Proceedings of the Conference on Modelling Fluid Flow
Budapest University of Technology and Economics, Hungary 2006

Edited by T. Lajos, J. Vad

Copyright © Department of Fluid Mechanics, Budapest University of Technology and Economics and the Authors
All Right Reserved

No part of the material protected by this copyright may be reproduced or utilized in any form or by any means, electronic or mechanical, including photocopying, recording or by any storage or retrieve system, without written permission from the copyright owner.

Volume I. ISBN 963 06 0361 6
Volume II. ISBN 963 06 0382 9

Published by Department of Fluid Mechanics
Budapest University of Technology and Economics
H–1111 Budapest, Bertalan L. u. 4–6, Hungary
Tel: +36 1 463 40 72
Fax: +36 1 463 34 64
e-mail: lajos@ara.bme.hu
www.ara.bme.hu
VORTEX DYNAMICS AND GRID-BASED SIMULATIONS FOR LOW REYNOLDS NUMBER FLOW PAST A CYLINDER

R. Ivan LEWIS¹, László BARANYI²

¹ School of Mechanical and Systems Engineering, University of Newcastle upon Tyne, UK, E-mail: r.i.lewis@ncl.ac.uk
² Corresponding Author, Department of Fluid and Heat Engineering, University of Miskolc, H-3515 Miskolc-Egyetemváros, Hungary. Tel.: +36 46 565 154, Fax: +36 46 565 471, E-mail: aramb@uni-miskolc.hu

ABSTRACT

Computational Fluid Dynamics models range from the Lagrangian style vortex cloud simulation technique to the finite difference type grid-based method for solving the Navier-Stokes equations. This paper undertakes a comparison of these two methods for the classical datum case of flow past a stationary circular cylinder at low Reynolds numbers in the range 10 to 220. Comparisons include time-history, time-mean and root-mean-square values of oscillating drag and lift coefficients, frequency of vortex shedding and related vortex street wake flow patterns. Particularly close agreement was obtained for Strouhal number versus Reynolds number, and good agreement for time-mean value of drag coefficients; comparison was also made with experimental results. Attempts are also made to calculate the skin friction and surface pressure components of the cylinder drag, revealing the significance of skin friction drag within this range and its relative insignificance above a Reynolds number of 220.

Keywords: CFD, grid-based method, stationary circular cylinder, Strouhal number, vortex cloud method

NOMENCLATURE

\( C_D \) [-] drag coefficient, \( 2D / (\rho U^2 d) \)
\( C_{Df} \) [-] drag coefficient due to skin friction
\( C_L \) \([m]\) lift coefficient, \( 2L / (\rho U^2 d) \)
\( d \) \([m]\) cylinder diameter, length scale
\( D \) \([-N]\) dilation, drag force
\( f_s \) \([1/s]\) vortex shedding frequency
\( k(s,\sigma) \) \([1/m]\) coupling coefficients linking points \( m \) and \( n \) on a body
\( L \) \([N]\) lift force
\( p \) [-] pressure, non-dimensionalised by \( \rho U^2 \)
\( Re \) [-] Reynolds number, \( Ud/\nu \)

\( St \) [-] Strouhal number, \( f_d/\nu \)
\( t \) [-] time, non-dimensionalised by \( d/\nu \)
\( u, v \) [-] velocities in \( x,y \) directions, non-dimensionalised by \( U \)
\( U \) \([m/s]\) free stream velocity, velocity scale
\( U_{90} \) \([m/s]\) free stream velocity in the \( x \) directions
\( U_{90x}, V_{90} \) \([m/s]\) velocity components at \( m \) due to a unit strength vortex element at \( j \)
\( v_n \) \([m/s]\) body surface velocity at point \( n \)
\( x, y \) [-] Cartesian coordinates, (grid-based method), non-dimensionalised by \( d \)
\( \beta_m \) [-] body profile slope at point \( m \)
\( \gamma \) \([m/s]\) surface sheet vorticity strength
\( \Delta T \) \([m^2/s]\) discrete vortex element strength
\( \Delta t \) [-] time step (non-dimensionalised by \( d/\nu \))
\( \omega \) \([kg/m^3]\) fluid density
\( \nu \) \([m^2/s]\) kinematic viscosity
\( \omega \) \([1/s]\) vorticity

Subscripts

c critical
\( D \) drag
\( f \) skin friction
\( L \) lift
\( m, n \) points on a body surface
\( mean \) mean value
\( rms \) root-mean-square value

1. INTRODUCTION

An interest in the formation of vortex streets behind bluff bodies has fascinated large numbers of researchers since the early experiments of Strouhal [1] in 1878 concerning the generation of 'aeolian tones' and the famous 1911 paper by Theodore von Kármán [2] on the phenomenon later called the 'Kármán vortex street'. Serious early attempts at discrete vortex modelling were made by Rosenhead [3], who studied the related phenomenon of the Kelvin-Helmholtz instability of vortex sheets. Abernathy and Kronauer [4] extended Rosenhead's model to simulate instabilities of a parallel pair of
vortex sheets of equal and opposite strength subjected to an initial sinusoidal perturbation along their length, a flow akin to two separating vortex sheets released by a bluff body. Also asserted by these authors was the constancy of mean drag coefficient and Strouhal number for a circular cylinder over a wide range of Reynolds numbers, \( R_e = 300 \) to 100,000, demonstrating the negligible influence of viscosity upon these flows. Below \( R_e = 300 \) viscosity is very important, while above this range three-dimensional instabilities due to convection set in. Within this range however there are considerable variations of both Strouhal number and cylinder drag coefficients, which are one interest of the present study.

This problem is of such practical importance that there are numerous studies dealing with flow past cylinders that are fixed, oscillating, rotating, or in orbital motion. Among these, the fixed cylinder is usually the starting point of investigations, as it is relatively simple to carry out experiments, and thus also for numerical studies it is favoured because comparison with experimental data is possible to confirm validity of the computer method used.

A huge number of researchers have investigated flow around a single circular cylinder through experimental, theoretical, and numerical approaches. Despite its simple geometry, the problem is not only extremely complex but also one with many applications. Knowledge of flow patterns around bluff bodies is important in the design of large structures such as smokestacks or bridges, which in winds are often subjected to large amplitude oscillation due to alternating vortex shedding, sometimes causing collapse of the structure. Slender struts or tubes are likewise subject to vibration due to vortex shedding, which can also cause noise generation in both external aerodynamics and internal aerodynamic. Roshko [5], Norberg [6], and Bearman [7] are among those who have provided invaluable experimental data on flow around a fixed cylinder. Computations on the same problem have been performed by many researchers [8,9].

The main purpose of this study is to compare two very different CFD methods against the important datum case of flow past a stationary circular cylinder at low Reynolds numbers. The first author has developed a vortex cloud analysis that follows the creation and diffusion from the body surface in Lagrangian fashion, Lewis [10], and that is well suited for modelling separated flows where boundary layer modelling is less key. This analysis is reviewed briefly in section 2.1. The second author on the other hand has developed a finite difference method based on Eulerian description with grid transformation that facilitates simulation of the flow of a viscous fluid past a cylinder. This was developed to study the flow past a cylinder in oscillating or orbital motion, Baranyi [11], and his method is reviewed briefly in section 2.2 for application to a stationary cylinder. Both methods deliver solutions of the Navier-Stokes equations for two-dimensional flow and are thus suited to simulation of cylinder wake studies in the low Reynolds number range 10 to 220.

2. OVERVIEW OF THE TWO METHODS OF ANALYSIS

This section gives an overview of the vortex cloud and grid-based methods for simulating a low Reynolds number flow around a circular cylinder placed in a uniform stream.

2.1. Overview of the present vortex cloud CFD method

The fundamental basis of vortex element representation of the Navier-Stokes equations has been given by the first author [10]. According to this method the entire flow is controlled by vorticity continuously being created at the body surface, diffused by viscosity and convected. The numerical method is Lagrangian in character and involves discretisation of the surface vorticity into discrete free-vortex elements which are released from the body surface at successive small time steps and subjected to convection and viscous diffusion, the latter being undertaken by random walks as described by Chorin [12], and Porthouse [13].

![Figure 1. Numerical model for vortex cloud analysis](image)

The slip flow at the body surface is created by the vorticity sheet \( \gamma(s) \) where, for the boundary condition of zero internal velocity inside the body,
the surface velocity $v_m$ is equal to the local surface vorticity $\gamma(s_m)$.

In a viscous fluid the surface vorticity $\gamma(s)$ is being continuously created in the boundary sublayer and diffused away from the surface where it is also subject to convective processes. For practical computations Eq. (1) may be expressed in the following equivalent numerical form

$$
\sum_{j=1}^{M} \frac{K(s_j, s_m) \gamma(s_m)}{\lambda} = -U_m \cos \beta_m - \sum_{j=1}^{M} \Delta \Gamma_j \left(U_m \cos \beta_m + V_m \sin \beta_m\right),
$$

where, as illustrated in Fig. 1, the body surface is represented by $M$ elements.

Over a sequence of small time steps $\Delta t$ vortex cloud theory assumes that these vortex sheets are shed as discrete vortex elements of strength $\Delta \Gamma = \gamma(s_m) \Delta s_m$. As indicated in Fig 1, greater resolution may be achieved by shedding this vorticity from $n_{sub}$ sub-elements, where, for the example of three illustrated, the vortex sheet strengths would be $\Delta \Gamma_n = \gamma(s_{n}) \Delta s_{n}/3$. In addition to this vorticity creation and shedding process for each time step $\Delta t$ and in accordance with the Navier-Stokes equations, the vortex cloud is subject to convection and diffusion, the latter being accomplished by imposing random walks.

**2.1.1. Lift and drag forces**

Lift and drag forces comprise two components due to surface pressure and shear stresses. As shown elsewhere [10], the pressure distribution on the cylinder surface at any instant may be calculated from the equation

$$
\Delta p = -\rho \gamma(s_m) \Delta s_m / \Delta t = -\rho n_{sub} \Delta \Gamma / \Delta t,
$$

where $\Delta p$ is the pressure change along the surface of element $\Delta s_n$ during the time step $\Delta t$. Pressure forces $P_x$ and $P_y$ on the cylinder in the $x$ and $y$ directions then follow from

$$
P_x = \sum_{n=1}^{M} \Delta p_n \sin(\beta_n), \quad P_y = -\sum_{n=1}^{M} \Delta p_n \cos(\beta_n).
$$

Frictional shear stresses may be estimated for each surface element by considering the shear stress after time $t$ on a plane wall parallel to the x-axis shedding surface vorticity $\gamma(s_m)$ at time $t=0$. As shown elsewhere [10,14], the frictional forces $F_x$ and $F_y$ on the cylinder may then be approximated by

$$
F_x = \rho \sqrt{\frac{V}{\pi \Delta t}} \sum_{n=1}^{M} \gamma(s_n) \Delta s_n \cos(\beta_n),
$$

The lift and drag forces then follow from

$$
L = P_x + F_x, \quad D = P_y + F_y.
$$

One problem of vortex cloud analysis is the presence of significant numerical noise due to the use of random walks for simulation of viscous diffusion. Smoothing can be achieved by using a running average of say three time steps before and after the current one when evaluating lift and drag, the practice adopted here.

**2.2. Overview of the grid-based method of CFD analysis**

The second author’s earlier studies dealt with computation of the flow around a fixed circular cylinder at different Reynolds numbers, from $Re = 10$ to 1000, and up to $Re = 180$ for an oscillating cylinder, e.g. Baranyi and Shirakashi [15], and good agreement was obtained with experimental data for the variation of Strouhal number and time-mean drag coefficient with Reynolds number, especially up to $Re = 190$ above which three-dimensional instabilities occur. Further features of flow are investigated for a fixed cylinder in Baranyi [16], and the time-mean value of base pressure coefficient, which influences the near-wake structure, compares well with the experimental data of Roshko [17]. In [16] the energy equation for a stationary circular cylinder with constant surface temperature placed in a uniform flow is also solved from $Re = 50$ to 180 and the predicted dimensionless heat transfer coefficient or Nusselt number agrees well with available experimental results. In Baranyi and Lakatos [18], computational results obtained by this method are compared in the Reynolds number domain of $Re = 50$ - 180 with existing experimental results for the root-mean-square value of lift coefficients shown in Norberg [19], and a very good agreement is observed. Additional evidence for the reliability of the method has been given by comparison with other computational methods. De Sampaio [20], reports that his results for time-mean drag coefficient, obtained by the finite element method, agree well with those in [16].

At the time [15] was published the computing power of computers was more limited. In this paper some of the earlier computations are repeated with much finer temporal and spatial resolution, and new computations have also been added and compared with those of the vortex cloud method.

**2.2.1. Governing equations**

Constant property incompressible Newtonian fluid flow is assumed. When deriving the governing equations, which in non-dimensional forms are the
Navier-Stokes equations, the equation of continuity and a Poisson equation for pressure:

\[
\begin{align*}
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} &= -\frac{\partial p}{\partial x} + \frac{1}{Re} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right), \\
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} &= -\frac{\partial p}{\partial y} + \frac{1}{Re} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right), \\
\frac{\partial D}{\partial t} + \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} &= 0, \\
\nu^2 p &= \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + 2 \left( \frac{\partial u}{\partial x} \frac{\partial v}{\partial y} - \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} \right). 
\end{align*}
\]

The gravity force is included in the pressure terms in Eqs. (7) and (8). Although the dilaton $D$ is theoretically zero by continuity (9), it is advisable to retain the term $\partial D/\partial t$ to avoid computational instability.

On the cylinder surface, no-slip boundary condition is used for the velocity and a Neumann type boundary condition is used for the pressure. At the far region, potential flow is assumed.

**2.2.2. Transformation of domain and applied numerical methods**

To be able to impose the boundary conditions precisely, a boundary-fitted coordinate system is used. The physical domain and the governing equations with boundary conditions are all transformed to the computational plane with an equidistant mesh, see [16]. This mapping (see Figure 2) provides a very fine mesh near the cylinder surface to capture the fluid-structure interaction in the near wake, but is gradually reduced to a coarse mesh in the far field to reduce computational cost. The equidistant mesh for the computational plane is beneficial for computational ease. The transformed equations are solved by finite difference method, [16]. 4th order central difference is used for the diffusion term and pressure derivatives, and the convective term is handled by a 3rd order upwind scheme [8]. Equations of motion (7) and (8) are integrated explicitly in time, and the S.O.R. method is used for solution of the pressure Poisson equation. Continuity (9) is satisfied at every time step. The code has been extensively tested against experimental results for stationary cylinders and good agreement has been found.

For this study of flow around stationary cylinders the non-dimensional time step was 0.0005 and the number of grid points for the majority of runs was 301x177, but at the lowest three $Re$ numbers 201x118, 251x148, and 270x159 were used, respectively, for convergence reasons. Even for the highest $Re$ investigated, the solution was mesh independent. The computational domain shown in Fig. 2 is characterised by $R_2/R_1=40$. For further details see [15,16].

**3. COMPARISON OF $C_L/C_D$, STROUHAL NUMBER AND WAKE FLOW PATTERN VERSUS REYNOLDS NUMBER**

As we shall see, over the chosen range of Reynolds number $Re$ there are considerable variations of both lift coefficient $C_L$ and drag coefficient $C_D$. Vortex street development is associated with large oscillations of $C_L$ and $C_D$. The high $Re$ range has also been explored, and the results are given in section 3.1. The drag on a circular cylinder originates from two sources, namely pressure forces normal to the body surface or "form drag" and viscous shear stresses parallel to the surface or "skin friction drag" $C_D$. For this low $Re$ range there are also considerable variations of these and comparisons of the predicted values are given in section 3.2. As a consequence of this the fluid motion is dominated by viscous effects at the low values $Re$ in this range and by convective effects at high $Re$. This has considerable influence over the wake flow patterns and will be illustrated in section 3.3.

**3.1. Comparison of lift and drag coefficients and of Strouhal number versus Reynolds number**

In this section comparisons are given of predicted lift coefficient and Strouhal number for the two CFD methods. To begin with in Figure 3, comparisons are given of the predicted lift and drag coefficients for a sample mid-range value of $Re=120$, assuming a uniform stream $U=1.0$, a cylinder diameter of unity and an overall elapse time of $t=400$, ample to establish periodic motion.

As may be seen from Figs. 3, the lift coefficient settles down quickly into oscillating von Kármán vortex shedding with a mean value of zero. The predicted Strouhal number for $Re=120$ is of value 0.173 from the vortex cloud analysis and 0.175 from the grid-based method which compare well. As is well known, the frequency of the drag coefficient signal is double that of the lift coefficient as predicted by both methods. There are, however, considerable differences in the predicted time-history curves obtained by the two methods.

![Figure 2. Physical and computational planes](image-url)
The grid-based analysis predicts that the motion settles down into a very regular periodic motion in which both $C_l$ and $C_D$ oscillate with constant amplitudes. The vortex cloud method, on the other hand, exhibits some randomness in these amplitudes, although the mean values are in reasonable agreement as can be seen from the dashed curves in Fig. 3a. The predicted variations of amplitude of $C_l$ and $C_D$ are probably due to the random walk method for simulation of viscous diffusion in vortex dynamics. However, good agreement between the methods was obtained for the two performance parameters that most affect the excitation characteristics of cylinder wake flows, the Strouhal number and the root-mean-square value of the lift coefficient fluctuations $C_{Lrms}$. As further evidence of this, comparisons of the predicted values of $St$, $C_{Dmean}$, and $C_{Lrms}$ versus $Re$ for the two methods are given below in Figures 4 & 5.

![Figure 4. Comparison of predicted Strouhal numbers](image)

The second author investigated the critical Reynolds number for the onset of vortex shedding. However, grid method computations in the vicinity of this critical $Re$ require a lot of CPU time to reach the quasi-steady state of vortex shedding (although the possibility exists that introducing turbulence into the free stream may reduce the time needed for the establishment of vortex shedding). That is why in this study the case pertaining to $Re = 47.2$ was the nearest Reynolds number investigated above the critical one. By extrapolating the present $C_{Lrms}$ values to zero the critical Reynolds number is about $Re_c = 46.8$. Other studies have found similar values. For sufficiently large aspect ratios Norberg [23] found experimentally that the critical value of
Reynolds number where periodic vortex shedding starts is about \( Re = 47 \). By using the Stuart-Landau model Thompson and Le Gal [24] found that the supercritical Hopf bifurcation characterising the onset of periodic vortex shedding is at \( Re = 46.4 \).

![Figure 5. Variation of rms lift and drag coefficient fluctuations with Reynolds numbers](image)

Fig. 5 shows the variation of the \( \text{rms} \) values of lift and drag coefficient over the chosen \( Re \) range, comparing the two CFD methods. It can easily be seen that predicted \( C_{\text{L rms}} \) values are much larger than \( C_{\text{D rms}} \) values above the critical Reynolds number of \( Re \approx 47 \), especially for the grid-based method. There is a good measure of agreement in \( C_{\text{L rms}} \) between the two CFD methods in the supercritical region, albeit with a scatter in the vortex cloud results. On the other hand the predicted \( C_{\text{D rms}} \) values are excessively high for the vortex cloud method.

3.2. Comparison of predicted form and skin friction drag coefficients

![Figure 6. Comparison of predicted total and skin friction drag coefficients with published experimental values](image)

The final comparison of the two CFD methods is shown in Figure 6, namely predicted values of both the total drag coefficient \( C_D \) and the skin friction component of this \( C_D \) compared also with experimental results published by Massey [25] for the selected \( Re \) range.

As Massey's results show (Fig. 6) the total drag coefficient drops considerably over the range \( 10 \leq Re \leq 200 \) from about 2.75 to 1.25 (a ratio of 2.2) and both methods predict this trend well, the grid method particularly so. It is encouraging to see this mutual agreement over such an important widely varying \( Re \) range of twenty to one, within which the shift transfers from the dominance of viscous effects to those of convective processes (as will be illustrated by other considerations in Section 3.3 and Table 1 below). Thus also from Massey's data the skin friction drag component falls from roughly \( C_{\text{DF}} = 0.75 \) to 0.14 over this \( Re \) range (a ratio of 5.42) and is thus only a small fraction of the total drag, decreasing to negligible values at the top end of the \( Re \) range. The two CFD methods here produce predictions that lie on either side of Massey's experimental results, reflecting the general dramatic trends while not in such good agreement with each other.

3.3. Selected flow patterns for low \( Re \) predicted by the vortex cloud method

![Figure 7. Predicted vortex shedding flow patterns for Re range 20 to 220](image)
One advantage of the vortex cloud method is the economy in modelling wake patterns by the cloud of discrete vortices, which can then also be viewed in movie style. Application of this method for the selected Re range produced the comparisons presented in Figure 7 revealing the developing significant changes in wake pattern.

As shown by these results, the onset of periodic vortex shedding only begins to appear above Re=20, below which viscous diffusion dominates sufficiently over convection to avert this. At the higher value of Re=60 however a very regular wake periodicity is established which continues right through the higher Re range up to Re=220. Through Re=100-140 the actual vorticity wake sheet thins down to a fairly constant thickness in the near wake oscillations. At the higher range of Re=180-220 however, one observes that as convection begins to dominate, the near wake oscillations begin to roll up into distinct vortex cores while remaining still in a regular vortex street pattern.

A simple analysis that bears up this expectation follows from a comparison of the average convective and diffusive displacements of a typical vortex element as a function of Re. Assuming an average convective velocity of say \( \frac{U_0}{\Delta t} \), the average convective displacement \( \Delta x_{cv} \) of a vortex cloud element during a time step \( \Delta t \) would thus be \( \Delta x_{cv} = \frac{U_0}{\Delta t} \Delta t = U_0 \Delta t \). As also shown elsewhere \([10,14]\), the diffusion average random walk may be expressed

\[
\Delta x_{sr} = \sqrt{\frac{4D}{\Delta t}} = 2\sqrt{\frac{(U d \Delta t ln 2)}{Re}}
\]

**Table 1. Vortex cloud average convective and diffusive shifts for \( U=1.0, d=1.0, \Delta t=0.05 \)**

<table>
<thead>
<tr>
<th>Re</th>
<th>Ave. convective shift ( \Delta x_{cv} )</th>
<th>Ave. diffusive shift ( \Delta x_{sr} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>0.025</td>
<td>0.117741</td>
</tr>
<tr>
<td>20</td>
<td>0.025</td>
<td>0.083255</td>
</tr>
<tr>
<td>60</td>
<td>0.025</td>
<td>0.049868</td>
</tr>
<tr>
<td>100</td>
<td>0.025</td>
<td>0.037230</td>
</tr>
<tr>
<td>140</td>
<td>0.025</td>
<td>0.031468</td>
</tr>
<tr>
<td>180</td>
<td>0.025</td>
<td>0.027752</td>
</tr>
<tr>
<td>220</td>
<td>0.025</td>
<td>0.025101</td>
</tr>
<tr>
<td>260</td>
<td>0.025</td>
<td>0.023091</td>
</tr>
<tr>
<td>300</td>
<td>0.025</td>
<td>0.021496</td>
</tr>
</tbody>
</table>

Table 1 compares these predicted average convective and diffusive shifts \( \Delta x_{cv} \) and \( \Delta x_{sr} \) for \( 10 \leq Re \leq 300 \). The extremely interesting observation to be made here is that for a typical vortex element \( \Delta x_{cv} \) and \( \Delta x_{sr} \) are equal when \( Re=220 \). Thus convective processes may be expected to dominate above this \( Re \) value and diffusive processes below it, in agreement with the vortex wake motions presented in Fig. 7. For the lowest value \( Re=10 \) tabulated here however, the average diffusive shift of the vortex cloud random walk process is almost 5 times the average convective shift. It should be mentioned that for this Reynolds number the numerical vortex cloud scheme arranges for the random walk simulation of viscous diffusion to be undertaken in five sub-steps to maintain equal resolution with the convective process.

**4. CONCLUSIONS**

Two completely different CFD simulation techniques based on vortex dynamics and a grid-based method have been compared for the classic datum case of flow past a stationary circular cylinder over the low Reynolds number range \( 10 \leq Re \leq 220 \) with the following conclusions:

1. Good agreement was obtained in prediction of the Strouhal number which varies between 0.12 < St < 0.2 over the Re range considered, and both methods predict the formation of a von Kármán vortex street above Re ≈ 47. Using the grid method and extrapolating the \( C_{D_{rms}} \) values to zero, the critical Re for the onset of vortex shedding was found to be Re ≈ 46.8.

2. Both methods are in reasonable agreement with the experimental measurements published by Massei [25] and with one another for the total drag coefficient \( C_D \). Predictions of the contribution to this due to skin friction \( C_D_f \) are in less good agreement but show similar trends.

3. Predictions of the rms fluctuations \( C_{D_{rms}} \) and \( C_{L_{rms}} \) are in less good agreement. The grid CFD method predicts zero values of these below the critical level Re=47 due to the total absence of a vortex street and rapidly rising values 0.0 < \( C_{D_{rms}} < 0.46 \) for 47 < Re < 190. Vortex cloud results agree with the latter reasonably well but predict excessive values of both \( C_{L_{rms}} \) and \( C_{D_{rms}} \) for subcritical Reynolds numbers Re=47.

4. With regard to the last mentioned item, it is noted that the grid CFD method used extremely small time steps \( \Delta t=0.0005 \) resulting in tight control of the discretisation errors in this time sensitive motion. The vortex cloud method on the other hand used time steps 100 times bigger, namely \( \Delta t=0.05 \) resulting in a much greater scale of "numerical turbulence". The main impact of this is felt within the body boundary layer due to the random walk method for simulation of viscous diffusion and its resulting impact on surface vorticity creation.

5. The average convective and diffusive shifts of a typical vortex element during a time step become equal at about Re=220. Above this, convective processes become more dominant.

6. Vortex dynamics handles heavily convective wake motions extremely well providing easy graphical presentation of the vortex street wake patterns. Furthermore the method has proved extremely flexible for adaptation to any other stationary or moving body flow and has been useful for simulating such other phenomena as rotating stall in turbomachinery blade rows.
ACKNOWLEDGEMENTS

The support provided by the Hungarian Research Foundation (OTKA, project No. T 42961) is gratefully acknowledged by the second author. We also thank Mr. S. Ujvárosi for preparing Fig. 6.

REFERENCES


