NUMERICAL CALCULATION OF LAMINAR VORTEX-SHEDDING FLOW PAST CYLINDERS

R. FRANKE, W. RODI and B. SCHÖNUNG
Sonderforschungsbereich 210, University of Karlsruhe, Kaiserstrasse 12,
D-7500 Karlsruhe (F.R.G.)
(Received January 5, 1990)

Summary

The paper presents numerical calculations of laminar vortex-shedding flows past circular and square cylinders for $Re \leq 5000$ in the former and $Re \leq 300$ in the latter case. The calculations were performed by solving the unsteady 2D Navier-Stokes equations with a finite volume method incorporating the third-order-accurate discretization scheme QUICK. The resulting Reynolds number dependence of the Strouhal number and of the drag coefficient is compared with experiments and with previous numerical results, showing good agreement for the lower Reynolds numbers at which fully laminar flow can be expected ($Re < 1000$). For higher Reynolds numbers the calculations deviate from the measurements, and this is blamed on the beginning influence of stochastic fluctuations. For the circular cylinder the time development of the flow towards periodic vortex shedding is illustrated by a series of streamline pictures, and for both geometries the time development of a number of important flow parameters is also presented and discussed.

1. Introduction

The periodic vortex shedding behind bluff bodies exposed to uniform flow has fascinated researchers ever since the days of Leonardo da Vinci. The occurrence of this flow phenomenon is due to instabilities and depends on the geometry of the bluff body and the Reynolds number; it has often been the cause of failure of flow-exposed structures in various fields of engineering. Of interest to engineers are not only the integral parameters such as Strouhal number, drag and lift coefficient but also the local dynamic loading of a bluff body placed in a wind or water stream. Especially in the field of civil engineering, detailed knowledge on the flow field can help to predict, for example, wind loads on facade elements or the dynamic response of construction elements.

A large number of experimental studies have been carried out on vortex-shedding flows (see e.g. Bearman [1]), but detailed experimental knowledge on the unsteady flow field is rather limited owing to the considerable effort involved in taking unsteady measurements in such flows. A rare exception is the work of Cantwell and Coles [2] who studied the turbulent flow past a
circular cylinder at a Reynolds number of 140,000 with the aid of a flying hot-wire. These authors succeeded in separating the periodic and turbulent fluctuations and presented phase-averaged values for various phases during one period not only for the velocity components but also for the Reynolds stresses.

Vortex-shedding flow past cylindrical structures has also attracted the attention of numerical analysts. Already 20 years ago, Son and Hanratty [3] published numerical solutions of the unsteady two-dimensional Navier–Stokes equations for the flow around a circular cylinder for $Re < 500$. Their and later publications of other authors reported promising results of such solutions. However, most of these earlier numerical calculations suffered from the limited computer resources available then and were restricted to fairly coarse numerical grids. This situation has changed in recent years, and two of the latest numerical studies should be mentioned explicitly. Braza et al. [4] used a finite volume method to perform a detailed numerical study of the vortex shedding past circular cylinders for Reynolds numbers below 1000, and Lecointe and Piquet [5] calculated the flow around the same geometry for steady and unsteady oncoming flow with a finite difference method. The latter authors employed the stream function–vorticity approach, which cannot be extended directly to three-dimensional problems. Both publications give detailed evaluations of the calculation results and comparisons with experiments. Somewhat disturbing is the fact the Braza et al. [4] had to introduce perturbations to obtain vortex shedding in their numerical simulations and that the choice of unsuitable perturbations can influence the numerical results. Davis and Moore [6] employed a finite volume method incorporating the QUICK-EST discretization scheme to study numerically the flow around a square cylinder. For this geometry, numerical problems are more severe owing to the extreme velocity gradients prevailing at the sharp corners of the square cylinder. This may be responsible for the fact that numerical and experimental results do not compare satisfactorily in all details. One problem common to most numerical studies of vortex-shedding flow past cylinders is the presence of numerical diffusion, which effectively reduces the Reynolds number and may even prohibit the self-excitation of vortex shedding. A trustworthy numerical method should be able to predict the occurrence of periodic vortex shedding by itself.

The great advantage of a numerical simulation is the availability of details on all aspects of the flow for every stage of the flow development. In particular, the transition from an initially quasi-steady flow to the final periodic vortex-shedding flow is difficult to study in detail in an experiment. One aim of the present paper is to examine in numerical experiments the development of periodic vortex shedding as the oncoming flow velocity is increased from zero to a terminal value. This corresponds of course to the situation when a body is accelerated from rest to a certain speed. The work reported here constitutes the first step of the development of a calculation method for three-dimensional
turbulent vortex-shedding flows. The numerical method employed allows the
direct incorporation of a turbulence model and also a direct extension to three-
dimensional situations. In this paper the method is presented and validated
for two-dimensional laminar vortex-shedding flows. To this end, calculations
of the flow around both square and circular cylinders are presented and com-
pared as far as is possible with measurements and with previous numerical
simulations. Work on the calculation of higher Reynolds number turbulent
flows using various turbulence models and on the extension to three dimen-
sions is in progress and will be reported in later papers.

2. Mathematical model

The continuity and momentum equations governing the two-dimensional
laminar flow past cylinders can be written for the square cylinder in cartesian
coordinates and for the circular cylinder in polar coordinates as follows.

**Cartesian coordinates**

continuity

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$  \hspace{1cm} (1)

x-momentum

$$\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} \right) = - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left( \mu \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)$$  \hspace{1cm} (2)

y-momentum

$$\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} \right) = - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left( \mu \frac{\partial v}{\partial y} + \frac{\partial u}{\partial x} \right) + 2 \frac{\partial}{\partial y} \left( \mu \frac{\partial v}{\partial y} \right)$$  \hspace{1cm} (3)

**Polar coordinates**

continuity

$$\frac{\partial}{\partial r} (ru) + \frac{\partial v}{\partial \Theta} = 0$$  \hspace{1cm} (4)

r-momentum

$$\rho \left( \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial r} + \frac{v \partial u}{r \partial \Theta} \right) = - \frac{\partial p}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} \left( \mu \frac{\partial u}{\partial r} \right) + \frac{1}{r} \frac{\partial}{\partial \Theta} \left( \mu \frac{\partial u}{\partial \Theta} \right)$$

$$+ \frac{v^2}{r} + \frac{1}{r \partial r} \left( \mu \frac{\partial v}{\partial r} \right) + \frac{1}{r \partial \Theta} \left\{ \mu \frac{\partial (ve)}{\partial r} \right\} - 2 \frac{\mu}{r^2} \left( \frac{\partial v}{\partial \Theta} + u \right)$$  \hspace{1cm} (5)
The computational domains in which the equations were solved and the outer boundary conditions are given in Fig. 1 for both configurations. At the cylinder wall the no-slip condition was applied. The equations were solved numerically for these boundary conditions with a modified version of the programme TEACH by Gosman and Pun [7]. This employs a finite volume method for solving the equations in primitive variables on a two-dimensional staggered grid. The coupling between continuity and momentum equations was achieved with the SIMPLEC predictor–corrector algorithm of van Doormal and Raithby [8], which is an improved version of the SIMPLE algorithm incorporated in the original TEACH programme. The central–upwind hybrid spatial discretization scheme in the original TEACH programme was replaced by the QUICK scheme (quadratic upwind interpolation for convective kinematics) proposed by Leonard [9]. This scheme combines the high accuracy of a third-order scheme with the stabilizing effect of upwind weighting. A disadvantage of the scheme is its unboundedness, which may cause over- and undershoots. For time discretization the fully implicit first-order Euler scheme was chosen. It provides high stability but requires small time steps in order to obtain accurate solutions (more than 100 time steps per period were used). The resulting system of linear difference equations was solved by the strongly implicit method of Stone [10]. A more detailed description of the numerical method is given in Franke and Schönung [11].

Figure 2 shows typical numerical grids for both square and circular cylinder configurations. The accuracy of the numerical results depends strongly on the resolution of the boundary layer near the cylinder walls. Therefore non-uniform grids were chosen with a constant expansion factor. The number of grid points used for the various calculations are given in Tables 1 and 2. Grid independence and the influence of the location of the outer boundary were tested extensively. Refinement of the numerical grid and of the time step by a factor of 2 changed the Strouhal number and the amplitudes of the force coefficients by less than 5%. It was found that the distance of the first grid point away from the wall has a particularly strong influence on the results. If near-wall resolution is too coarse, the shedding frequencies are predicted too high. For the square cylinder, distances of $y_{wall}/D \approx 0.004$ were found to be sufficient, and for the circular cylinder of $y_{wall}/D \approx 0.001$. Further refinement did not change the numerical results in the Reynolds number ranges covered.

The time-marching calculations were started with the fluid at rest, and the

$$
\rho \left( \frac{\partial v}{\partial t} + u \frac{\partial v}{\partial r} + v \frac{\partial v}{\partial \theta} \right) = - \frac{1}{r \partial \theta} \frac{\partial}{\partial \theta} \left( \frac{\mu}{r} \frac{\partial v}{\partial \theta} \right) + \frac{1}{r \partial r} \left( \frac{\mu}{r} \frac{\partial v}{\partial r} \right) + \frac{1}{r \partial r} \left( \frac{\mu}{\partial \theta} \right)
$$

$$
+ \frac{1}{r \partial r} \left( \frac{\mu}{r} \frac{\partial v}{\partial r} \right) + \frac{1}{r \partial r} \left( \frac{\partial u}{\partial r} \right) + \frac{1}{r \partial r} \left( \frac{\partial u}{\partial r} \right) + \frac{1}{r \partial r} \left( \frac{\partial (r \mu)}{\partial r} \right) - \frac{\rho uv}{r} \quad (6)
$$

The computational domains in which the equations were solved and the outer boundary conditions are given in Fig. 1 for both configurations. At the cylinder wall the no-slip condition was applied. The equations were solved numerically for these boundary conditions with a modified version of the programme TEACH by Gosman and Pun [7]. This employs a finite volume method for solving the equations in primitive variables on a two-dimensional staggered grid. The coupling between continuity and momentum equations was achieved with the SIMPLEC predictor–corrector algorithm of van Doormal and Raithby [8], which is an improved version of the SIMPLE algorithm incorporated in the original TEACH programme. The central–upwind hybrid spatial discretization scheme in the original TEACH programme was replaced by the QUICK scheme (quadratic upwind interpolation for convective kinematics) proposed by Leonard [9]. This scheme combines the high accuracy of a third-order scheme with the stabilizing effect of upwind weighting. A disadvantage of the scheme is its unboundedness, which may cause over- and undershoots. For time discretization the fully implicit first-order Euler scheme was chosen. It provides high stability but requires small time steps in order to obtain accurate solutions (more than 100 time steps per period were used). The resulting system of linear difference equations was solved by the strongly implicit method of Stone [10]. A more detailed description of the numerical method is given in Franke and Schönung [11].

Figure 2 shows typical numerical grids for both square and circular cylinder configurations. The accuracy of the numerical results depends strongly on the resolution of the boundary layer near the cylinder walls. Therefore non-uniform grids were chosen with a constant expansion factor. The number of grid points used for the various calculations are given in Tables 1 and 2. Grid independence and the influence of the location of the outer boundary were tested extensively. Refinement of the numerical grid and of the time step by a factor of 2 changed the Strouhal number and the amplitudes of the force coefficients by less than 5%. It was found that the distance of the first grid point away from the wall has a particularly strong influence on the results. If near-wall resolution is too coarse, the shedding frequencies are predicted too high. For the square cylinder, distances of $y_{wall}/D \approx 0.004$ were found to be sufficient, and for the circular cylinder of $y_{wall}/D \approx 0.001$. Further refinement did not change the numerical results in the Reynolds number ranges covered.

The time-marching calculations were started with the fluid at rest, and the
inflow velocity was then increased following a sine function until the preselected free-stream velocity was achieved. Earlier calculations had shown that an impulsive start of the flow generated disturbances which were damped out.
only slowly by physical and numerical diffusion. When accurate numerical schemes are employed, the fully periodic phase of the solution is reached faster when the fluid is accelerated more slowly. A typical computing time on an 88×94 grid was 40 min on an IBM 3090 computer for one flow period in the fully periodic stage.
### Table 1

Numerical parameters and results for the flow around the circular cylinder

<table>
<thead>
<tr>
<th>$Re$</th>
<th>Grid</th>
<th>$\Delta t$ (s)</th>
<th>$\Delta t/T \times 10^{-3}$</th>
<th>$\Delta y_{wall}/D$</th>
<th>Strouhal number</th>
<th>Drag coefficient</th>
<th>Lift coefficient</th>
<th>Point of separation</th>
<th>Point of stagnation amplitude</th>
</tr>
</thead>
<tbody>
<tr>
<td>50</td>
<td>$88 \times 144$</td>
<td>0.05</td>
<td>5.8</td>
<td>0.0001</td>
<td>0.116</td>
<td>1.39 0.94 0.45</td>
<td>0 0 0</td>
<td>123.0 0.0 0.0</td>
<td>—</td>
</tr>
<tr>
<td>66</td>
<td>$88 \times 94$</td>
<td>0.05</td>
<td>6.8</td>
<td>0.0001</td>
<td>0.135</td>
<td>1.35 0.95 0.40</td>
<td>±0.12 ±0.10 -0.02</td>
<td>120.2 ±1.0 ±0.5</td>
<td>1.71</td>
</tr>
<tr>
<td>80</td>
<td>$140 \times 184$</td>
<td>0.05</td>
<td>7.6</td>
<td>0.0001</td>
<td>0.152</td>
<td>1.35 0.97 0.38</td>
<td>±0.26 ±0.22 -0.04</td>
<td>118.9 ±3.5 ±0.9</td>
<td>1.56</td>
</tr>
<tr>
<td>200</td>
<td>$140 \times 144$</td>
<td>0.05</td>
<td>9.7</td>
<td>0.0001</td>
<td>0.194</td>
<td>1.31 1.06 0.25</td>
<td>±0.65 ±0.60 -0.06</td>
<td>111.5 ±5.7 ±2.1</td>
<td>1.14</td>
</tr>
<tr>
<td>300</td>
<td>$88 \times 144$</td>
<td>0.05</td>
<td>10.3</td>
<td>0.0001</td>
<td>0.205</td>
<td>1.32 1.11 0.21</td>
<td>±0.84 ±0.77 -0.07</td>
<td>109.1 ±8.2 ±2.5</td>
<td>1.02</td>
</tr>
<tr>
<td>1000</td>
<td>$140 \times 144$</td>
<td>0.012</td>
<td>2.95</td>
<td>0.0001</td>
<td>0.236</td>
<td>1.47 1.34 0.13</td>
<td>±1.36 ±1.30 -0.06</td>
<td>108.5 ±44.8 ±3.5</td>
<td>0.76</td>
</tr>
<tr>
<td>2000</td>
<td>$200 \times 94$</td>
<td>0.012</td>
<td>3.06</td>
<td>0.0001</td>
<td>0.245</td>
<td>1.58 1.49 0.09</td>
<td>±1.73 ±1.68 -0.05</td>
<td>108.0 ±62.5 ±4.9</td>
<td>0.66</td>
</tr>
<tr>
<td>5000</td>
<td>$140 \times 144$</td>
<td>0.012</td>
<td>3.06</td>
<td>0.0001</td>
<td>0.245</td>
<td>1.68 1.63 0.05</td>
<td>±1.91 ±1.87 -0.04</td>
<td>106.7 ±69.5 ±6.6</td>
<td>0.55</td>
</tr>
</tbody>
</table>
### 3. Results

#### 3.1. Steady Flow

For a Reynolds number of 40 the flow around a circular cylinder is steady and was studied quite extensively in both measurements and numerical calculations. Therefore test calculations for this case are well suited to generate confidence in the numerical method employed. The present calculations for this case yielded a length of the separation bubble of 2.86 cylinder diameters, a drag coefficient of 1.52 and the separation to occur at an angle of 126.2° from the front stagnation point. These figures are in good accord with previous findings. Figure 3 compares the pressure distribution along the cylinder wall and the velocity distribution across the wake at \(x/D = 1\) with experimental data. The agreement can be seen to be very good. The calculation of the flow around a square cylinder at the same Reynolds number (\(Re = 40\)) also gave a steady solution with reasonable results (see Table 2). For this case no experimental data are available for comparison.

At higher Reynolds numbers, when unsteady vortex shedding occurs, a solution of the steady Navier–Stokes equations (time-dependent terms in eqns. (2) and (3) dropped) does not yield reasonable results for the time-averaged behaviour of vortex-shedding flow. This is demonstrated in Fig. 4, where the upper half shows the streamlines for the flow around a square cylinder at \(Re = 150\) as calculated with a steady procedure, while the lower half shows the streamline picture obtained by time averaging the unsteady solution. The steady approach leads to a considerably too long separation bubble because important moment exchange processes due to the interaction of the shed vortices are neglected. These also influence the base pressure, and the drag coefficient predicted by the steady approach is about 18% smaller than calculated by the unsteady method.

**Table 2**

<table>
<thead>
<tr>
<th>(Re)</th>
<th>Grid</th>
<th>(\Delta t (s))</th>
<th>(\Delta t/T \times 10^{-3})</th>
<th>(\Delta y_{wall}/D)</th>
<th>Strouhal number</th>
<th>Drag coefficient</th>
<th>Lift coefficient</th>
<th>Eddy size</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td>Total Due to friction</td>
<td>Total Due to friction</td>
<td></td>
</tr>
<tr>
<td>40</td>
<td>76×64</td>
<td>-</td>
<td>0.0063</td>
<td>-</td>
<td>1.98</td>
<td>1.69</td>
<td>0.029</td>
<td>-</td>
</tr>
<tr>
<td>70</td>
<td>88×76</td>
<td>0.025 3.3</td>
<td>0.0038 0.133</td>
<td>1.69</td>
<td>1.57</td>
<td>0.12</td>
<td>±0.19  ±0.16  ±0.03 2.3</td>
<td></td>
</tr>
<tr>
<td>100</td>
<td>88×76</td>
<td>0.025 3.8</td>
<td>0.0038 0.154</td>
<td>1.61</td>
<td>1.55</td>
<td>0.06</td>
<td>±0.27  ±0.24  ±0.03 2.0</td>
<td></td>
</tr>
<tr>
<td>150</td>
<td>88×76</td>
<td>0.025 4.1</td>
<td>0.0038 0.165</td>
<td>1.56</td>
<td>1.57</td>
<td>-0.01</td>
<td>±0.38  ±0.34  ±0.04 1.96</td>
<td></td>
</tr>
<tr>
<td>200</td>
<td>186×156</td>
<td>0.012 2.0</td>
<td>0.0013 0.157</td>
<td>1.60</td>
<td>1.65</td>
<td>-0.05</td>
<td>±0.62  ±0.57  ±0.05 2.20</td>
<td></td>
</tr>
<tr>
<td>250</td>
<td>88×76</td>
<td>0.025 3.5</td>
<td>0.0038 0.141</td>
<td>1.67</td>
<td>1.72</td>
<td>-0.05</td>
<td>±1.15  ±1.07  ±0.08 2.38</td>
<td></td>
</tr>
<tr>
<td>300</td>
<td>186×156</td>
<td>0.025 3.3</td>
<td>0.0013 0.130</td>
<td>1.83</td>
<td>1.89</td>
<td>-0.06</td>
<td>±1.92  ±1.84  ±0.08 2.40</td>
<td></td>
</tr>
</tbody>
</table>
Fig. 3. Comparison between experimental and numerical data for the flow around a circular cylinder at $Re=40$; (a) pressure distribution at cylinder wall; (b) velocity distribution at cross-section $x/D=1$ behind the cylinder.
3.2. Unsteady Flow

In the following, results of unsteady calculations are presented for Reynolds numbers for which the flow shows unsteady vortex shedding. It should be emphasized that no disturbance was introduced into the calculations. The periodic vortex shedding is a natural outcome of the numerical solutions triggered by round-off errors.

3.2.1. Circular Cylinder

The flow around a circular cylinder was calculated for the Reynolds number range $50 < Re < 5000$. Details on the numerical grids and time steps used and various resulting integral parameters are compiled in Table 1. Figures 5 and 6 show respectively the dependence of the Strouhal number and of the drag coefficient on the Reynolds number. Experimental data and results from the numerical calculations of Braza et al. [4] and Lecointe and Piquet [5] are also included. The predictions of these parameters agree fairly well with the mea-
measurements up to a Reynolds number of 300, and for the Strouhal number even up to $Re=1000$. The deviations at the higher Reynolds number, which are similar for Lecointe and Piquet's results, are likely to be due mainly to the beginning influence of the stochastic turbulent fluctuations. These are not accounted for in the calculations. According to Braza et al. [4] and Bloor [15], turbulent fluctuations start to influence the flow in the wake of a circular cylinder in the Reynolds number range 400–1400. In order to account for these effects in a two-dimensional calculation, a turbulence model has to be introduced. At the higher Reynolds numbers there is also an increased uncertainty about the numerical accuracy, since grid independence tests could not be afforded and the reduced viscous damping may lead to numerical oscillations.

In addition to an evaluation of the Strouhal number and the time-mean drag coefficient, the numerical results allow a detailed study of the time-dependent nature of the flow. During the development of the periodic vortex shedding, the flow goes through various phases. Figure 7 shows this development by providing streamline pictures at various times after the onset of the motion, and Fig. 8 shows the time development of various typical flow parameters. Shortly after the acceleration of the oncoming flow has started, a symmetric separation bubble develops behind the cylinder (Fig. 7(a)) which grows as the oncoming velocity increases. In this phase the drag coefficient grows rapidly as is shown.
Fig. 7. Development of vortex shedding past a circular cylinder; final Reynolds number is $Re = 200$. 

- **a)** $t = 2$, $c_D = 1.34$
- **b)** $t = 10$, $c_D = 1.00$
- **c)** $t = 28$, $c_D = 0.90$
- **d)** $t = 32$, $c_D = 0.89$
- **e)** $t = 42$, $c_D = 0.88$
- **f)** $t = 46$, $c_D = 0.88$
- **g)** $t = 50$, $c_D = 0.87$
- **h)** $t = 52$, $c_D = 0.87$
- **i)** $t = 54$, $c_D = 0.88$
- **j)** $t = 56$, $c_D = 0.89$
- **k)** $t = 58$, $c_D = 0.90$
- **l)** $t = 60$, $c_D = 0.93$
- **m)** $t = 62$, $c_D = 0.93$
- **n)** $t = 64$, $c_D = 0.93$
Fig. 8. Time behaviour of various parameters for flow past circular cylinder at $Re = 200$: (a) drag coefficient; (b) lift coefficient; (c) position of stagnation point (angle); (d) position of separation angle on top half of the cylinder; (e) position of separation angle on bottom half of the cylinder; (f) size of attached vortex at positions 1, 2 and 3.
in Fig. 8(a). After the oncoming velocity has reached its final value and the acceleration has stopped, the drag coefficient falls again (Fig. 8(a)), but the separation bubble still continues to grow (Figs. 7(b) and 7(c)). A quasi-steady state develops with no lift force (Fig. 8(b)) and with steady positions of the stagnation and separation points (Figs. 8(c)-8(e)). The Föppl vortices behind the cylinder oscillate slightly during this phase (Figs. 7(c)-7(f)), and out of this, vortex shedding and the vortex street develop (Figs. 7(g)-7(n)). After the start of the periodic vortex shedding, the time-mean drag coefficient increases again, and the other flow parameters such as the lift coefficient (Fig. 8(b)), the positions of the stagnation point (Fig. 8(c)) and the points of separation (Figs. 8(d) and 8(e)) oscillate periodically after a short period of transition. This initial behaviour was observed for all the Reynolds numbers considered and for both cylinder geometries, but the stability of the quasi-steady flow phase decreases as the Reynolds number increases.

The dynamics of the flow behind the cylinder and the formation and shedding of vortices can be studied to advantage by tracking the time development of some suitable parameters characterizing the size and location of the vortices. To this end, along three lines denoted as positions 1, 2 and 3 in Fig. 9 the distance from the wall is determined where the velocity component parallel to the wall changes sign for the first time. If the position line goes through the centre of the vortex, this distance will correspond to the radius of the vortex as long as this is still attached to the rear cylinder wall. At the upper and lower positions 1 and 3, tracking of this eddy size parameter allows one to observe the growth and finally the separation (shedding) of the vortices developing on either side of the symmetry line. At the centre position 2 the parameter is very sensitive to flow asymmetries and is therefore a good indicator of the development of such asymmetries. Figure 8(f) shows the time development of the eddy size parameters at the three positions. As was discussed already, the initially almost symmetrical upper and lower vortices first increase in size, growing fairly slowly during the quasi-steady phase (see positions 1 and 3), while at the middle position 2 the oscillation of the Föppl vortices can be seen to start fairly early. In the fully periodic phase the behaviour at the upper position 1 is shifted 180° relative to that of the lower position 3 as the vortices grow, and the final shedding happens alternately on both cylinder sides. The lower peaks developing at these positions stem from the vortices originating from the opposite side but extending in their final stages before separation to the position line on the other side.

Figure 10 displays the streamlines for three times during one period of the fully periodic flow. The lowest picture represents a time when the vortex originating from the upper cylinder side is just about to separate from the cylinder surface and has reached its maximum size as an attached vortex, which is represented by the peaks at position 2 in Fig. 8(f). This vortex is also responsible for the higher peaks at position 1, while the lower peaks are caused by a vortex.
Fig. 9. Definition of eddy size (e.s.) of the attached vortices: (a) circular cylinder; (b) square cylinder.

originating from the other side, as seen in the top part of Fig. 10. Owing to the strong interaction between the vortices originating from either side, the area of recirculating fluid is considerably smaller than during the quasi-steady phase, which can also be seen from Fig. 7. This shows once more why a steady calculation must overpredict the length of the separation bubble.

The dependence of various flow parameters on the Reynolds number is summarized in Table 1, showing that the point of separation moves towards the front of the cylinder and the vortex size is reduced when the Reynolds number increases. The table shows further that the forces on the cylinder are not af-
3.2.2. Square Cylinder

Calculations of the flow around a square cylinder were performed for the Reynolds number range $40 < Re < 300$, and details on the numerical grids and time steps employed as well as on the predicted integral parameters are compiled in Table 2. The flow around a square cylinder behaves in a very similar way to the flow around a circular cylinder; the main difference is the fact that the points of separation are fixed at the sharp corners of the cylinder. Compared with the circular cylinder, there are only a few experimental data available. In Fig. 11 the Strouhal number is plotted versus the Reynolds number, including results from the present predictions and from the predictions of Davis and Moore [6] as well as experimental results of Davis and Moore [6] and Okajima [16]. The relatively large discrepancies between the various results point to experimental as well as numerical uncertainties. In particular, the sharpness of the cylinder corners in the experiments and the numerical treatment of these corners can influence the shedding frequency. Owing to the high
velocity gradients in the corner regions, the present method employing the unbounded QUICK scheme is particularly prone to over- and undershoots in this region.

For $Re < 150$ the separation has been observed to happen at the rear corners of the cylinder, and up to this Reynolds number the size of the vortices is reduced when the Reynolds number increases, which is associated with a reduction of the drag coefficient. The opposite behaviour is observed for $Re > 150$, when separation occurs at the front corners. The size of the vortices now increases with increasing Reynolds number. At $Re = 150$ the calculated separation jumps between the front and the rear corner as can be seen from the streamlines presented in Fig. 12. It can further be noted that the vortex sizes are roughly twice as large as for the circular cylinder.

The time development of various flow parameters is shown for the case of $Re = 200$ in Fig. 13. Similarities to Fig. 8 are obvious, and the flow can be seen to go through the same phases as in the case of the circular cylinder. There are, however, a few differences in the behaviour of the vortex sizes shown in Fig. 13(d). The maximum vortex sizes are larger than for the circular geometry, as was noted before. Also, in contrast to the circular cylinder, the vortices originating on one side never extend to the position line on the other side (see Fig. 9) so that the double-peak nature observed in Fig. 7 does not occur here. Up to $Re = 200$ the behaviour of lift and drag forces is indeed quite similar for the circular and the square cylinder. This changes for the higher Reynolds num-

Fig. 11. Strouhal number versus Reynolds number for the square cylinder.
Fig. 12. Streamlines of flow past square cylinder for two times during one period ($Re = 150$).

Fig. 13. Time behaviour of various parameters for flow past square cylinder at $Re = 200$: (a) drag coefficient; (b) lift coefficient; (c) frequency analysis of lift coefficient; (d) size of attached vortex at positions 1, 2 and 3.
bers. Figure 14 displays the time behaviour of the lift and drag coefficients as well as the corresponding power spectra for $Re = 250$. For this case and also for $Re = 300$ a second frequency is observed for the drag coefficient, which occurs in addition to the mean mode. These additional frequencies ($f = 0.06$ for $Re = 250$, $f = 0.09$ for $Re = 300$) have not been reported by other researchers. On the other hand, for the lift coefficient the shedding frequency is still dominant since the power spectrum for this quantity shows only very small additional peaks. From this difference in the behaviour of the drag and lift coefficient, it can be deduced that the higher mode periodicity appears only in the wake of the cylinder and is not present in the separation bubbles at the cylinder sides occurring at the higher Reynolds numbers. For $Re = 250$, Davis and Moore [6] did not predict these additional frequencies, but for $Re = 1000$ they report what they call "subharmonics" without giving the actual frequencies. The shift in Reynolds number for the onset of this phenomenon between their and the present study corresponds to a shift in the Reynolds number dependence of the Strouhal number between the two numerical simulations (see Fig. 11). In experiments, any additional frequencies may have been obscured or suppressed by the onset of superimposed turbulent fluctuations.

Fig. 14. Time behaviour of various parameters for flow past square cylinder at $Re = 250$: (a) drag coefficient; (b) frequency analysis of drag coefficient; (c) lift coefficient; (d) frequency analysis of lift coefficient.
4. Conclusions

Numerical calculations of vortex shedding past circular and square cylinders have been reported for situations up to \( Re = 5000 \) for the circular and up to \( Re = 300 \) for the square geometry. The calculations were obtained by solving the unsteady 2D Navier–Stokes equations without incorporation of a turbulence model with the aid of a finite volume method. This employed a higher order spatial discretization scheme and produced and maintained the oscillating shedding motion without introducing an artificial disturbance. For the lower Reynolds numbers (say \( Re < 1000 \)), grid- and timestep-independent solutions could be obtained, as was tested extensively and is reported in more detail in ref. 11.

Satisfactory results were obtained for \( Re \) up to 1000 and \( Re \) up to 300 in the cases of circular and square cylinders respectively. This conclusion is supported by the fairly good agreement with experiments and with other numerical results. In this Reynolds number range the influence of stochastic fluctuations can still be expected to be negligible, while for larger Reynolds numbers this influence becomes significant and a turbulence model must be used. For the truly laminar Reynolds number range the calculation method described is capable of producing reasonably accurate results for the main practically relevant parameters such as Strouhal number, drag and lift coefficient. For practical applications one problem is the relatively high computing time. The great potential of the numerical method for studying all details of the unsteady flow development was demonstrated; many more results on the flow details that could not be included here are given in the back-up report [11].

The work presented here is only a first step towards the development of a more general calculation method. In practice, flows past bluff bodies usually occur at much higher Reynolds numbers so that they are definitely influenced by stochastic turbulent fluctuations, and they are also often three-dimensional, e.g. when the approach flow is not uniform. Extensions to include the effect of turbulent fluctuations through a turbulence model and to three-dimensional flows are in progress.

Acknowledgments

The work reported here was supported by the Deutsche Forschungsgemeinschaft. The calculations were carried out on the IBM 3090 computer of the University of Karlsruhe computer centre. The authors are grateful to Mrs. R. Zschernitz for her expert typing of the manuscript.
References