APPLICATION OF THE VORTEX METHOD TO CENTRIFUGAL PUMP IMPELLER

Ednei Pedro Gomes Putini, ednei_pedro@yahoo.com.br
Luiz Antonio Alcântara Pereira, luizantp@unifei.edu.br
Universidade Federal de Itajubá, Instituto de Engenharia Mecânica, CP 50, Av. BPS 1303, Itajubá, MG, CEP. 37500-903

Miguel Hiroo Hirata, hirata@fat.uerj.br
FAT/UERJ, Campus Regional de Resende, Estrada Resende–Riachuelo, Resende, RJ

Abstract. *The vortex cloud method was developed primarily for the simulation of viscous flow through turbine or compressor cascades. The previous works presented a simple scheme for the numerical simulation of the flow in an infinite linear cascade. Due the periodicity, presented by the cascade, only one element – the reference airfoil – needed be considered if the interference effects are taken into account. In the present study, without introducing the periodicity of the blade-to-blade stream, the two-dimensional, unsteady flow through a centrifugal impeller of pump is simulated using vortex method. The results show instantaneous vortex flow pattern of the flow through the impeller. The aim of this study was to predict the presence of rotating stall in blade. This Lagrangian manner offers future scope for study of centrifugal impeller rotating in volute casing for which Euler methods may be less adaptable.*

Keywords*: vortex cloud analysis, centrifugal pump impeller, rotating stall, vortex dynamics*

1. INTRODUCTION

Flows around moving boundaries at high Reynolds number occur in many areas of engineering. A typical example of this kind is the flow within a turbomachinery. For a pump configuration the moving boundaries are the blades of impeller which rotate with respect to the stator and/or to the spiral casing. Understanding of the vortex-shedding flow behind blades of impeller is of great fundamental and practical importance to design of turbomachines. The impeller-viscous wake interaction originates from the impingement and the convection of the wakes shed from the preceding blades in the relative motion. A large number of detailed experimental investigations and theoretical or numerical studies are reported in the literature to unsteady flow investigation in turbomachines. Most of these efforts concern the unsteadiness in axial turbomachines.

In order to solve the turbomachines problem numerically, we can use either Eulerian or Lagrangian methods. In the past three decades, the Vortex Methods have been developed and applied for analysis of complex vortical flows and simulation of unsteady flows related with problems in various engineering fields, because they are standing on simple algorithm based on physics of flows and their numerical stability is usually quite well, and it should be noted that grid generation in the flow field is not necessary (Kamemoto, 2004).

The first application of Vortex Method to turbomachinery blade rows, including the prediction of rotating stall in compressors and vibrations induced by blade row wake interaction, was published by Lewis and 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 the paper presented by Alcântara Pereira *et al.* (2004), the surface of the reference airfoil was represented by straight-line panels, with a constant density vortex distribution on them. A new approach to the pressure calculation was presented to that one used by Lewis (1989). This approach represents an enhancement to the previous ones not only due to the more accurate computed values, but also because it allows one to compute the pressure distribution on the body surface as well as in the whole fluid domain; this feature can be of fundamental importance in many engineering problems. In order to take into consideration the effects of the phenomena that take place in the micro scales a new methodology was presented. With this methodology one can consider the effects of turbulence in the flow that develops in and around complex geometry structures. The Vortex Method, a particle or Lagrangian method was used in combination with a sub grid scale modeling for turbulence that employs a second-order velocity structure function of the filtered field, see Alcântara Pereira *et al.* (2002).

In the all the above studies, for cascades the assumption was made that each blade flow was identical, a not unreasonable assumption for unstalled turbine or fan blade rows. For off-design angles of attack leading to stall it is know that there can significant blade-to-blade variations. Lewis (2004) extended the analysis to deal with such situations.

On the other hand, Zhu *et al.* (1998) presented a new procedure to simulate the vorticity transport near a centrifugal impeller boundary using Vortex Method. In their paper, a number of nascent vortices were introduced according to diffusion and convection vorticity near the boundary. The two-dimensional unsteady features of the whole flow field were solved without introducing the periodicity of a blade-to-blade flow. The results showed how the separation of boundary layer develops into a strong vortex structure and how the vortex is periodically shed and moves and produces an oscillatory flow in a blade-to-blade passage.

In this paper, the Vortex Method developed by Recicar *et al.* (2006) is extended to simulate the vorticity transport near a blades boundary without introducing the periodicity of a blade-to-blade flow (Zhu *et al.*, 1998). The present methodology introduces geometrical simplifications to prediction of the two-dimensional unsteady flow established in a radial flow centrifugal pump. More specifically, the blades are represented by NACA 0012 base profile.

Vortex cloud modeling offers great potential for numerical analysis of important problems in fluid mechanics. 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 Methods (Chorin, 1973; Leonard, 1980; Sarpkaya, 1989; Sethian, 1991; Lewis, 1999; Alcântara Pereira *et al.*, 2002; Kamemoto, 2004). Vortex Methods offer 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.

2. FORMULATION OF THE PHYSICAL PROBLEM

2.1. Definitions

The problem to be considered is that of incompressible flow (of a Newtonian fluid) and two-dimensional past a centrifugal impeller of pump. The analysis introduces geometrical simplifications to prediction of the two-dimensional unsteady flow established in a radial flow centrifugal pump. Here, we represent each blade for the simplest situation of a symmetrical aerofoil, NACA 0012. The blades rotate anti-clockwise with angular velocity of the impeller, λ. The volumetric flow rate per unit breadth of the impeller is established considering a point source located at (x,y)=(0,0).

This is represented, in Fig. 1, by four NACA 0012 aerofoils immersed in a radial flow with velocity V_r. Note that the (x, o, y) is the inertial frame of reference and the (ξ, o, η) is the coordinate system fixed to the impeller; this coordinate system rotate anti-clockwise, being $\varphi_t=\lambda t$.

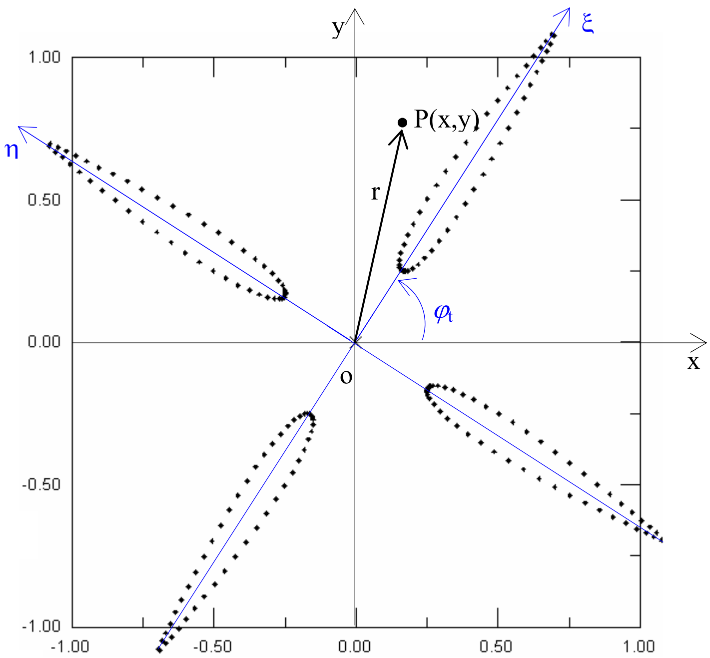


Figure 1 – Definitions of the physical domain

The boundary S of the fluid domain is $S = S_b \cup S_\infty$; being S_∞ the far away boundary, which can be viewed as $r = \sqrt{x^2 + y^2} \to \infty$, and S_b the blades surface.

In the impeller fixed coordinate system, the surface S_b is defined by

$$\begin{Bmatrix} \xi \\ \eta \end{Bmatrix} = \begin{Bmatrix} \cos(\varphi_t) & \sin(\varphi_t) \\ -\sin(\varphi_t) & \cos(\varphi_t) \end{Bmatrix} \begin{Bmatrix} x \\ y \end{Bmatrix} . \tag{1}$$

2.2. Governing Equations

For an incompressible fluid flow the continuity is written as

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

where $\mathbf{u} \equiv (u, v)$ is the velocity vector.

If, in addition, the fluid is Newtonian with constant properties the momentum equation is represented by the Navier-Stokes equation as

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

Re stands for the Reynolds number defined as $Re = \frac{D_2 V_0}{\upsilon}$, where V_0 is defined as radial velocity at the entrance of the impeller for the design point and D_2 is defined as outer diameter of the impeller.

On the impeller surface the adherence condition has to be satisfied. This condition is better specified in terms of the normal and tangential components as

$$(\mathbf{u} \cdot \mathbf{n}) = (\mathbf{v} \cdot \mathbf{n}) \text{ on } S_b \text{, the impenetrability condition} \tag{4}$$

$$(\mathbf{u} \cdot \boldsymbol{\tau}) = (\mathbf{v} \cdot \boldsymbol{\tau}) \text{ on } S_b \text{, the no-slip condition} \tag{5}$$

here $\mathbf{n}$ and $\boldsymbol{\tau}$ are unit normal and tangential vectors and $\mathbf{v}$ is the blade surface velocity vector.

Far from the impeller (for $r \to \infty$, in Fig. 1) one assumes that the perturbation due to the rotation impeller fades away, that is

$$|\mathbf{u}| \to 0 \ . \tag{6}$$

One should mention that the above boundary value problem was made non-dimensional using V_0 and D_2 as characteristic quantities. Normalized non-dimensional time t is defined as $t = \frac{TV_0}{D_2}$, where T is the time elapsed after revolution of the impeller starting at a constant speed.

3. THE VORTEX METHOD

3.1. Viscous Splitting Algorithm (Chorin, 1973)

Taking the curl of the Navier-Stokes equation and with some algebraic manipulations one gets the vorticity equation which presents no pressure term. In two-dimensions this equation reads

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

which is an scalar equation since ω is the only component of the vorticity vector $\boldsymbol{\omega} = \nabla \times \mathbf{u}$.

The left hand side of the above equation carries all the information needed for the convection of vorticity while the right hand side governs the diffusion. Following Chorin (1973) we use the viscous splitting algorithm, which, for the same time step of the numerical simulation, says that

Convection of vorticity is governed by

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

Diffusion of vorticity is governed by

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

3.2. Convection and diffusion of vorticity

The standard numerical strategy is to represent the vorticity in the fluid domain by a large number N of small discrete vortices $\Delta\Gamma_k$. The numerical analysis is conducted over a series of small discrete time steps Δt for each of which a discrete vortex element $\Delta\Gamma_k$ is shed from each body surface element. The intensity $\Delta\Gamma_k$ of these newly generated vortices is determined using the no-slip condition, see Eq. (5).

For the convection of the discrete vortices of the cloud, Eq. (8) is written in its Lagrangian form as

$$\frac{dx^{(i)}}{dt} = u^{(i)}(x, y, t) , \tag{10a}$$

$$\frac{dy^{(i)}}{dt} = v^{(i)}(x, y, t) , \tag{10b}$$

being (i) = 1, N.

A second order solution to this equation is given by the Adams-Bashforth formula (Ferziger, 1981)

$$x^{(i)}(t + \Delta t) = x^{(i)}(t) + \left[1.5u^{(i)}(t) - 0.5u^{(i)}(t - \Delta t)\right]\Delta t \tag{11a}$$

$$y^{(i)}(t + \Delta t) = y^{(i)}(t) + \left[1.5v^{(i)}(t) - 0.5v^{(i)}(t - \Delta t)\right]\Delta t . \tag{11b}$$

The diffusion of vorticity is taken care of using the random walk method (Lewis, 1999). The random displacement $Z_d \equiv (x_d, y_d)$ for vortex (i) is defined as

$$x_d^{(i)} = \left[\cos(2\pi Q)\right]\sqrt{\frac{4\Delta t}{\mathrm{Re}} \ln\left(\frac{1}{P}\right)} \tag{12a}$$

$$y_d^{(i)} = \left[\sin(2\pi Q)\right]\sqrt{\frac{4\Delta t}{\mathrm{Re}} \ln\left(\frac{1}{P}\right)} \tag{12b}$$

where P and Q are random numbers in the range 0.0 to 1.0. Therefore the final displacement is written as

$$x^{(i)}(t + \Delta t) = x^{(i)}(t) + \left[1.5u^{(i)}(t) - 0.5u^{(i)}(t - \Delta t)\right]\Delta t + x_d^{(i)} \tag{13a}$$

$$y^{(i)}(t + \Delta t) = y^{(i)}(t) + \left[1.5v^{(i)}(t) - 0.5v^{(i)}(t - \Delta t)\right]\Delta t + y_d^{(i)} . \tag{13b}$$

3.3. Numerical Implementation

The $u^{(i)}$ and $v^{(i)}$ components of the velocity induced at the location of the vortex (i) can be written as

$$u^{(i)} = ur^{(i)} + ub^{(i)} + uv^{(i)} \tag{14a}$$

$$v^{(i)} = vr^{(i)} + vb^{(i)} + vv^{(i)} \tag{14b}$$

where, $\mathbf{ur}^{(i)} \equiv [ur^{(i)}, vr^{(i)}]$ is the velocity vector of the radial direction,

$\mathbf{ub}^{(i)} \equiv [ub^{(i)}, vb^{(i)}]$ is the velocity vector induced by the impeller at the location of vortex (i),

$\mathbf{uv}^{(i)} \equiv [uv^{(i)}, vv^{(i)}]$ is the velocity vector induced at the vortex (i) due to the vortex cloud.

The $ur^{(i)}$ and $vr^{(i)}$ calculations present no problems and they follow the usual Vortex Method procedures.

To be considered impeller rotation, however, the blades boundary conditions can not be transferred from the actual position to the mean position (Moura *et al.*, 2007). As the each blade surface is simulated by MB straight line panels on which singularities are distributed (Panels Method) it is convenient to calculate the impeller induced velocity in the moving coordinate system. For that one has to observe the following

- The fluid velocity on the each blade surface is written as

$$\mathbf{u}(\xi, \eta; t) = (ur + u_\lambda)\mathbf{i} + (vr - v_\lambda)\mathbf{j}\,. \tag{15}$$

As a consequence of impeller rotation components of the right hand side of the fluid velocity (in the above expression) one gets an additional singularities distribution on the each blade surface. Of course, the induced velocity due to this additional singularities distribution fades away from the each blade.

- The velocity induced by the impeller, according to the Panels Method calculations, is indicated by $[ub(\xi,\eta), vb(\xi,\eta)]$; this is the velocity induced at the vortex (i), located at the point $[\xi(t), \eta(t)]$; thus

$$ub^{(i)}(x, y; t) = ub(\xi, \eta; t) - u_\lambda \tag{16a}$$

$$vb^{(i)}(x, y; t) = vb(\xi, \eta; t) + v_\lambda \tag{16b}$$

where the following relations remains

$$\begin{Bmatrix} x \\ y \end{Bmatrix}^{(i)} = \begin{Bmatrix} \cos(\varphi_t) & -\sin(\varphi_t) \\ \sin(\varphi_t) & \cos(\varphi_t) \end{Bmatrix} \begin{Bmatrix} \xi \\ \eta \end{Bmatrix}^{(i)}. \tag{17}$$

- The velocity induced due to the vortex cloud is computed using Biot-Savart´s Law in the fixed coordinate system.

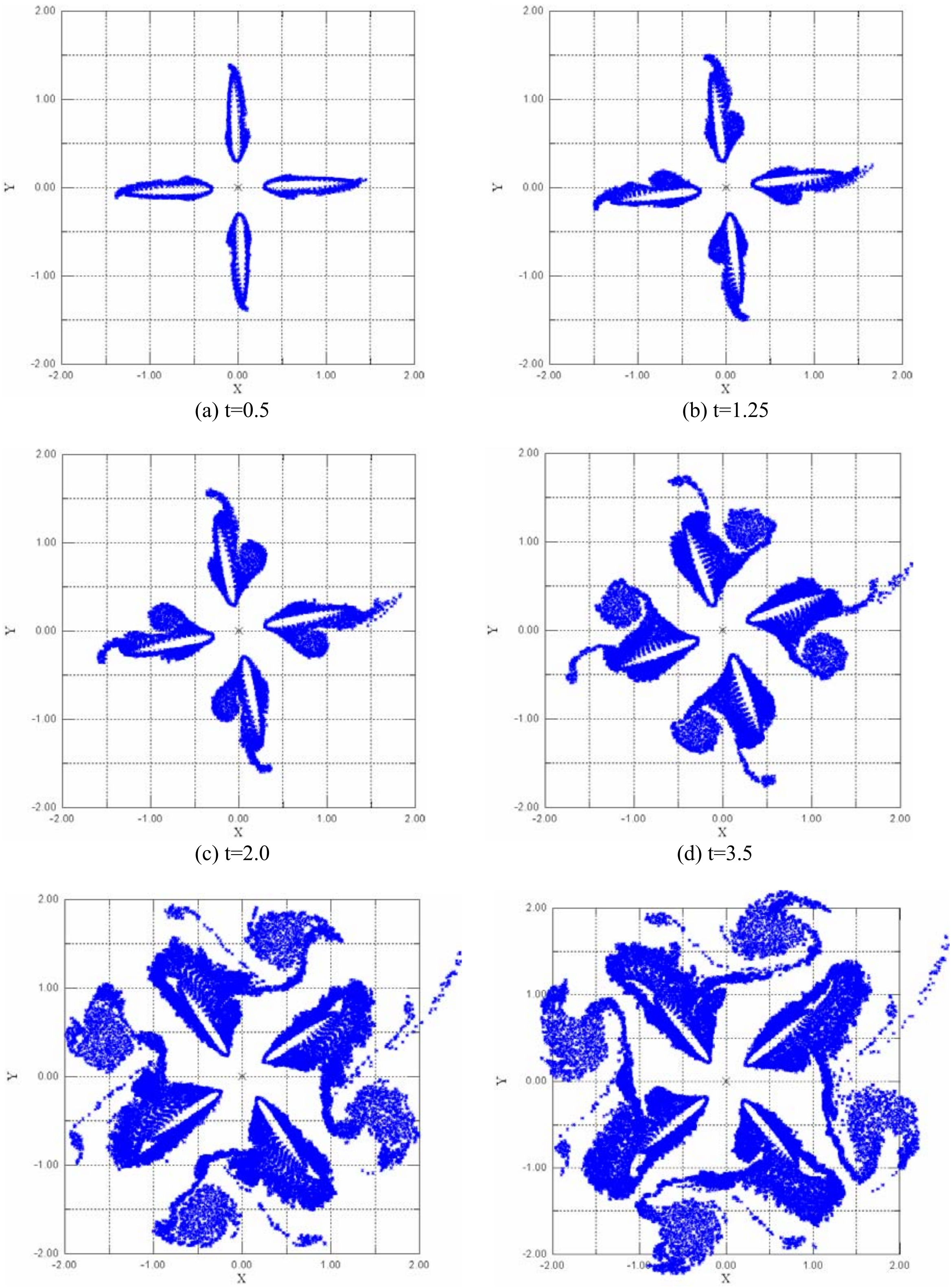
4. RESULTS AND DISCUSSION

The numerical simulations were restricted to the interference effects between four profiles whose shape consists of NACA 0012 aerofoils. Each boundary S_j, j = 1, 4, of Fig. 1 was modeled by fifty (MB=50) straight-line source panels with constant density. The inlet diameter of the impeller was defined as D_1=0.3D_2. The time increment (Δt=0.05) was evaluated according to $\Delta t=2\pi k/M$, $0<k\leq1$ (Mustto *et al.*, 1998). In each time step the nascent vortices were placed into the cloud through a displacement $\varepsilon=\sigma_0=0.0009D_1$ normal to the panels. For infinite Reynolds number viscous effects would have a little influence. For the impeller Reynolds number of 1×10^5 selected here on the other hand, viscous diffusion will differ considerably due to radial velocity diffusion.

Figure 2 shows the development of the vortex structure in a centrifugal pump impeller with four blades. The angular velocity of the impeller and the volumetric flow rate per unit breadth of the impeller were λ=0.1 and Q=2.0 respectively. In this example the impeller was rotated anti-clockwise with zero prewhirl but with sufficient angular velocity.

As can be seen from the predicted flow pattern after 40 time steps, Figure 2(c), there is clear evidence of stall with some circumferential variation from blade to blade. After a period of time as the motion proceeds, we observe the development of large stall cells in Figure 2(f) which propagate in the radial direction. The reason for this behaviour can be deduced from the vortex dynamics simulation demonstrating the power of this CFD technique for prediction and diagnosis.

In the case of a centrifugal fan it can expected that these stall cells would cause serious fluctuating interference with the stator and be a serious source of unwanted vibrations. Numerical techniques to predict unsteady flow characteristics of an impeller have been desired since before, for design and improvement of operation performance. The vortex code has been developed shows potentialities to predict flow separation in a blade-to-blade passage.



(a) t=0.5 (b) t=1.25

(c) t=2.0 (d) t=3.5

(e) t=6.0 (f) t=7.0

Figure 2 – Vortex cloud analysis of a centrifugal impeller

Figure 3 presents the predicted flow pattern for speed of rotation λ=1.0 and Q=2.0 at t=7.0. It is clarified that there are differences among the flows around individual profile, whereas the whole flow field is calculated without introducing the approximation of the periodicity of the blade-to-blade flow. Choice of the group of NB blades is then a matter of computational limitations including available memory, time of execution and numerical accuracy available for vortex code.

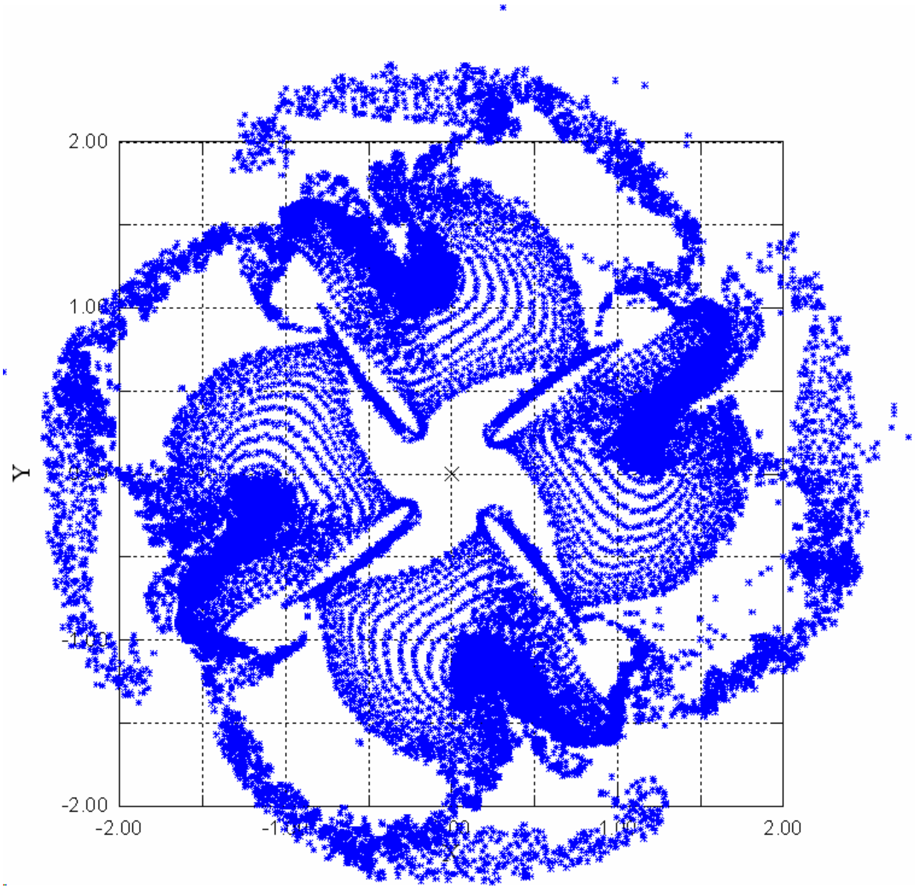


Figure 3 – Vortex flow pattern at t=7.0, Q=2.0

Because the distributed vorticity of the mainstream flow has been replaced in the numerical model by a cloud of discrete vortices, the CPU time for vortex-vortex interaction turns expensive. No attempts to simulate the flow for MB greater than 50 were made since the operation count of our algorithm is proportional to the square of N. As MB increases N also tends to increase, and the computational efforts becomes expensive. This is a major source of difficulties, and it can only be handled through the utilization of faster schemes for the induced velocity calculations, such as the multipole technique (Greengard and Rokhlin, 1987) and/or parallel computers to run long simulations (Takeda *et al.,* 1999). Finally, the results are promising and encourage performing additional tests in order to explore the phenomena in more details.

5. CONCLUSIONS

The main observation to note here is that the present method can be applied to analysis of unsteady characteristics and occurrence of unsymmetrical flow through a centrifugal impeller of pump. In this paper, two-dimensional unsteady features of the flow were computed without introducing the periodicity of a blade-to-blade flow. Using this representation, a grid-free (Lagrangian) numerical method was derived based on the coupling of the boundary element and vortex particle methods. The main objective of the work was to implement the algorithm and to get some insight into the potentialities of the model developed; this was accomplished since the results show that the behavior of the rotating stall cells propagation in a centrifugal impeller is the expected one. It is clearly demonstrated in Figure 3 that the flow becomes completely non-axis-symmetrical and some of blade-to-blade passages seem to be blocked with separation bubbles.

The use of a fast summation scheme to compute the velocities of the fluid elements, such as the multiple expansions, allows an increase in the number of vortices and a reduction of the time step, which increases the resolution of the simulation, in addition to a reduction of the CPU time, which allows a longer simulation time to be carried out. In order to solve the pressure Poison equation, a simple method based on the boundary element method will be carried out (Uhlman, 1992; Kamemoto, 1993; Shintani and Akamatsu, 1994). A new method to simulated diffusion vorticity will be carried out (Rossi, 2006). Future work will investigate the influence of shape of one-circular-arc camber and the thickness of the blades. Then, our method will be applied extensively to the complex problems of unsteady interaction between impeller-blades and guide vanes and/or volute casing.

6. ACKNOWLEDGEMENTS

The authors would like to acknowledge CNPq 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.

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

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", 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.

Leonard, A., 1980, "Vortex Methods for Flow Simulations", J. Comput. Phys., Vol. 37, pp. 289-335.

Lewis, R.I., 2004, "Study of Blade to Blade Flows and Circumferential Stall Propagation in Radial Diffusers and Radial Fans by Vortex Cloud Analysis", Journal of Computational and Applied Mechanics, Vol. 5, no.2 , pp. 323-335.

Lewis, R.I., 1999, "Vortex Element Methods, the Most Natural Approach to Flow Simulation - A Review of Methodology with Applications", Proceedings of 1st Int. Conference on Vortex Methods, Kobe, Nov. 4-5, pp. 1-15.

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.

Moura, W.H., Alcântara Pereira, L.A. and Hirata, M.H., 2006, "Analysis of the Flow around an Oscillating Circular Cylinder in Ground Effect", 11th Brazilian Congress of Thermal Sciences and Engineering, Proceedings of ENCIT 2006, December 5-8, Curitiba, Brazil.

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, June.

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.

Recicar, J.N., Alcântara Pereira, L.A. and Hirata, M.H., 2006, "Harmonic Oscillations of a Circular Cylinder Moving with Constant Velocity in a Quiescent Fluid", 11th Brazilian Congress of Thermal Sciences and Engineering, Proceedings of ENCIT 2006, December 5-8, Curitiba, Brazil.

Rossi, L.F., 2006, "A Comparative Study of Lagrangian Methods Using Axisymmetric and Deforming Blobs", SIAM J. Sci. Comput., Vol. 27, No. 4, pp. 1168-1180.

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

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.

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.

Takeda, K., Tutty, O.R. and Nicole, D.A., 1999, "Parallel Discrete Vortex Method on Commodity Supercomputers; an Investigation into Bluff Body Far Wake Behaviour", ESAIM: Proceedings, Third International Workshop on Vortex Flows and Related Numerical Methods, Vol. 7, pp. 418-428.

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

Zhu, B.S, Kamemoto, K. and Matsumoto, H., 1998, "Direct Simulation of Unsteady Flow through a Centrifugal Pump Impeller Using a Fast Vortex Method". Computational Fluid Dynamics Journal, Vol. 7, No. 1, April, pp. 15-26.

8. RESPONSIBILITY NOTICE

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