STUDY OF LINEAR CASCADE-VORTEX CLOUD INTERACTION

Luiz Antonio Alcântara Pereira
Instituto de Engenharia Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
luizantp@unifei.edu.br

Miguel Hiroo Hirata
FAT/UERJ, Campus Regional de Resende, Estrada Resende-Riachuelo, Resende, Rio de Janeiro, Brasil
hirata@fat.uerj.br

Abstract. *In this paper the vortex method is applied to the unsteady, two-dimensional and incompressible flow that occurs during a linear cascade-vortex cloud interaction. As the flow in a linear cascade of airfoils is periodic in the y direction, the discrete vortex shedding need only be considered for a reference airfoil. In the numerical simulation a cloud of Lamb vortices is used in order to simulate the vorticity, which is generated on the body surface, develops in the boundary layer and is carried out into the viscous wake. A second vortex wake is generated next to the linear cascade, in a row of points, which moves in the y direction with constant speed. The free vortices are generated in a row of points with strength alternatively positive and negative. The pressure distribution is obtained using an integral equation derived from the pressure Poisson equation, which was first developed for a single body. The flow characteristics around the NACA 65-410 series airfoils are analyzed in this study.*

Keywords*: discrete vortex method, pressure distribution, deflection angle, linear cascade-vortex cloud interaction*

1. Introduction

The study of aerodynamic loading variations has many engineering applications, including complex structures like buildings, helicopter rotor blades, wind turbines and blades in turbomachines. Understanding the vortex shedding flow in these complex structures is of fundamental and practical importance in the field of engineering as they can create destructive effect and change the aerodynamic performance. The aerodynamic loading acting on the group structures and their vortex shedding patterns are different from those of a single structure since their interaction may result in complex and different flow phenomena.

The flow in these structures is complex and many of its aspects can be considered as open questions, deserving additional investigations. Much of this complexity is associated to the phenomena occurring inside the boundary layer which strongly interacts with the near field viscous wake. To overcome the difficulties of the analysis much effort is required. The development of new techniques and a fresh approach to the solution of the problems are urgently needed.

The ever changing characteristics of the flow inside the boundary layer and the separation are, by far, the most important phenomena occurring there. As the flow develops these phenomena does not remain constant and so does the flow in the regions where they occurs; these aspects must be considered in the analysis. Obviously the viscous body wake is affected, since it concentrates the vorticity generated on the body surface and that are carried downstream. It is of fundamental importance, therefore, to understand and to be able to analyse the vorticity generation and its dynamics as it develops into the viscous wake.

The main goal of the research line, where this paper is inserted, consists of the analysis of the flow in a turbomachines. This kind of flow is characterized by the interaction of the vortices generated by a rotor blade with the vortices generated by another blade of the same rotor. The viscous wake thus created interacts strongly with the casing surface. It is also frequently found the situation where the rotor blades are immersed in a viscous wake – from now on referred to as a shear flow – generated by an upstream rotor (stator).

A cascade of identical airfoils represents a simple model, suitable for the analysis of the main characteristics of this flow. Many researchers using the potential flow theory have exploited this model. The first application of vortex cloud modeling to turbomachines blade rows was published by Lewis and Porthouse (1983), Porthouse (1983) and Sparlat (1984). Lewis (1989) presented a basic scheme for vortex cloud modeling of cascades assuming that the boundary layers and wakes developed by the blades of an infinite cascade are identical. As the coupling coefficients are periodic in the y direction, surface elements and discrete vortex shedding need only be considered for the reference airfoil. The surface of the reference airfoil was represented by straight-line elements, with a point vortex located at the pivotal point. The vorticity diffusion that occurs in the wake was simulated using random walk method. The pressure on the airfoil surface was calculated according inviscid flow analysis. Although predicted surface pressure agrees well with experiment for the turbine cascade, losses are over-predicted. Due to "numerical stall", Lewis (1989) approach proves inadequate to deal with the compressor cascade. He and Su (1998) showed that the results for the pressure calculation could be improved by considering the nonlinear acceleration terms.

Recently, Alcântara Pereira *et al.* (2004) represented the surface of the reference airfoil by straight-line elements, with constant-strength vortex distribution including an additional feature, the turbulence modeling, see Alcântara Pereira *et al.* (2002). An improvement was also introduced in the convective step of the simulation; by using the anti symmetry property of the vortex-vortex velocity induction, the computational effort was considerably reduced;

this is an important feature, since the vortex-vortex velocity induction calculation is the most time consuming part of the simulation. In their work they presented a new approach to the pressure distribution calculation. With this new approach the pressure can be more accurately computed values, not only on the body surface but also in the whole fluid domain, which is of great importance for many engineering problems.

In the present paper, the Vortex Method is employed to simulate the linear cascade-vortex cloud interaction, see Fig. 1; the turbulence modeling is not included. The main feature of the paper is the presence of the vorticity in the oncoming stream flow that has two important characteristics. The first is that the vorticity in the oncoming stream is continuously generated in a plane perpendicular to the main flow and the second characteristic is the possibility of having a time variation of the vorticity carried out by this oncoming stream, see Silva de Oliveira *et al.* (2004). It is believed that with these two features the shear flow due to the upstream rotor could be simulated in a more realistic way; after all, the rotor always moves in relation to the upstream rotor or stator.

To simulate the vorticity in the oncoming stream, a cloud of free Lamb vortices is generated next to the linear cascade of airfoils. For this one can imagine a row of vortex generating points that are aligned in a previously fixed position. This row of vortices moves in a direction perpendicular to the x axis at a pre-set velocity V, see Fig. 1. As an application, a simulation of the interaction between vorticity in the oncoming stream and a stationary linear cascade of airfoils is also discussed.

As mentioned, the model is analyzed using the Vortex Method which is a meshless numerical method or a particle method. A cloud of free vortices is used in order to simulate the vorticity which is generated on the reference airfoil surface and on the row of vortex generating points and develops into the boundary layer and the viscous wake. 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 (e.g. references Chorin, 1973; Sarpkaya, 1989; Lewis, 1991; Alcântara Pereira *et al.*, 2002 and Kamemoto, 2004). With the Lagrangian formulation a grid for the spatial discretization of the fluid region is not necessary. Thus, special care to handle numerical instabilities associated to high Reynolds numbers is not needed. Also, the attention is only focused on the regions of high activities, which are the regions containing vorticity; on the contrary, Eulerian schemes consider the entire domain independent of the fact that there are sub-regions where less important, if any, flow acticity can be found. With the Lagrangian tracking of the vortices, one need not take into account the far away boundary conditions. This is of important in the wake regions (which is not negligible in the flows of present interest) where turbulence activities are intense and unknown, a priori.

2. Governing equations and vorticity dynamics

The problem to be considered is that of the flow past an infinite linear cascade of airfoils at high Reynolds numbers. A second cloud of free vortices is generated, next to the linear cascade, in a row of points, which moves in the y direction with constant velocity V; at every time increment a new vortex group is generated in each of points of the row, see Fig. 1. The vortices of this cloud are generated alternately with strength positive and negative. The flow is assumed to be incompressible and two-dimensional, and the fluid to be Newtonian, with constant properties. The unsteady flow that develops as it goes through the airfoils surface generates an oscillatory wake downstream the linear cascade, which interacts strongly with that vortex wake generated in each of the points of the row.

The boundary of the fluid region Ω is defined by the surface S, which, by its turn, is viewed as $S = S_k \cup S^*_{-\infty} \cup S^*_{+\infty}$, $k = 0, \pm1, \pm2, ..., \pm\infty$ defines each airfoil surface in the cascade; see Fig. 1.

The flow, depicted in Fig. 1, is governed by the continuity and the Navier-Stokes equations, which can be written in the form

$$\nabla \cdot \mathbf{w} = 0 \tag{1}$$

$$\frac{\partial \mathbf{w}}{\partial t} + \mathbf{w} \cdot \nabla \mathbf{w} = -\nabla p + \frac{1}{Re} \nabla^2 \mathbf{w} \,. \tag{2}$$

In the equations above $\mathbf{w}$ is the velocity vector field and p is the pressure. As can be seen the equations are non-dimensionalized in terms of W_1 and b (a reference length). The Reynolds number is defined by

$$Re = \frac{bW_1}{\upsilon} \tag{2a}$$

where υ is the fluid kinematics viscosity coefficient.

For a complete definition of the problem, on S_k, $k = 0, \pm1, \pm2, ..., \pm\infty$, the impenetrability and no-slip conditions are written as

$$w_n = \mathbf{w} \cdot \mathbf{e}_n = 0 \tag{3a}$$

$$w_\tau = \mathbf{w} \cdot \mathbf{e}_\tau = 0 \tag{3b}$$

where $\mathbf{e}_n$, $\mathbf{e}_\tau$ and $\mathbf{w}$ are unit normal vector, unit tangential vector and fluid particle velocity vector, respectively. One assumes that far away the perturbation caused by the infinite linear cascade of airfoils fades away as

$$\mathbf{w}(-\infty, y) = \mathbf{w}_1 \text{ in } S^*_{-\infty} \tag{3c}$$

$$\mathbf{w}(+\infty, y) = \mathbf{w}_2 \text{ in } S^*_{+\infty} . \tag{3d}$$

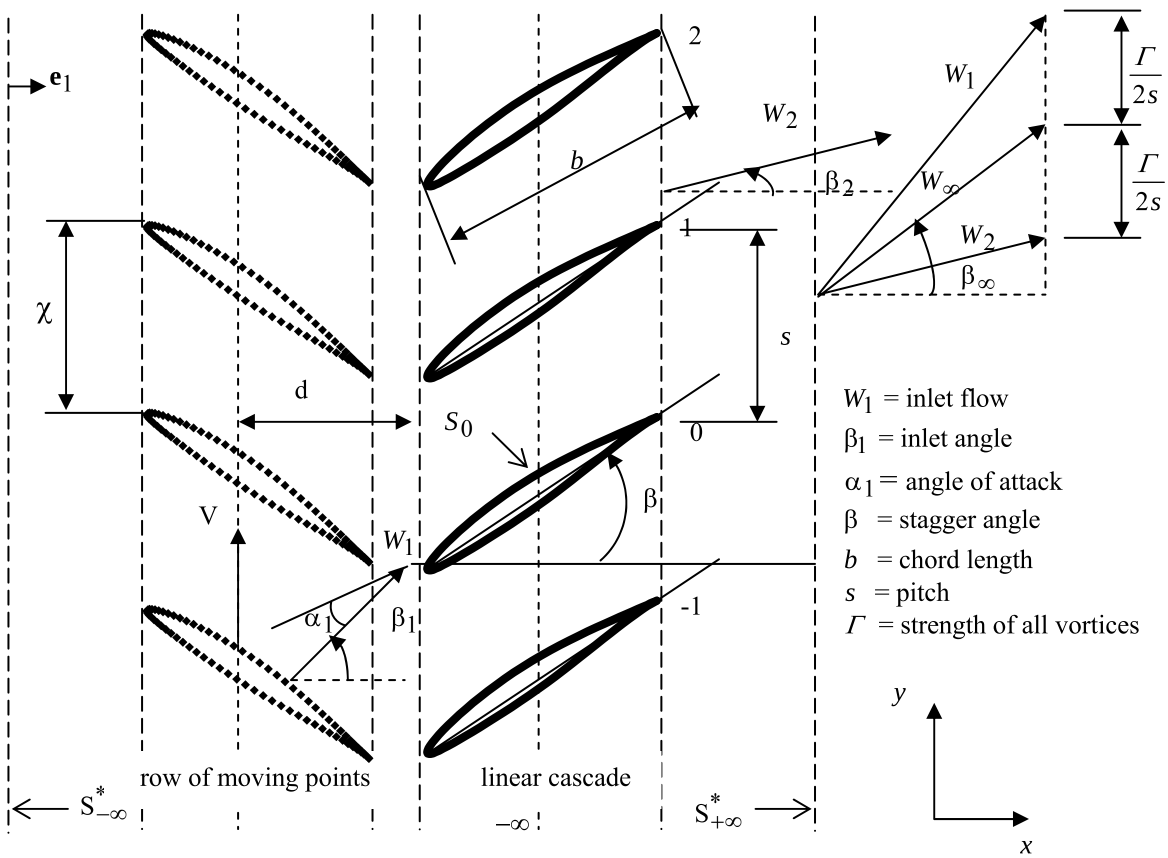


Figure 1. Definitions.

The dynamics of the fluid motion, governed by the boundary-value problem (1), (2) and (3), can be alternatively studied by taking the curl of Eq. (2), obtained the vorticity transport equation

$$\frac{\partial \omega}{\partial t} + \mathbf{w} \cdot \nabla \omega = \frac{1}{Re} \nabla^2 \omega \tag{4}$$

in this equation ω is the only non-zero component of the vorticity vector.

3. Vortex method

According to the viscous splitting algorithm (Chorin, 1973) it is assumed that in the same time increment the convection of the vorticity can be simulated independently of its diffusion and are governed by

$$\frac{\partial \omega}{\partial t} + \mathbf{w} \cdot \nabla \omega = 0 \tag{5}$$

$$\frac{\partial \omega}{\partial t} = \frac{1}{Re} \nabla^2 \omega \tag{6}$$

The vorticity, generated on the reference airfoil surface, is simulated by a cloud of discrete vortices. In each time increment of the simulation M new discrete vortices are generated and incorporated to the vortex cloud. The M newly nascent vortices are placed on a line normal to the panel, at a distance ε. No-slip boundary condition, Eq. (3b), is used for the calculation of the strength of the nascent vortices at reference airfoil surface. Also, at every time increment new discrete vortices are generated, alternately with strength positive and negative, in each of points of the row, see Fig. 1. Once generated, all the new discrete vortices are incorporated to the vortex cloud which, in turn, are subjected to the convection and diffusion process.

Convection is governed by Eq. (5) and the velocity field is given by (Alcântara Pereira *et al.*, 2004)

$$W_{t_k}(t) = W_\infty + \sum_{j=1}^{M} W_{p_{k,j}}(t) + \sum_{\substack{j=1 \\ j \neq k}}^{N} W_{v_{k,j}}(t) \tag{7}$$

where

$$W_\infty = W_1 - i\frac{\Gamma}{2s} \ , \ \ \Gamma = \sum_{j=1}^{M} \gamma_j \Delta s_j + \sum_{n=1}^{N} \Delta\Gamma_n \ , \tag{8}$$

$$\overline{W}_{p_{k,j}}(Z_{c_k}) = i \ \gamma_j \frac{e^{-i\alpha_j}}{2\pi} \ln\left\{ \frac{\sinh\left[\frac{\pi}{s}\left(Z_{c_k} - Z_j\right)\right]}{\sinh\left[\frac{\pi}{s}\left(Z_{c_k} - Z_{j+1}\right)\right]} \right\}, \ \ W_{v_{k,j}} = i\frac{\Delta\Gamma_j}{2s}\left\{ \frac{\sinh\frac{2\pi}{s}\left(x_k - x_j\right) - i\sin\frac{2\pi}{s}\left(y_k - y_j\right)}{\cosh\frac{2\pi}{s}\left(x_k - x_j\right) - \cos\frac{2\pi}{s}\left(y_k - y_j\right)} \right\} \tag{9}$$

Here, W_∞ is the mean value of the inlet and exit velocities W_1 and W_2, see the Fig. 1. The summation of M integral terms comes from the M panels distributed on the airfoil surface; $\overline{W}_{p_{k,j}}$ is the complex velocity induced by linear cascade and α_j defines airfoil profile slope. The second summation represents the velocity induced by the cloud of N Lamb free vortices.

Each vortex of the cloud is followed in time according to the Adams-Bashforth second-order formula (Ferziger, 1981)

$$\mathbf{r}(t + \Delta t) = \mathbf{r}(t) + [1.5\mathbf{w}(t) - 0.5\mathbf{w}(t - \Delta t)]\Delta t + \xi \tag{10}$$

in which $\mathbf{r}$ is position of a fluid particle, Δt is the time increment and ξ is the random walk, representing diffusion of vorticity (Lewis, 1991).

The pressure calculation starts with the Bernoulli function, defined by Uhlman (1992), as

$$Y = \frac{p}{\rho} + \frac{w^2}{2}, \ w = |\mathbf{w}| \ . \tag{11}$$

Using the same function and starting from the Navier-Stokes equations Kamemoto (1993) was able to write a Poisson equation for the pressure. This equation was solved using a finite difference scheme. Here the same Poisson equation was derived and its solution was obtained through the following integral formulation (Shintani and Akamatsu, 1994)

$$\lambda \overline{Y}_i - \int_{S_\circ} \overline{Y} \nabla G_{i_\circ} \cdot \mathbf{e}_n dS = \iint_{\Omega_\circ} \nabla G_{i_\circ} \cdot (\mathbf{w} \times \boldsymbol{\omega}) d\Omega_\circ - \upsilon \int_{S_\circ} \left(\nabla G_{i_\circ} \times \boldsymbol{\omega} \right) \cdot \mathbf{e}_n dS \tag{12}$$

where

$$\lambda = \begin{cases} 0, x_i \notin \Omega_\circ \\ 1, x_i \in \Omega_\circ \text{ (in the fluid domain)} \\ \frac{1}{2}, x_i \in S_\circ \text{ or } x_i \in S^*_{\pm\infty} \text{ (on } S_\circ \text{ and } S^*_{\pm\infty} \text{ surfaces)} \end{cases} , \tag{13}$$

$$G_{i_0} = \frac{1}{2\pi}\ln\left(\frac{1}{r_{i_0}}\right), \; r_{i_0} = \sqrt{\left(x_0 - x_i\right)^2 + \left(y_0 - y_i\right)^2} . \tag{14}$$

Equation (12) is valid if there is only one isolated body in the fluid region, $\Omega_\circ$. $G_{i_\circ}$ is the fundamental solution of Laplace equation and r_{i_0} the distance from i-th vortex element to the field point. Note that Eq. (12) is particularly suited for using with the Vortex Method since the integration are restricted to the domains where vorticity exists; this equation needs no grid generation and contains no time-dependent term (Shintani & Akamatsu,1994).

Alcântara Pereira *et al.* (2004) applied Eq. (12) to an infinite linear cascade of airfoils (see Fig. 1). The pressure distribution is, thereby, computed as

$$\lambda \overline{Y}_i - \int_S \overline{Y}\nabla\left(\sum_{k=-\infty}^{+\infty} G_{i_k}\right)\cdot \mathbf{e}_n dS = \iint_\Omega \nabla\left(\sum_{k=-\infty}^{+\infty} G_{i_k}\right)\cdot(\mathbf{w}\times\omega)d\Omega - \upsilon\int_S \nabla\left(\sum_{k=-\infty}^{+\infty} G_{i_k}\right)\times\omega\cdot\mathbf{e}_n dS \tag{15}$$

being

$$\sum_{k=-\infty}^{+\infty} G_{i_k} = -\frac{1}{2\pi}\sum_{k=-\infty}^{+\infty} \ln r_{i_k} = \varphi_i , \tag{16}$$

$$\varphi_i = -\frac{1}{2\pi}\ln\frac{\sqrt{\cos^2\frac{\pi}{s}\left(y_i - y\right)\sinh^2\frac{\pi}{s}\left(x_i - x\right) + \sin^2\frac{\pi}{s}\left(y_i - y\right)\cosh^2\frac{\pi}{s}\left(x_i - x\right)}}{\frac{\pi}{s}} , \tag{17}$$

and $S = S_k \cup S^*_{-\infty} \cup S^*_{+\infty}, k = 0, \pm1, \pm2, ..., \pm\infty$.

According to Eq. (17), the discrete vortex shedding need only be considered for a reference airfoil of the linear cascade. Then Eq. (15) is valid at the domain $\Omega_{\circ R}$ for the reference airfoil

$$\lambda Y_i - Y_{-\infty} - \int_{S_{\circ R}} Y\left(\nabla\varphi_i + \frac{\mathbf{e}_1}{2s}\right)\cdot\mathbf{e}_n dS = \iint_{\Omega_{\circ R}} \left(\nabla\varphi_i + \frac{\mathbf{e}_1}{2s}\right)\cdot(\mathbf{w}\times\omega)d\Omega - \frac{1}{Re}\int_{S_{\circ R}} \left[\left(\nabla\varphi_i + \frac{\mathbf{e}_1}{2s}\right)\times\omega\right]\cdot\mathbf{e}_n dS \tag{18}$$

Here, the airfoil surface is defined by $S_{\circ R}$. Numerically, Eq. (18) is solved by mean of a set of simultaneous equations for the pressure Y_i . It is worth noting that if $s \to \infty$ one gets the corresponding formulation for a single airfoil (Shintani and Akamatsu, 1994). One can also observe that if the pressure distribution, on the fluid domain, is needed one has to change the λ value according to Eq. (13).

4. Results and discussion

The performance of NACA 65-series compressor blade sections in cascade was investigated systematically in a low-speed cascade tunnel by Emery *et al.* (1957). The effects of inlet angle β_1 , angle of attack α_1 , solidity b/s and blade shape was studied. The stagger angle was defined as $\beta = \beta_1 - \alpha_1$ and the deflection angle as $\theta = \beta_1 - \beta_2$ (Fig. 1).

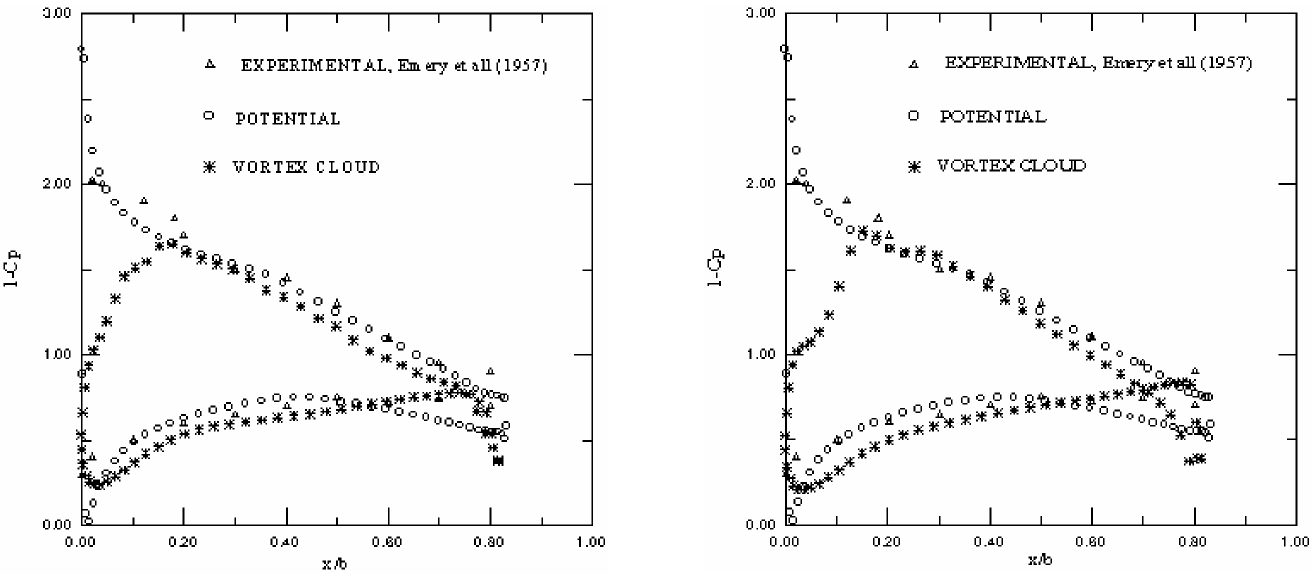
The flow past an NACA 65-410 airfoil cascade at Reynolds number $4.45\text{x}10^5$ is simulated to confirm the ability of the present method to predict pressure distributions on the airfoil surface and the linear cascade deflection angle. All run were performed with 300 time steps of magnitude $\Delta t = 0.05$, with an inlet angle $\beta_1 = 45.00^\circ$ and solidity $b/s = 0.50$. The blade profile was represented by $M = 80$ straight-line vortex panels with constant density. In each time step, M new vortex elements are shed into the cloud through a displacement $\varepsilon = \sigma_0 = 0.03b$ normal to the straight-line panels; these new vortices are added to the vortex cloud.

Table 1 presents results obtained for an NACA 65-410 airfoil cascade; these results refer to numerical simulation without the row of moving points.

Table 1. Linear cascade of NACA 65-410 airfoils: $\beta_1 = 45.00^\circ$ and Re=4.45x10^5.

Case	α_1	β	$\theta = \beta_1 - \beta_2$
I	15.60°	29.40°	10.60°
II	11.40°	33.60°	9.10°
III	7.70°	37.30°	7.00°

The pressure distribution computations on the airfoil surface starts at t=5. Figure 2(a) shows the experimental results, which are compared with the ones obtained using the potential flow simulation and vortex method simulation. As expected the potential flow results show a high-pressure peak at the leading edge; the vortex method results do not show this behavior. In the neighbourhood of the leading edge, the potential flow results have a better behavior (if compared to the experimental data) than the present numerical results, this is especially evident at the low-pressure surface. In the present numerical simulation, therefore, the free vortices generated near the leading edge of an airfoil are not able to comply with the local acceleration. This seems to indicate that the higher value of M combined with a smaller time step would improve the resolution and probaly produce a better simulation with respect to the pressure distribution.



(a) computed without row of moving points

(b) computed with row of moving points

Figure 2. Predicted pressure distribution on a linear cascade of NACA 65-410 airfoils. Case II, Re=4.45x10^5, M=80, Δt=0.05, $\beta_1 = 45.00^\circ$, β =33.60° and $\varepsilon = \sigma_0 = 0.03b$.

In general, however, the pressure distribution numerically predict does agree well with the experimental data up to the point that they are available, however, the potential flow results depart a little bit form the experimental values in the low pressure surface.

A new implementation to simulate the interaction of an reference airfoil with a vortex group generated alternately with strength positive and negative in each of points of the row (see Fig. 1) was tested for an NACA 65-410 reference airfoil. Therefore, the three conditions of Table 1 were again numerical simulated, now considering the proposed row of moving points. For illustration, the flow pattern, represented by the instantaneos free vortices positions is presented in Fig. 3; this flow pattern refers to case II, Table 1, at t=7.5. Each vortex generated at row of moving points has dimensionless strength $\Delta\Gamma$=± 0.001. The center of row with length s and V=0.065 W_1 was located at distance d=b, see Fig. 1. The vertical distance between two shedding points of the row was equal to χ=2b. To avoid the influence of numerical transients, we start the simulation with the reference airfoil in an inlet flow without vortex cloud generated at row of moving points and we insert the row only after the numerical simulation has been established (t=5).

Computed values for the distribution of the mean pressure coefficient in presence of row of moving points is shown in Fig. 2(b).

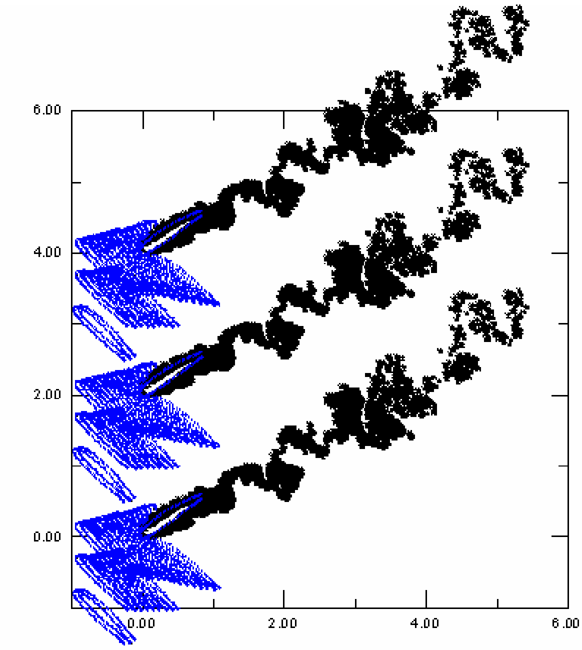


Figure 3. NACA 65-410 airfoil: Vortex wake structure at t=7.5. Case II, Re=4.45x10^5, M=80, Δt=0.05, $\beta_1 = 45.00°$, β =33.60° and $\varepsilon = \sigma_0 = 0.03b$.

The linear cascade deflection angle, one of the more important parameter for the designer, is presented in Fig. 4 as a function of the angle of attack. This figure shows the vortex method values, calculated with and without the row of moving of points, which are compared to the results, obtained using potential flow theory with experimental values. The potential flow theory overpredicts the value of the deflection angle and it departs from the experimental values, as well as (probably because one gets closer to separation phenomenon). On the other hand, the vortex method predictions get closer to the experimental values even when the angle of attack increases and the flow behave as if the airfoil were a bluff body. As expected the deflection angle is insensitive to presence of a row of moving points.

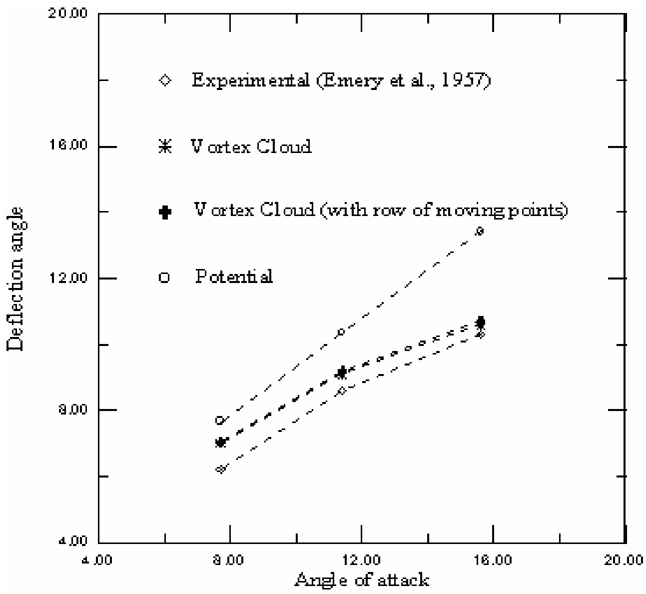


Figure 4. Deflection angle in a linear cascade of NACA 65-410 airfoils, Re=4.45x10^5, M=80, Δt=0.05 and $b/s = 0.50$.

5. Concluding remarks

This work presented the investigation of the behaviour of a linear cascade of airfoils in the presence of a cloud of free vortices. The cloud of free vortices is generated alternately with strength positive and negative next to the linear cascade, in a row of points, which move in the y direction with constant velocity V.

The main objective of the work with the implementation and initial test of a numerical vortex cloud model used to predict the aerodynamics of a two-dimensional linear cascade-row wakes interaction, has been achieved. The methodology was used to simulate the flow around a cascade of NACA 65-410 airfoils. The results indicate the small influence of the second vortex wake generated next to the linear cascade in the numerical solution of the deflection angle. The distance between the linear cascade and the row, the row velocity, the distance between two vortices at row and the value of the vortex strength generated at row are extremely important in determining the degree of interaction. The influence of these parameters will be carried out. The sub-grid turbulence modeling is of significant importance for the numerical simulation. The results of this analysis, taking into account the sub-grid turbulence modeling, are being generated and will be presented in due time, elsewhere.

From the present study, it is confirmed that the methodology used in the present study presents promising features and predicts well the main global parameters of interest to the designer.

6. Acknowledgements

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

7. References

Alcântara Pereira, L.A., 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., 92, pp. 477-491.

Alcântara Pereira, L.A., Ricci, J.E.R., Hirata, M.H. and Silveira-Neto, A., 2002, "Simulation of Vortex-Shedding Flow about a Circular Cylinder with Turbulence Modeling", Intern'l Society of CFD, vol. 11, no. 3, October, pp. 315-322.

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

Emery, J.C., Herrig, L.J., Erwin, J.R and Felix, A.R., 1957, "Systematic Two-Dimensional Cascade Tests of NACA 65-Series Compressor Blades at Low Speeds", NACA TN 1318.

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

He, F. and Su, T.C., 1998, "A Numerical Study of Bluff Body Aerodynamics in High Reynolds Number Flows by Viscous Vortex Element Method", J. Wind Eng. Ind. Aerodyn. 77/88, pp. 393-407.

Kamemoto, K., 2004, "On Contribution of Advanced Vortex Element Methods Toward Virtual Reality of Unsteady Vortical Flows in the New Generation of CFD", Proceedings of the 10th Brazilian Congress of Thermal Sciences and Engineering-ENCIT 2004, Rio de Janeiro, Brazil, Nov. 29-Dec. 03, Invited Lecture-CIT04-IL04.

Kamemoto, K., 1993, "Procedure to Estimate Unstead Pressure Distribution for Vortex Method" (In Japanese), Trans. Jpn. Soc. Mech. Eng., vol. 59, no. 568 B, pp. 3708-3713.

Lewis, R. I., 1991, "Vortex Element Method for Fluid Dynamic Analysis of Engineering Systems", Cambridge Univ. Press, Cambridge, England, U.K..

Lewis, R.I., 1989, "Application of the Vortex Cloud Method to Cascades", Int. J. Turbo & Jet Engine, vol. 6, pp. 231-245.

Lewis, R.I. and Porthouse, D.T.C., 1983, "Numerical Simulation of Stalling Flows by an Integral Equation Method", AGARD Meeting. Viscous Effects in Turbomachines, AGARD-CPP-351, Copenhagen.

Porthouse, D.T.C., 1983, "Numerical Simulation of Airfoil and Bluff Body Flows by Vortex Dynamics", University of Newcastle upon Tyne, England, UK, Ph. D. Thesis.

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

Shintani, M. and Akamatsu, T, 1994, "Investigation of Two Dimensional Discrete Vortex Method with Viscous Diffusion Model", Computational Fluid Dynamics Journal, vol. 3, no. 2, pp. 237-254.

Silva de Oliveira, L., Alcântara Pereira, L.A., Hirata, M.H. and Manzanares Filho, N., 2004, "Numerical Modelling of the Airfoil-Vortex Cloud Interaction", Proceedings of the 10th Brazilian Congress of Thermal Sciences and Engineering-ENCIT 2004, Rio de Janeiro, Brazil, Nov. 29-Dec. 03, Paper CIT04-0253.

Sparlat, P.R., 1984, "Two Recent Extensions of the Vortex Method. A.I.A.A.", 22nd Aerospace Sciences Meeting, Reno, Nevada, A.I.A.A. Paper 84-034.3.

Uhlman, J.S., 1992, "An Integral Equation Formulation of the Equation of an Incompressible Fluid", Naval Undersea Warfare Center, T.R. 10-086.

8. Responsibility notice

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