Particle based flow solvers for incompressible flows in two dimensions: impulsively started flow past a circular cylinder

Prabhu Ramachandran[1] , M.Ramakrishna[2] and S.C.Rajan[3]

Department of Aerospace Engineering, IIT-Madras, Chennai 600 036

Abstract

Vortex methods are effective in providing high-resolution flow solutions for incompressible, viscous fluid flow problems. These methods are grid free and self adaptive by construction. They are typically used for very high-resolution bluff body flow computations. In spite of their simplicity and grid free nature they are not trivial to implement. In this work a sample implementation of a vortex method that uses a random walk technique for diffusion is described. The method is validated using the flow past a circular cylinder as a benchmark. For a comparison, the high-resolution results for a similar flow, obtained by Koumoutsakos and Leonard [13] are considered. Some of the results and problems with the random walk method are discussed.

Nomenclature

C_D	= Drag coefficient
D	= Drag force
$\vec{\boldsymbol{I}}$	= Vortex momentum
$\boldsymbol{K}_{\delta}$	= Desingularised velocity kernel
L	= Length of a vortex sheet
N	= Number of particles
$\vec{\boldsymbol{r}}$	= position vector
R	= Radius of cylinder
t	= Time
U	= Free stream velocity
u	= x component of velocity
v	= y component of velocity
$\vec{\boldsymbol{v}}$	= Velocity vector
x	= x co-ordinate
y	= y co-ordinate
ω	= Vorticity
ν	= Kinematic viscosity
δ	= Core radius of vortex blob
γ	= Strength of vortex sheet
φ_{δ}	= Desingularisation function
ρ	= Density of fluid

Introduction

Vortex methods are a class of techniques that are used to study viscous, incompressible flows. The advantages of these methods are that they are grid free, self adaptive and mathematically well researched. The solution procedure is highly intuitive. These advantages result from the fact that for this class of flows, the entire phenomenon can be represented in terms of vorticity. The vorticity is the curl of the velocity field. By discretising this vorticity and tracking the resulting particles of vorticity (called vortex blobs) one can in principle solve the flow entirely. In spite of the simplicity in the description of these methods, the numerical implementation of such schemes is usually non-trivial. This is because there are a large number of important

[1] Ph. D. Research Scholar
[2] Associate Professor
[3] Assistant Professor

issues that need to be addressed before a useful solver can be created. Obtaining high resolution results using vortex methods involves very large computational constraints. However, it is also possible to obtain very good engineering results by using simpler implementations. Dutta[9] addresses several of the issues related to such implementations. The present work focuses on obtaining high resolution results for the flow past a circular cylinder at various Reynolds numbers. Currently vortex methods are usually applicable to incompressible single-phase flows. Compressible vortex dynamics and other areas are still open. The current work is restricted to incompressible, viscous fluid flows in two dimensions.

There are two steps in the numerical implementation of vortex methods, the convection and diffusion steps. During convection, the vorticity is convected due to the local velocity field (by integrating an ODE in time). In the diffusion step an equation analogous to the heat equation is applied to the vorticity field. Convection involves the computation of the velocity field due to the vorticity on itself plus the effect of any other potential field on it (like the effect due to a free stream or a solid boundary). The accuracy of a vortex method is directly proportional to the number of particles, N, that are used to discretise the vorticity field. If the velocity induced by the N particles on each other is $O(N^2)$ process. This is extremely inefficient when N is large. It is imperative, therefore, to use a scheme that reduces this operation count. There are a large number of schemes [3, 1, 11, 4, 2, 22, 8, 15] that have been developed in the recent past that reduce this $O(N^2)$ computation to either an O(N logN) or O(N) one. Each method has its own set of advantages and disadvantages. In the present work the adaptive fast multipole method due to Carrier et al [4] is used. This method belongs to the class of methods called fast multipole techniques and requires O(N) operations. The basic philosophy is to represent a cluster of particles that are sufficiently far away by a single computational unit. Once this is done, a hierarchical tree of particles is created. For each cluster the relevant far away, intermediate and nearby clusters are identified and computations are performed appropriately. The effect due to clusters that are nearby are computed directly. It can be shown [4] that any order of accuracy that is desired can be obtained using such methods. By using such techniques the effect of the vortex blobs on each other can be computed efficiently. The effect of the free stream can be found trivially. If a solid boundary is present one is required to solve the potential flow past this body in the presence of the vorticity. This can be done in several ways. For relatively simple geometries it is possible to use the method of images in either the half plane or in the plane of a cylinder and use appropriate conformal mappings to obtain the flow past various solid boundaries. Even fairly complex body geometries have been solved in the past using such techniques. References [9, 5, 16, 18] are good examples. It is also possible to use a grid based Laplace solver to satisfy the boundary condition, but since this technique is grid based and has associated problems it is usually not used. The other technique is to use a boundary element method or a panel method. In the present work, a linear vortex panel method is used to satisfy the no-penetration condition on the boundary. This enables one to solve for more general geometries.

The second step involves the diffusion of the vorticity. The diffusion equation is nothing but the heat equation applied to the vorticity. The equations for this will be discussed later. There are several techniques to diffuse the vorticity. The techniques can be classified into two broad categories, non-deterministic and deterministic schemes. The first technique developed was the random vortex method (RVM) by Chorin [6]. This technique is easy to implement for simple geometries and requires one to displace the vortex elements with a random displacement. Recently, quite a few novel techniques have been developed to solve the diffusion equation using deterministic schemes. Such schemes eliminate the noise inherent in non-deterministic ones. A few of the notable techniques are the core spreading algorithm[12], the particle strength exchange technique (PSE)[13], and the vorticity redistribution technique (VRT)[19]. These methods enable one to get unprecedented resolution using vortex methods. In the current work the random vortex method is employed. In spite of the fact that solutions using such a non-deterministic technique are inherently noisy, the method proves to be very useful for obtain good engineering solutions. Recently Lin et al [14] have developed a simplified version of the RVM. Taylor and Vezza[21], [20] use this technique to solve for flow past transversely oscillating square cylindrical sections and obtain very good results. The deterministic schemes are usually more complicated to implement.

In the present work the random vortex method is used to obtain the flow past an impulsively started cylinder for a range of Reynolds numbers (550, 1000, 3000, and 9500). This problem has been studied extensively before but recently, Koumoutsakos and Leonard[13] have produced results of extraordinary accuracy and resolution. Their results are used to benchmark the current work, and understand the problems and limitations of a random vortex method.

Mathematical Preliminaries

In this section, the basic equations that govern vortex-based techniques are discussed. The fluid considered is incompressible, viscous and two-dimensional. In such a case it can be shown that in order to solve the flow it is sufficient to solve for the vorticity field. The governing equation for the vorticity, ω is given as

$$\frac{D\omega}{Dt} = \nu \nabla^2 \omega \qquad (1)$$

where $\omega = \nabla \times \vec{v}$ is the curl of the velocity field, $\vec{v}$ is the velocity vector and ν is the viscosity of the fluid. The term

$$\frac{D}{Dt} = \frac{\partial}{\partial t} + \vec{v} \cdot \nabla$$

represents the material derivative. It is common practice to split eqn.(1) into two steps as follows

$$\frac{D\omega}{Dt} = 0 \qquad (2)$$

$$\frac{\partial \omega}{\partial t} = \nu \nabla^2 \omega \qquad (3)$$

This is called operator or viscous splitting. Eqn. (2) is the one that advects the vorticity as per the velocity field and eqn. (3) is the one that diffuses the vorticity. Each step is performed separately in the course of one time step (a fractional time step technique).

Computationally, to convect the vorticity, it is necessary to first discretise the vorticity field into units of vorticity. These are called vortex blobs. The vorticity is discretised as follows

$$\omega(\vec{r}, t) = \sum_i \varphi_\delta(r - r_i(t))\Gamma_i$$

(4)

where Γ_i is the circulation of the vortex blob located at $\vec{r}_i$ at time t. $\varphi_\delta(r)$ is called a smoothing function, which is an approximate delta function which approaches $\delta(r)$ as the parameter $\delta \to 0$. Any appropriate smoothing function, $\varphi_\delta(r)$ can be chosen and the various kinds of blobs used in the literature correspond to different smoothing functions. The parameter δ is called the core radius of the blob. Using the Biot-Savart equation one can compute the velocity of each vortex blob at an arbitrary point as follows

$$\vec{v}(\vec{r}, t) = \int K_\delta(\vec{r} - \vec{r}\,')\omega(\vec{r}\,', t)\, d\vec{r}\,' \qquad (5)$$

$$K_\delta(\vec{r}) = \left(\int_0^r r\varphi_\delta(r)\, dr\right)\frac{1}{2\pi r^2}(y, -x) \qquad (6)$$

where x, y are the x and y components of the position vector $\vec{r}$. Reference [16] provides an excellent discussion and detailed overview of vortex methods.

The diffusion equation, eqn. (3), is in fact the heat equation and has an exact solution given by

$$\omega(\vec{r}, t) = \frac{1}{4\pi\nu t}\int e^{(\vec{r} - \vec{\eta})^2/4\nu t}\omega(\vec{\eta})\, d\vec{\eta} \qquad (7)$$

This equation can be numerically solved using several techniques. The present work focuses on the random vortex method (RVM). The idea behind the random vortex method is as follows: diffusion can be approximated by giving the particles a random displacement with a zero mean and total variance 2νt. During a time step, each particle must be given an independent random displacement with mean 0 and variance 2νΔt, where Δt is the time step. The method has been numerically shown to have an $O(\sqrt{\nu / N})$ rate of convergence.

In the boundary layer region there are further simplifications that arise and in order to simulate the vorticity in such regions Chorin[7] devised a vortex sheet algorithm. In this scheme one uses the fact that near a boundary a vortex blob behaves differently due to the presence of its image vortex. There is also a notion of a numerical layer that surrounds the body. If a blob enters this region is converted to a sheet and vice versa. The height of this numerical layer is a multiple of $\sqrt{2\nu\Delta t}$. The implementation of such a scheme for the flow past a circular cylinder is straightforward but for complex shapes it is extremely hard to implement. Puckett [16] provides a detailed discussion on these schemes. If the strength of a sheet is given as γ, its length is L and if the sheet is parallel to the x-axis and starting at the origin then the velocity induced by it at a point (x, y) is given as

$$(u,v)=(\gamma,0)\quad 0<x<L;\quad y<0$$
$$(u,v)=(0,0)\quad otherwise \tag{8}$$

When a sheet is converted to a blob the core radius of the converted blob is set to be such that $\delta=L/\pi$. In the present computation, the blob due to Chorin[16] is used. It is also to be noted that the sheets are to be given a random displacement for the diffusion only in the direction of the local normal to the body surface. This is because of the nature of the boundary layer equations.

The actual computation proceeds as follows: The slip due to all the existing vorticity and free stream is computed on the surface of the body. Sheets are then introduced just at the surface of the body such that this slip is offset. Each sheet has a fixed magnitude of strength and this parameter is called γ_{max}. Therefore, given some magnitude of slip, sufficient numbers of sheets are introduced to satisfy the no slip boundary condition. These sheets are then diffused and convected as per equation (3) and (2) respectively. As the vortices diffuse and convect the particles are converted to blobs and sheets as required. This process is repeated for each time step. The computation can be continued in this fashion for as long as required.

Computational results

As discussed in the introduction, this work implements a fast multipole algorithm to solve for the velocity field due to the blobs on themselves. A panel method that uses a linear vorticity distribution is used to solve for the potential boundary condition on the body surface. The flow past a circular cylinder is considered as a benchmark problem and the results are compared with those obtained by Koumoutsakos and Leonard [13]. They have performed high-resolution studies of the flow past the circular cylinder at various Reynolds numbers and use a very large number of particles. As a result the simulations are only for small times. They also indicate that the drag coefficient is a very good measure of the accuracy of the scheme. Hence, in the present computations, the coefficient of drag, C_D, versus time is plotted and the resulting curves are compared with those obtained in [13]. The coefficient of drag C_D is computed using the vortex momentum of the fluid. The vortex moment in a fluid flow at a time t is defined as

$$\vec{I}=\sum_{i=1}^{N}\vec{r}_i\times\vec{\omega}_i\ . \tag{9}$$

It is known that the force $\vec{F}$ on the body is given as $d\vec{I}/dt$. Therefore, given a set of vortex blobs, $\vec{I}$ can be obtained from eqn.(9). The force $\vec{F}$ can be calculated by performing a central difference of the vortex momentum. C_D is computed using the equation

$$C_D=\frac{D}{\rho U^2 R} \tag{10}$$

where U is the free stream velocity, R is the radius of the cylinder and D is the drag force. Since the random vortex method introduces noise into the positions of the blobs, the curve for $\vec{I}$ versus t (time) will have oscillations about a mean curve. If such a curve is to be differentiated as it is, large errors will be obtained in the C_D curve. In order to avoid this the vortex momentum curve is smoothed using a spline and the resulting curve is differentiated to obtain the drag curve. Koumoutsakos and Leonard plot C_D versus a non-dimensional time, T = Ut/R, where t is the actual time and the other symbols are as mentioned above. The same is done for the present computations.

The various computational parameters have to be chosen carefully. The parameters involved are Δt, γ_{max}, δ, the numerical layer height and the number of panels, N. The present computations use the Chorin blob, which has compact support. Since a panel method is being used for the no penetration condition, it is imperative that the vortex blobs do not penetrate the panels. This condition can be ensured by making sure that the core radius, δ, is smaller than the numerical layer height. The numerical layer height is given by k $\sqrt{2\nu\Delta t}$, where k is a constant. Hence it is required that $\delta \le k\sqrt{2\nu\Delta t}$. In the present computations k is chosen between 1 and 5. As mentioned in an earlier section, the core radius is related to the length of a vortex sheet via the equation $\delta=L/\pi$, where L is the length of the sheet. Given N equal sized panels for the circle, if each panel is to release sheets, it is clear that $L=2\pi R/N$. Hence, $\delta=2R/N$. Thus, given a Reynolds number and a cylinder radius R, except for k and γ_{max}, only one parameter, Δt, needs to be chosen. Puckett[16], also provides a simple stability criterion for Δt, such that the sheet displacement should not be more than its length, L. In the case of the flow past a circular cylinder, $2U\Delta t < L$. Hence, the present computations use a Δt such that the above criteria are satisfied. For the Re=9500 case, Δt is chosen to be 0.0025 seconds, R=1m and 400 panels are used for the body. The value of γ_{max} is chosen to be 0.05m/s for all simulations.

Figure 1 plots C_D versus T for various Reynolds numbers. The circular symbols are the results due to Koumoutsakos and Leonard and the lines are the present results. The random vortex method that is presently used for the computations is not accurate for highly viscous flows. This is clearly seen from the plots. As the Reynolds number increases the computed curves are closer to the results of Koumoutsakos and Leonard. For the case of Re=550 the trends are clearly very similar but the curve is slightly shifted. For the case of Re=1000 the curve agrees even more with that given by Koumoutsakos and Leonard and for the case Re=3000 the agreement is even better. For the case when Re=9500 the results match very well for small times but subsequently the results are very different. The prime reason for this is the fact that the random vortex method introduces noise into the simulation.

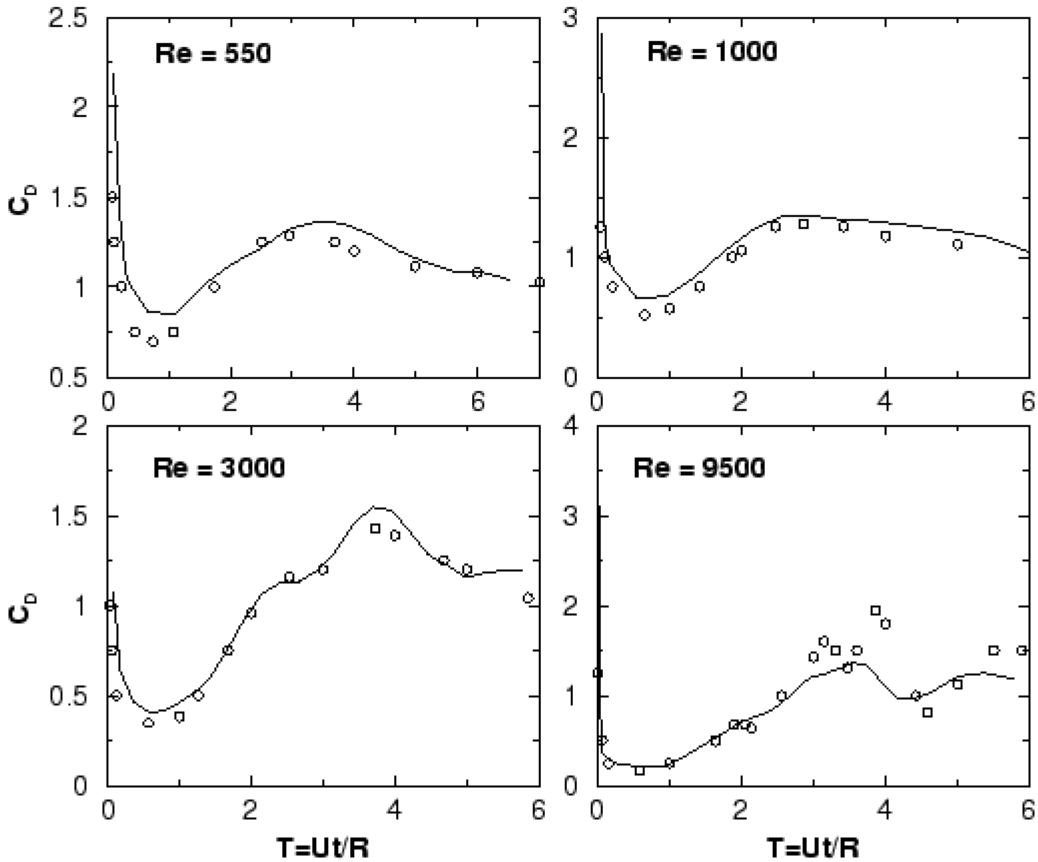
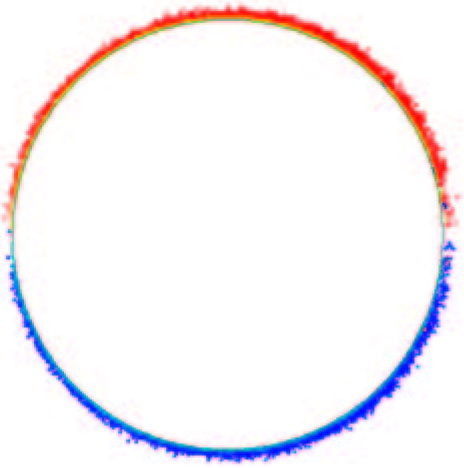
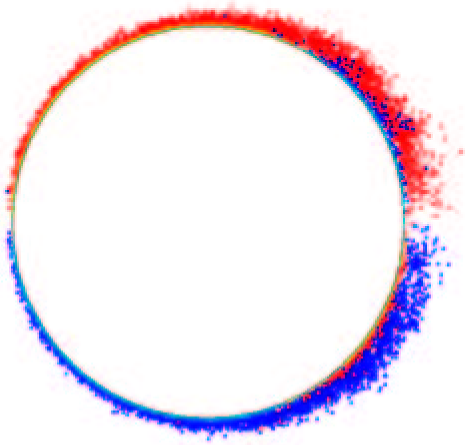


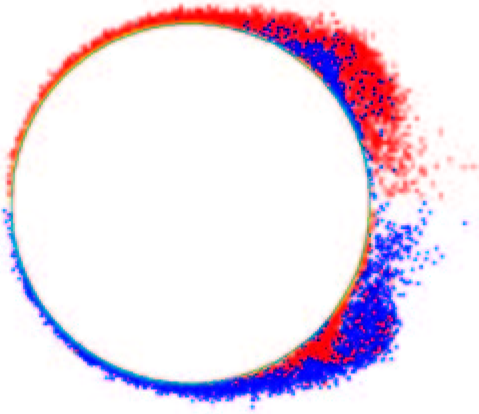
***Fig. 1** C_D versus T curves for various Reynolds numbers. The circles are the values obtained by Koumoutsakos and Leonard and the lines represent the present computations.*



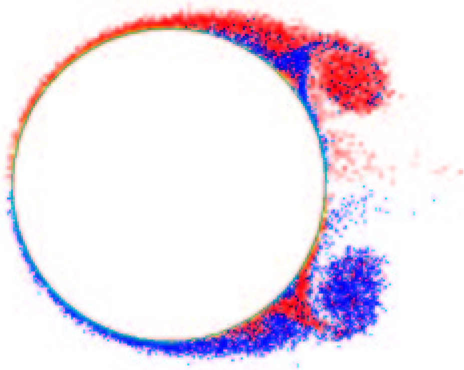
[a] 0.3 seconds



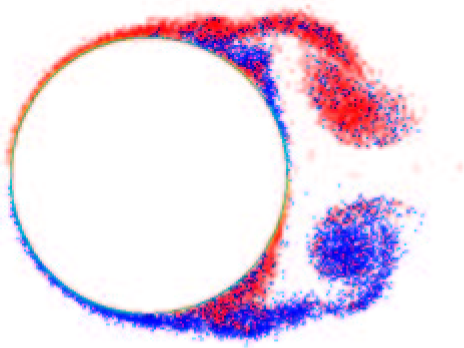
[b] 1.2 seconds



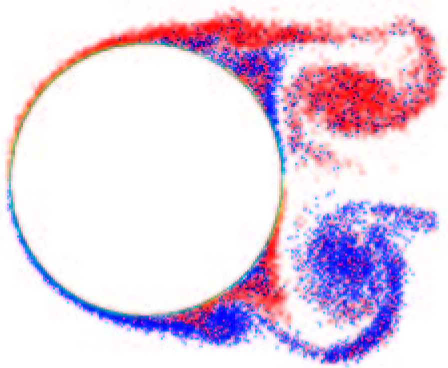
[c] 2.1 seconds



[d] 2.7 seconds

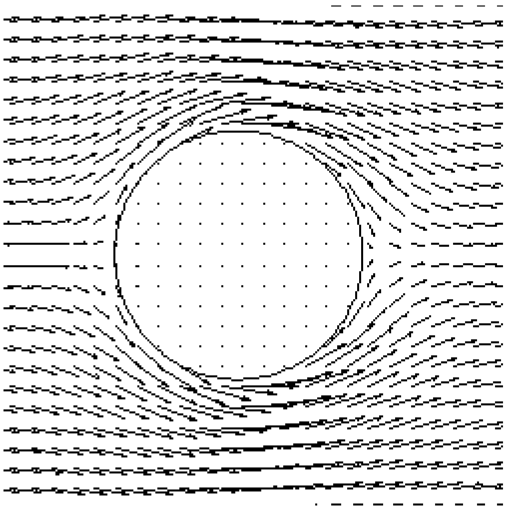


[e] 3.9 seconds

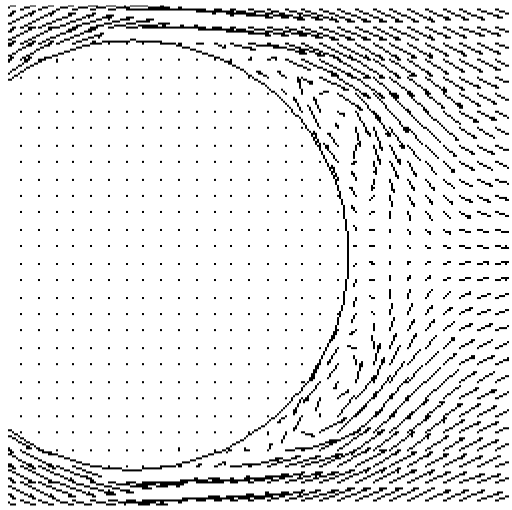


[f] 4.8 seconds

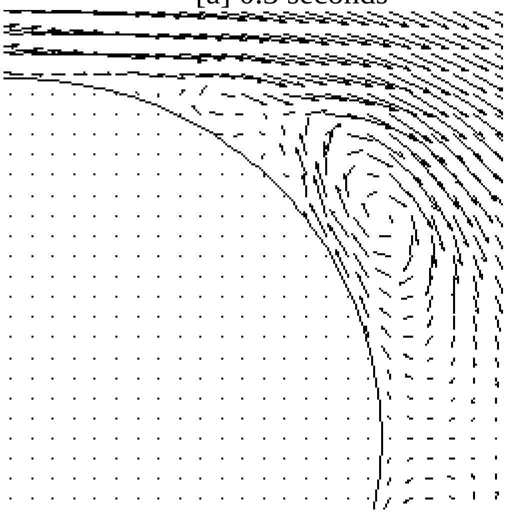
Fig.2 Vorticity plot for various times for the Reynolds number 3000. Red dots represent vortex blobs that have a clockwise circulation and blue dots represent anti-clockwise circulation. Sheets having clockwise circulation are coloured yellow and sheets with anti-clockwise circulation are coloured cyan.



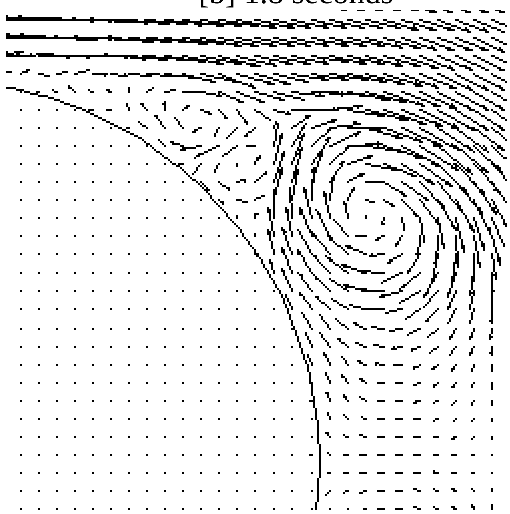
[a] 0.3 seconds



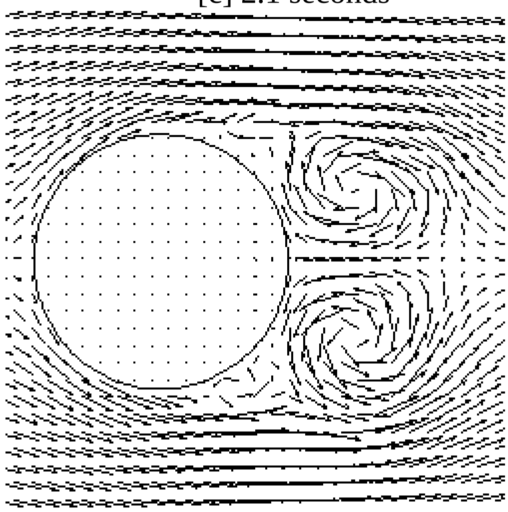
[b] 1.8 seconds



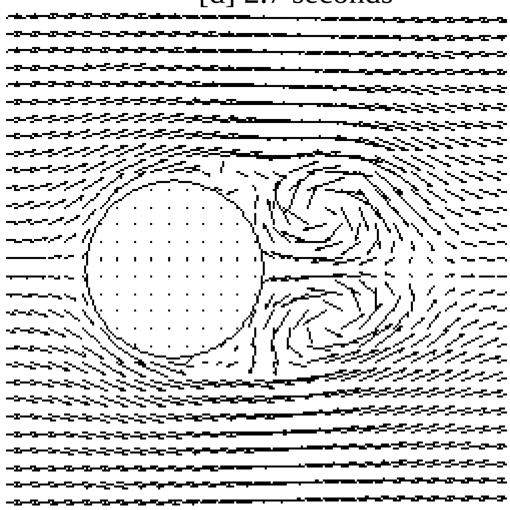
[c] 2.1 seconds



[d] 2.7 seconds



[d] 4.8 seconds



[f] 6.0 seconds

Fig.3 Velocity vector plots for the flow past an impulsively started cylinder. Reynolds number is 3000.

The technique used by Koumoutsakos and Leonard is a deterministic one (particle strength exchange - PSE) and hence does not introduce random noise into the flow. Due to the high Reynolds number, the flow is unstable and the noise in the RVM causes the vortex distribution in the wake of the cylinder to become asymmetric. The PSE technique, used by Koumoutsakos and Leonard, does not cause this asymmetry and hence the present results differ from theirs.

The problem due to noise in the RVM is a common one and it is for this reason that deterministic diffusion schemes are in vogue today. In spite of this difference in the results it is known that in realistic situations, there are small perturbations, therefore the random vortex method is still useful from an engineering standpoint. From the above results it is clear that for small times the agreement with extremely accurate results is good. Even for fairly low Reynolds numbers the results are acceptable and capture the trends reasonably accurately.

Figures 2(a) to 2(f) plot the vortex blobs and sheets at different times. The Reynolds number is 3000. Each red dot represents a single vortex blob that has a clockwise circulation. Blue dots represent blobs with anti-clockwise circulation. Sheets having clockwise circulation are coloured yellow and sheets with anti-clockwise circulation are coloured cyan. It is clear that the method is able to capture interesting features. At t=0.3 seconds the boundary layer has formed and t=1.2 seconds the rear of the flow has started to separate. At t=2.1 seconds the formation of the secondary vortex is seen clearly. At 2.7 seconds a small tertiary vortex is also seen clearly. A small asymmetry in the flow is also seen. This is due to the random nature of the diffusion process. At 3.9 seconds the primary vortex, has moved significantly towards the aft of the cylinder. After 4.8 seconds the primary vortex is still close to the rear of the cylinder. The asymmetry in the solution is clearly seen at this time.

Figures 3(a) to 3(f) plot velocity vectors for various times for a Reynolds number of 3000. The plots show the development of the flow in the wake of the cylinder for small times. The fine scale structure of the flow is captured. Secondary and even tertiary vortex structures are seen.

Conclusions

In this paper the random vortex method technique coupled with a fast multipole method is introduced. Such a methodology can be used to simulate two-dimensional, incompressible, viscous flows. Some of the numerical issues are discussed and results are shown for the flow past a circular cylinder for a range of Reynolds numbers. These results are compared with those obtained by Koumoutsakos and Leonard [13]. For the case when Re=1000 and Re=3000 the agreement with their results is good but due to the noise introduced by the random vortex method and the consequent asymmetry in the vorticity distribution the results for the case when Re=9500 do not agree for longer times. This failure of the scheme is well known and it is for this reason that deterministic diffusion schemes have been developed. However, it is known that for engineering solutions in realistic situations the random vortex method technique is ideal. Since such a solver has been developed and successfully validated for high resolutions using the flow past a cylinder as a benchmark problem it is now possible to study more realistic flows. Further validations using criteria different from drag curves are also essential and these will be pursued in the future.

References

[1] Christopher R. Anderson. A method of local corrections for computing the velocity field due to a distribution of vortex blobs. ***Journal of Computational Physics***, 62:11-123, 1986.

[2] Christopher R. Anderson. An implementation of the fast multipole method without multipoles. ***SIAM J. Sci. Stat. Comput.***, 13(4):923-947, 1992.

[3] J. Barnes and P. Hut. A hierarchical o(nlogn) force-calculation algorithm. ***Nature***, 324:446-449, 1986.

[4] J. Carrier, L. Greengard, and V. Rokhlin. A fast adaptive multipole algorithm for particle simulations. ***SIAM J. Sci. Stat. Comput.***, 9(4):669-686, 1988.

[5] A. Y. Cheer. Unsteady separated wake behind an impulsively started cylinder in slightly viscous fluid. ***Journal of Fluid Mechanics***, 201:485-505, 1989.

[6] A. J. Chorin. Numerical study of slightly viscous flow. ***Journal of Fluid Mechanics***, 57:785-796, 1973.

[7] A. J. Chorin. Vortex sheet approximation of boundary layers. ***Journal of Computational Physics***, 27:428-442, 1978.

[8] C. I. Draghicescu and M. Draghicescu. A fast algorithm for vortex blob interactions. ***Journal of Computational Physics***, 116:69-78, 1995.

[9] P. K. Dutta. ***Discrete vortex simulation of high Reynolds number flows.*** Ph.D. Thesis, Indian Institute of Science, Bangalore, April 1988.

[10] A. F. Ghoniem and Y. Cagnon. Vortex simulation of laminar recirculating flow. ***Journal of Computational Physics***, 68:346-377, 1987.

[11] L. Greengard and V. Rokhlin. A fast algorithm for particle simulations. ***Journal of Computational Physics***, 73:325-348, 1987.

[12] Teruhiko Kida. Theoretical and numerical results of a deterministic two-dimensional vortex model. ***Sadhana***, 23(5,6):419-441, Oct and Dec 1998.

[13] P. Koumoutsakos and A. Leonard. High-resolution simulations of the flow around an impulsively started cylinder using vortex methods. ***Journal of Fluid Mechanics***, 296:1-38, 1995.

[14] H. Lin, M. Vezza, and R. A. McD. Galbraith. Discrete vortex method for simulating unsteady flow around pitching aerofoils. ***AIAA Journal***, 35(3):494-499, March 1997.

[15] Junichiro Makino. Yet another fast multipole method without multipoles - pseudo-particle multipole method. ***Journal of Computational Physics***, to appear.

[16] E. G. Puckett. Vortex methods: An introduction and survey of selected research topics. In R. A. Nicolaides and M. D. Gunzburger, editors, ***Incompressible Computational Fluid Dynamics - Trends and Advances***. Cambridge University Press, 1991.

[17] J. A. Sethian and A. F. Ghoniem. Validation study of vortex methods. ***Journal of Computational Physics***, 74:283-317, 1988.

[18] S. Shashidhar. ***Vortex methods for simulation of two-dimensional flows.*** M.S. Thesis, Department of Aerospace Engineering, Indian Institute of Technology, Madras, 1998.

[19] Shankar Subramaniam. ***A new mesh-free vortex method.*** PhD thesis, The Florida State University, FAMU-FSU College of Engineering, 1996.

[20] I. Taylor and M. Vezza. Calculation of the flow around a square section cylinder undergoing forced transverse oscillations using a discrete vortex method. ***Journal of Wind Engineering and Industrial Aerodynamics***, 82:271-291, 1999.

[21] I. Taylor and M. Vezza. Prediction of unsteady flow around square and rectangular cylinders using a discrete vortex method. ***Journal of Wind Engineering and Industrial Aerodynamics***, 82:247-269, 1999.

[22] Leon van Dommelen and Elke A. Rundensteiner. Fast, adaptive summation of point forces in two-dimensional poisson equation. ***Journal of Computational Physics***, 83:126-147, 1989.