Reproducibility of flow past two-dimensional rectangular cylinders in a homogeneous turbulent flow by LES

H. Nodaa,⁎, A. Nakayamab

a Technical Research Institute, Mitsui Construction Co., Ltd., 518-1 Komaki Nagareyama, 270-0132 Chiba, Japan
b Graduate School of Science and Technology, Kobe University, Japan

Abstract

A large-eddy simulation technique has been applied for prediction of flow past rectangular cylinders influenced by free-stream turbulence. A stochastic method was used to generate homogeneous turbulence with specified power and cross spectra in the oncoming flow. It has been found that the buffeting effects due to the inflow turbulence on front surface of cylinders are predicted well. In case of a cylinder of small length-to-height ratio \( B/D = 1.0 \), just Strouhal number increases slightly and for large \( B/D = 2.5 \), enhanced turbulent mixing causes reattachment of separated shear flow on the cylinder side. These are in good agreement with the results of previous experimental studies, and the present method appears to be sufficient for practical applications.

Keywords: Large eddy simulation; Rectangular cylinder; Inflow turbulence

1. Introduction

Though many computational fluid dynamics (CFD) studies of flows past bluff bodies have been made, most of them are on flows past bluff bodies in uniform smooth flow. Actual structures like buildings and bridges, on the other hand, are in natural wind that is turbulent and is not uniform. Characteristics of wind load on structures in smooth flow and in turbulent flow are quite different. Therefore, in
investigating flows around structures, simulations need to be carried out with turbulent inflow.

Turbulence in the free stream may be generated by a stochastic method of generating random process or by conducting actual flow calculation over some distance upstream of the body to let it develop to turbulence with desired properties. The latter method was tried by Nozu et al. [1] and Nozawa et al. [2] but it requires quite a large grid system that is not very economical. There are two types that may be classified as the stochastic methods. One is the spectral representation method for which general description is given by Hoshiya [3] and application to turbulence simulation is shown by Shinozuka et al. [4]. The other is the time-series method [5]. The spectral method has been applied to the generation of inflow for turbulence simulation and has been tried by Kondo et al. [6] and Maruyama et al. [7]. Use of this method for simulations of inflow turbulence, however, is relatively new and the method is not quite established. While time-series methods are computationally more efficient, they can be numerically unstable.

In the present work, we first show an improved stochastic method of generating homogeneous turbulence with a prescribed spectrum. Then a large-eddy simulation (LES) is carried out for flow past a square and rectangular cylinders with and without the free-stream turbulence to examine if the LES can reproduce the free-stream turbulence effects properly. The simulated results are compared with existing experimental results including those obtained by the present authors [8].

2. Calculation method

The numerical aspects of the LES technique used here are more or less standard and have been verified in the simulation of smooth flow past a square cylinder [9–10]. It is not of high resolution to resolve the laminar sublayer near the body and uses the standard Smagorinsky model but the results are found to reproduce the effects of the free-stream turbulence very well both qualitatively and quantitatively.

Flow field to be simulated is that around a rectangular cylinder of length \(B\) and height \(D\) placed in a uniform flow of \(U_{in}\) as shown in Fig. 1. The coordinates and the boundary conditions are also shown. The cross section of the cylinder is either square (\(B/D = 1.0\)) or rectangular (\(B/D = 2.5\)). The inflow at the far left plane is generated by the method described in the next section. For the sub-grid stress, we use the conventional Smagorinsky model with the values of the constant \(C_S = 0.13\).

LES method used here is summarized in Table 1. Calculations were done for the four cases shown in Table 2. These cases correspond to the experiments conducted by the present authors [8] and the Reynolds number \(Re\) based on the mean velocity of the inflow \(U_{in}\) and the height \(D\) of the cylinder is 50,000.
Table 1

Summary of computational method

<table>
<thead>
<tr>
<th>Spatial difference scheme</th>
<th>Conv. terms: 3rd-order upwind Visc. &amp; turb. terms: 2nd-order diff.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Time advancing</td>
<td>Conv. Terms: Adams–Bashforth v. &amp; turb. terms: implicit Euler</td>
</tr>
<tr>
<td>Pressure coupling</td>
<td>HSMAC</td>
</tr>
<tr>
<td>Sub-grid scale model</td>
<td>Standard Smagorinsky ($C_s = 0.13$)</td>
</tr>
<tr>
<td>Calculation domain</td>
<td>$B/D = 1.0:26.4D \times 21.3 \times 4D$</td>
</tr>
<tr>
<td></td>
<td>$B/D = 2.5:27.9D \times 21.3 \times 4D$</td>
</tr>
<tr>
<td>Grid size</td>
<td>$B/D = 1.0:200 \times 117 \times 21$</td>
</tr>
<tr>
<td></td>
<td>$B/D = 2.5:230 \times 117 \times 21$</td>
</tr>
<tr>
<td>Minimum grid spacing</td>
<td>$\Delta x = 0.05D, \quad \Delta y = 0.05D, \quad \Delta z = 0.05D,$</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>$Re = 50,000$</td>
</tr>
<tr>
<td>Time step</td>
<td>$\Delta t = 0.01$</td>
</tr>
<tr>
<td>Averaging time</td>
<td>$T = 100$</td>
</tr>
</tbody>
</table>

Table 2

Computational case

<table>
<thead>
<tr>
<th>Case</th>
<th>$B/D$</th>
<th>Inflow turbulence</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td><strong>Intensity</strong></td>
</tr>
<tr>
<td>1</td>
<td>1.0</td>
<td>Non-turbulent</td>
</tr>
<tr>
<td>2</td>
<td>1.0</td>
<td>5%</td>
</tr>
<tr>
<td>3</td>
<td>2.5</td>
<td>Non-turbulent</td>
</tr>
<tr>
<td>4</td>
<td>2.5</td>
<td>5%</td>
</tr>
</tbody>
</table>

Fig. 1. Definition of flow field.
3. Generation of inflow turbulence

Stochastic method [3] was used to generate homogeneous inflow turbulence with specified power and cross-spectral density in the oncoming flow. Similar method has been used in various engineering applications including seismic vibrations. Use of this method for simulations of inflow turbulence, however, is relatively new and the method is not quite established. There are still some problems to be solved. We show here appropriate modification to alleviate these problems.

In the basic method of Hoshiya [3], \( m \) random processes \( X_i(t) \) that are correlated are given by a linear combination of trigonometric functions with random amplitudes and phase angles

\[
X_i(t) = \sum_{p=1}^{i} \sum_{k=1}^{N} [a_{ip}(k) \cos(\omega_k t - \alpha_{ip}(k)) + b_{ip}(k) \sin(\omega_k t - \alpha_{ip}(k))],
\]

(1)

where \( i = 1, 2, \ldots, m \), \( a_{ip}(k) \) and \( b_{ip}(k) \) are the Gaussian random variables with following properties:

1. \( a_{ip}(k) \) and \( b_{ip}(l) \) are always independent except for the case \( i = j, p = q \) and \( k = l \), in which case they take the identical probability distribution,
2. \( a_{ip}(k) \) and \( a_{jq}(l) \) or \( b_{ip}(k) \) and \( b_{jq}(l) \) are correlated only when \( p = q \) and \( k = l \), with the covariance given by

\[
E[a_{ip}(k)a_{jp}(k)] = E[b_{ip}(k)b_{jp}(k)] = 2|H_{ip}(\omega_k)||H_{jp}(\omega_k)|\Delta\omega,
\]

(2)

where \( \Delta\omega = (\omega_u - \omega_l)/N \) and \( \omega_k = \omega_l + (k - \frac{1}{2})\Delta\omega \), \( k = 1, 2, \ldots, N \). \( H_{ip}(\omega_k) \) is related to the cross spectra \( S_{ij}(\omega_k) \) of \( X_i(t) \) and \( X_j(t) \) by

\[
S(\omega) = H(\omega)H^T(\omega),
\]

(3)

where \( S(\omega) \) is the matrix with \( S_{ij}(\omega) \) as its elements, \( H(\omega) \) is the matrix defined by

\[
H(\omega) = \begin{pmatrix}
H_{11}(\omega) & 0 \\
H_{21}(\omega) & H_{22}(\omega) \\
\vdots & \ddots & \ddots \\
H_{m1}(\omega) & H_{m2}(\omega) & \cdots & H_{mm}(\omega)
\end{pmatrix}
\]

(4)

and \( H^T(\omega) \) is the Hermitian transpose of \( H(\omega) \).

3. \( \alpha_{ip}(k) \) is the phase angle and is given by

\[
\alpha_{ip}(k) = \tan^{-1} \left\{ \frac{\text{Im} H_{ip}(\omega)}{\text{Re} H_{ip}(\omega)} \right\}.
\]

(5)

In using this technique to generate the fluctuating velocity components \( (u, v, w) \) at \( N \) spatial points, we assume that the power spectrum is of the Karman type.
as follows:

\[ S_{ui}(n) = S_{vi}(n) = S_{wi}(n) = \frac{4\sigma^2 L_x / U_i}{\{1 + 70.8(nL_x / U_i)^2\}^{3/6}} \]  

(6)

where \( L_x \) is the integral length scale, \( \sigma^2 \) is the variance, \( U_i \) is the mean velocity at point \( i \). Since we like to generate isotropic turbulence, the cross spectra \( S_{ij} \) between \( u \) at point \( i \) and \( v \) at point \( j \), \( v \) at \( i \) and \( w \) at \( j \) and \( w \) at \( i \) and \( u \) at \( j \) are all set to zero as follows:

\[ S_{ui,vj}(n) = S_{vi,wj}(n) = S_{wi,uj}(n) = 0, \]  

(7)

The cross spectra between \( u \)'s at points \( i \) and \( j \), \( v \)'s at points \( i \) and \( j \), or \( w \)'s at points \( i \) and \( j \) separated by \( \Delta s \) is assumed to take the form specified by the Davenport type root coherence as

\[ \sqrt{\text{Coh}(n)} = \frac{|S_{ij}|}{\sqrt{S_i \sqrt{S_j}}} = \exp\left\{ -\frac{kn\Delta s}{U} \right\}. \]  

(8)

where \( \Delta s \) is the separation distance between points \( i \) and \( j \), and \( k \) is a constant related to the decay rate. The root coherence between \( u \)'s at points \( i \) and \( j \) is given by setting \( k = 8 \), and \( k = 4 \) for the coherence between \( v \)'s and \( w \)'s, in which case the separation vector and velocity vector are perpendicular to each other. The reason for using Karman-type power spectra and Davenport-type root coherence is that the Karman spectrum is very common in approximating power spectra but formula for coherence is not given. From these given spectra and cross spectra, \( H_{ip}(w) \) is first calculated by solving Eq. (3). Then \( a_{ip}(k) \) and \( b_{ip}(k) \) that are needed in the equation for \( X_i(t) \) are generated by the Monte-Carlo method as

\[ a_{ip}(k) = \sqrt{2\Delta \omega} |H_{ip}(\omega_k)| \zeta_{pk}, \]  

(9)

\[ b_{ip}(k) = \sqrt{2\Delta \omega} |H_{ip}(\omega_k)| \eta_{pk}, \]  

(10)

where \( \zeta_{pk} \) and \( \eta_{pk} \) are the normal Gaussian random variables with zero mean and unit variance.

The inflow velocity must be generated at a huge number of positions and need to be defined at very closely spaced positions compared with other direct applications such as simulation of seismic waves, and it may cause a numerical instability.

In solving Eq. (3) for \( H_{ip}(\omega) \), numerical instability may occur when the cross spectra \( S_{ij} \) becomes very small. This situation occurs when cross correlations are defined very small or the number of generating velocity is huge. This may be avoided if the entire region is segmented into smaller regions and generating them in the first sub-region and then in other regions using the results of the values in the segments already calculated. If the positions where velocity is to be generated, are defined very
closely, numerical instability may occur as well, since the imaginary part of $S_{ij}$ is very small. To avoid this problem, we calculate only the real part of $S_{ij}$ assuming $\gamma_{ij}(k)$ is zero. In case of homogeneous turbulence, this assumption is quite suitable theoretically, and calculation using Eq. (1) is significantly simplified. When we assume $\gamma_{ij}(k)$ is zero, Eq. (1) can be written as

$$X_i(t) = \sum_{k=1}^{N} [A_{ip}(k) \cos \{\omega_k t\} + B_{ip}(k) \sin \{\omega_k t\}],$$

(11)

$$A_{ip}(k) = \sum_{p=1}^{i} a_{ip}(k),$$

(12)

$$B_{ip}(k) = \sum_{p=1}^{i} b_{ip}(k).$$

(13)

The order of generating these random fluctuating velocity components is such that $u, v, w$ defined for the two center columns of the computational grids adjacent to the centerline $y = 0$, are generated first. Then the values along the next two columns on each side are generated. The rest is obtained by repeating this process until all values in the entire $y-z$ plane are generated.

Generated velocity does not necessarily satisfy the continuity equation and momentum equations in three dimensions. Especially, when continuity equation is not satisfied, subsequent flow calculation load becomes enormous. In order to carry out subsequent flow calculation smoothly, the continuity equation must be satisfied at least approximately not only theoretically but also numerically. One way of approximately satisfying the continuity equation is to correct the generated velocity component time-series interpreting time as the streamwise distance using the Taylor hypothesis before starting the flow calculation. The solution algorithm used in the flow calculation may be used for this purpose. For example, if HSMAC method is used in the flow calculation, generated velocity should be made to satisfy the continuity equation using the HSMAC method, and if MAC method is used in the flow calculation, generated velocity should be satisfied by the MAC method before the subsequent flow calculation is started.

Examples of the spectra of velocity computed by the present LES using the generated velocity components as inflow condition after enforcing the continuity equation by HSMAC iteration are shown in Fig. 2. The pressures for inflow condition are also the results of HSMAC. The Karman spectrum that is the target prescribed in the generation is also shown. It is seen that $u$ and $v$ spectra are reproduced very well except in the high-frequency region while $w$ spectrum is generated to be slightly smaller than the target even in the low-frequency range. All spectra are seen to reduce their magnitudes with the streamwise distance from the plane of generation. This is due to the filtering effects of LES calculation. But they appear to settle within acceptable range from the prescribed one.
4. Calculation results and discussion

4.1. Instantaneous vorticity distribution

Typical instantaneous flow field of the calculation results are shown in Fig. 3 in terms of the contours of spanwise component of the vorticity. Fig. 3(a) and (c) are the “smooth flow” results without the free-stream turbulence and Fig. 3(b) and (d) are the “turbulent flow” results with the free-stream turbulence. In the cases of $B/D = 1.0$ (Cases 1 and 2) shown in Fig. 3(a) and (b), the separated shear layers off...
the upstream corners curve around the downstream corners of the cylinder resulting in the beginning of the vortex shedding. This formation of vortex shedding is blurred a little and the location of formation of vortex is shifted closer to the body in the turbulent flow case. Otherwise, the overall pattern of the flow with the free-stream turbulence is almost the same as that without the turbulence.

In the cases of $B/D = 2.5$, however, the flow without the free-stream turbulence shows formation of vortex shedding which is larger than the case of $B/D = 1.0$, while in Case 4 with turbulence, the separated shear layers reattach on the side surfaces and no clear vortex shedding is seen in the wake. The large structures are broken into smaller scale motions. This is in good agreement with the experimental observation.

4.2. Mean velocity profiles on the cylinder side surface

In order to examine more details of the simulated flows, particularly the boundary layers on the side surface, the calculated profiles of the mean streamwise velocity component for the cases of $B/D = 2.5$ are compared in Fig. 4. It is seen that the boundary layer is about the same up to the position $x/D = 0.5$ whether or not there is free-stream turbulence, but it is clearly seen that the boundary layer thickness is significantly thinner in the downstream half of the side surface when there is free-stream turbulence. It is further seen that the boundary layer does reattach at the
most downstream position. The results of the streamwise turbulence intensity at the same positions are shown in Fig. 5. The turbulence level in the boundary layer is strongly influenced by the free-stream turbulence even at the center of the cylinder $x/D = 0.5$ and is seen to increase downstream significantly. This agrees with the experiment of Nakamura and Ohya [11], which indicates that the free-stream turbulence enhances the entrainment of the separated shear flow.

4.3. Aerodynamic parameters

The mean drag coefficient and the RMS fluctuations of the instantaneous drag and lift coefficients are defined by

$$C_D = \frac{\bar{f}_D}{\frac{1}{2} \rho U_{in}^2 D},$$

(14)
where \( \overline{f_D} \) is the mean drag force per unit length of the cylinder and \( f_D' \) and \( f_L' \) are the r.m.s. of fluctuating drag and lift forces, respectively. The values of these parameters from the present simulation calculation and the experiments [8] are summarized in Table 3, together with the base-pressure coefficient \( C_{pb} \) and the Strouhal number \( St \).

The experimental results obtained in the “smooth” flow with the tunnel turbulence intensity of 0.2% are also shown. For the case of \( B/D = 1.0 \), calculated results of the Strouhal number \( St \), the mean drag and lift coefficient \( C_D \), \( C_L' \) and the base pressure coefficient \( C_{pb} \) are all seen to agree with the experimental results fairly well indicating the quality of the basic LES simulation. In the case of \( B/D = 2.5 \), the fluctuation of the drag coefficient \( C'_D \) agrees well with experiment, but the base pressure is predicted smaller and as a result, the mean drag coefficient is larger than the experiment. It is known from the experiments by Nakaguchi et al. [12] that the flow changes sharply near \( B/D = 2.8 \). Also, Okajima [13] showed that for high Reynolds number flow of \( Re > 10^4 \), the vortex shedding has two different frequencies when \( B/D \) is in the range from 2.0 to 3.0. Therefore, the flows of this case sensitively depend on \( B/D \) and the Reynolds number. In the present LES, the reattachment of the separated flow happens to be predicted smaller.

As to the effects of turbulence, the experimental results for the case of \( B/D = 1.0 \) indicate that the free-stream turbulence increases \( St \) slightly, increases \( C_{pb} \), decreases \( C_D \) and \( C_L' \). The calculation reproduces all these trends correctly at least qualitatively. Quantitatively, the pressure recovery is under predicted. The above effects are about the same for the larger \( B/D \) of 2.5, except that the fluctuating drag coefficient increases. These are all calculated correctly except that the fluctuating lift coefficient \( C_L' \) is predicted larger in the case with free-stream turbulence. This is
thought to be due to the widened effective width of the wake caused by the shorter reattachment length.

4.4. Mean and fluctuating pressure distributions

The mean surface-pressure coefficient $\overline{C_p}$ and the RMS pressure fluctuation coefficient $C_p'$ are shown in Fig. 6 together with the experimental results of Noda and Nakayama [8]. First, for the square cylinder case of $B/D = 1.0$, the overall mean pressure distribution is predicted correctly on all surfaces. The recognizable asymmetry indicates insufficient averaging time but it is only over small region near the front corner. As to the pressure fluctuation, the increased fluctuation on the front surface is predicted well both qualitatively and quantitatively. The reduction on both side surfaces is reproduced satisfactorily, but the reduction on the rear surface is not well predicted.

In case of $B/D = 2.5$, the effects of the turbulence on both mean and fluctuating pressure are predicted correctly except for the noticeable increase of fluctuation near the downstream end of the side surface. This is considered to be due to the difference of the reattachment positions of the calculation and experiment since the pressure distributions are very sensitive to the reattachment position. Almost uniform fluctuation distribution on the front surface for the turbulent flow case is seen to be well predicted.
4.5. Spanwise correlation of surface pressure

In order to study the effects of the free-stream turbulence on the spanwise coherence of the vortex-shedding mechanisms, the spanwise pressure correlation coefficient defined by

\[
C_{pp} = \frac{\overline{p_0' p_i'}}{\sqrt{\overline{p_0'^2}} \sqrt{\overline{p_i'^2}}}
\]  

(17)

where \(p_0'\) and \(p_i'\) are the fluctuating pressure at the fixed and point \(i\) has been obtained and plotted in Fig. 7.

In the case of \(B/D = 1.0\), the correlation on the side surface decreases slightly due to the turbulence. This is in agreement with Nakamura and Ohya’s [11] results. In all cases, the trends of the calculated effects of the free-stream turbulence agree with experiments. In other words, the correlation reduces on the side surfaces but increases on the rear surface. The cases of \(B/D = 2.5\) are predicted very well while those of \(B/D = 1.0\) show larger correlation than the experiments. The spanwise length of the calculated region is only \(4D\) which is insufficient for quantitative evaluation beyond \(z/D = 2.0\) but the results up to about \(z/D = 2.0\) may be considered meaningful. The spanwise resolution of the computational grid is \(0.3D\) which is rather coarse and it is the reason that the correlation for small \(z/D\) is predicted generally higher than experiments.

Fig. 7. Spanwise correlation coefficient of surface fluctuation pressure: (a) side surface \((B/D = 1.0)\); (b) rear surface \((B/D = 1.0)\); (c) side surface \((B/D = 2.5)\); (d) rear surface \((B/D = 2.5)\).
5. Conclusions

The present paper presents a method of generating homogeneous turbulence by a stochastic method and the LES calculation results of flows passing square and rectangular cylinders using the generated turbulence at the inflow. New methods to avoid numerical instability and to be able to calculate efficiently are proposed. Calculations conducted with no free-stream turbulence are also presented for comparison. These results are compared with experimental results and the following may be concluded.

The present stochastic method of generating turbulence with prescribed intensity and spectrum is sufficient to be used as the inflow condition for simulation of flows with free-stream turbulence.

The standard LES with standard Smagorinsky subgrid stress model appear to be sufficient for representing the free-stream turbulence effects. Particularly, the enhancement of boundary layer reattachment is well reproduced and the conditions for generation of vortex shedding are well predicted. The effects of the free-stream turbulence on the surface pressure are also well predicted but the magnitude of the pressure on the rear surface is slightly over-predicted in the negative direction and correspondingly the drag coefficient is slightly over-predicted. These slightly differences in the rear-surface pressure are the trend of most LES calculation with more refined subgrid models [9,14]. The present result is an indication that with moderately fine grid, LES with a low-level subgrid model can reproduce properties of turbulent flow around a bluff body with reasonable accuracy for engineering purposes.

References


