HEAT TRANSFER IN THE WAKE BEHIND A BODY USING A PARTICLE METHOD

Luiz Antonio ALCÂNTARA PEREIRA
Departamento de Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
luizantp@iem.efei.br

Miguel Hiroo HIRATA
Departamento de Mecânica, UNIFEI, CP. 50, Itajubá, Minas Gerais, 37500-903, Brasil
hirata@iem.efei.br

Abstract. *In this paper the vortex cloud method was extended to take into account the convection and diffusion heat transfer. Discrete heat particles were generated close to a wall surface in addition to nascent vortex elements. The unsteady flow and heat transfer were simulated around a circular cylinder in a uniform flow. The result of the surface and time-averaged Nusselt number at constant surface temperature showed reasonable agreement with that of experiment.*

Keywords: *vortex and heat element method, heat transfer, panel method, aerodynamic loads, lagrangian description.*

1. Introduction

The Vortex Methods is employed in the literature to a variety of problems from the view point of engineering application, such as shear layer, external flow, internal flow, flow with multiple bodies, etc. 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 (e.g. references Chorin, 1973; Sarpkaya, 1989; Sethian, 1991; Kamemoto, 1994; Lewis, 1999 and Alcântara Pereira et all, 2002). 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.

On the other hand, there are only a few examples of the simulations of vorticity and heat transport using a particle method. Ghoniem & Sherman (1985) investigated one-dimensional heat diffusion using random walk scheme (Chorin, 1973, 1978). They present a complete analysis of heat particles with different properties and the vorticity generation due to the heat transfers process. Ghoniem et all (1988) and Zhang & Ghoniem (1993) handles shear layers and a rise of a plume in two-dimensions. The diffusion process was simulated using the core spreading method (Leonard, 1980) and the density difference was considered, although heat transfer was not.

Smith & Stansby (1989) and Stansby & Dixon (1983) used a hybrid method to analyse the vorticity and the heat transport around a circular cylinder. Using the vortex in cell method incorporate with the random walk model, they introduced both vortex and temperature particles according to the similarity of equations of vorticity transport and energy. The vorticity generation due to heat and natural convection are not accounted for.

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

Ogami (2001) presented two models for creating vortices from temperature particles. Initially he modeled the vorticity equation as it is, and it is regarded as a natural extension of the method of Ghoniem & Sherman (1985). Next, a vortex pair (one positive and one negative) is generated from one temperature particle. In addition, the diffusion velocity method (Ogami & Akamatsu, 1991) was used in order to handle heat and vorticity diffusion. In the one-dimensional field, the models are compared with analytic solution, and the accuracy and validity are clarified. In the two-dimensional field, the application sample with the natural convection and the interaction between heat and vortex is shown.

The purpose of the present paper is to analyse the mechanism of heat transfer in the separated flow region behind a circular cylinder. As the formulation of energy equation is very similar to the vorticity transport equation for a two-dimensional and incompressible flow, discrete heat particles are introduced into the flow field close to a wall surface in addition to nascent vortex elements.

Straight-line panels, with constant-strength vortex distribution, simulate the body surface. The strength of the discrete vortices is obtained directly without going through any additional calculation. Using a primary diffusion process this vorticity is replaced by Lamb vortices located nearby the body surface. The dynamics of the body wake is computed using the convection-diffusion splitting algorithm; the convection process is carried out with the Lagrangian Adams-Bashforth time-marching scheme and the diffusion process is simulated using the random walk method.

The temperature is considered constant around the body surface. The heat transport from the body surface to the fluid nearby the body surface is determined by the temperature gradient at the surface. The thermal diffusion is simulated by the random walk method. In this approach, the effect of buoyancy is not considered because the present study is focused on the forced convection heat transfer.

2. Formulation of the Physical Problem

Consider the incompressible fluid flow of a newtonian fluid around a circular cylinder in an unbounded two-dimensional region. Figure (1) shows the incident flow, defined by free stream speed U, free stream temperature T_∞ and the domain Ω with boundary $S = S_1 \cup S_2$, S_1 being the body surface at constant temperature T_w and S_2 the far away boundary.

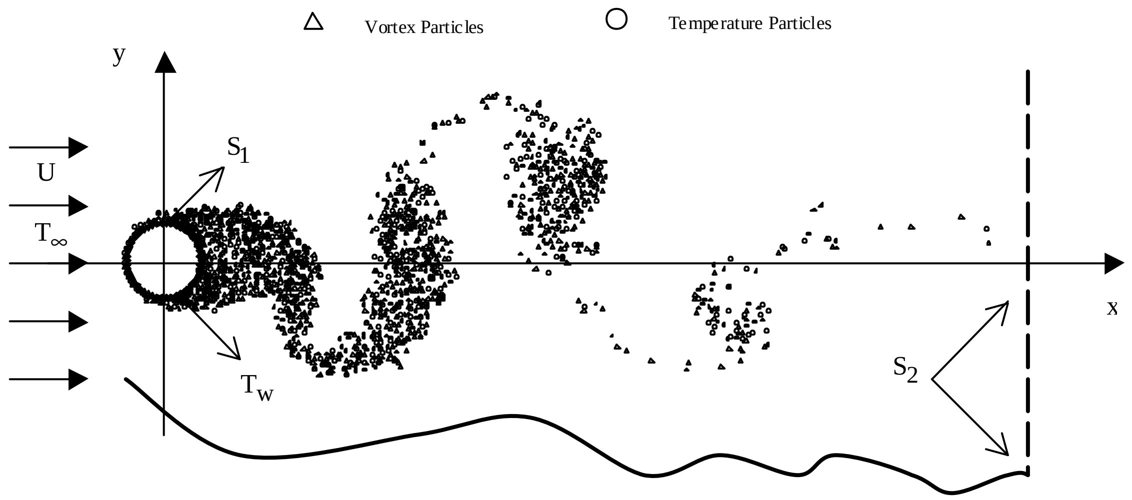

Figure 1. Flow around a bluff body.

The viscous and incompressible fluid flow is governed by the continuity and the Navier-Stokes equations, which can be written in the form

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

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

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

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

where υ is the fluid kinematics viscosity coefficient.

The impenetrability and no-slip conditions on the body surface are written as

$$u_n = \mathbf{u} \cdot \mathbf{e}_n = 0 . \tag{3a}$$

$$u_\tau = \mathbf{u} \cdot \mathbf{e}_\tau = 0 . \tag{3b}$$

where $\mathbf{e}_n$ and $\mathbf{e}_\tau$ being, respectively, the unit normal and tangential vectors. One assumes that, far away, the perturbation caused by the body fades as

$$|\mathbf{u}| \to 1 \quad \text{at } S_2 . \tag{3c}$$

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), obtaining the well-known 2-D vorticity transport equation

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

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

The energy equation of the forced convective heat transfer is expressed as

$$\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T = \alpha \nabla^2 T \tag{5}$$

where T is temperature and α is the thermal diffusivity, being

$$T = T_w \quad \text{at } S_1 \text{ and} \tag{5a}$$

$$T = T_\infty \quad \text{in } \Omega, \text{ at } t = 0 . \tag{5b}$$

3. Numerical Simulation

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

$$\frac{\partial \omega}{\partial t} + \mathbf{u} \cdot \nabla \omega = 0 . \tag{6}$$

$$\frac{\partial \omega}{\partial t} = \frac{1}{Re} \nabla^2 \omega . \tag{7}$$

Vorticity and heat transport are governed by the same equation. This fact seems to suggest that in solving Eq. (5) one can use an analogous splitting scheme as used for the vorticity equation, see Eq. (4).

Convection is governed by Eq. (6) and the velocity field is given by

$$u - iv = 1 + \frac{i}{2\pi} \sum_{n=1}^{M} \gamma(S_n) \int_{\Delta S_n} \frac{d}{dz} \ln(z - \zeta) d\zeta + \frac{i}{2\pi} \sum_{k=1}^{N} \frac{\Delta\Gamma_k}{z - z_k} . \tag{8}$$

Here, u and v are the x and y components of the velocity vector $\mathbf{u}$ and $i = \sqrt{-1}$. The first term in the right hand side is the contribution of the incident flow; the summation of M integral terms comes from the panels distributed on the body surface. The second summation is associated to the velocity induced by the cloud of N free vortices; it represents the vortex-vortex and heat particles-vortex interactions.

In order to remove the singularity in the second summation of Eq. (8) Lamb vortices are used , whose mathematical expression for the induced velocity of the k-th vortex with strength $\Delta\Gamma_k$ in the circumferential direction u_{θ_k} , is (Mustto et all, 1998)

$$u_{\theta_k} = \frac{\Delta\Gamma_k}{2\pi r} \left\{ 1 - \exp\left[-5.02572 \left(-\frac{r}{\sigma_0} \right)^2 \right] \right\} \tag{9}$$

where σ_0 is core radius of the Lamb vortex.

In this particular equation r is the radial distance between the vortex center and the point in the flow field where the induced velocity is calculated.

Each vortex and heat particle distributed in the flow field is followed during numerical simulation according to the Adams-Bashforth second-order formula (Ferziger, 1981)

$$z(t+\Delta t) = z(t) + [1.5u(t) - 0.5u(t-\Delta t)]\Delta t + \xi \tag{10}$$

in which z is the particle position of a particle, Δt is the time increment and ξ is the random walk displacement. According to Lewis (1991), the random walk displacement is given by

$$\xi = \sqrt{4ß\,\Delta t \ln\left(\frac{1}{P}\right)}\left[\cos(2\pi Q) + i\sin(2\pi Q)\right] \tag{11}$$

where $ß = Re^{-1}$ for the vortex particles and $ß = \alpha$ to the heat particles; P and Q are random numbers between 0.0 and 1.0.

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

$$Y = p + \frac{u^2}{2}, \; u = |\mathbf{u}| . \tag{12}$$

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 was obtained through the following integral formulation (Shintani & Akamatsu, 1994)

$$H\overline{Y_i} - \int_{S_1} \overline{Y}\nabla G_i \cdot \mathbf{e}_n dS = \iint_{\Omega} \nabla G_i \cdot (\mathbf{u}\times\omega)d\Omega - \frac{1}{Re}\int_{S_1} (\nabla G_i \times \omega)\cdot \mathbf{e}_n dS \tag{13}$$

where H is 1.0 inside the flow (at domain Ω) and is 0.5 on the boundary S_1. $G_i = (1/2\pi)\log R^{-1}$ is the fundamental solution of Laplace equation, R being the distance from ith vortex element to the field point.

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 in the cloud. Therefore it does not require any additional calculation at mesh points. Numerically, Eq. (13) is solved by mean of a set of simultaneous equations for pressure Y_i.

The surface heat flux is determined by Fourier's Law

$$\dot{q} = -\lambda \frac{dT}{dn} \tag{14}$$

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

$$\Delta Q_j = \alpha\Delta t \frac{(T_w - T_j)}{\varepsilon}\Delta S_j \tag{15}$$

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

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

$$T(z) = \sum_j \frac{\Delta Q_j}{\pi\sigma_T^2} \exp\left[-\frac{(z - z_j)^2}{\sigma_T^2}\right] \tag{16}$$

where σ_T is the core radius of the heat particles

The local Nusselt number Nu_i is evaluated from

$$Nu_i = \frac{\dot{q}_i \, b}{\lambda \Delta T}. \quad (17)$$

4. Discussion and Results

The numerical simulation was restricted to the flow around a non rotating circular cylinder, where the effects of natural convection and radiation on the heat transfer were neglected. The conditions of flow and calculations are as follows: Reynolds number $Re=10^4$, Prandtl number $Pr=0.71$, the temperature of body wall $T_W = 363$ K (constant), the temperature of approaching flow $T_\infty = 293$ K (constant), the number of vortex sheet panels distributed around a cylinder M=50. The simulation was performed up to 800 time steps with magnitude $\Delta t = 0{,}05$. In each time step the vortex and temperature particles are shedding into the flow through a displacement $\varepsilon = \sigma_0 = \sigma_T = 0.03b$ normal to the straight-line elements (panels).

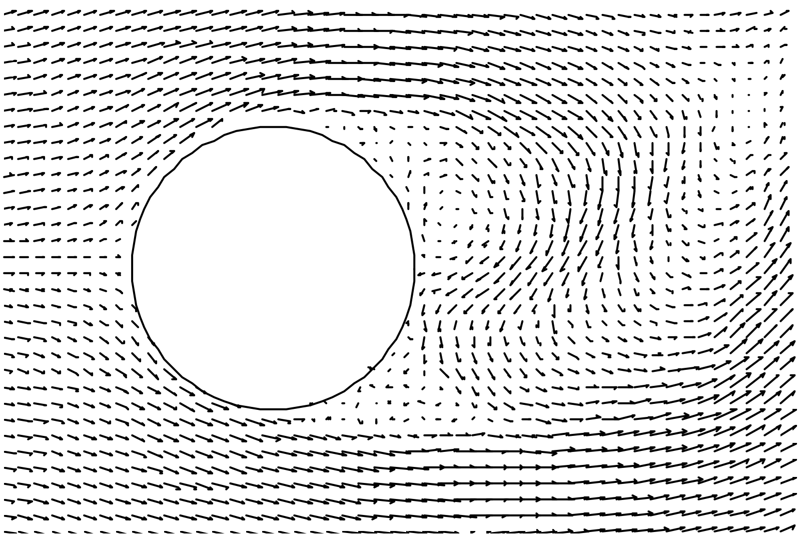

Figure 2. Instantaneous velocity field at t=40 around a circular cylinder.

Figure (2) shows the instantaneous velocity field at t=40. The aerodynamics loads computations starts at t=10. The results of the viscous flow numerical simulation are presented in Tab. (1). In this table one can find also experimental (with 10% uncertainty) and numerical results.

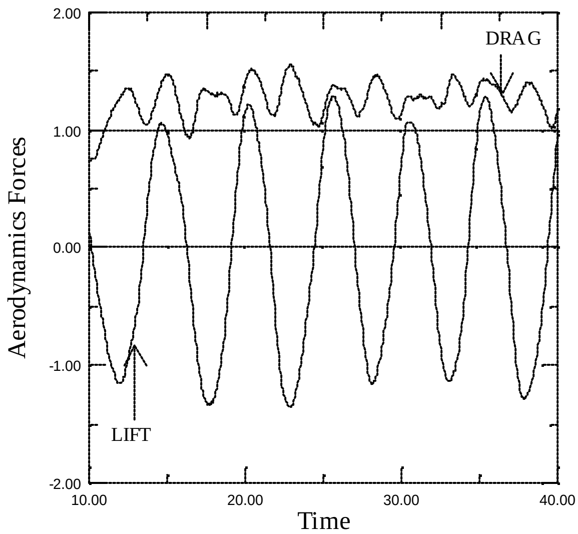

Figure 3. Time history of the aerodynamics loads at Reynolds number $Re=10^4$.

The agreement between the calculated and experimental results is very good for the Strouhal number. The drag coefficient shows a high value as compared to the experimental result. One should observe, however, that three-dimensional effects are non-negligible for the Reynolds number used in the simulation. Therefore one can expect that a two-dimensional computation of such a flow must produce higher values for the drag coefficient. On the other hand, the Strouhal number is insensitive to these three-dimensional effects. Figure (3) shows that the lift coefficient, C_l, oscillates around zero, as expected.

Table 1. Comparisons of the mean drag coefficient and Strouhal number with other numerical and experimental results for a circular cylinder.

$Re=10^4$	$\overline{C_d}$	$\overline{St}$
Schlichting (1979) [experimental]	-	0.20
Blevins (1984) [experimental]	1.10	-
Ogami & Ayano (1995) [numerical]	0.84	-
Present Simulation	1.26	0.21

The mean pressure coefficient along the cylinder surface is compared with experimental available in the literature. Figure (4) shows the pressure distribution calculated in the present simulation; it is referred to as "numerical". The experimental values (Goldstein, 1938) were obtained for Re=9900. The present results agree very well with the experimental ones. In the present numerical simulation the predict separation points occurs at around 86°, while the experimental value is around 80°.

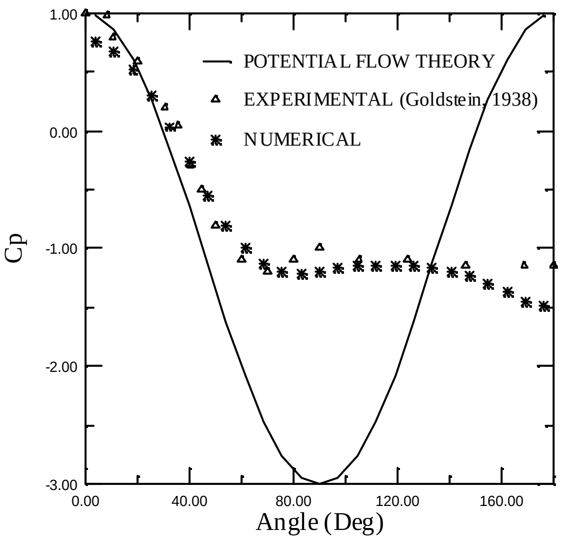

Figure 4. Predicted pressure distribution for circular cylinder at Reynolds number $Re=10^4$.

Figure (5) shows the heat particles in the cloud, after 800 times steps of the simulation.

Figure 5. Distribution of temperature elements for $Re=10^4$, Pr=0.71 and at t=40.

The time-averaged local Nusselt number distribution is shown in Fig. (6), which are compared with experimental result by Igarashi (1984). The numerical results reasonably coincide with the experiments. The time-averaged local Nusselt number computations starts at t=10.

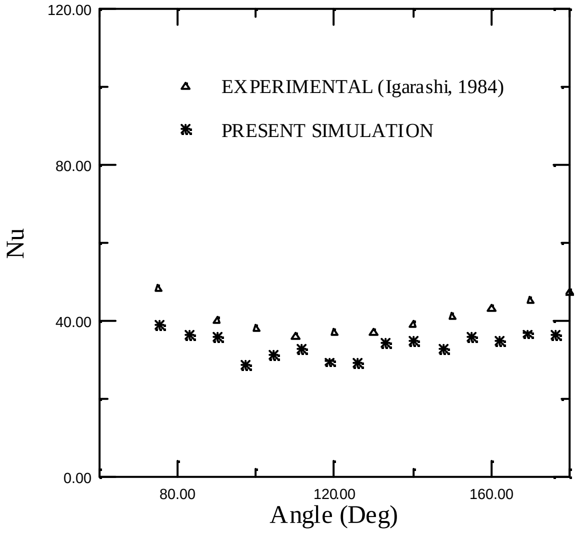

Figure 6. Time-averaged local Nusselt number distribution: $Re=10^4$ and $Pr=0.71$.

5. Conclusions

In the present study, a vortex and heat element method was presented for the analysis of unsteady heat transfer in a flow around a body. The time-averaged distribution of the local Nusselt number and the calculated values for aerodynamics loads around a non rotating circular cylinder show good agreement with the data from the literature. The differences encountered in the comparison of the results simulation with the experimental are attributed mainly to the inherent three-dimensionality of the real flow. Use of a larger number of panels distributed on the body surface can also improve the results, but for this is necessary a larger number of free vortex and temperature in the cloud and consequently a larger computational effort. The present calculation required 22 h of CPU time in a PENTIUM II/400 Mhz.

The main objective of the work with the implementation of a vortex and heat particles method for the analysis of unsteady and forced-convective heat transfer in a flow around a body has been achieved.

The present methodology, therefore, is able to provide good estimates for Strouhal number, lift and drag coefficients, pressure distribution and time-averaged Nusselt number, and is able to predict the flow correctly in a physical sense. As a future work, the effect of buoyancy will be carried out.

6. Acknowledgement

The work described in this paper was supported by the FAPEMIG (TEC-00025/02 Process).

7. References

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 / Computational Fluid Dynamics Journal, Vol. 11, No. 3, October, pp. 315-322.

Blevins, R.D., 1984, "Applied Fluid Dynamics Handbook", Van Nostrand Reinhold Co..

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

Chorin, A.J., 1978, "Vortex Sheet Approximation of Boundary Layers", J. Comp. Physics, Vol. 27, pp. 428-442.

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

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

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

Goldstein, S., 1938, "Modern Developments in fluid Dynamics", Cambridge University Press, Cambridge, England, U.K.

Igarashi, T., 1984, "Flow and Heat Transfer in the Separated Region around a Circular Cylinder" (In Japaese), Trans. JSME B 50-416, pp. 3008, 3014.

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

Kamemoto, K., 1994, "Development of the Vortex Methods for Grid-Free Lagrangian Direct Numerical Simulation", Proc. Third JSME-KSME, Sendai, Japan, pp. 542-547.

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

Leonard, A., 1980, “Vortex Methods for Flow Simulations”, J. Comp. Physics, Vol. 37, pp. 289-335.

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.

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”, AIAA Paper 98-2409, Proceedings of the 16th AIAA Applied Aerodynamics Conference, Albuquerque, NM, USA, June.

Ogami,Y. and Akamatsu,T., 1991, “Viscous Flow Simulation Using the Discrete Vortex Model -The Diffusion Velocity Method”, Computers & Fluids, Vol 19, pp. 433-441.

Ogami, Y. and Ayano, Y., 1995, “Flows around a Circular Cylinder Simulated by the Viscous Vortex Model the Diffusion Velocity Method ”, Computational Fluid Dynamics Journal, Vol. 4, No. 3, October, pp. 383-399.

Ogami, Y., 2001, “Simulation of Heat-Fluid Motion by the Vortex Method”, JSME International Journal, Series B, Vol. 44, No. 4, pp. 513-519.

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

Schlichting, H., 1979, “Boundary Layer Theory”, McGraw-Hill, 7a Edition.

Sethian, J.I., 1991, “A Brief Overview of Vortex Method, Vortex Methods and Vortex Motion”, SIAM Philadelphia, pp. 1-32.

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.

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

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

Uhlman, J.S., 1992, “An Integral Equation Formulation of the Equation of an Incompressible Fluid”, Naval Undersea Warfare Center, TR 10-086.

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