IX CONGRESSO BRASILEIRO DE
ENGENHARIA E CIÊNCIAS TÉRMICAS

9th BRAZILIAN CONGRESS OF THERMAL
ENGINEERING AND SCIENCES

Paper CIT02-0490

A NUMERICAL STUDY OF LINEAR CASCADE AERODYNAMICS IN HIGH REYNOLDS NUMBER FLOWS BY VORTEX METHOD WITH TURBULENCE MODEL

Luiz Antonio Alcântara Pereira
Escola Federal de Engenharia de Itajubá – Departamento de Engenharia Mecânica
Av. BPS 1303 – Itajubá, MG – 37500-903
luizantp@iem.efei.br

Miguel Hiroo Hirata
Escola Federal de Engenharia de Itajubá – Departamento de Engenharia Mecânica
Av. BPS 1303 – Itajubá, MG – 37500-903
hirata@iem.efei.br

Nelson Manzanares Filho
Escola Federal de Engenharia de Itajubá – Departamento de Engenharia Mecânica
Av. BPS 1303 – Itajubá, MG – 37500-903
nelson@iem.efei.br

Abstract. *In this paper the vortex cloud method is used in an attempt to simulate the viscous flow through linear cascades. As the flow is periodic in the y direction, the discrete vortex-shedding need only be considered for a reference airfoil. The Vortex Method is also modified to take into account the sub grid- scale phenomena; a second-order velocity structure function model is adapted to the Lagrangian scheme. The dynamics of the airfoil wake is computed using the convection-diffusion splitting algorithm, where the convection process is carried out with a Lagrangian second order Adams-Bashforth time-marching scheme, and the diffusion process is simulated using the random walk scheme. In this study, the pressure distribution is obtained using the integral equation derived from the pressure Poisson equation. Flows around NACA 65-410 serie airfoils are calculated and comparisons are made with experimental studies for pressure distribution and linear cascade deflection angle.*

Keywords: *vortex method, sub grid-scale modeling, linear cascade, pressure distribution, deflection angle.*

1. Introduction

Understanding of the vortex-shedding flow behind a linear cascade is of fundamental and practical importance. Vortex cloud modeling offers great potential for numerical analysis of important problems in turbomachines. A cloud of free vortices is used in order to simulate the vorticity, which is generated on the body surface 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 (Chorin, 1973; Sarpakaya, 1989; Kamemoto, 1994; Lewis, 1999). 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, and no explicit choice of the outer boundaries is needed a priori; (d) no boundary condition is required at the downstream end of the flow domain.

The first application of vortex cloud modeling to turbomachinery blade rows, including the prediction of rotating stall in compressors and vibrations induced by blade row wake interaction, was published by Lewis & Porthouse (1983) and Porthouse (1983) followed by some fairly comprehensive studies by 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.

In a recent paper Alcântara Pereira & Hirata (2000) represented the surface of the reference airfoil by straight-line elements, with constant-strength vortex distribution. 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. The pressure calculation adopted by the authors was identical to the one used by Lewis (1989). He & Su (1999) showed, however, that the results for the pressure calculation could be improved by considering the nonlinear acceleration terms.

In the present paper we introduce two important new features for the numerical simulation of the flow in a linear cascade: (a) the inclusion of the turbulence aspects, through a sub grid-scale modeling which employs a second-order velocity structure function of the filtered field; (b) a new development for the pressure calculation which is based on the full Navier-Stokes equations.

2. Formulation of the Physical Problem

The problem to be considered is that of a flow past an infinite linear cascade of airfoils at high Reynolds numbers; see Fig. (1). 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 cascade.

The boundaries of the fluid region is defined by the surface W, which by its turn is viewed as $W = S_k \cup S_{-\infty} \cup S_{+\infty}$, $k = 0, \pm 1, \pm 2, \ldots, \pm\infty$

The governing equations are

$$\frac{\partial \overline{w_i}}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{w_i}}{\partial t} + \overline{w_j}\,\frac{\partial \overline{w_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + 2\frac{\partial}{\partial x_j}\left[(\upsilon + \upsilon_t)\overline{S_{ij}}\right] \tag{2}$$

where the Einstein's summation convention applies. The above governing equations was filtered ($w_i = \overline{w_i} + w_i{}'$, $w_i{}'$ denotes the fluctuation field), υ is the fluid kinematic viscosity coefficient, υ_t is the eddy viscosity coefficient, ρ is the fluid density, $\overline{S_{ij}}$ is the deformation tensor of the filtered field and p is the pressure.

The large structures are governed by Eq. (2) and the eddy-viscosity assumption (Boussinesq's hypothesis) is used to model the sub grid-scale tensor $T_{ij} = \overline{w_i w_j} - \overline{w_i}\,\overline{w_j}$, see Alcântara Pereira et al. (2000).

For a complete definition of the problem, on $S_k, k = 0, \pm 1, \pm 2, \ldots, \pm\infty$, the impenetrability and non-slipping 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

$$\mathbf{w}(-\infty, y) = \mathbf{w}_1 \text{ in } S_{-\infty} \tag{3c}$$

$$\mathbf{w}(+\infty, y) = \mathbf{w}_2 \text{ in } S_{+\infty} \tag{3d}$$

In order to take into account the local activity of turbulence, Métais & Lesieur (1992) considered that the small scales may not be too far from isotropy and to define the eddy viscosity υ_t they proposed to use the local kinetic-energy spectrum $E(k_c)$ at the cutoff wave number k_c. Using a relation proposed by Batchelor (1963) the local spectrum at k_c is calculated in terms of the local second-order velocity structure function $\overline{F_2}$ of the filtered field (Lesieur & Métais, 1996)

$$\overline{F_2}(\mathbf{x},D,t)=\overline{\left\|\overline{\mathbf{w}}(\mathbf{x},t)-\overline{\mathbf{w}}(\mathbf{x}+\mathbf{r},t)\right\|^2_{\|\mathbf{r}\|=D}} \tag{4}$$

From the Kolmogorov spectrum the eddy viscosity can be written as a function of $\overline{F_2}$

$$\upsilon_t(\mathbf{x},D,t)=0.105\,C_k^{-\frac{3}{2}}D\sqrt{\overline{F_2}(\mathbf{x},D,t)} \tag{5}$$

where $C_k = 1.4$ is the Kolmogorv constant. The great computational advantage of this formulation over the Smagorinsk (1963) model, vis a vis the Vortex Method, is that in Eq. (4) the notion of velocity fluctuations (differences of velocity) is used instead of the rate of deformation (derivatives). The velocities $\mathbf{w}(\mathbf{x}+\mathbf{r})$ are calculated over the surface of a sphere of radius D.

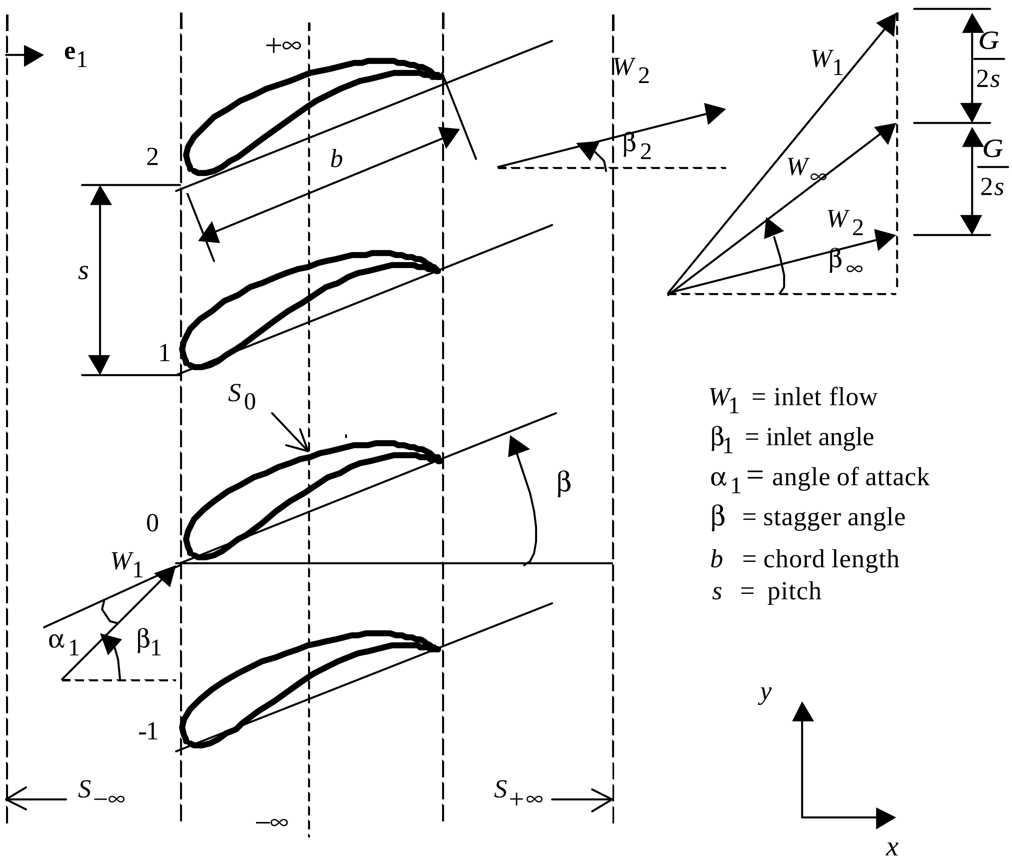

Figure 1. Flow in a linear cascade of airfoils.

Alcântara Pereira et al. (2000) adapted the second-order velocity structure function $\overline{F_2}$ to the Lagrangian scheme as

$$\overline{F_2}=\frac{1}{NV}\sum_{l=1}^{NV}\left\|\mathbf{w}(\mathbf{x})-\mathbf{w}(\mathbf{x}+\mathbf{r}_l)\right\|_l^2\left(\frac{\sigma_0}{\|\mathbf{r}_l\|}\right)^{\frac{2}{3}} \tag{6}$$

In Eq. (6), NV is the number of discrete vortices found in the region defined by the distances $r_1=(0.1)\sigma_0$ and $r_2=(1.0+f_2)\sigma_0$ from the center of the reference vortex. Here σ_0 is the core radius of a Lamb vortex used as a model for the discrete vortices of the cloud, see Fig. (2). The correction factor $\left(\sigma_0/\|\mathbf{r}_l\|\right)^{2/3}$ is necessary due to the fact that the NV vortices are not located at equal distance from the centre of the reference vortex.

The filtered Eq. (2) are nondimensionalized in terms of W_1 and b. The Reynolds number is defined as

$$\mathrm{Re} = \frac{bW_1}{\upsilon} \tag{7}$$

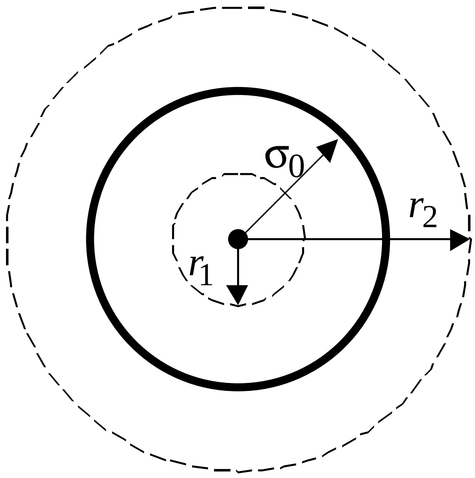

Figure 2. Region defined by distances r_1 and r_2 from the center of the reference vortex.

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

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

In this equation ω is the only non-zero component of the vorticity vector and **w** is the velocity field. At this point, it is convenient to define the turbulent Reynolds number as

$$\mathrm{Re}_c = \frac{bW_1}{\upsilon + \upsilon_t} \tag{9}$$

3. Numerical Approach: The Discrete Vortex Method with Turbulence Model

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

$$\frac{\partial\omega}{\partial t} + \mathbf{w}\cdot\nabla\omega = 0 \tag{10}$$

$$\frac{\partial\omega}{\partial t} = \frac{1}{\mathrm{Re}_c}\nabla^2\omega \tag{11}$$

The vorticity, generated on the airfoils surface, is simulated by a cloud of discrete vortices. Non-slipping boundary condition, Eq. (3b), is used for the calculation of the strength of the nascent vortices. Once generated, the new discrete vortices are incorporated to the vortex cloud which, in turn, is subjected to the convection and diffusion process. Convection is governed by Eq. (10). The convection velocity field is given by (Alcântara Pereira, 2002)

$$W_{t_k}(t) = W_\infty + \sum_{j=1}^{M} W_{p_j}(t) + \sum_{\substack{m=1 \\ m\neq n}}^{N} W_{v_{m,n}}(t) \tag{12}$$

$$W_\infty = W_1 - i\frac{G}{2s}\,,\ G = \sum_{j=1}^{M}\gamma_j \Delta s_j + \sum_{n=1}^{N}\Delta\Gamma_n \tag{12a}$$

$$\overline{W}_{p_j}\left(Z_{c_k}\right) = i\ \gamma_j\ \frac{e^{-i\alpha_j}}{2\pi}\ln\left\{\frac{\sinh\left[\frac{\pi}{s}\left(Z_{c_k} - Z_j\right)\right]}{\sinh\left[\frac{\pi}{s}\left(Z_{c_k} - Z_{j+1}\right)\right]}\right\} \tag{12b}$$

$$W_{v_{mn}} = i\,\frac{\Delta\Gamma_n}{2s}\left\{\frac{\sinh\frac{2\pi}{s}\left(x_m - x_n\right) - i\sin\frac{2\pi}{s}\left(y_m - y_n\right)}{\cosh\frac{2\pi}{s}\left(x_m - x_n\right) - \cos\frac{2\pi}{s}\left(y_m - y_n\right)}\right\} \tag{12c}$$

where $i = \sqrt{-1}$ and the complex variable notation $z = x + iy$ is used. Here, W_∞ is the velocity vector that represent the mean of the inlet and exit velocities W_1 and W_2, see Fig. (1). The summation of M integral terms comes from the M panels distributed on the airfoil surface; $\overline{W}_{p_j}$ is the conjugate complex velocity induced by linear cascade (Giesing, 1964) and α_j defines airfoil profile slope. The second summation is associated to the velocity induced by the cloud of N free vortices.

To simulate the vorticity convection, each vortex of the cloud is followed in time according to the Adams-Bashforth second-order formula (Ferziger, 1981)

$$\mathbf{r}(t + \Delta t) = \mathbf{r}(t) + \left[1.5\mathbf{w}(t) - 0.5\mathbf{w}(t - \Delta t)\right]\Delta t + \xi \tag{13}$$

where $\mathbf{r}$ is the position of a fluid particle, $\mathbf{w}$ is the fluid velocity at the particle location, Δt is the time increment. In this expression the random walk displacement, ξ, which is used to simulate the vorticity diffusion, is added. This displacement is defined as

$$\xi = \sqrt{\frac{4\Delta t}{\mathrm{Re}}\left(1 + \frac{\upsilon_t}{\upsilon}\right)\ln\left(\frac{1}{P}\right)}\left[\cos(2\pi Q) + i\sin(2\pi Q)\right] \tag{14}$$

P and Q are random numbers between 0.0 and 1.0.

The core radius of a Lamb vortex originally recommended by Mustto et al. (1998) is modified to

$$\sigma_{0_{\upsilon_t}} = 4{,}48364\sqrt{\frac{\Delta t}{\mathrm{Re}}\left(1 + \frac{\upsilon_t}{\upsilon}\right)} \tag{15}$$

The pressure calculation starts with the Bernoulli function, defined by Uhlman (1993), as

$$Y = p + \frac{w^2}{2}\ ,\ w = \left|\mathbf{w}\right| \tag{16}$$

Kamemoto (1993) used the same function and starting from the Navier-Stokes equations 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 for a linear cascade was obtained through the following integral formulation (Alcântara Pereira, 2002)

$$\alpha Y_i - Y_{-\infty} - \int_{S_0} Y\left(\nabla\lambda_i + \frac{\mathbf{e}_1}{2s}\right)\cdot\mathbf{e}_n dS = \iint_{W_0}\left(\nabla\lambda_i + \frac{\mathbf{e}_1}{2s}\right)\cdot\left(\mathbf{w}\times\mathbf{\grave{u}}\right)dW - \frac{1}{\mathrm{Re}}\int_{S_0}\left[\left(\nabla\lambda_i + \frac{\mathbf{e}_1}{2s}\right)\times\mathbf{\grave{u}}\right]\cdot\mathbf{e}_n dS \tag{17}$$

where α takes the value 1.0 in the domain W_0 and 0.5 on its boundary S_0 and

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

It is worth to observe that this formulation is specially suited for a Lagrangian scheme because it utilizes the velocity and vorticity field defined at the position of the vortices of the cloud. Therefore it does not require any additional calculation at mesh points. Numerically, Eq. (17) is solved by mean of a set of simultaneous equations for the pressure Y_i. Additionally, if $s \to \infty$ one gets the corresponding formulation for a single body (Shintani & Akamatsu, 1994).

4. Discussion of Results

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 is defined as $\beta = \beta_1 - \alpha_1$ and the deflection angle as $\theta = \beta_1 - \beta_2$. These parameters were used for the numerical simulations and for comparisons.

Table (1) presents the results of the calculation of three cases carried out for NACA 65-410 airfoil, without turbulence modeling. All run were performed with 200 time steps of magnitude $\Delta t = 0.05$, with an inlet angle β_1=45.00° and solidity $b/s = 0.50$. The blade profile was represented by $M = 80$ surface vortex elements. The nascent vortices were initially displaced, normal to the panel, by $\varepsilon = \sigma_0 = 0.03b$.

Table 1. Linear cascade of NACA 65-410 airfoils, without turbulence modeling; $\mathrm{Re} = 4.45x10^5$.

Case	α_1	β	$\%N$
I	15.60°	29.40°	0.6741
II	11.40°	33.60°	0.7135
III	7.70°	37.30°	0.7096

$\%N$ denotes free vortices percentage in the cloud at $t = 10$.

Figure (3) shows the flow pattern over the cascade for case II, at $t = 10$. Similar patterns can be observed if the numerical simulation takes into account the turbulence modeling.

Consider a discrete vortex located at point L. The value of the second-order velocity structure function, $\overline{F_2}$, which measures the turbulent manifestations, is statically sound only if the neighborhood of L is sufficiently populated with other points vortices. After numerical experiments with the flow around the linear cascade presented in the Fig. (3), it was assumed that this happens if $NV/A \geq 5000$, where NV is the number of point vortices in the region, of area A, defined by two circumferences centered in L with radius $r_1 = (0.1)\sigma_0$ and $r_2 = (1.50)\sigma_0$, see Fig. (2). Thus the ratio NV/A acts as a trigger, which is fired every time that it assumes a value equal or above 5000.

Figure (4) shows the positions where the flow is turbulent according to the above criterion; this happens mainly in the boundary layer, specially near the trailing edge and the lower surface, as well as in the wake near field.

The pressure distribution and deflection angle computations starts at $t = 9$. Figure (5a) presents the experimental and numerical results; the later were obtained using potential flow theory and vortex cloud prediction without turbulence modeling. Figure (5b) presents the same results, but turbulence modeling was included in the calculations. The results for the potential flow model behaves well except near the leading edge where it predicts the stagnation pressure and in the lower surface. The results for the regions near the trailing edge does not behave well. The vortex cloud method, even without turbulence modeling, is able to improve the trailing edge and lower surface predictions, however the leading edge pressure is under predicted. As could be expected from Fig. (4), the turbulence modeling was unable to modify the behavior of the pressure distribution on the upper surface, near the leading edge. On the other hand the predictions in the lower surface and near the trailing edge was enhanced considerably.

The cascade deflection angle is one of the most important parameter for a turbomachinery designer. Computed values for this angle are compared with experimental values and presented in the Fig. (6). As expected, potential flow calculations behaves badly, specially for the maximum deflection angle. Vortex Method simulations produce results that are in close agreement with the experimental ones; the results obtained without turbulence modeling are closer, to the experimental values, then the ones obtained with turbulence modeling.

The present methodology, therefore, is able to provide good estimates for pressure distribution and deflection angle, and able to predict the flow correctly.

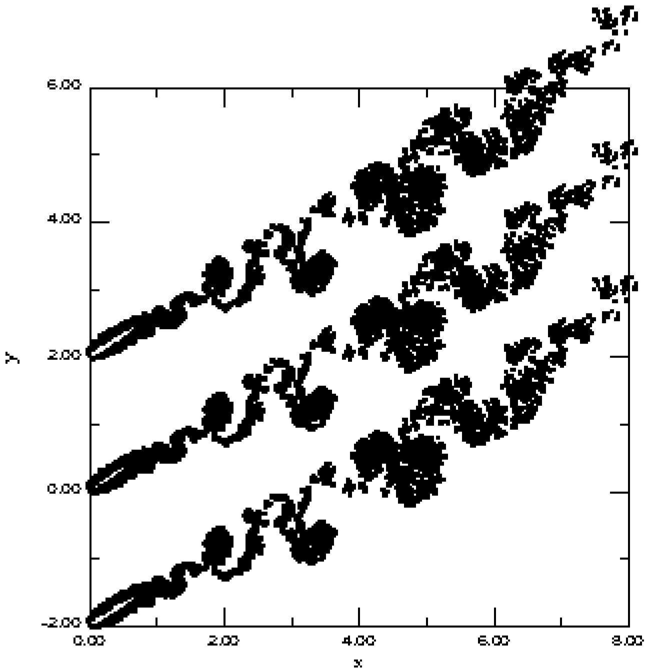

Figure 3. NACA 65-410 airfoil: Vortex wake structure at $t = 10$
Case II: $\mathrm{Re} = 4.45x10^5$, $M = 80$, $Dt = 0{,}05$, $\beta_1 = 45.00^0$ and $\beta = 33.60^0$.

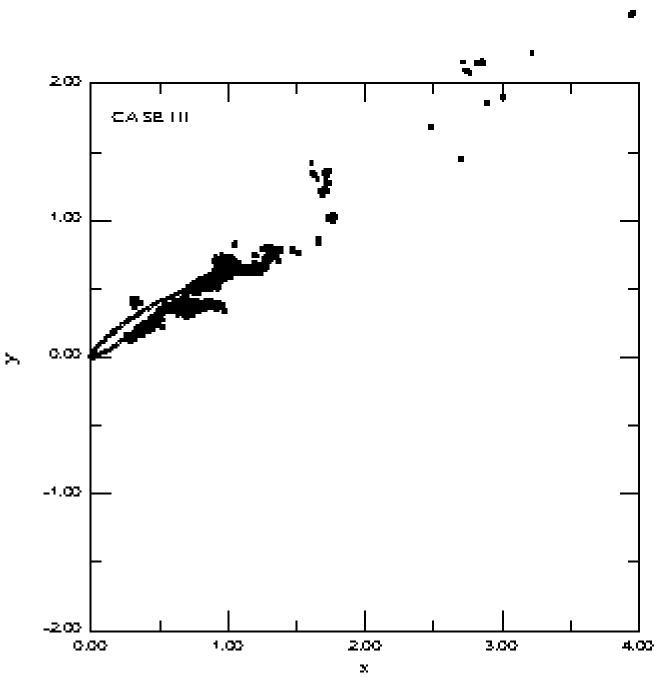

Figure 4. Position of the vortices in the cloud simulating turbulence at $t = 10$
Case III: $\mathrm{Re} = 4.45x10^5$, $M = 80$, $Dt = 0{,}05$, $\beta_1 = 45.00^0$ and $\beta = 37.30^0$.

These calculation required 1 hr of CPU time on a Pentium II/400 Mhz without to consider turbulence modeling and 20-30 min of CPU time plus with turbulence modeling.

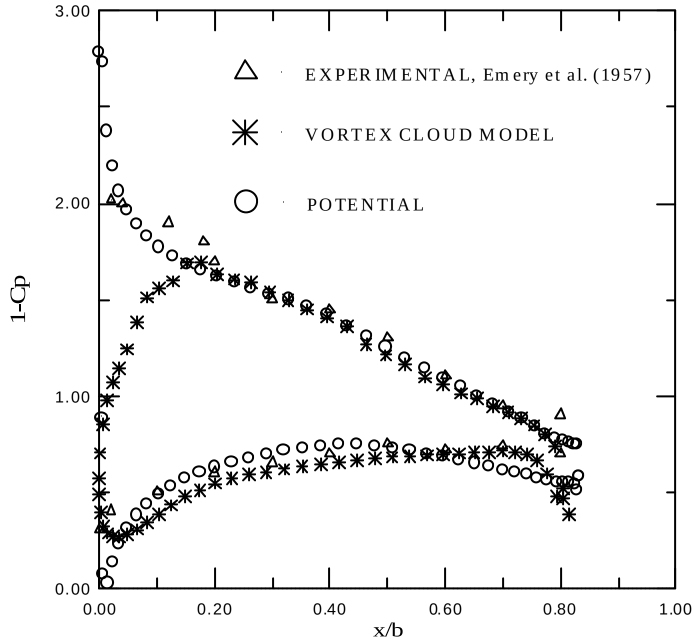

Figure 5(a). Computed without sub grid-scale modeling

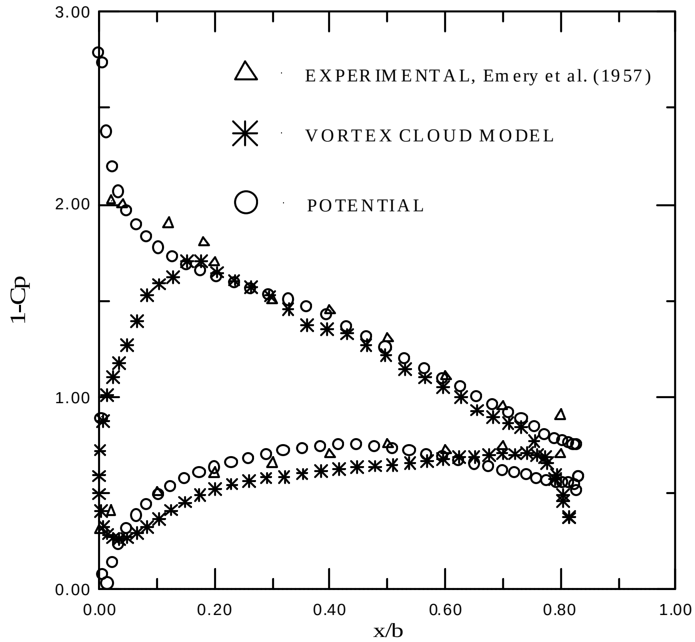

Figure 5(b). Computed with sub grid-scale modeling

Figure 5. Predicted pressure distribution on a linear cascade of NACA 65-410 airfoils

Case II: $Re = 4.45x10^5$, $M = 80$, $Dt = 0,05$, $\beta_1 = 45.00^0$ and $\beta = 33.60^0$.

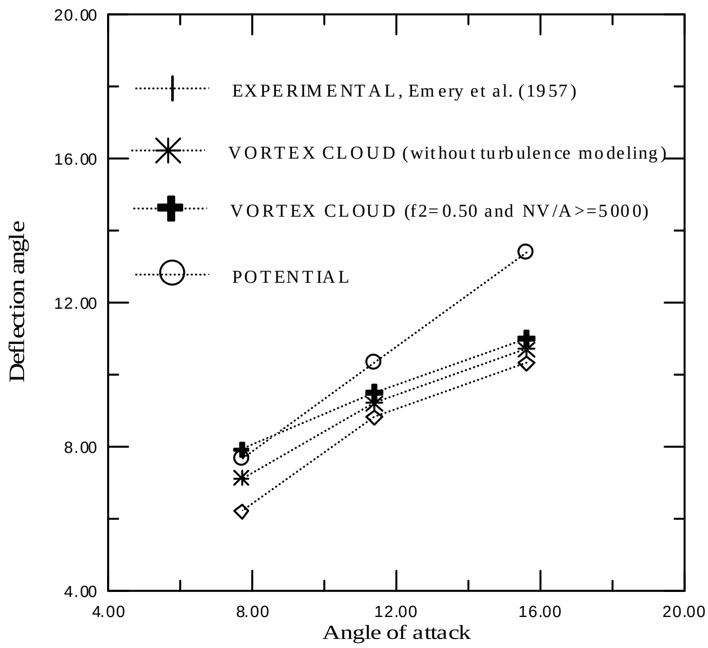

Figure 6. Deflection angle in a linear cascade of NACA 65-410 airfoils, $\mathrm{Re} = 4.45x10^{5}$, $M = 80$, $Dt = 0{,}05$ and $b/s = 0.50$.

5. Conclusions

The vortex-shedding flow in a linear cascade of airfoils was modeled based on the Navier-Stokes equations with a sub grid-scale turbulence modeling. The Navier-Stokes equation was solved numerically; to simulate the sub grid structures of the flow, the turbulence sub grid modeling uses a second-order velocity structure function adapted to the Lagrangian scheme. A new methodology for the pressure distribution calculation, in a linear cascade, was presented; this methodology is specially adapted to the vortex method. The differences encountered in the comparison of the numerical simulation with the experimental results are attributed mainly to the inherent three-dimensionality of the real flow for such a value of the Reynolds number, which is not modeled in the simulation.

The methodology developed here presents promising features and predicts well the main global parameters of interest to the designer. The analysis shows that the Vortex Method with turbulence modeling can improve the previously obtained results without turbulence modeling. Use of a larger number of panels distributed on the airfoil surface can also improve the results, but for this it is necessary a larger number of free vortices in the cloud and consequently a larger computational effort.

6. Acknowledgement

The authors would like to acknowledge CAPES and CNPq for its financial support during the time of this project. The authors are also grateful to Dr. A. Silveira-Neto at the Universidade de Uberlândia, whose detailed comments helped to improve the implementation of the sub grid-scale models.

5. 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. Escola Federal de Engenharia de Itajubá, Departamento de Mecânica, Itajubá, MG, Brasil, Ph. D. Thesis.

Alcântara Pereira, L.A. e Hirata, M.H., 2000, “Análise do escoamento em perfis aerodinâmicos em grades de turbomáquinas utilizando o método de vórtices discretos”. VIII ENCIT, Porto Alegre, RS, Brasil.

Alcântara Pereira, L.A., Hirata, M.H. e Silveira-Neto, A., 2000, “Simulação numérica do escoamento de um fluido newtoniano usando uma nuvem de vórtices e modelagem sub-malha de turbulência”. IV SIMMEC, Maio, Uberlândia, MG, Brasil, pp. 465-472.

Batchelor, G.K., 1963, The theory of homogeneous turbulence. Cambridge Univ. Press, Cambridge, England, U.K.
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, 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.
Giesing, J.P., 1964, Extension of the Douglas Neumann program to problems of lifting, infinite cascades. Douglas Aircraft Co., Inc., Report n° LB-31653.
He, F. and Su, T.C., 1999, "An improved offset model for vorticity shedding from a solid boundary in discrete vortex element method". Department of Mechanical Engineering, FAU: Florida Atlantic University, Boca Raton, FL 33431, pp. 1-24.
Kamemoto, K., 1993, "Procedure to estimate unstead pressure distribution for vortex method" (In Japonese). Trans. Jpn. Soc. Mech. Eng., vol. 59, n° 568 B, pp. 3708-3713
Kamemoto, K., 1994, "Development of the vortex methods for grid-free lagrangian direct numerical simulation". Proc. Third JSME-KSME, Sendai, Japan, pp. 542-547.
Lesieur, M. and Metais, O., 1996, "New trends in large-eddy simulation of turbulence". An.Review in Fluid Mechanics, vol. 28, pp. 45-82.
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.
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., 1991, Vortex element method for fluid dynamic analysis of engineering systems. Cambridge Univ. Press, Cambridge, England, U.K..
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, Nov. 4-5, pp. 1-15.
Métais, O. and Lesieur, M., 1992, "Spectral large-eddy simulations of isotropic and stably-stratified turbulence", J. Fluid Mech. , vol. 239, pp. 157-94.
Mustto, A.A., Hirata, M.H. and Bodstein, G.C.R., 1998, "Discrete vortex method simulation of the flow around a circular cylinder with and without rotation". A.I.A.A. Paper 98-2409, Proceedings of the 16th A.I.A.A. Applied Aerodynamics Conference, Albuquerque, NM, USA.
Porthouse, D.T.C., 1983, Numerical simulation of aerofoil 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, n° 2, pp. 237-254.
Smagorinsky, J., 1963, "General circulation experiments with the primitive equations." Mon. Weather Rev.. vol. 91, n° 3, pp. 99-164.
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.