Large eddy simulation of pulsating flow over a circular cylinder at subcritical Reynolds number

Chunlei Liang, George Papadakis *

Experimental and Computational Laboratory for the Analysis of Turbulence (ECLAT), Department of Mechanical Engineering, King’s College London, Strand, London WC2R 2LS, UK

Received 18 March 2005; received in revised form 1 October 2005; accepted 24 October 2005
Available online 19 January 2006

Abstract

The pulsating cross-flow over a single circular cylinder at the subcritical Reynolds number \( \text{Re}_D = 2580 \) is studied with the large eddy simulation (LES) technique using the standard Smagorinsky model as well as a dynamic model in which the test filtered quantities are evaluated through a truncated Taylor series expansion. The filtered equations are discretised using the finite volume method in an unstructured, collocated grid arrangement with a second-order accurate method, in both space and time. The predictions are compared against very detailed experiments for mean velocities and Reynolds stresses that were performed in a duct of cross-section 72 mm × 72 mm using the PIV technique. The effects of mesh refinement close to the cylinder as well as of subgrid scale model are also examined. The numerical predictions are in very good agreement with the measurements in terms of mean as well as turbulence quantities. The instantaneous flow patterns of the flow field are examined and the effect of the external flow pulsation on the wake characteristics such as vortex formation length, vortex strength, Strouhal number as well as the lift and drag coefficients is quantified. The vortex formation length is decreased while the mean drag, as well as the rms values of the drag and lift coefficients increase significantly under pulsating flow conditions. The performance of the LES technique is analysed in the light of the wake characteristics.

© 2005 Elsevier Ltd. All rights reserved.

1. Introduction

The understanding of vibrations induced by bluff body vortex shedding is of great practical importance for the design of structures such as heat exchangers, offshore platforms, power cables, etc. The uniform cross-flow over a circular cylinder is a classical example of bluff body flow. Although the configuration is simple, the flow is characterised by very rich and complex phenomena for the subcritical Reynolds number \( \text{Re}_D = 2580 \) examined in this work: thin and attached laminar boundary layers in the front part of the cylinder, separating shear layers, development of Kelvin–Helmholtz instabilities along these layers that lead to transition, as well as small scale streamwise and spanwise vortices in the turbulent wake that interact with large scale Karman vortices. A comprehensive review of the flow characteristics for a wide range of Reynolds numbers is given by Williamson [1].

During the past decade, many large eddy simulations (LES) have been performed for subcritical Reynolds numbers, especially at \( \text{Re}_D = 3900 \), mainly due to the availability of the experimental results of Lourenco and Shih [2] and Ong and Wallace [3]. The calculations have been performed in either structured [4–7] or unstructured meshes [8–10]. Recently, Piomelli et al. [11] performed LES calculations using the Lagrangian dynamic model [12] and the immersed boundary method [13]. In the aforementioned studies, the effect of grid resolution, subgrid scale model, as well as discretisation schemes for the convection terms have been examined in great detail. There is a general consensus that low-order upwind schemes are not able to predict correctly the base suction coefficient, separation angles and the size and structure of the recirculation zone behind the cylinder. A variety of discretisation methods and numerical solution techniques have been used, for example
the finite volume method, the Galerkin B-Spline or the spectral/hp method. This flow has also been used as a benchmark case to examine theoretical issues pertaining to the application of LES in general curvilinear coordinate systems. For example, Jordan [14] examined the sequence of two spatial operations, namely coordinate transformation of the basic Cartesian form of the governing equations and filtering and recommended to transform the full-resolution equation system first and then filter the result.

None of the aforementioned LES calculations considered the effect of inlet flow conditions, although they have a great impact on the wake flow structure. Invariably, all experiments contain some level of freestream turbulence. For the interested reader, Scholten and Murray [15,16] provide detailed information on the effect of inlet turbulence level. Prescribing time-dependent turbulent inlet flow conditions at the upstream boundary is an important and challenging task [17] but it requires a separate solution of the filtered equations in a simpler arrangement, usually between parallel flat plates. During this simulation, the instantaneous velocities in a cross-stream plane are stored on the disk and then used as inlet conditions for the simulation of the turbulent flow.

The aforementioned inlet conditions do not contain a discrete frequency i.e. the energy of the fluctuations is spread along a wide range of frequencies, which is typical of fully turbulent flows. On the other hand, there are many practical applications for which a sinusoidal wave is superimposed on a uniform mean bulk flow. For example, all biological flows through peristaltic pumps are pulsating. Pulsation also occurs in many engineering applications such as the discharge from a reciprocating pump, the flow in the intake and exhaust manifolds of an internal combustion engine, and the flow in hydraulic and pneumatic lines. It has been shown that pulsating approaching flow enhances mixing [18], heat and mass transfer [19,20] and also offers the opportunity for active flow control by tuning the frequency and amplitude of pulsation [21,22].

Pulsating approaching cross-flow at a frequency around twice the natural vortex shedding frequency introduces the phenomenon of vortex shedding lock-on. The phenomenon is accompanied by a number of changes in the cylinder body forces and the wake flow and it is therefore expected to result in significant changes of heat transfer as well. Experimental data of Barbi et al. [23], Armstrong et al. [24,25] and Konstantinidis et al. [26] reveal that lock-on modifies significantly the near-wake flow structure. The numerical results of Papadakis and Bergeles [22] have shown that lock-on enhances heat transfer after the separation point at low Reynolds numbers.

Although detailed simulations of the steady flow around a single cylinder have been reported in the past as already mentioned, similar simulations for pulsating flows have, to the best of the authors’ knowledge, yet to appear in the literature for turbulent flows. All the aforementioned simulations for pulsating approaching flows are for two-dimensional laminar flows only. However, as the Reynolds number increases, streamwise vortices form and the flow becomes at first three-dimensional, while subsequently instabilities appear in the laminar separating shear layers that eventually lead to transition to a turbulent wake. The interaction of the externally imposed pulsation, the separating and unsteady shear layers and the Karman vortices constitutes a very interesting problem of fluid mechanics that is tackled in this paper using LES.

The aim of the present paper is twofold. First, to analyse the effect of flow pulsation on the mean and instantaneous flow patterns. To this end, the instantaneous vortex shedding patterns are examined, the lift and drag coefficients are correlated with the observed patterns and the changes on the vortex shedding frequency are quantified. For comparative purposes, runs have also been made for a steady approaching flow in the same Reynolds number. Second, to fill the aforementioned gap and examine the suitability of LES to predict accurately this type of flow. To this end, the effect of grid size and subgrid-scale model on the predicted time-averaged mean flow and turbulence quantities under pulsating approaching conditions is examined. The results are validated against detailed experimental data for mean and turbulence quantities. The predictive ability of LES is also analysed in the context of the observed coherent structures.

2. Geometry and conditions

The computational domain is described in a fixed Cartesian coordinate system (x, y, z). The x-axis is along the streamwise mean flow direction, the z-axis is parallel to the cylinder axis (spanwise direction), while the y-axis is perpendicular to both x- and z-axis (transverse or cross-wake direction). The origin of the coordinate system and the size of the computational domain are shown in Fig. 1. The diameter of the cylinder is 7.2 mm. The experiments, against which the simulations are compared, were carried out using the PIV technique in a stainless steel water tunnel with cross-section 72 mm × 72 mm i.e. the blockage ratio was 10%. Many experimental conditions were examined and part of the data have appeared in Konstantinidis et al. [27,28]. The actual data were provided kindly by Konstantinidis [29].

The time-averaged approaching velocity is $u_\infty = 0.36$ m/s and the corresponding Reynolds number is $Re_D = 2580$. A sinusoidal profile is superimposed on $u_\infty$ and thus the velocity $u(t) = u_\infty + \frac{u_\infty}{\pi f} \sin(2\pi f_t, t)$ is prescribed at the inlet plane. This velocity is assumed to be uniform across the entrance to the computational domain. The operating conditions for all cases examined are shown in Table 1.

The size of the domain in the cross-stream direction y matches that of the experimental arrangement i.e. it is equal to 72 mm or 10 cylinder diameters, as shown in Fig. 1. The size in the spanwise direction is usually estimated from prior knowledge of the sizes of the streamwise vortex structures. However, details of these structures are available only for steady approaching flows around free cylinders. The wavelength of the streamwise structures in
The near wake of a free circular cylinder scales as $k_z = D/\lambda_2^{1/4}$ [30]. For Reynolds number 2580, the estimated wavelength $k_z/D$ is equal to around 0.5D. Further downstream, larger scale structures have been reported with wavelength $k_z/D \approx 1$. It is expected that the blockage will tend to decrease rather than increase the scale of these structures. Breuer [7] as well as Kravchenko and Moin [6] for the majority of their calculations for steady approaching flow for Reynolds number 3900 used a spanwise length equal to $L_z = \pi D$. In the absence of experimental information regarding the size of streamwise structures in the examined conditions, the spanwise length used in the present study was the same as the one employed by the aforementioned investigators. The predictions obtained with this length are then validated against detailed experimental data for both mean as well as turbulence quantities.

### 3. Modelling approach and solution method

#### 3.1. Governing equations and sgs modelling

Applying a low-pass spatial filter to the Navier–Stokes equations and assuming that the filtering and differentiation operations commute, the following equations for the filtered quantities written in Cartesian form, are obtained:

\[ \frac{\partial \bar{u}_i}{\partial x_i} = 0, \]

\[ \frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = - \frac{1}{\rho} \frac{\partial P}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} + \nu \frac{\partial^2 \bar{u}_i}{\partial x_i \partial x_j}, \]

where the index $(i = 1, 2, 3)$ represents each direction in the Cartesian coordinate system and $P$ is the filtered pressure. The incompressible form of the subgrid scale Reynolds stress is

\[ \tau_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j. \]  

To model the above term, two modelling approaches are used, which are described below.

**3.1.1. Smagorinsky eddy viscosity model**

The standard Smagorinsky model [31] employs the Boussinesq approximation,

\[ \tau_{ij} - \delta_{ij} \frac{2}{3} \tau_{kk} = -2 \nu_T \bar{S}_{ij} = -\nu_T \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right), \]

where $\bar{S}_{ij}$ is the resolved-scale strain rate tensor. The trace of the subgrid-scale stresses $\tau_{kk}$ is incorporated in the pressure resulting in a modified pressure term. The eddy viscosity is modelled as

\[ \nu_T = (C_s A_s)^2 |\bar{S}|, \]

where the Smagorinsky constant $C_s$ used is 0.1, as suggested by Deardorff [32] and Breuer [7]. The magnitude of the strain rate tensor $|\bar{S}|$ is $\sqrt{2S_{ij}S_{ij}}$.

The LES calculation is sensitive to the modelling of the near cylinder wall region, because the separation depends on the details of the attached boundary layer whether laminar or turbulent (in the present conditions laminar). In order to avoid the resolution of the thin boundary layers,

![Fig. 1. Physical configuration of the cylinder flow.](image-url)
one can use wall functions [33,34]. However, this is a delicate task when the separation point is not fixed by the geometry and is far from being settled. For the present computations, the non-slip condition was applied at the cylinder surface as a local refinement technique was used to obtain high resolution in the near wall region as explained below. The van-Driest damping function should be used together with the Smagorinsky model in order to reproduce the correct asymptotic behaviour of the sgs viscosity close to the wall [35]. However it was not used in the present investigation, mainly because this approach lacks generality as it involves the distance of a cell centroid to the wall which is difficult to evaluate in complex geometries. Instead, it was decided to use a dynamic model that not only reproduces the correct near-wall behaviour, but also evaluates dynamically the coefficient of the Smagorinsky model based on the local flow field as explained below.

3.1.2. Dynamic eddy viscosity model

Dynamic models [12,36] are sensitive to the local state of the flow and thus predict more accurately transition or re-laminarization and have the correct near-wall behaviour as opposed to the constant-coefficient Smagorinsky model. In the present simulations, a dynamic model which is based on a truncated Taylor series expansion that approximates the test filtering operator is used. The model, which was proposed by Chester et al. [37], has been tested for simplified cases like isotropic turbulence and turbulent flow between flat plates where Cartesian meshes are used. In the present investigation, the model was implemented in an unstructured grid arrangement and tested against detailed experimental data for the pulsating flow around a circular cylinder. As explained below, the model is based on velocity derivatives, which are readily available in the cell centroids of unstructured meshes, and therefore it is suitable for this type of grids.

This dynamic model also uses the Germano identity,

\[ L_{ij} = T_{ij} - \widetilde{\tau}_{ij} \tag{6} \]

which relates the subgrid-scale stresses computed at grid-filter level (denoted using a tilde, \( \tilde{\cdot} \)) and test-filter level (denoted using a hat, \( \hat{\cdot} \)). The individual terms in identity (6) are defined as

\[ L_{ij} = \hat{u}_i \hat{u}_j - \tilde{\tau}_{ij} \tag{7} \]

\[ T_{ij} = \hat{u}_i \hat{u}_j - \hat{u}_i \hat{u}_j \tag{8} \]

\[ \tilde{\tau}_{ij} = \tilde{u}_i \tilde{u}_j - \tilde{u}_i \tilde{u}_j. \tag{9} \]

Similar to the evaluation of \( \tau_{ij} \) using Eqs. (4) and (5), \( T_{ij} \) is computed using the length of the compound filter i.e. the effective filter obtained by applying sequentially the grid and test filters. This length was taken to be equal to \( xA \) where \( A \) is the grid-filter width and \( x \) is a parameter. Thus \( T_{ij} \) is given by

\[ T_{ij} = -2(C_s xA)^2 \tilde{S}_{ij}/S_{ij}. \tag{10} \]

The error associated with use of the model into the Germano identity can be minimized using the same approach as Germano et al. [36] and Lilly [38], by which the Smagorinsky coefficient is evaluated using

\[ (C_s)^2 = \frac{(L_{ij}M_{ij})}{(M_{mm}M_{mm})}, \tag{11} \]

where the bracket \( \langle \cdot \rangle \) denotes spatial averaging over the homogeneous direction. The tensor \( M_{ij} \) is given by

\[ M_{ij} = 2A^2(\tilde{S}_{ij} - x^2S_{ij}). \tag{12} \]

The idea of the model is to evaluate \( L_{ij} \) and \( M_{ij} \) by expanding locally the function to be test filtered in a truncated Taylor series. The resulting approximations to \( L_{ij} \) and \( M_{ij} \) are denoted by \( L_{ij}^{\prime} \) and \( M_{ij}^{\prime} \), respectively, while the subscript “′” is used to denote a Taylor series expansion. Using a box filter and keeping only terms which are second-order in terms of \( \Delta \), the following expressions are obtained:

\[ L_{ij}^{\prime} = \frac{\langle \Delta \rangle^2}{12} \frac{\hat{u}_i \hat{u}_j}{\hat{c}_m \hat{c}_m}, \tag{13} \]

\[ M_{ij}^{\prime} = 2A^2 \left\{(\tilde{S}_{ij}S_{ij} - x^2S_{ij}S_{ij}) + \frac{(\Delta)^2}{24} \hat{c}_m^2 (\langle S_{ij} \rangle_S^2) \right. \]

\[ \left. -x^2(S_{ij}S_{ij} \hat{c}_m S_{ij}) \right\}, \tag{14} \]

where \( \langle S_{ij} \rangle_S \) is a Taylor series approximation to \( \tilde{S}_{ij} \), which is given by

\[ \langle S_{ij} \rangle_S = \left[ 2 \left( \tilde{S}_{ij}^{\prime} + \frac{(\Delta)^2}{24} \hat{c}_m^2 S_{ij} \right) \right]^{1/2}. \tag{15} \]

The above equations are substituted in Eq. (11) to obtain \( C_s \). The index \( m \) denotes the homogenous direction, \( z \). As can be seen, the explicit test filtering operation is avoided. Instead, test-filtered quantities are replaced by expressions involving grid-filtered quantities and their first and second-order derivatives, which can be easily evaluated in unstructured grids. The parameter \( z \), the ratio of the compound filter width to the grid filter width, was taken to be equal to 2. Negative values of the square of Smagorinsky coefficient evaluated using Eq. (11) are simply clipped and no stability problems were encountered. It has been shown by Chester et al. [37], that the model reproduces the correct asymptotic behaviour of the coefficient \( C_s \) in the near wall region. The model takes about 30–40% more time compared to the standard Smagorinsky model.

3.2. Computational details

The finite volume method applied on an unstructured, collocated grid arrangement is employed to discretize the governing equations. All spatial terms in the momentum equations are discretized using the second-order central dif-
ferencing scheme (CDS) while the second-order accurate Crank–Nicolson method is employed to advance the terms in time, with the exception of the pressure term which is treated fully implicitly, i.e. it was evaluated at the new time step, instead of being split into two terms, one on the old and one on the new time step. The PISO algorithm [39] is used to deal with the pressure–velocity coupling between the momentum and the continuity equations. In order to avoid the generation of a check-board pressure field, the velocity interpolation method at the cell faces proposed by Rhie and Chow [40] is employed.

As shown in Fig. 2 and Table 1, two meshes are used. The coarser one has 294,912 cells while the finer has 746,688 cells and is produced by locally refining the coarser one in the near cylinder region (subdivision of each cell into four smaller cells in the $x$–$y$ plane). In the present simulations, 32 layers are used for both meshes in the spanwise direction that has length $L_z = \pi D$ as already mentioned. This number of layers is identical to the one used by Breuer [7] for the majority of his calculations for steady approaching flow. For each layer 128 or 256 cells are placed along the periphery of the cylinder for the coarse or the finer mesh respectively. A constant expansion factor of 1.11 is used for the cell spacings in the radial direction away from the cylinder wall. For the finer mesh, the smallest cell spacing in the radial direction is $\Delta r_{\text{min}}/D = 1.75 \times 10^{-3}$, slightly higher than the one of Beaudan and Moin’s [4] finest mesh ($\Delta r_{\text{min}}/D = 1.25 \times 10^{-3}$) for a Reynolds number of 3900.

The resolution of the boundary layer with the locally refined mesh was also examined. In order to facilitate comparison with the values reported by the aforementioned investigators, the thickness of the vorticity layer ($\delta_{\text{vort}}$) and the number of cells across it in the radial direction at two peripheral locations were recorded. The first location is at $10^\circ$ from the front stagnation point and the second at $90^\circ$, which is very close to the separation point. According to [4], the value of $\delta_{\text{vort}}$ is defined as the distance from the wall in the radial direction at which the spanwise vorticity magnitude is equal to 1% of its maximum value. The number of cells within the vorticity layer is shown in Table 1. The corresponding number of cells used by Beaudan and Moin [4] for $Re = 3900$ at $\theta = 10^\circ$ and $90^\circ$ varies from 5 to 14 and 11 to 30, respectively for grids of different densities. Therefore it can be seen that the present resolution is close to the finest one of [4]. The predicted values of $\delta_{\text{vort}}$ along with the values reported at [4] are summarised in Table 2.

The resolution of the separating shear layers was also checked. Values of the thickness of the separating shear layer at $x/D = 0.5$ for Reynolds numbers 4800 and 9500 are roughly identical as reported in [4] and equal to 0.15$D$. Since the examined Reynolds number is lower than 4800, the thickness of the layers is expected to be slightly

![Fig. 2. Cross-section of mesh in $X$–$Y$ plane (a) coarse mesh, (b) fine mesh.](image-url)
The normalised time step size is \( \Delta T = 0.025 \) for all cases. The maximum CFL number is around 4 for the finer mesh and 2.5 for the coarse mesh while the mean is around 0.8 for the finer mesh and 0.35 for the coarse mesh. Around 20 iterations are required for convergence of the equations within each time step to within a prescribed tolerance of \( 10^{-3} \) for the normalised residuals.

A convective boundary condition \([41]\) \( \frac{\partial \phi}{\partial x} + U_{\text{conv}} \frac{\partial \phi}{\partial x} = 0 \) is used for the exit boundary, where \( U_{\text{conv}} \) is the convective velocity normal to the outlet boundary, and \( \phi \) is any physical variable convected out through the outlet. For the flow fields presented in this paper, \( \phi \) stands for the velocity components \( u_i \) in each direction. Zero velocity boundary conditions are used for the top, bottom and cylinder walls. Periodic boundary conditions are applied in the spanwise direction. The normal derivative for the pressure correction is set to zero at all boundaries.

The simulations start with a zero velocity and pressure field. The flow is then allowed to develop for several shedding periods so that all transients are convected out of the exit. Statistics are then collected for seven shedding cycles. The corresponding value reported in [4] varies from 5 to 13, depending on the grid density. Therefore the number of cells used is again comparable, taking into account that the examined Reynolds number is smaller.

The normalised time step size is \( \Delta T = 0.025 \) for all cases. The maximum CFL number is around 4 for the finer mesh and 2.5 for the coarse mesh while the mean is around 0.8 for the finer mesh and 0.35 for the coarse mesh. Around 20 iterations are required for convergence of the equations within each time step to within a prescribed tolerance of \( 10^{-3} \) for the normalised residuals.

A convective boundary condition \([41]\) \( \frac{\partial \phi}{\partial x} + U_{\text{conv}} \frac{\partial \phi}{\partial x} = 0 \) is used for the exit boundary, where \( U_{\text{conv}} \) is the convective velocity normal to the outlet boundary, and \( \phi \) is any physical variable convected out through the outlet. For the flow fields presented in this paper, \( \phi \) stands for the velocity components \( u_i \) in each direction. Zero velocity boundary conditions are used for the top, bottom and cylinder walls. Periodic boundary conditions are applied in the spanwise direction. The normal derivative for the pressure correction is set to zero at all boundaries.

The simulations start with a zero velocity and pressure field. The flow is then allowed to develop for several shedding periods so that all transients are convected out of the exit. Statistics are then collected for seven shedding cycles. The value is close to that employed by other investigators. The number of cells used is again comparable, taking into account that the examined Reynolds number is smaller.

4. Results and discussion

In this section, the results of the simulations will be presented and discussed. Comparisons were made against detailed experimental data for mean and turbulent quantities provided by Konstantinidis [29] for pulsating cross-flow with frequency \( f_e = 21.6 \) Hz, which is close to twice the natural shedding frequency, \( f_n \). The non-dimensional value of \( \Delta u \), which is equal to twice the amplitude of pulsation, was equal to \( \Delta u/\bar{u}_\infty = 0.1 \).

### 4.1. Turbulence statistics in the near wake: impact of grid resolution and subgrid scale model

As shown by Kravchenko and Moin [6], insufficient grid resolution can cause early transition in the shear layers separating from the cylinder which leads to inaccurate predictions of the near-wake statistics. A grid independence study was performed using the coarser mesh for Cases C0 and C1 as well as the finer mesh for Cases B0 and B1, with steady and pulsating flow respectively. The Smagorinsky model was used for all cases. The grid parameters as well as the external flow conditions are shown in Table 1. Figs. 3 and 4 show comparisons with the PIV experimental data of the time-averaged streamwise and cross-stream velocity in the near wake region for Cases B1 and C1. The two meshes give very similar results for all examined locations and for both velocity components. The agreement with experimental data was very similar for the turbulence quantities as well, although supporting graphs are not shown for brevity of presentation. The resolution of the finer mesh is therefore considered adequate for the pulsating flow cases and in the forthcoming sections results are shown only for the finer mesh.

Figs. 5 and 6 show comparison with the experiments of the predicted time-averaged normalised streamwise and cross-wake velocity respectively in the very near cylinder wake obtained using the Smagorinsky as well as the dynamic model. It can be seen that the differences between the two subgrid-scale models are small and both give results which are in very good agreement with the experimental data. Figs. 7 and 8 show the normalized time-averaged streamwise and cross-wake Reynolds stresses respectively. The peaks in the streamwise component that correspond to the two shear layers are well predicted while the peak of the cross-stream component at the centreline is slightly overpredicted at \( x/D = 1 \) and 1.5. Predictions of the shear stress are shown in Fig. 9. In general, both models give good agreement with the experiments for the examined second-order statistics. Some small asymmetries in the time-averaged values are due to the fact that averaging was not performed in the spanwise direction. All such asymmetries are small and are localized to areas where the examined first or second-order quantity has very small magnitude.

Table 2 summarises the main surface and wake data for the finest mesh such as the length of the recirculation zone, angle of separation, thickness of the vorticity layer in the radial direction, Strouhal number and mean drag coeffi-
cient. It can be seen that the predicted length of the recirculation bubble is around 1.1$D$, while for steady approaching flow, the corresponding figure is 1.75$D$ for this Reynolds number, which shows that the external pulsation reduces the size of the recirculation zone. This result is in agreement with the experimental findings of Konstantinidis et al. [26]. The small differences between the two models indicate that for these flow conditions the effect of subgrid scale modelling is not crucial. It is expected that the effect of the subgrid scales will be more pronounced on the velocity spectra, but unfortunately such experimental information is not available to confirm this. Nevertheless, several investigators (for example [4,7,6]) have also reported the relatively small effect of the subgrid scale model for simulations of steady cross-flow over a cylinder at $Re_D = 3900$, which is in the same Reynolds number regime as the one examined in this paper. For pulsating approaching flow

![Diagram](image1)

**Fig. 3.** Time-averaged streamwise velocity of the pulsating cross-flow, (- - -) Case C1; (——) Case B1; (○ ○ ○) Experiment [29]. The dotted line (⋯) represents the zero location of the shifted curves.

![Diagram](image2)

**Fig. 4.** Time-averaged transverse velocity of the pulsating cross-flow, (- - -) Case C1; (——) Case B1; (○ ○ ○) Experiment [29]. The dotted line (⋯) represents the zero location of the shifted curves.

![Diagram](image3)

**Fig. 5.** Normalized streamwise mean velocity behind the cylinder of the pulsating cross-flow. (——) Dynamic model, Case A1; (- - -) Smagorinsky model, Case B1; (○ ○ ○) Experiment. The dotted line (⋯) represents the zero location of the shifted curves.

the effect is even smaller, as will be shown in the next section, where the instantaneous velocity and vorticity patterns are compared between the two flow inlet conditions. The dynamic features of the wake will also provide an explanation for the good predictions of mean and turbulence quantities.
4.2. Instantaneous flow field

Having examined the effect of grid resolution and sub-grid scale and having validated the numerical results through comparison with detailed experimental data, the next step is the study of the dynamic features of the wake. The instantaneous flow field characteristics are examined with the help of pressure traces on selected points on the surface of the cylinder as well as instantaneous velocity vectors and vorticity contour plots. In order to understand the
effect of pulsation, corresponding results are also presented for the case of steady approaching flow.

Fig. 10 shows the variation of instantaneous surface pressure with time, for steady (case B0) and pulsating approaching flow (case B1) at one characteristic point before separation, at 60°. For both graphs, the variation of the lift coefficient is also superimposed, while for the pulsating case, the variation of the external velocity is included as well. The values of pressure and external velocity have been scaled with appropriate factors, as shown in the figures, so as to fit in the same graph with the lift coefficient. Note also that the scale on the vertical axis is different for the two flow cases examined. For the steady approaching flow, the variation of pressure upstream of the separation point is solely due to the vortex shedding process. For the pulsating case, the pressure is modulated with the external frequency. In both flow cases, the pressure is out of phase with the lift coefficient as expected for this angle. The amplitude of the lift coefficient is also affected by the external pulsation, as will be discussed later.

It is not possible to compare the instantaneous velocity and vorticity patterns in the wake of the cylinder for steady and pulsating flow without some sort of common reference signal. For the pulsating flow, the external velocity provides such a reference, but this is absent for the steady case. In order to compare the two flow patterns, the variation of the lift coefficient was used as a reference signal. This coefficient has also been used in the past, for example by Beaudan and Moin [4], in order to calculate the contributions of the random and coherent components of the Reynolds stresses. In the present work, it also provides a useful common reference for comparing the instantaneous flow patterns for steady and pulsating approaching flows. Its variation with time for cases B0 and B1 is shown in Fig. 11. The four phases for which the instantaneous flow variables have been recorded are shown in the same figure as well with letters A–D.

Before comparing the phases one by one, it is useful to examine how the vorticity and pressure fields are related to the lift coefficient. The phase selected for comparison is phase D, which corresponds to a lift coefficient close to a minimum as can be seen from Fig. 11(a). Similar conclusions can be obtained by comparing other phases as well. Fig. 12 presents the instantaneous spanwise isovorticity and isopressure contours for cases B0 and B1. The maximum and minimum values of spanwise vorticity in the cylinder wake for the pulsating case are about twice as large compared with the steady case. Note that for the evaluation of the minimum and maximum values only cells located in the near wake of the cylinder, i.e. within the section shown in the graph, were used. It can be seen that the values of vorticity are higher when the flow is pulsed, which indicates that stronger vortices are shed. The isopressure contours correlate quite well with the isovorticity ones. For both flow cases, the emitted vortices create a low pressure region that affects the pressure field in the lower part of the cylinder. The negative pressure, at the lower part of the cylinder, results in a negative lift coefficient as shown in Fig. 11. Note that the stronger emitted vortices result in minimum pressure values that are also higher (in absolute terms) for the pulsating case. The low pressure area is also located closer to the cylinder and this combination of vortex topology and strength results in smaller values of the lift coefficient compared with the steady approaching flow case.

Fig. 13 present four snapshots in one shedding period for cases B0 and B1. The velocity vectors are colour coded with the local values of the spanwise vorticity in order to facilitate the identification of vorticity in areas where the flow does not recirculate. For all graphs, the minimum and maximum values of vorticity were kept the same in order to facilitate comparison. The graphs show that the wake behind the cylinder at this Reynolds number is extremely complicated, and apart from the dominating von-Karman vortices, many other smaller vortices exist that make a detailed description of the flow and vorticity patterns a very difficult task. Nevertheless, an attempt will be made to describe the broad features of the two flow patterns and pinpoint the major differences between the wakes resulting from steady and pulsating approaching flows. In phase A, which corresponds to high positive values of the lift, a vortex is shed from the top part of the cylinder for

![Figure 10](image1.png)

![Figure 11](image2.png)

Fig. 10. Variation of instantaneous pressure at angle 60° from the front stagnation point with time for (a) Case B0 and (b) Case B1.
Fig. 11. Variation of (a) lift and (b) drag coefficient for Cases B0 and B1.

Fig. 12. Contour plots of local instantaneous (a) spanwise vorticity and (b) pressure for Cases B0 and B1 at phase D.
both cases, but when the approaching flow is pulsating, it is located closer to the cylinder. In phase B, the vortex is convected downstream (thus the lift becomes smaller), but for case B1 the top and bottom shear layers are much shorter and appear to start disintegrating. A rolling-up of the bottom shear layer close to the cylinder is also evident for the same phase. In phase C, the vortex from the top of the cylinder is further convected downstream and is now located close to the right end of the graph, while the process for the formation of the bottom vortex is well under way. Indeed, the shear layer of case B0 forms a vortex rotating counter-clockwise while for case B1 the roll-up of the bottom shear layer has progressed so much that the vortex is about to be released. The close proximity of this strong vortex at the bottom part of the cylinder results in a lift coefficient that is very close to its minimum value. In phase
D, the bottom vortex has been released in both cases. It is this phase that was examined in the previous paragraph where it was found that the vorticity is higher for the pulsating case B1.

From the previous description, it can be noticed that the pulsation of the external approaching flow leads to shorter separating shear layers that fluctuate strongly in the transverse direction and emit stronger vortices that are formed closer to the cylinder surface generating regions of lower pressure. It is gratifying to note that this observation agrees with the result found experimentally by Armstrong et al. [24,25]. This also explains the observed behaviour of the lift coefficient. Fig. 11(b) presents the time variation of the drag coefficient with time. Since the emitted vortices create regions of lower pressure compared to the steady case, the resulting instantaneous drag coefficient is increased. Note also that the flow pulsation results in an increase in the time averaged value of the drag coefficient. Unfortunately, measurements of the time averaged value of the drag coefficients are not available for comparisons, but it is worth noting that this result agrees with the experimental findings of Armstrong et al. [24,25].

The aforementioned flow features can explain the observed behaviour described in the previous section i.e. the smaller size of recirculation zone compared to the steady approaching flow, the effect of the subgrid scale model as well as the quality of LES predictions. At the Reynolds number examined, the separating shear layers are laminar and transition to turbulence takes place further downstream in the wake. Since it is well known that the standard Smagorinsky model is not suitable for transitional flows because it predicts excessive damping of the resolved structures [36], one would expect that this model would not perform well, at least very close to the cylinder. Nevertheless, for the pulsating flow, it was shown earlier (Figs. 7 and 9) that the second-order statistics very close to the cylinder, that are affected by the shear layers, are well predicted with both the standard and the dynamic version of the Smagorinsky model. This is most likely due to the fact that the separating shear layers are excited by the external pulsating flow and become turbulent faster compared to the steady approaching flow. This is supported by Fig. 13 that shows that there is significant large scale fluctuation of the shear layers close to the cylinder. Apart from that, as the vortices become stronger, the contribution of the coherent component of the Reynolds stress increases and thus the effect of small scale fluctuations is reduced. It is for this reason that the LES predictions that resolve well the strong vortices agree well with the measurements even for the second-order turbulent statistics and the results reveal that the subgrid scale model is not critical for the quality of the predictions.

Attention is now turned to velocity spectra. Fig. 14 shows the streamwise velocity spectra for case B0 and B1 obtained from the velocity signal at the point (x/D = 0.86 and y/D = 0.54), which is located at the top separating shear layer. A clear peak at 11.47 Hz, that corresponds to Strouhal number 0.23, is observed. When the external flow is pulsed at 21.6 Hz, i.e. with a frequency which is close (but not equal) to double the natural shedding frequency (which is 22.94 Hz), the peak is moved to 10.87 Hz (Strouhal number 0.216). This is exactly half of the external frequency i.e. the wake is locked to the external perturbation. Note also that the amplitude is increased, which suggests that more energy is carried now by the emitted vortices and the flow has become more coherent, a result which agrees with the previous analysis of the instantaneous flow patterns. Fig. 15 shows that the transverse

![Fig. 14. Streamwise velocity spectra for (a) Case B0 and (b) Case B1 in the location x = 0.86D, y = 0.54D.](image-url)
velocity spectra for case B0 and B1 obtained from the velocity time series at the same location in the shear layer. The same behaviour is apparent for the $v$ component as well, the difference being that for this component the pulsation leads to a much higher increase in the amplitude. This is consistent with the strong fluctuations that appear in the transverse direction in the instantaneous velocity and vorticity plots (Fig. 13) when the flow is pulsed.

The velocity autocorrelation plots confirm the above findings. Fig. 16(a) presents the autocorrelation of the streamwise velocity at the same location ($x/D = 0.86$ and $y/D = 0.54$). The $u$ velocity becomes periodically correlated with peaks for case B1 higher than case B0, suggesting stronger periodic coherence for this component. The period of the autocorrelation function of the locked-on case is slightly longer due to the small shift of the vortex shedding frequency shown in Fig. 14. Fig. 16(b) shows the autocorrelation of the transverse velocity. The peaks from the case B1 are now much higher than the ones from case B0. The dominant vortex shedding frequency shown in the figure corresponds to the peak shown in Fig. 15. No significant peaks in the autocorrelation function of the transverse velocity are detected in case B0, indicating that the shear layer fluctuation is small in the transverse direction.

5. Concluding remarks

The turbulent cross-flow past a circular cylinder at a Reynolds number $Re_D = 2580$ was simulated and the effect of external flow pulsation on the wake characteristics was examined. The LES predictions for the mean velocities and turbulence quantities agreed quite well with PIV measurements for pulsating cross-flow at a frequency close to twice the natural shedding frequency and $\Delta u/u_\infty = 0.1$. It
was found that the cross-stream component is more strongly affected due to pulsation compared to the streamwise component in the separating shear layer. The pulsations lead to stronger vortices that are formed closer to the rear of the cylinder which are associated with smaller pressure. This vorticity pattern leads to shorter recirculation bubble, increased mean and rms drag coefficient as well as increased rms lift coefficient. The wake characteristics provide a good explanation for the good performance of the LES approach for this kind of flow.

Acknowledgment

The authors gratefully acknowledge financial support by EPSRC (GR/R04256/01). Chunlei Liang also acknowledges support by an ORS award.

References