Direct Numerical Simulation of Flow around a Surface-Mounted Square-Section Cylinder with AR=4

Mohammad Saeedi\textsuperscript{1}, Philip LePoudre\textsuperscript{2} and Bing-Chen Wang\textsuperscript{3}

\textsuperscript{1,3}Dept. of Mechanical Engineering, Univ. of Manitoba, Winnipeg, MB, R3T 5V6, Canada
Email: umsaeedi@cc.umanitoba.ca

\textsuperscript{2}Dept. of Mechanical Engineering, Univ. of Saskatchewan, Saskatoon, SK, S7N 5A9, Canada

ABSTRACT

Turbulent flow over a wall mounted square cylinder with aspect ratio 4 was studied using Direct Numerical Simulation (DNS) based on an in house computer code. Second order energy conservative finite difference schemes were used for discretizing the governing equations. All computations were accomplished using a 220-core computer cluster over more than 80,000 CPU hours. The first- and second-order turbulence statistics were thoroughly analyzed and compared against the experimental data. Turbulent flow structures including lateral and tip vortex shedding at different frequencies and coherent structures downstream the obstacle were investigated. The recirculation bubble behind the cylinder was also studied, and it was observed that the boundary of the recirculation region extends streamwise as the elevation decreases due to the downwash flow induced by the tip vortices from the free cylinder end.

1. INTRODUCTION

Direct numerical simulation (DNS) of turbulent flow and structures dynamically evolving around a square cylinder submerged in a boundary layer represents an interesting and challenging topic in computational fluid dynamics. Owing to the need to resolve fine-scale viscous motions in the vicinity of all solid surfaces and to precisely calculate the dynamic interactions between the tip and Karman vortices, the computational cost involved in a DNS procedure is typically demanding which makes the use of parallel processing necessary. As the aspect ratio (AR) of the square cylinder increases, the vortical motion of the flow becomes more energetic and complex, which demands a drastic increase in the size of the computational domain in order to properly capture the temporal and spatial evolution of all energy containing eddies. In the remainder of this section, previous contributions through experimental and numerical approaches will be briefly reviewed.

1.1 Previous Progresses in Experimental Studies

The flow physics characteristic of a surface-mounted cylinder are of significant practical value and have been extensively studied in literature. For instance, the air flow around buildings is of keen interests to structural engineers. Thorough understanding of the pattern of the low pressure vortices induced by the building structure is crucial in the design in order to avoid any resonance between the vortex shedding frequency and natural frequency of the building. Okajima [1] investigated the velocity field and Strouhal number of rectangular cylinder flows. He studied a wide range of Reynolds number varying from 70 to $2 \times 10^4$ in a water channel and a wind tunnel. During his experiments, Okajima observed that there existed a special range of Reynolds numbers and AR values, within which the flow becomes unstable and changes from laminar to turbulent patterns accompanied with a discontinuity in the Strouhal number. Martinuzzi et al. [2] studied the dynamics of bluff bodies mounted in thin and thick boundary layers based on the hotwire measurements. They observed that for both boundary layers, the normalized ground plane pressure distribution in the wake region may be scaled using an attachment length measured from upstream origin of the separated shear layer to the reattachment point of the ground point. Based on their particle image velocimeter (PIV) measurements, Borgeois et al. [3] investigated the effects of the aspect ratio and shape of the cylinder cross sections on the quasi periodical
vortical flow structures in the wake region of the obstacle. Araugo et al. [4] studied the effects of asymmetrically mounted trip wires on the flow behind circular cylinder of different aspect ratios. It was observed that the presence of the tripped wire would cause delay in boundary layer separation and thus alter the wake structures. Lim et al. [5] experimentally investigated the effect of Reynolds number on turbulent flow characteristics over a cubical bluff body. They found out that the mean flow quantities are independent of Reynolds number while the fluctuating quantities are Reynolds number dependent and the correction of wind tunnel measurement are needed if turbulent fluctuations are of interest. Wang et al. [6] conducted a couple of experiments using hot-wire anemometer and particle image velocimeter to analyze the wake region of square cylinder with different aspect ratios. They revealed three types of vortices, i.e. tip, base and span-wise, contributing in the wake flow characteristics. Martinuzzi et al. [7] in another experiment investigated the vortex shedding from two surface mounted cubes in a tandem configuration using hot-wire anemometer. They revealed that the Strouhal number based on the distance between obstacles is geometrically locked and the shedding frequency decreases as the distance between cubes is increased.

1.2 Previous Progresses in Numerical Simulations

Besides the achievements in experimental studies reviewed above, the past two decades have witnessed a significant progress in numerical simulation of the surface-mounted bluff body flows in the context of turbulent boundary layers. Shah et al. [8] studied the flow over a wall-mounted cube of aspect ratio one at high Reynolds number using large eddy simulation (LES) approach. Their main goal was to show the capability of large eddy simulation to solve complex 3-D flows and good agreement of turbulent statistics from numerical simulation and experiment was reported. Yakhot et al. [9,10] studied the flow around a wall-mounted cube in a fully developed channel flow using immersed boundary method and direct numerical simulation. They showed the unstable interaction between two side horse-shoe vortices and one arch-type vortex behind the cube. Villalba et al. [11] conducted a thorough LES study of the flow over a modeled natural obstacle. They studied the separated flow over an axisymmetric hill at Reynolds number 130000 based on the hill height and free stream velocity. The hill was subjected to growing boundary layer whose thickness was half of the hill height. It was shown to be streamwise acceleration combined with transverse circulation in the leeward of the hill. Lee et al. [12] investigated the 3-D turbulent boundary layer roughened with a staggered array of cubes and compared their results with boundary layer flow over 2-D rod roughened wall. It was shown in their study that the friction velocity over wall with 3-D roughness was smaller than 2-D and the downward shift of logarithmic law was greater in 2-D roughness flow. Schmidt et al. [13] studied the capability of different turbulence models to resolve the complex features of 3-D flow over wall mounted cubes and demonstrated the capability of detached eddy simulation to resolve the dominant flow patterns.

In the present research, turbulent flow over a wall mounted square section cylinder of aspect ratio 4 will be directly simulated and turbulence statistics are compared with experimental results of Martinuzzi et al. as the challenge problem of CFD 2012.

2. Numerical Simulation and Boundary Conditions

The present work aims at solving the challenge problem prescribed by the CFD 2012 conference using a rigorous DNS approach. The turbulent flow around a surface-mounted square-section obstacle of aspect ratio 4 will be investigated using DNS. The first- and second-order moments of the velocity and pressure fields will be thoroughly validated using the experimental data provided by the CFD 2012 conference.

Numerical simulations have been performed using a three-dimensional (3D) in-house parallel code developed using the FORTRAN 90/95 programing language and message passing interface (MPI). A second-order energy conservative finite difference method based on a staggered grid system was applied to the discretization of the governing equations. A fully implicit four-level fractional step method coupled with a second-order Crank-Nicolson scheme was used to advance the velocity field over a single time step. In the following context, the numerical algorithm is briefly described in a semi-discretized form.

\[
\frac{u_i^n - u_i^*}{\Delta t} + \frac{1}{4} \frac{\partial (u_j^n + u_j^*) (u_i^n + u_i^*)}{\partial x_j} = \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\nu}{2} \frac{\partial^2 (u_i^n + u_i^*)}{\partial x_j \partial x_j}
\]

(1)

\[
\frac{u_i^* - u_i^*}{\Delta t} = \frac{1}{2\rho} \frac{\partial p^n}{\partial x_i}
\]

(2)
Here, \( u^n \) and \( u^{n+1} \) are velocity components at the previous and current time steps respectively, \( u^* \) and \( u'^* \) are two intermediate velocity components, and \( p^n \) and \( p^{n+1} \) are also old and new pressures, respectively. In the first step, an intermediate velocity based on the pressure of previous time step, and then in the second step, it is further modified to the second intermediate velocity by removing half the old pressure. In the third step, the Poisson equation is solved to obtain the new pressure field. Finally, half the new pressure is used to update the velocity field.

In order to conduct DNS, all scales of flow motions (ranging from the smallest Kolmogorov scales to the largest energy containing eddy size) need to be calculated. This requirement dictates the dimension of the domain to be large enough to resolve the largest eddies of the flow (which is of the order of the obstacle size in this case). In view of this, a relatively large computational domain is chosen for conducting the simulations. The obstacle is located along the central streamline of the domain, 3d away from each side boundary of the domain in spanwise direction, 5d below the free surface of the flow in the vertical direction, 4d downstream of inlet and 18d upstream of the outlet. In order to resolve the small scale motions of the flow near the solid surfaces, the wall coordinate value for \( x^* \), \( y^* \), \( z^* \) for the first node off all solid surfaces is kept less than one.

The schematic of the domain, surface-mounted cylinder and coordinate system are shown in Fig. 1. At the inlet of the flow domain, the measured data of the mean velocity and turbulence kinetic energy are used as boundary conditions. At the outlet, zero gradient boundary condition is used. Periodical boundary conditions are applied to the spanwise direction. No slip boundary condition is applied to all solid surfaces. The domain is discretized using 16.9 million computational cells (440x200x192 in \( x \), \( y \) and \( z \) directions respectively). In order to perform parallel computing, the computational domain is divided into 11x5x4 sub-domains (in \( x \), \( z \) and \( y \) directions respectively). Correspondingly, 220 processors are used to solve the flow field at each time step. All the computations are performed using the 252-core cluster system of CFD lab (based on Intel Xeon X5650 2.66 GHz chip technology) at the University of Manitoba. The total CPU-hours spent to solve the flow field is over 80,000 CPU hours.

3. RESULT AND DISCUSSION

In this section, some interesting qualitative and quantitative results obtained from numerical simulation will be analyzed including the instantaneous pressure contours, time-averaged streamlines, contours of Q criterion, and first- and second-order turbulence statistics.

3.1 Qualitative Results

Since the geometry and flow conditions are all symmetrical with respect to the central \( x \) axis, the mean flow field will be symmetrical too. In order to obtain a statistically stationary mean flow field, the sampling time for collection of turbulence statistics should be long enough to take into account of all turbulent scales. In the present work, the sampling time is around 12 shedding period reported by Martinuzzi et al. [3]. Figure 2 shows the time-averaged streamlines and streamwise velocity (U) contours (top view) in the \( x-z \) plane at elevation \( y/d=3 \). As shown in this figure, the time averaged velocity field is symmetric with respect to the central streamline, which is an expected behavior. There are two large counter rotating vortices in the rear region of the obstacle and two small vortices adjacent to the side walls.

Figure 3 shows the instantaneous pressure field in the \( x-y \) plane located at the middle of the domain. This figure clearly shows the tip vortices generated on the top of the obstacle (free end) can entrain the wake region where the Karman vortex street and recirculation flow pattern dominate. The complex interaction between the tip and Karman vortices imposes special challenges to DNS, especially when the AR value is large. This is because the interaction between these two types of vortices increases significantly as the AR value increases. In terms of pressure regions, three distinct regions are visible: firstly, the stagnation region which has the highest
static pressure and is formed right in front of the obstacle; secondly, the wake region into which the tip vortices are shed, is located immediately behind the obstacle and features the lowest pressure of the flow field; and finally, the outer region which is located above the recirculation region has a pressure almost identical to free stream pressure. Because of the presence of these three regions with different pressures, suction of fluid flow from high pressure to low pressure regions causes a net drag force acting on the obstacle and produces complex flow pattern around the obstacle.

Figures 4a and 4b show the pressure field at two instantaneous moments. The vortex shedding phenomenon is evident in these figures. The low pressure vortices shed into the wake region at the frequency of 125 Hz. Corresponding to this characteristic frequency, the Strouhal number (based on the inlet velocity and cylinder diameter) is around 0.106, which is in agreement with the reported result of Martinuzzi et al. [3].

Figure 5a and 5b show the spatial evolution of the coherent structures based on the so-called "Q criterion" in x-z plane located at y/d=3 and also in the whole three dimensional domain. The Q criterion defined in equation (5) is a useful tool for studying the coherent structures in turbulent flows especially from a qualitative point of view. This criterion identifies vortices as flow regions with positive second invariant of velocity gradient. It is actually a local measure of the excess rotation rate relative to the strain rate [15].

\[ Q = 0.5(r_{ij}r_{ij} - s_{ij}s_{ij}) \]  

where \( r_{ij} = 0.5\left(\frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i}\right) \) and \( s_{ij} = 0.5\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \) are rotation and strain rate tensors respectively, and \( u_i \) represents instantaneous velocity field. These contour plots show extension of coherent turbulent structures downstream of the cylinder in all directions which is a consequence of interaction of tip vortices from the cylinder free end, Karman vortices shed from side walls and upwash hairpin structures from the ground on which the cylinder is mounted.

### 3.2 Quantitative Results

Figures 6a and 6b show the mean streamwise and spanwise velocity profiles (non-dimensionalized using free stream velocity) at 1.45d behind the cylinder (x/d=1.95) in comparison with the experimental results of Martinuzzi et al. As evident in these figures, numerical and experimental profiles are in good agreement with each other. For the mean streamwise velocity (U) profile, outside the range -1<z/d<1, the mean streamwise velocity is slightly overpredicted by numerical simulation. This over prediction is likely due to the limited domain size in the spanwise direction, which should be expanded to be large enough to simulate the boundary layer wind tunnel dimension. However, this step is not feasible at this moment because of computational limitations.

In the wake region behind the cylinder, the streamwise velocity is negative and has a very small absolute value which reflects the recirculation effect. As shown in Figs. 6a and 6b, both the streamwise and spanwise profiles are symmetrical about the central streamline (z/d = 0), and DNS predictions are in agreement with the experimental data. In fig. 6b, the mean spanwise velocity is symmetrical about the central streamline (z/d = 0), and the existence of two
positive and negative peaks are evident. The main peaks show the deviation of streamlines from straight path in the side regions of the obstacle and the small peaks show the presence of counter rotating vortices in the rear region. The small peaks are one order of magnitude smaller than the large peaks which are consistent with the relatively low level of the velocity magnitude in the wake region.

Figures 7a and 7b show the RMS values of the streamwise and spanwise velocities behind the cylinder. In general the numerical results are in good agreement with the experimental data. The RMS values normalized by bulk mean velocity also show turbulence intensity distribution behind the cylinder. The locations corresponding to the highest turbulence intensity for both streamwise and spanwise velocities are at $z/d=1$ and $z/d= -1$ (0.5d away from the cylinder side walls), and the maximum RMS value is around 0.3 for the streamwise and 0.25 for the spanwise components.

Figure 8 shows the mean Reynolds stress distribution normalized by square of the free stream velocity at $x/d=1.95$ and $y/d=3$. As expected, the profile is symmetrical about the central streamline of the domain ($z/d = 0$). Excellent agreement between numerical and experimental graphs is observed in Fig. 8. The maximum Reynolds stress occurs at the same location of maximum turbulence intensity where the shear production is the strongest due to boundary-layer development and separation from both side walls (associated with Karman vortices).
Fig. 6: Time-averaged streamwise and spanwise velocity profile behind the obstacle at \(x/d=1.95\) and \(y/d=3\).

Fig. 7: RMS values of streamwise and spanwise velocities at \(x/d=1.95\) and \(y/d=3\).

Fig. 8: Profile of the mean Reynolds stress component (\(uv/u^2\)) at \(x/d=1.95\) and \(y/d=3\).

In figures 9, 10 and 11, the spanwise streamwise velocity profiles at different locations downstream of the obstacle are shown which are for \(y/d = 1, 2\) and \(3\) respectively. By analyzing these figures, the boundary of recirculation region behind the obstacle can be identified. As previously shown in Fig. 2, there is a recirculation bubble in the rear region of the obstacle in which two large counter rotating vortices are present. Inside the recirculation bubble and along the central streamline \(z/d=0\), there is a reverse flow with negative values of streamwise velocity. Further away from the cylinder, the magnitude of the reverse velocity is decreased, and right at the boundary of recirculation bubble, the value of streamwise velocity at \(z/d=0\) is zero. This boundary can be different for different elevations. For \(y/d=1\), as shown in fig. 9, the value of streamwise velocity is zero at \(z/d=0\) and \(x/d = 3.5\), which implies that the boundary of the recirculation bubble is at \(x/d=3.5\) at elevation \(y/d = 1\).
As shown in Figs. 10 and 11, for elevations \( y/d = 2 \) and \( y/d = 3 \), this boundary is located at \( x/d = 3.2 \) and \( x/d = 2.3 \), respectively. As evident in the trend of recirculation bubble size at different elevations, it has a larger streamwise size at lower elevations and smaller streamwise size at higher elevations. At zero elevation, this boundary which is the location of reattachment point is at \( x/d = 3.8 \). This trend can be interpreted as the effect of tip vortices shed from the free end of the cylinder and being extended along with the downwash flow. The above analysis is consistent with the instantaneous pressure contour shown previously in Fig. 5.

Figure 12 schematically shows the recirculation boundary behind the obstacle as calculated from previous figures at different elevations. As mentioned before, the boundary of recirculation region is extended from higher elevation to lower elevation through downwash flow mechanism.

4. CONCLUSION

Direct Numerical Simulation of turbulent flow over a wall mounted square cylinder of aspect ratio 4 has been performed based on an in-house computer code developed using the FORTRAN 90/95 programing language and MPI library for parallel computing. 220 processors were employed and more than 80,000 CPU hours were spent to perform the unsteady numerical simulation. Turbulence statistics obtained from the DNS have been carefully compared against the experimental data of Martinuzzi et al. (provided by CFD2012 conference as a challenge case dataset). Our comparative study includes a thorough examination of the mean velocity profiles, Reynolds shear stress distribution, vortex shedding frequency, and the characteristics of the recirculation region.

The spatial extension of the recirculation bubble has been carefully investigated. It was observed that the streamwise length of the recirculation region increases as the elevation decreases. It was also observed in the simulation that at elevation \( y/d = 1 \),...
the boundary of the recirculation bubble is located at $x/d = 3.5$. At the floor level, the reattachment point is located at $x/d = 3.8$. Von-Karman and tip vortices were well captured and the Strouhal number of the flow based on the lateral vortex shedding frequency was 0.106 in the DNS, which is in good agreement with the experimental result.

REFERENCES