FLOW PAST ROTATING CYLINDERS AT HIGH REYNOLDS NUMBERS USING HIGHER ORDER UPWIND SCHEME

MANOJ T. NAIR, TAPAN K. SENGUPTA† and UMENDRA S. CHAUHAN

Department of Aerospace Engineering, Indian Institute of Technology, Kanpur 208 016, India

(Received 5 December 1996; in revised form 24 December 1997)

Abstract—Flow past a translating and spinning circular cylinder at different Reynolds numbers and different rotation rates is studied by solving the incompressible two-dimensional Navier–Stokes equation using a third-order upwind scheme. A detailed error analysis is made before choosing a grid that minimises numerical dispersion and dissipation which are known to affect the higher order schemes. The resultant grid-independent method is not restricted by numerical instability at high Reynolds numbers. The results are compared with the available experiments to establish the effectiveness of the scheme in solving high Reynolds number problem at different rotation rates. This finite-difference method is found to be accurate and fast as compared with other published results. A specific case for \( Re = 3800 \) and \( \Omega = 2.0 \) is investigated next. The pressure solution for this case is obtained by solving the pressure Poisson equation.

1. INTRODUCTION

The possibility of generating a transverse force by imparting rotation and hence circulation around a body has been of interest for more than a century. In the literature, this is referred to as the Magnus effect after the discoverer of this effect (Batchelor [1] and Prandtl and Tietjens [2]). Unfortunately the increase of lift is associated with a high drag penalty thus producing little aerodynamic efficiency and hence has not been favoured in practical devices. However, it has also been shown by Prandtl that the effect of rotation has the beneficial effect of inhibiting boundary layer separation and of suppressing it completely at sufficiently high rotational rates. The dual benefit of inhibiting separation and at the same time producing high lift has prompted many researchers (Tennant et al. [3] and Modi et al. [4]) to suggest the use of a rotating cylinder as boundary layer control device on the flaps of V/STOL aircrafts, upstream of the ship rudder and in sub-sonic diffusers. Consequently understanding the nature of such flows and in particular to ascertain the level of rotation to delay separation is of utmost importance.

The two-dimensional flow field generated by an infinitely long circular cylinder of diameter \( D \) translating with an uniform velocity \( U_{\infty} \) at right angles to its axis and rotating about this axis with constant angular velocity \( \Omega^* \) has been studied. The cylinder attains its rotational and translational speeds impulsively. The two important parameters of this study are the Reynolds number given by \( Re = U_{\infty}D/v \), with \( v \) as the coefficient of kinematic viscosity and the ratio of peripheral to translational speed \( \Omega = \Omega^*D/2U_{\infty} \).

In the early attempts, the Magnus effect had been explained by inviscid theory (Batchelor [1]) and the resultant lift was found to be proportional to the rotation rate. This is not the case for real fluids with boundary layer separation. It is to be pointed out that for moderate-to-high Reynolds number flows the surface roughness and other three-dimensional effects play a significant part. However, it is beyond the scope of the present day computers and attention is focussed on two-dimensional flows. In the present paper, results are obtained numerically and compared with the experimental results of Coutanceau and Menard [5] who have stated that

†Corresponding author.
their experimental results exhibited two-dimensional flow field. Diaz et al. [6], in presenting their measured velocity field at Re = 9000, have stated that the measured flow quantities did not exhibit three-dimensionality. However, we have not computed this flow field as at this higher Reynolds number the flow is turbulent in the near wake.

For the two-dimensional flow field, Glauert [7] considered the development of steady flow for high Reynolds number for both large and small values of rotation rates on the basis of boundary layer theory. Similar studies were undertaken by Wood [8] and Moore [9]; all of which showed that lift increases linearly with rotation rate. Some of the early computations of the Navier–Stokes equations were restricted to low Reynolds numbers, where the flow field remains steady. This restriction on Reynolds numbers is due to poor convergence rate and other numerical stability problems. These results are due to Townsend [10], Ingham and Tang [11] and Tang and Ingham [12].

The unsteady flow development at low rotation rate is of interest because of its importance in understanding flow separation. The first meaningful computation of such a flow field is due to Badr and Dennis [13] from the numerical solution of the two-dimensional Navier–Stokes equation. The results reported were for Re = 200 and 500 and rotation rates of 0.5 and 1.0. In Badr et al. [14], the developed numerical method was extended to Re = 10 000 and rotation rate to 3.25 and compared with experimental results due to Coutanceau and Menard [5]. The method is based on a hybrid spectral–finite difference formulation of Navier–Stokes equation in boundary layer coordinate system that changes with time. The starting solution was obtained by using a formal perturbation technique with time as the small parameter. However, the equation giving the initial wall vorticity is reported incorrectly—the correct expression appears in Chew et al. [15]. Kimura et al. [16] have studied the problem of rotating and translating cylinder by discrete vortex method. The reported results have a limited applicability as there were only 14 surface segments on the cylinder. Chen et al. [17] have produced results using a finite-difference–pseudo-spectral method for solving the Navier–Stokes equation. Although the results are of good quality, results have only been obtained for Re = 200 for various rotation rates. The emphasis of the paper was to show that vortices continue to shed when viewed from a reference frame moving with a vortex.

One of the earliest experimental results for the nominally two-dimensional flow past a translating and rotating circular cylinder can be seen in Prandtl and Tietjens [2]. The other flow visualization results are due to Taneda [18,19] and Koromilas and Telionis [20]. In Coutanceau and Menard [5], flow visualization results are presented for different Reynolds numbers and rotation rates at different time instants that could be used for qualitative comparison with computations. The Reynolds number at which experiments were performed were 200, 500 and 1000, while the rotation rate varied from 0.28 to 3.25 with most of the results for Re = 200. Diaz et al. [6] conducted an experimental investigation at Re = 9000, where near-wake velocities were measured by a hot wire probe. The reported results for rotating cylinder showed the measured Reynolds stress near the midplane of the wake to be much lower when the rotation rate increases as compared with the Reynolds stress for a translating cylinder.

The numerical method used for the present exercise has earlier been used in Sengupta and Sengupta [21] for flow past a translating cylinder at high Reynolds numbers. When the same method was used for the rotating cylinder case, it exhibited some convergence and grid dependence problems. In the present exercise, we have used a much stricter convergence criteria for the solution of the boundary value problem. In general, higher-order schemes suffer from a grid dependence of the obtained solution. In this paper, we have analysed this and developed a new strategy for avoiding grid dependence. The results are presented for the cases shown in Fig. 1. Some of the results are compared with other computational and experimental results reported in Coutanceau and Menard [5] and Badr et al. [14]. Additional results are presented for Re = 3800 and Ω = 2.0.

In Section 2, the governing equations and associated boundary conditions are discussed. In Section 3, the numerical method is presented, and finally in Section 4, the results and discussion are given.
2. GOVERNING EQUATIONS AND BOUNDARY CONDITIONS

The unsteady Navier–Stokes equation for incompressible two-dimensional flows are solved here in stream function and vorticity formulation and are given by:

\[ \nabla^2 \psi = -\omega \]  

and

\[ \frac{\partial \omega}{\partial t} + \nabla \cdot (\omega \mathbf{V}) = \frac{1}{Re} \nabla^2 \omega \]  

where \( \omega \) is the vorticity defined by

\[ \omega = (\nabla \times \mathbf{V}) \cdot k \]  

and the velocity is related to the stream function by

\[ \mathbf{V} = \nabla \times \Psi \]  

where \( \Psi = (0, 0, \psi) \).
The computations are performed in the translational frame in an O-grid topology. The transformed coordinates are related to the physical coordinates by:

\[ x = r(\eta) \cos(2\pi \xi) \]
\[ y = r(\eta) \sin(2\pi \xi) \]  

(5)

where \( \xi \) and \( \eta \) are the normalized transformed coordinates along the circumference and radius, respectively.

The grids are uniformly spaced in the \( \xi \)-direction, while the radial spacing is chosen such that the grid related dispersion and dissipation errors are minimum and is given by:

\[ r(\eta) = r_0 + \frac{\eta}{\Delta \eta} \left[ \Delta r_0 + \frac{d}{2} \left( \frac{\eta}{\Delta \eta} - 1 \right) \right] \]  

(6)

where \( r_0 \) indicates the radius of the cylinder, \( \Delta r_0 \) is the spacing of the first radial grid line and \( d \) is the increment of the successive grid line spacing. This particular choice of grid removes alternation of basic convection and lowest order dissipation. The rational is discussed in Section 3 following Equation (17). The other details of the formulations are the same as in Sengupta and Sengupta [21]. The steady-state inviscid solution is taken to be the initial condition for the impulsively started cylinder problem.

The governing Poisson equation for pressure is obtained by taking the divergence of the Navier–Stokes equation in primitive variables. For an orthogonal curvilinear coordinate system this is given as:

\[ \frac{\partial}{\partial \xi} \left( \frac{h_2}{h_1} \frac{\partial p}{\partial \xi} \right) + \frac{\partial}{\partial \eta} \left( \frac{h_1}{h_2} \frac{\partial p}{\partial \eta} \right) = \frac{\partial}{\partial \xi} \left( h_2 \Omega \right) - \frac{\partial}{\partial \eta} \left( h_1 \Omega \right) \]  

(7)

For the boundary conditions on the cylinder wall, the no-slip condition is used and is given by:

\[ \frac{\partial \psi}{\partial \eta} \bigg|_{\text{body}} = h_2 r_0 \Omega \]  

(8)

and

\[ \psi = \text{constant} \]  

(9)

where \( h_1 \) and \( h_2 \) are the scale factor (defined in Section 3).

While Equations (8) and (9) are used in calculating the wall vorticity, Equation (9) is used only in solving the stream function Equation (1). The additional periodic boundary conditions is applied at the cut (see definition in Fig. 1), while at the outer boundary uniform flow conditions are applied. The corresponding far field boundary conditions are either:

\[ \frac{\partial \omega}{\partial \eta} = 0 \]  

(10)

or

\[ \frac{\partial^2 \omega}{\partial \eta^2} = 0. \]  

(11)

Most of the results presented in this paper are from using the boundary condition given in Equation (10).

For the pressure Poisson equation [Equation (7)] the Neumann boundary condition, as obtained from the normal (\( \eta \)) momentum equation, is used on the surface and outer boundary, whereas periodic boundary condition is applied at the cut (Fig. 1).

3. NUMERICAL METHOD

The stream function equation Equation (1), is retained in the following self-adjoint form while solving it numerically in the transformed plane,
\[ \frac{\partial}{\partial \zeta} \left( \frac{h_2}{h_1} \frac{\partial \psi}{\partial \zeta} \right) + \frac{\partial}{\partial \eta} \left( \frac{h_1}{h_2} \frac{\partial \psi}{\partial \eta} \right) = -h_1 h_2 \omega \]  

(12)

where \( h_1 \) and \( h_2 \) are the scale factors of the transformation given by

\[ h_1 = \sqrt{x_0^2 + y_0^2} \]
\[ h_2 = \sqrt{x_0^2 + y_0^2} \]

The vorticity transport equation when written in the transformed plane is given by:

\[ h_1 h_2 \frac{\partial \omega}{\partial t} + h_2 u \frac{\partial \omega}{\partial \zeta} + h_1 v \frac{\partial \omega}{\partial \eta} = \frac{1}{Re} \left\{ \frac{\partial}{\partial \zeta} \left( \frac{h_2}{h_1} \frac{\partial \omega}{\partial \zeta} \right) + \frac{\partial}{\partial \eta} \left( \frac{h_1}{h_2} \frac{\partial \omega}{\partial \eta} \right) \right\} \]

(13)

where the components of the velocity vector are given by

\[ h_2 u = \frac{\partial \psi}{\partial \eta} \]
\[ h_1 v = -\frac{\partial \psi}{\partial \zeta} \]

(14)

These equations and their solution method have been described in details in Sengupta and Sengupta [21] and are quoted here in brief. The finite-difference form of various derivatives in Equations (12) and (13) are obtained by employing central differencing, excepting the convection terms in Equation (13), which are discretized using a third-order upwind scheme [22] given by:

\[ \left( \frac{\partial \omega}{\partial \eta} \right)_i = u_i \frac{u_{i+2} + 8(u_{i+1} - u_{i-1}) + u_{i-2}}{12 \eta} + |u_i| \frac{u_{i+2} - 4u_{i+1} + 6u_i - 4u_{i-1} + u_{i-2}}{4 \eta} \]

(15)

The reader is referred to Appendix A of Sengupta and Sengupta [21] for a detailed analysis of the various finite-difference schemes. The upwinding produces a truncation error proportional to the fourth derivative of the vorticity with respect to the transformed variable. These error terms when projected back in the physical plane give rise to numerical dispersion, added dissipation and altered convection, which produce grid-dependent results. In the present work, the particular choice of the grids as given by Equations (5) and (6) minimizes dispersion and dissipation but does not alter the convection terms at all. Discretizing the convection term \( h_1 v (\partial \omega / \partial \eta) \) by the third-order upwinding results in a truncation error:

\[ \frac{\Delta \eta^3}{4} (h_1 v) \frac{\partial^4 \omega}{\partial \eta^4} \]

(16)

This scheme was originally proposed by Kawamura et al. [22] where the authors stated that the truncation error is a higher-order dissipation term which stabilizes computations at high Reynolds numbers and does not affect the physical dissipation. This is true only when one works in the physical plane. Working on a general transformed plane, Equation (16) can be rewritten in the physical cartesian plane as:

\[ \frac{h_1 v \Delta \eta^3}{4} \left\{ A + B + C + D + E + F + G \right\} \]

(17)

where the quantities in the parenthesis are given by

\[ A = \frac{\partial^4 x}{\partial \eta^4} \frac{\partial \omega}{\partial x} + \frac{\partial^4 y}{\partial \eta^4} \frac{\partial \omega}{\partial y} \]
A similar set of terms will be obtained for the truncation error terms associated with the other convection term. It is easy to see that because of the large flow gradients in the normal direction close to the body, only the truncation error terms as given in Equation (16) will be dominant. In Equation (17), the term $A$ represents alteration in physical convection. The terms $B$, $D$ and $F$ represent pure dissipation, dispersion and higher-order dissipation, respectively. While the terms $C$ and $G$ give rise to mixed dissipation, the term $E$ represents mixed dispersion. These truncation error terms alter the quality of the solution if proper care is not exercised. It is to be noted that if one chooses the constant radial lines to be stretched in a geometric progression, as is usually performed in boundary layer analyses, then all the components of the truncation error will be present. This type of stretching was used in Sengupta and Sengupta [21]. The grid transformation given by Equations (5) and (6) does not alter the pure convection via the term $A$. In a similar way the second-order dissipation (both pure and mixed) will be greatly reduced which are due to $\frac{\partial^3 x}{\partial \eta^3}$ and $\frac{\partial ^3 y}{\partial \eta^3}$ in $B$ and $C$. In addition, it is easy to see that the coefficients of the dispersion terms $(D \text{ and } E)$ are three to four orders of magnitude smaller.

Flows past a rotating cylinder is a perfect case for studying the receptivity of bluff body flows as compared with the non-rotating cylinder case. Computationally, the translating cylinder either does not exhibit vortex shedding (Braza et al. [23]) or it shows asymmetry much later as compared with experiments (Nair and Sengupta [24]). One of the reasons cited for such a behaviour is the inability to prescribe the disturbance environment completely. For the rotating cylinder in uniform flow the rotational effects would dominate the background inherent disturbance field. Furthermore the results at $Re = 200$ and 1000 seem to be insensitive to initial conditions. As pointed out earlier, Badr and Dennis [13] have not used the correct expression for the initial wall vorticity, but showed good agreement with the experimental results. In the present effort the initial wall vorticity is calculated by discretizing the stream function equation while satisfying the no-slip condition.

We would also like to comment on the convergence criteria that is often used in iterative solvers for elliptic equations. It has been noted by Roach [25] and Ferziger [26] that comparing successive iterates to check for convergence lead to erroneous results. Instead one can use the solution residue for checking the convergence. In this work, the solution residue at every grid point is made to fall below a preassigned value (0.00001). This method of checking convergence at every point is a stronger method than what is often used in other methods (see, for example,
Mittal and Tezduyar [27]) where an average over the whole flow domain is computed and compared for convergence at each level of iteration.

For the solution of the Poisson equations [Equations (12) and (7)], the alternating direction implicit (ADI) method of Peaceman and Rachford (as given in Ames [28]) and the modified strongly implicit procedure (MSIP) of Schneider and Zedan [29] are used. In general, the equations are discretized at the cell nodes while the pressure equation is discretized at the cell centres for the boundary points. The time discretization for Equation (13) is by two point Euler backward scheme.

4. RESULTS AND DISCUSSION

The computations are performed on a O-type grid with 251 points in the azimuthal and 300 points in the radial direction for \( Re = 200 \) and 1000 cases. A finer grid with 400 points in the radial direction has been used for the case of \( Re = 3800 \). Time integration is performed by Euler backward two point discretization with a time step of 0.0001. The first azimuthal line is 0.001 distance away from the cylinder. The outer boundary is located 12 diameters from the cylinder (as used in Chen et al. [17]). The present grids are finer compared with the \((128 \times 120)\) grid used by Chen et al. [17]. Badr and Dennis [13] and Badr et al. [14] took a value of \( \Delta z \) of 0.05 in initial stages and 0.1 beyond \( t = 1.5 \), where \( z \) is the radial distance non-dimensionalized by viscous length scale (usually defined for unsteady boundary viscous flows). Because they used a fixed maximum value of \( z \) (a quantity which varies as \( t^{-0.5} \)), the grid resolution in the physical space becomes inferior as time progresses. Furthermore, the use of central differencing with exponentially stretched grid will have larger spurious dispersion, as discussed in Section 3. The higher-order spatial accuracy of the present scheme allows resolving high wavenumber components of the solution. A constant time step of 0.0001 is used here as compared with the variable time steps taken by Chen et al. [17] (between 0.0001 and 0.01) and Badr and Dennis [13] (between 0.05 and 0.1).

The results reported here are for the cases shown in Fig. 1. It was conjectured by Chen et al. [17] that the flow will be predominantly unsteady and two-dimensional for high Reynolds numbers (greater than 50) and small rotation rates. For the \( Re = 200 \) case, the flow will be unsteady and the corresponding results are shown in Figs 2 and 3. The qualitative agreement of the computed results with the experimental results of Coutanceau and Menard [5] and the computed results of Badr et al. [14] and Chen et al. [17] is excellent.

For the reported computations for \( Re = 1000 \), the Poisson equation for the stream function required three ADI cycles involving three acceleration parameters. The computing time required for each time step is roughly 5 s on an HP 735/100 workstation for the case of \( Re = 1000 \) and \( \Omega = 3.0 \). This figure is one-third for the lower Reynolds number and lower rotation rate cases. This time compares very favourably with the reported computation time of 5 CPU seconds per time step on a CRAY 2 in Chen et al. [17] for the case of \( Re = 200 \) and \( \Omega = 0.5 \) and 1.0, although the present computations were performed on a \((251 \times 300)\) grid as compared with \((128 \times 120)\) grid used by Chen et al. The main aim of the present paper is to demonstrate the ability of the present numerical method to produce very accurate results with very little computational efforts. As discussed earlier the accuracy of the solution is due to:

1. higher-order spatial accuracy of the scheme;
2. choice of the grid along the radial direction [Equation (6)];
3. very fine mesh that has been used;
4. very fine time step.

The initial flow field for the present computation for the impulsively started rotation and translation is given by the potential flow. Detailed perturbation results have been produced by Badr et al. [14] and Chew et al. [15] for initial solution of Navier–Stokes equation which they have used for starting their computations. This is performed following the procedure originally given in Collins and Dennis [30]. Few comments are in order here on the initial condition. Although most of the computations reported for the rotating cylinder cases use impulsively started condition, the corresponding laboratory experiments reveal some differences. For the experimental set-up described by Coutanceau and Menard [5], it is easy to see that the character-
Fig. 2(a–d)—Caption on page 10.
Flow past rotating cylinders

Fig. 2(e–h)—Caption overleaf.
Fig. 2. Comparison of experimental [5] and computational streamlines for flow past a translating and rotating circular cylinder at $Re = 200$ and $\Omega = 0.5$. 
Fig. 3(a–d)—Caption overleaf.
istic time scale evaluated from the free stream speed and the diameter of the cylinder ranges between 0.8 and 12 s for the chosen translational velocities (0.5–5 cm/s) and diameter of the cylinders (4 or 6 cm). Comparing this time with the tunnel start up time of about 0.1 s, the assumption of impulsive start seems quite adequate for the present computations. The plotted streamlines are in the same frame of reference as in the experiment, i.e. the visualization is in a translating frame of reference.

Figure 2(a–j) shows the comparison of experimental results of Coutanceau and Menard [5] with our computation for $Re = 200$ and $\Omega = 0.5$ for different times. The time scale adopted in the present computations are same as that in Coutanceau and Menard [5], while the results reported in Badr et al. [14] requires a division by two of the corresponding times therein.

Fig. 3. Comparison of experimental [5] and computational streamlines for flow past a translating and rotating circular cylinder at $Re = 200$ and $\Omega = 1.0$. 
There is an excellent account of the flow topology and specifically the separation pattern and vortex shedding from a spinning cylinder in Coutanceau and Menard [5]. It is pointed out in Coutanceau and Menard [5] that the saddle points for the cases of a rotating cylinder will be orthogonal which can also be ascertained from the figures. Furthermore, it is stated that the loss of symmetry due to rotation will lead to the disappearance of eddies forming on the lower side of the cylinder for an anticlockwise rotation. The presence of a thin rotating layer of fluid close to the cylinder pushes the stagnation point from the cylinder as can be seen from the plotted streamline patterns. The streamlines ABC [as in Fig. 2(f)] with the progress of time reveals the alleyway formation around the primary vortex.

Figure 3(a–g) shows the similar comparison between experimental visualization and computed streamline for the case of $Re = 200$ and $\Omega = 1.0$. Once again, our results are indistinguishable from the computations of Badr and Dennis [13] except for $t = 3.0$ where it is seen that neither of the computations match with the experimental result to all details of the flow field. The match is, however, quite good at subsequent times. The solution changes in character at around this time and the elliptic solver needs larger number of iterations for convergence. As compared with the case of $\Omega = 0.5$, here the second and third vortices are released much later. The transpositioning of saddle points and alleyway formation are absent here. The nascent second and third vortices appear sometime between $t = 4.0$ and 4.5. In the next frame at $t = 5.0$, these two vortices merge to form a single recirculating zone. These can be seen better in the vorticity contours as plotted as a function of time in Fig. 4, which are also compared with the computations of Chen et al. [17].

In Fig. 5, we show the computed streamline contours for the case of $Re = 1000$ and $\Omega = 3.0$. These are also compared with experimental results and other computations wherever possible. Once again, the agreement is quite encouraging except for later times. While our calculations match with the results of Badr et al. [14] for all times shown, both the calculations differ from the corresponding experimental results for $t = 3.0$ and beyond. The larger time results show remarkable similarity with the steady-state inviscid results (as can be seen in Batchelor [1]) and the experimental results reported in Prandtl and Tietjens [2]. Badr et al. [14] have stated that the lack of agreement is due to either three-dimensionality or the flow becoming turbulent. In the absence of any three-dimensional calculations, it is not possible to pinpoint the reasons. However, we would like to conjecture that this could also be due to vibration of the cylinder and other disturbance sources ingested from the shear layer on the wind tunnel wall at large rotation rates for high translational velocities of the towed model in the tunnel. This is plausible since the agreement becomes poor only close to the body; also, the presented results in Badr et al. [14] for $Re = 1000$ and $\Omega = 2.0$ shows a very good agreement even for $t = 6.5$. It is known from the experimental results of Diaz et al. [6] that the Reynolds stress reduces near the plane of symmetry in the wake significantly with increase in rotation rate and hence the effect of turbulence will be more pronounced for $\Omega = 2.0$ than for 3.0—which is at odds with the conclusions of Badr et al. [14]. The present numerical scheme can resolve much higher wavenumbers for high Reynolds number flows and this was the reason that the original proponents of this scheme [22] claimed that a direct numerical simulation (DNS) is possible with this method. However, without simulating the full three-dimensional flows this cannot be tested for the present flow situation. This will be an attractive proposition for computing higher Reynolds number flows keeping in view the observation of Diaz et al. [6] for reducing Reynolds stresses and eventually relaminarizing the flow.

To investigate this issue further we have also computed the case for $Re = 3800$ and $\Omega = 2.0$. The results are presented in Fig. 6 and it is quite clear that even at $t = 20$, the flow has not reached the steady-state. This case was computed as Tokumaru and Dimotakis [31] have provided a lift value for this case. From Figure 3 of Tokumaru and Dimotakis [31], it is seen that the lift value for this rotation rate is insensitive to Reynolds number and the ratio of cylinder span and diameter. The results for three different rotation rate cases at $Re = 3800$ are provided in Mittal et al. [32].

Here the computed flow field reveals some interesting differences with the $Re = 1000$ and $\Omega = 3.0$ case. We know from the potential flow theory that a closed ring of fluid forms around the cylinder for a rotation rate greater than 2.0; however, one can note the closed ring of fluid...
forming for this case. This ring of fluid retains its identity and then again disappears because of various vortex interactions. From Fig. 6, one can note the formation of such rings at $t = 3.5$, 8.5 and 14.5 and which disintegrates at $t = 5.0$, 10.5 and 16.5, respectively. The present sets of computations are performed only up to $t = 20$ and within this period, the flow reveals some sort of periodicity. The important flow features revealed through these computations are the following: initially, one can notice the formation of a pair of vortices near the shoulder of the cylinder which continuously interact with each other. By $t = 2.0$, they merge with each other.
Fig. 5(a,b)—Caption on page 17.
Fig. 5(c–e)—Caption opposite.
Fig. 5. Comparison of experimental and computational [14] streamlines with the present computations for flow past translating and rotating circular cylinder at $Re = 1000$ and $\Omega = 3.0$. 

(f) $t=3.0$

(g) $t=4.0$

Present computation

Computational result [2]
Fig. 6. Streamline contours for flow past a translating and rotating circular cylinder at $Re = 3800$ and $\Omega = 2.0$. 
Fig. 7(a).
Fig. 7. Isovorticity contours for flow past a translating and rotating circular cylinder at $Re = 3800$ and $\Omega = 2.0$. 
At $t = 5.5$, the closed ring of fluid disintegrates and the streamline coming from the top forms a tongue-shaped recirculation zone and wraps around the cylinder like an alleyway. The above recirculation zone keeps growing in length and width with time, and at $t = 8.5$ it detaches from the body along with the formation of the closed ring of fluid close to the boundary. Notice that this ring also encompasses a small vortex and because of this the ring takes a distorted form—which is never seen for the $Re = 1000$ and $\Omega = 3.0$ cases. At $t = 6.5$, the above-mentioned recirculation zone gives rise to a bubble near the shoulder of the cylinder and in the process the dividing streamline turns by $270^\circ$ and remains so until $t = 8.0$, after which this situation is not seen to recur. A similar sequence leading to formation of a tongue-like recirculation zone occurs at $t = 17.0$ and this remains intact till the end of computation at $t = 20$. From the vorticity contours shown in Fig. 7, one can discern a pattern of shed vortices after $t = 10$. One sees the formation of two anticlockwise vortex to a single clockwise vortex. While the anticlockwise vortices seem to form on the lower lee side of the cylinder, the clockwise vortices form in the vicinity of the shoulder of the cylinder.

Figure 8 shows the lift, drag and moment for this particular case as a function of time up to $t = 20$. The loads and moment are calculated from the surface pressures as obtained from the solution of the pressure Poisson equation at discrete times. The corresponding $c_p$ distribution around the cylinder is plotted as a function of time in Fig. 9.

From Fig. 8, it is observed that the lift coefficient increases rapidly up to $t = 4$ after which it is approximately periodic with a slight increase in the mean phase averaged lift. The moment about the leading edge decreases from a small positive value up to $t = 4$ and then is approximately periodic—with qualitatively similar behaviour as observed for lift. The drag coefficient decreases in an oscillatory manner. Notice the small negative values of $c_d$ at those times where the $c_p$ distribution exhibits a trough at around $\theta = 240^\circ$.

5. CONCLUDING REMARKS

In this paper, the initial time development of flow past a translating and rotating cylinder is studied computationally by a third-order upwind scheme. A detailed error analysis allowed us to pinpoint the problem associated with computation of such a flow field by third-order upwind schemes which led to a choice of a particular grid system. This grid minimizes numerical dispersion and dissipation. In addition, the developed methods are seen to be much faster and very accurate while not being restricted by numerical instability at high Reynolds numbers. The computed results match well with the experimental flow visualization results of Coutanceau and Menard [5] for $Re = 200$ and low rotation rates ($\Omega = 0.5$ and 1.0). For $Re = 1000$ and $\Omega = 3.0$ the computed results match well at smaller times while it approaches a steady state at larger time—which is similar to that predicted in inviscid theory and also found in early experiments.
of Prandtl and Tietjens [2]. Some new results are provided for $Re = 3800$ and $\Omega = 2$ and they exhibit some entirely new flow phenomena as compared with the other cases. Nonetheless, it will be extremely beneficial to do a three-dimensional calculation for the high Reynolds number case with the higher-order upwind scheme to resolve some of these issues.

Acknowledgements—We would like to thank Professor Coutanceau for permitting reproduction of some of the experimental photographs taken from their published results.

REFERENCES


32. Mittal, S., Nair, M. T. and Sengupta, T. K., Numerical Simulation of flow past rotating and translating circular cylinder. Accepted for presentation at the 7th Asian Congress of Fluid Mechanics, December 1997, Madras.