of advantages over the more traditional Eulerian schemes: (a) the absence of a mesh avoids stability problems of explicit schemes and mesh refinement problems in regions of high rates of strain; (b) the Lagrangian description eliminates the need to explicitly treat convective derivatives; (c) all the calculation is restricted to the rotational flow regions; (d) no boundary condition is required at the downstream end of the flow domain.

On the other hand, the lagrangian vortex method also has some disadvantages such as the CPU time necessary to compute the flow velocity field which precludes the use of more accurate vorticity diffusion schemes to replace the classical random walk method developed by Chorin (1973).

The aim of the present study is to make feasible the use of core spreading method developed by Rossi (1996) to take into account the vorticity field diffusion. It is known that the lateral wind influences the problem and the wake developed from the aircraft wings interacts with the airport track. However, in the present work none of these influences are considered, what means that the focus is only on vorticity advection and diffusion. In fact, the purpose here is to implement the fast multipole method (Greengard and Rokhlin, 1987; Nishimura, 2002; Wolf, 2011; Yokota and Barba, 2013; Ricciardi, 2016; Ricciardi *et al.*, 2017a; Ricciardi *et al.*, 2017b) to reduce the computational time to calculate the velocity field. The fast multipole method, FMM, is listed as one of the top 10 algorithms of the twentieth century (Cipra, 2000) and the success of its implementation with the vortex method will make feasible the use of the core spreading method to solve the vorticity diffusion equation more accurately.

Finally, it is important to mention that the authors of this paper have developed the present vortex method to simulate the macro scale phenomena, therefore the smaller scale ones are taken into account through the use of a second order velocity function (Alcântara Pereira *et al.*, 2002; Bimbato *et al*., 2012). In the present paper, the effects of small scale are still not considered.

## 2. GOVERNING EQUATIONS

In Figure 2 is shown the fluid domain $\Omega$, where the vortex structures detached from the airplane wings at a height *h*, have a circulation $\pm\Gamma$, which is related to the lift force, $F_L$, through the Kutta-Joukowsky equation, during a landing or a take-off procedure:

$$\Gamma = \frac{F_L}{\rho U w} \tag{1}$$

In Eq. 1 $\rho$ is the fluid density, $U$ is the fluid velocity and $w$ is the wingspan.

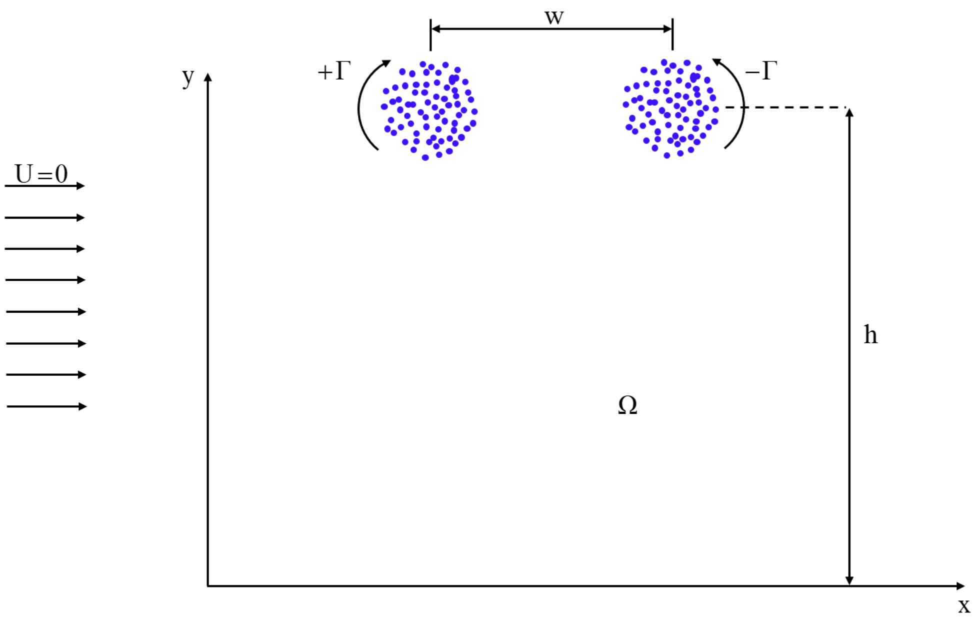



Figure 2. Model used in the numerical simulation.

The incompressible flow of a Newtonian fluid with constant properties in a two-dimensional region $\Omega$ is governed by the continuity and the Navier-Stokes (N-S) equations, which can be written in the form:

$$\nabla^* \cdot \mathbf{u}^* = 0 \tag{2}$$

$$\frac{\partial \mathbf{u}^*}{\partial t^*} + \mathbf{u}^* \cdot \nabla^* \mathbf{u}^* = -\frac{1}{\rho} \nabla^* p^* + \nu \nabla^{*2} \mathbf{u}^* \tag{3}$$

In Eqs. (2) and (3) $\boldsymbol{u}$ is the velocity vector field, $p$ is the pressure field, $\nu$ is the fluid kinematic viscosity coefficient and the symbol $*$ means dimensional quantities.

In order to make Eqs. (2) and (3) non-dimensional, the following characteristic quantities are used:

$w \equiv$ characteristic length;
$\Gamma/w \equiv$ characteristic velocity.

Thus, the continuity and the Navier-Stokes equations become:

$$\nabla \cdot \mathbf{u} = 0 \tag{4}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{u} \tag{5}$$

The Reynolds number is defined as

$$\mathrm{Re} = \frac{\Gamma}{\nu} \tag{6}$$

Taking the curl of N-S equations the pressure term is eliminated and the non-dimensional 2-D vorticity transport equation is obtained:

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \frac{1}{\mathrm{Re}} \nabla^2 \omega \tag{7}$$

In Eq. (7) $\omega$ is the only non-zero component of the vorticity vector. So, $\partial\omega/\partial t$ represents the local vorticity variation rate, $\boldsymbol{u} \cdot \nabla\omega$ represents the advective vorticity variation rate and $\nabla^2\omega/Re$ is the diffusive vorticity variation rate. As will be explained in the next section, since the problem is solved using a lagrangian numerical method, it is not necessary to deal with the non-linear term, $\boldsymbol{u} \cdot \nabla\omega$.

## 3. SOLUTION METHOD

The solution method applied in this paper uses the advection-diffusion splitting algorithm (Chorin, 1973). According to it, one assumes that in the same time increment the advection and the diffusion of the vorticity can be independently handled and are governed by

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = \frac{D\omega}{Dt} = 0 \tag{8}$$

$$\frac{\partial \omega}{\partial t} = \frac{1}{\mathrm{Re}} \nabla^2 \omega \tag{9}$$

Advection is governed by Eq. (8), which is computed using the material derivative, making evident the Lagrangian characteristic of the solution method. Since there is no solid boundaries neither lateral wind, the velocity field is calculated by

$$\mathbf{u}(\mathbf{x}, t) = \mathbf{uv}(\mathbf{x}, t) \tag{10}$$

In Eq. (10), the velocity $\boldsymbol{uv}(\boldsymbol{x},t)$ is due to the vortex cloud. The vortex-vortex interaction requires an expensive convolution step of $O(Z^2)$ calculations, where $Z$ is the total number of discrete vortices present in the computational domain. It is obvious that this feature imposes a heavy limitation on lagrangian methods to use them to solve practical engineering problems. In the present paper the authors implemented a fast multipole method based on the work of Ricciardi (2016) in order to reduce the CPU time of the velocity field computations.

The algorithm of the fast multipole method consists of clustering the influence of elements close to each other into multipole expansions, and then evaluating their interaction at distant locations. Thus, the influence among different groups of discrete vortices is computed faster than the direct computation done by Biot-Savart law.

In order to cluster the influence of particles (discrete vortices) close to each other into multipole expansions, the computational domain is divided into $4^L$ boxes ($L$ is the maximum refinement level); the core of the FMM method is to promote more interactions among boxes than interactions among particles. This characteristic makes the computation faster. So, lists of well-separated boxes and their neighbors are determined in a pre-processing step. It is possible to create two interaction lists: considering $B$ as the parent of a box $b$, $B$ can have up to 9 boxes sharing a node, being 8 neighbors and $B$ itself. Each one of these 9 boxes have 36 children boxes, which can be classified as $b$ itself, the near neighbors of $b$ (8 boxes) and the 27 well-separated boxes in the interaction list of $b$.

The first step (called particle-to-multipole) is the creation of multipole expansion, $a$, in the center of the box $b$, in the finest level $L$, using a Taylor's series truncated after $p$ terms, for $n$ discrete vortex with intensity $\Gamma_i$, with a complex distance of $z_i$ from the center of the box (Eq. 11). The sum of all vortex particle intensities is given by Eq. (12).

$$a(b, k, L) = \sum_{i=1}^{n} -\Gamma_i \frac{z_i^k}{k} \tag{11}$$

$$Q(L)=\sum_{i=1}^{n}\Gamma_i \tag{12}$$

In the sequence (step called multipole-to-multipole), the influences are translated to the center of the parent box $B$, at the level $l-1$, resulting in $a(B,k,l-1)$- see Eq. (13). The intensity of the multipoles in level $l-1$, $Q(l-1)$, is the sum of the children's intensities, $Q(l)$ – see Eq. (14).

$$a(B,k,l-1)=\sum_{i=1}^{4}\left\{\left[\sum_{kk=1}^{k}a_i(kk,l)z_i^{k-kk}\binom{k-1}{kk-1}\right]-\left[Q_i(l)\frac{z_i^k}{k}\right]\right\} \tag{13}$$

$$Q(l-1)=\sum_{i=1}^{4}Q_i(l) \tag{14}$$

The multipole-to-multipole steps are made up to level 2, which is the lowest level possible to have multipole-to-local calculations (Eq. 15). The interaction list of a box $b$ contains 27 boxes $j$ that interact via multipole-to-local with objective box $B$, resulting in the multipole-to-local variable $b(B,kk,l)$.

$$b(B,kk,l)=\sum_{j=1}^{nbox}\left[\frac{\sum_{k=1}^{p}a_j(k,l)\left(\frac{-1}{z_j}\right)^k\binom{kk+k-1}{kk-1}}{z_j^{kk}}-\frac{Q_j(l)}{kk\,z_j^{kk}}\right] \tag{15}$$

In Eq. (15), $nbox$ represents all the non-empty boxes from the interaction list of $B$. In this equation, $z_j$ is the complex distance between the box with the multipole representation to that with the local representation, and $a$ and $Q$ are obtained from the previously calculated multipole expansion from box $j$ in the interaction list of $b(l)$.

The local-to-local step represents the translation of the influence coefficients from the boxes that are in the interaction list of $B$ to a child box $b$. The influence of all distant boxes of $b$ is the sum of the multipole-to-local steps from the boxes of its own interaction list and also from the local-to-local steps from its parent ($B$). The local-to-local influence in a box in the level $l+1$ from its parent at level $l$ is given by Eq. (16); $b(k,l)$ is the local representation of the far field multipole expansions at the parent box and $z_i$ is the complex distance from the parent's center to its children´s center.

$$c(b,k,l+1)=\sum_{i=1}^{4}\left\{\sum_{kk=k}^{p}\left[b(k,l)(-z_i)^{kk-k}\binom{kk}{k}\right]\right\} \tag{16}$$

When all contributions are evaluated in a box, they are translated by another Taylor's series to all particles within the box. Hence, all the interactions from distant particles are evaluated with the steps previously described, while the near-field interactions are computed through the Biot-Savart law. Eq. (17) gives the translation from the center of a box to a particle, resulting in the induced velocity by the near field and distant particles in the vortex $i$:

$$uv_i=vv+\sum_{k=1}^{p}b(k,l)k\,z_i^{k-1} \tag{17}$$

In Eq. (17) $vv$ is the vortex-vortex interaction, $b(l)$ is the sum of local-to-local and multipole-to-multipole steps in a box in the highest refinement level $l$ and $z_i$ is the complex distance from the discrete vortex $i$ to the center of its box. All the calculations performed here to compute the velocity field used the lamb vortex model.

## 4. RESULTS

In order to evaluate the performance and the accuracy of the algorithm, no solid boundaries are presented in the fluid domain. The key idea is to focus just on the velocity field calculation due to the vortex cloud.

Thus, two identical vortex clouds positioned as shown in Figure 2 were used to simulate the vortices detached from airplane wings during a landing or a take-off procedure. Each vortex cloud is composed by $nz = 1.5 \times 10^4$ discrete vortices with intensity $\pm 1/nz$, totalizing $Z = 3.0 \times 10^4$ discrete vortices in all the numerical simulations. In order to create the clouds, the $nz$ discrete vortices were initially concentrated on the origin of the reference system; then they were submitted to a diffusion dislocation using random numbers until the more distant discrete vortex reach a distance equal to $0.1w$ (see Figure 2). This way, one ensures a gaussian distribution of the vorticity contained in the clouds as if it were a big lamb vortex. The Reynolds number used in all the simulations presented here is $7.5 \times 10^4$ and an Euler first order time marching scheme is used.

The velocity field is computed using two approaches: (i) only through the Biot-Savart law and (ii) using the FMM algorithm up to a maximum refinement level of 6. This strategy allows the comparison between both methods.

Figure 3 shows how the error in the velocity field computation increases with the refinement level, since there are more operations between boxes than operations between particles for higher refinement levels. The analysis of Figure 3 is done using fifteen terms in the Taylor series expansion.

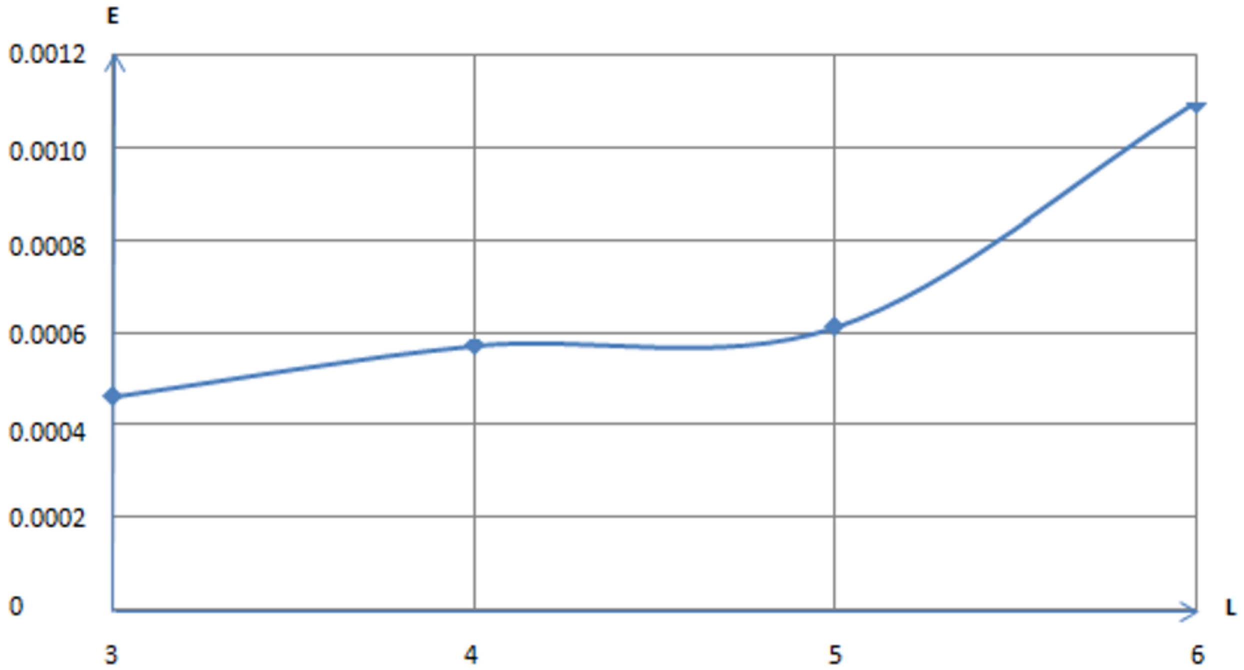



Figure 3. The influence of the refinement level in the FMM algorithm (Euler; $\Delta t = 0.025$; $p = 15$; $Z = 3.0 \times 10^4$; $Re = 7.5 \times 10^4$).

The error definition is based on Carrier *et al.* (1988) and is computed as:

$$E = \left( \frac{\sum_{i=1}^{Z} \left( u_{FMM_i} - u_{BS_i} \right)^2}{\sum_{i=1}^{Z} u_{BS_i}^2} \right)^{1/2} \quad (18)$$

In Eq. (18) $E$ is the error, $u_{FMM_i}$ is the velocity computed in discrete vortex $i$ position through the FMM method and $u_{BS_i}$ is the velocity computed in discrete vortex $i$ position through the Biot-Savart law.

As it can be seen in Figure 4, the computational time decreases as the refinement level increases until $L = 6$. So, for this number of discrete vortices present in the computational domain ( $Z = 3.0 \times 10^4$ ) and taking into account the numerical errors shown in Figure 3, it is not necessary to use more than five refinement levels to represent the region $\Omega$. This behavior happens because the total computational cost of fast multipole method algorithms is composed by near-field and far-field costs. As the refinement level increases, the near-field operations (use of Biot-Savart law) decrease, but higher refinement levels require much more FMM operations. So, there is an equilibrium between far-field and near-field costs that one's need to reach in order to optimize the velocity field calculation. As a reference, the red constant line in Figure 4 shows the CPU time necessary to compute the velocity field exclusively through the Biot-Savart law 1,000 times.

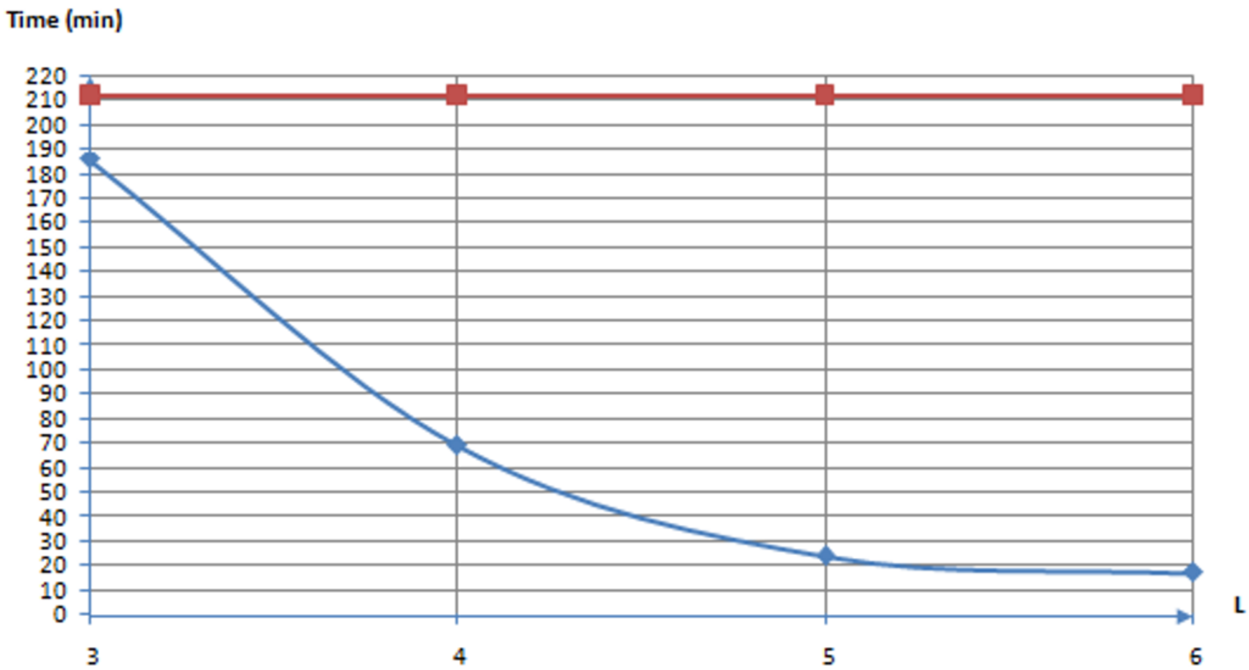



Figure 4. Computational time necessary to calculate de velocity field (Euler; $\Delta t = 0.025$; $p = 15$; $Z = 3.0\times10^4$; $Re = 7.5\times10^4$).

One of the most important use of the fast multipole methods when it is associated to vortex methods is to reduce de exponential increasing of the CPU time, which is a typical characteristic of lagrangian approaches. However, it is expected that the error in the FMM algorithms accumulates for simulations with many time steps as the ones performed with the discrete vortex method. Figure 5 shows how the error in the computation of the velocity field increases for a simulation performed up to 1,000 time steps with five levels of refinement.

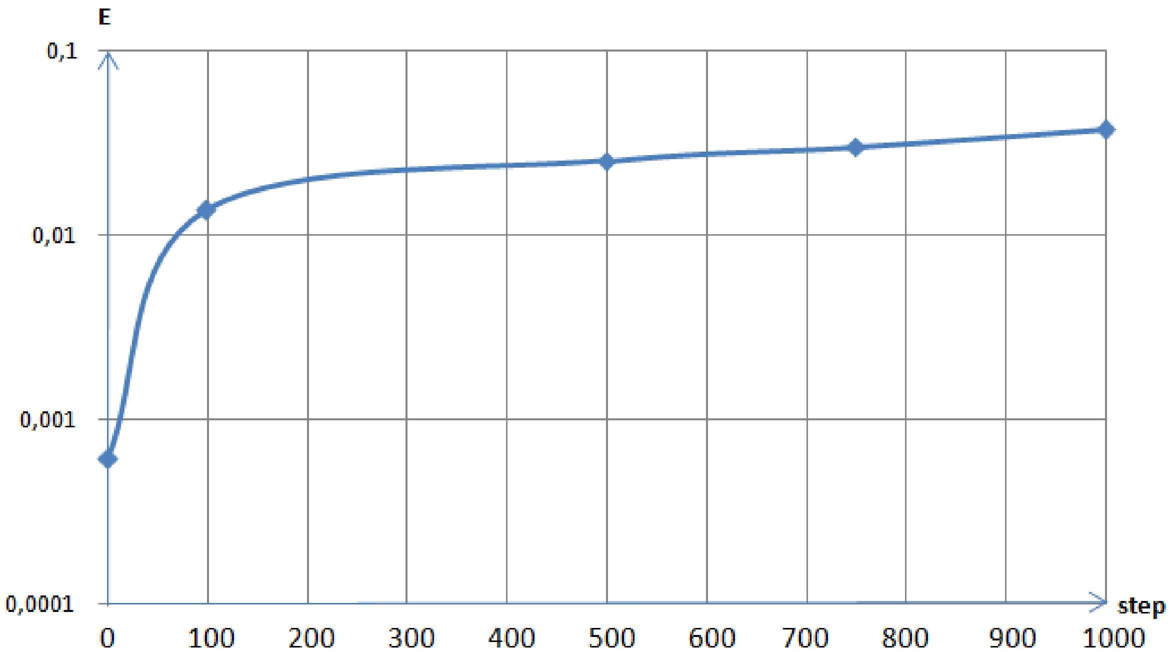
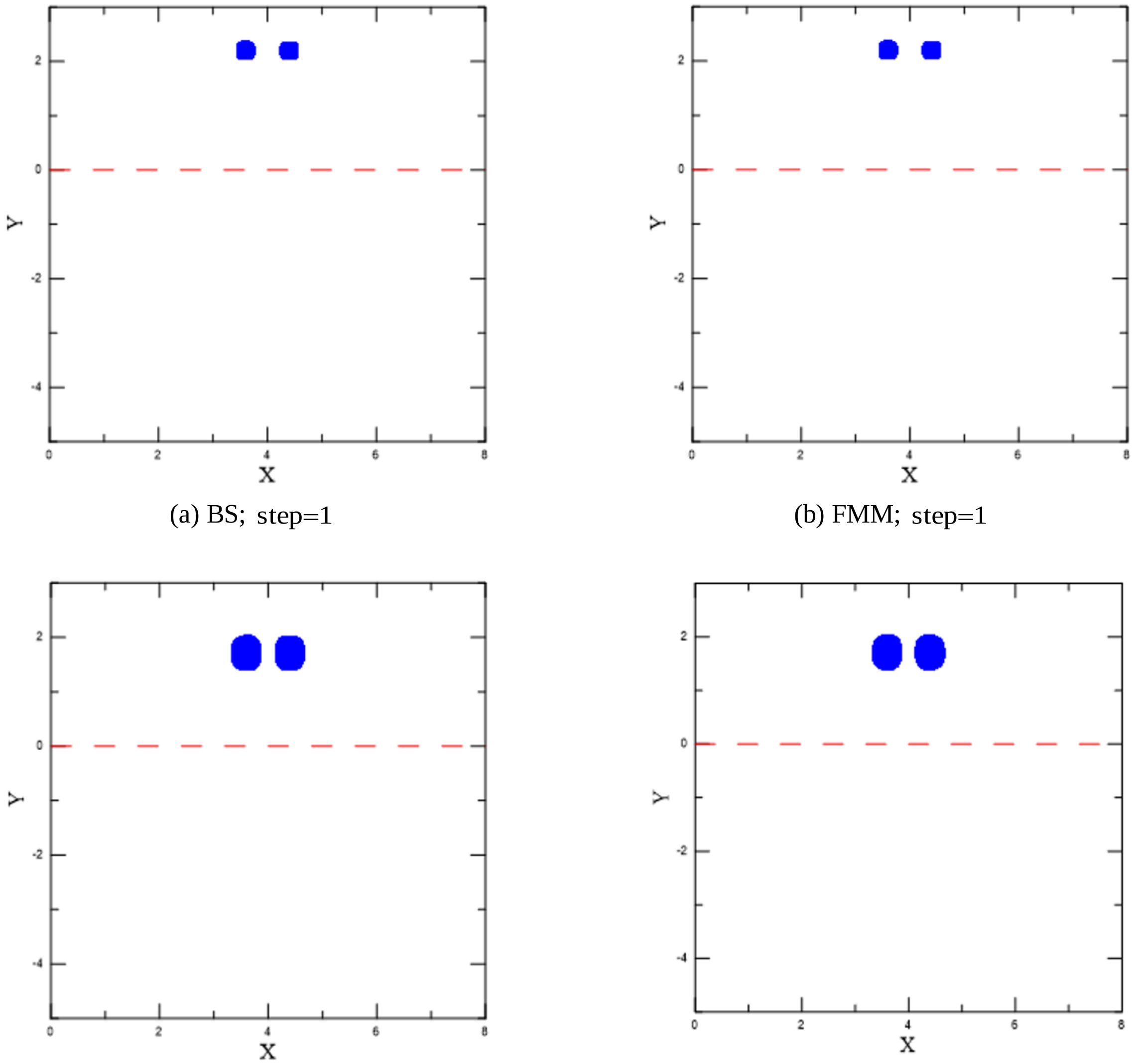


Figure 5. Accumulated error to compute de velocity field using the fast multipole method (Euler; $\Delta t = 0.025$; $p = 15$; $L = 5$; $Z = 3.0 \times 10^4$; $Re = 7.5 \times 10^4$).

It can be seen from Figure 5 that the numerical errors increase a lot, which is unacceptable because the simulations involving the discrete vortex method deals with 5,000 time steps or even more. This means that a careful investigation about the FMM algorithm is needed, especially with regard to the number of terms used in Taylor series, time marching schemes and also to the time steps used in the simulations.

Figure 6 shows the position of the vortex structures during the simulation computed through the direct summation (Biot-Savart law) and through the fast multipole method algorithm. The development of such structures is almost the same.

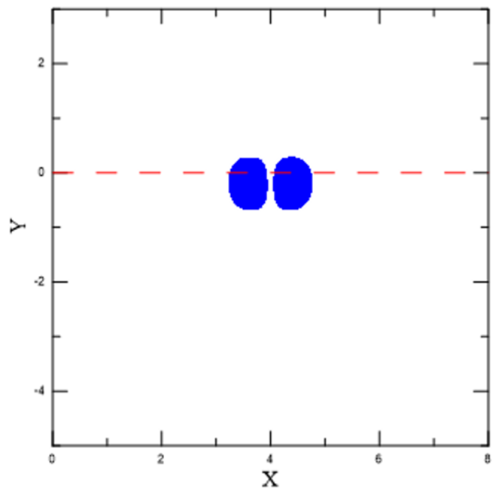



(a) BS; step=1

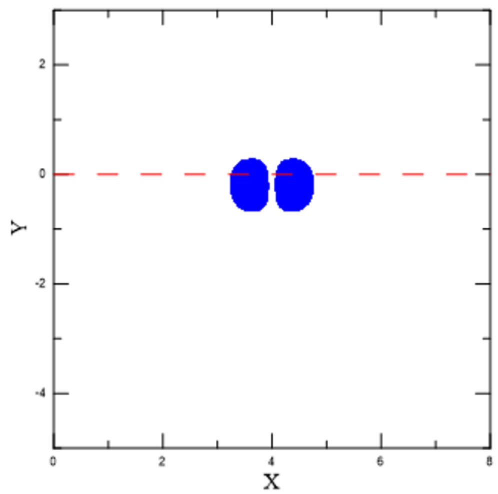
(b) FMM; step=1

(c) BS; step=100

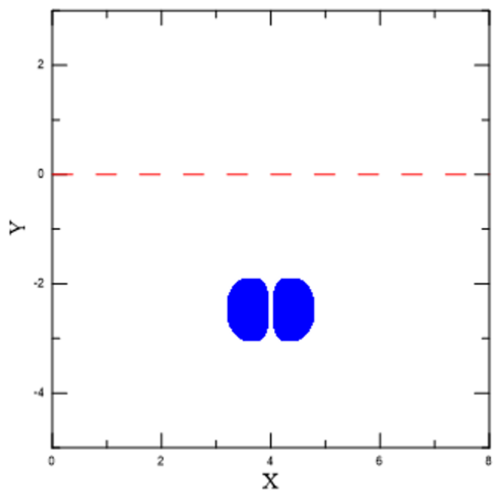
(d) FMM; step=100



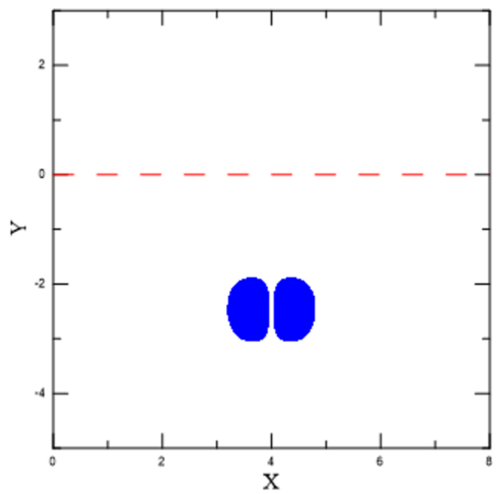

(e) BS; step=500




(f) FMM; step=500




(g) BS; step=1000




(h) FMM; step=1000

Figure 6. Vorticity field evolution computed through the Biot-Savart law, BS, and through the fast multipole method, FMM (Euler; $\Delta t = 0.025$; $p = 15$; $L = 6$; $Z = 3.0 \times 10^4$; $Re = 7.5 \times 10^4$).

## 5. CONCLUSIONS

An algorithm of the fast multipole method based on the work developed by Ricciardi (2016) was implemented to study the possibility to make feasible the use of the modified core spreading method to solve the vorticity diffusion equation, which requires much higher CPU time than that presented in this work.

This first analysis about such numerical implementation is promising, since the CPU time decreased almost $88.7\%$ (for $L = 5$) and the numerical errors varies from approximately $6{,}0 \times 10^{-4}$ in the beginning of the simulation until approximately $1{,}7 \times 10^{-1}$ in its end; it is notice that velocity vectors whose module are approximately zero makes the errors higher. Anyway, it is necessary to perform simulations with a different number of discrete vortices present in computational domain in order to evaluate the behavior of near-field and far-field costs and even the numerical errors, since the mean number of discrete vortices per box may cause influence on numerical precision.

Finally, it is important to investigate in future works how the number of terms used in the Taylor series expansion modifies the numerical errors. Besides that, it is necessary to perform an analysis of the influence of time marching schemes and time step, and also to run simulations involving vorticity generation at each time step in which the focus is on the determination of aerodynamic coefficients.

## 6. ACKNOWLEDGEMENTS

The authors would like to acknowledge São Paulo Research Foundation (FAPESP), Proc. 2020/01212-5.

## 7. REFERENCES

Alcântara Pereira, L.A., Ricci, J.E.R., Hirata, M.H. and Silveira Neto, A., 2002. "Simulation of the Vortex-Shedding Flow about a Circular Cylinder with Turbulence Modeling". *CFD Journal*, Vol. 11, No. 3, pp. 315-322.

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

Bimbato, A.M., Alcântara Pereira, L.A. and Hirata, M. H., 2012. "Corrected Lagrangian LES model for vortex method". In *Proceedings of 8th Spring School on Transition and Turbulence* – EPTT 2012. São Paulo, Brazil.

Bimbato, A.M., Alcântara Pereira, L.A. and Hirata, M.H., 2018. "Development of a new Lagrangian vortex method for evaluating effects of surfaces roughness". *European Journal of Mechanics B-Fluids*, Vol. 74, pp. 291-301.

Carrier, J., Greengard, L. and Rokhlin, V., 1988. "Multipole Algorithm For Particle Simulations". *SIAM Journal of Scientific Statistics and Computation*, Vol. 9, No. 4, pp. 669-686.

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

CIPRA, B.A., 2000. "The best of the 20th century: Editors name top 10 algorithms". *SIAM News*, Vol. 33, pp. 1–2.

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

Greengard, L. and Rokhlin, V., 1987. "A fast algorithm for particle simulations". *Journal of Computational Physics*, Vol. 73, pp. 325–348.

Kamemoto, K., 2004. "On Contribution of Advanced Vortex Element Methods Toward Virtual Reality of Unsteady Vortical Flows in the New Generation of CFD". In *Proceedings of the 10th Brazilian Congress of Thermal Sciences and Engineering* - ENCIT 2004. Rio de Janeiro, Brazil.

Lewis, R.I., 1999. "Vortex Element Methods, the Most Natural Approach to Flow Simulation - A Review of Methodology with Applications". In *Proceedings of 1st Int. Conference on Vortex Methods*. Kobe, Japan.

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

Nishimura, N., 2002. "Fast multipole accelerated boundary integral equation methods. *Applied Mechanics Reviews*, Vol. 55, pp. 299–324.

Ricciardi, T.R., 2016. *Fast multipole discrete vortex method applied to unsteady flow simulations*. M.Sc. Dissertation, University of Campinas.

Ricciardi, T.R., Wolf, W.R. and Bimbato, A.M., 2017. "A fast algorithm for simulation of periodic flows using discrete vortex particles". *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, Vol. 39, pp. 45-55.

Ricciardi, T.R., Wolf, W.R. and Bimbato, A.M., 2017. "Fast multipole method applied to Lagrangian simulations of vortical flows". *Communications in Nonlinear Science and Numerical Simulation*, Vol. 51, pp. 180-197.

Rossi, L.F., 1996. "Resurrecting Core Spreading Vortex Methods: A New Scheme that is both Deterministic and Convergent. *SIAM J. Sci. Comput.*, Vol. 17, No. 2, pp. 370-397.

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

Statista, 2020. https://www.statista.com/statistics/564769/airline-industry-number-of-flights/#:~:text=The%20number%20of%20flights%20performed,to%2016.4%20million%20in%202020. Accessed 30 november 2020.

Stock, M.J., 2007. *Summary of Vortex Methods Literature (A lifting document rife with opinion)*.

Wolf, W.R., 2011. *Airfoil Aeroacoustics: LES and Acoustic Analogy*. Ph.D. Thesis, Stanford University.

Yokota, R. and Barba, L.A., 2013. "Fmm-based vortex method for simulation of isotropic turbulence on gpus, compared with a spectral method". *Comput Fluids*, Vol. 80, pp. 17–27.

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.

## 8. RESPONSIBILITY NOTICE

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