LES OF FLOWS AROUND
A CIRCULAR CYLINDER
IN THE CRITICAL REYNOLDS NUMBER REGION

Yoshiyuki ONO*, Tetsuro Tamura†

*Fluid Engineering Department, Technical Research Institute, Obayashi Corporation
4-640 Simokiyoto, Kiyose, Tokyo, Japan
e-mail: ono.yoshiyuki@obayashi.co.jp,
†Department of Environmental Science and Technology, Tokyo Institute of Technology
4259 Nagatsuta, Midori-ku, Yokohama, Japan
e-mail: tamura@depe.titech.ac.jp

Keywords: LES, Circular cylinder, The critical Reynolds-number region, Vortex structure

Abstract. This paper discusses the applicability of the LES method to flow around a circular cylinder in the critical Reynolds number region. There have been many investigations on the applicability of various numerical models to this situation. However, almost all previous computations have shown an ambiguous decrease in the drag coefficient and an insufficient increase in vortex shedding frequency in comparison with the experiment. Furthermore, the details of the complicated flow structure in the critical region have never been numerically captured.

In this research, a sophisticated numerical model is introduced. The flow around a circular cylinder at very high Reynolds number (Re=6x10^5) is investigated using the LES method, focusing particularly on the complicated flow structure near the flow separation region.

1 INTRODUCTION

Recent advancement of numerical techniques has made it possible to easily simulate the flow around a bluff body. Some LES methods have succeeded in simulating large-scale wake structures associated with flow separation on a circular cylinder at sub-critical Reynolds numbers. On the other hand, in the critical Reynolds number region, the flow represents an intricate combination of laminar separation, transition, reattachment and turbulent separation of the boundary layers on the cylinder [1]-[4]. This difference between the flow characteristics in the sub-critical and the critical Reynolds number regions results in a discontinuous drop in drag and a jump in vortex shedding frequency.
Until now, many numerical models have been applied to complicated flow in the critical Reynolds number region. However, previous LES computations show an ambiguous decrease in the drag coefficient and an insufficient increase in vortex shedding frequency in comparison with the experiments, due to too much numerical dissipation and inadequate grid resolution. Furthermore, the details of the complicated flow structure in the critical region, such as the behavior of separation bubbles, have never been numerically captured.

While previous experimental researches have shown that separation bubbles are formed after separation in the critical Reynolds number region. It has also been observed that separation bubbles tend to be broken as the Reynolds number increases in the critical region. The CFD technique is expected to become a powerful tool in investigating the behavior of separation bubbles in the critical Reynolds number region.

The present research introduces a sophisticated numerical model. The flow around a circular cylinder at very high Reynolds number (Re=6x10⁵) is investigated using the LES method, focusing particularly on the behavior of the separation bubble. First, an adequate numerical model is investigated. The influences of (1) numerical dissipation, (2) SGS model and (3) grid resolution on the aerodynamic quantities, pressure coefficient and skin friction are examined. Next, the behavior of the separation bubble is investigated using flow visualization.

2 PROBLEM FORMULATION

The governing equations are the incompressible Navier-Stokes and the continuity equations. In this simulation, the original governing equations are transformed to a curvilinear coordinate system. To advance the solutions of velocities and pressure in time, a fractional step method is employed. The time integral of the momentum equation is hybrid, that is to say, the Crank-Nicolson scheme is applied to the viscous terms and the explicit third-order Runge-Kutta method is used for convective terms. Spatial derivatives of variables are treated as second-order central differences. Convective terms are approximated using the higher-order interpolation method. To avoid numerical oscillation, very slight numerical dissipation is added to convective terms by controlling the numerical dissipation parameter (αND). Thus, the convective terms are approximated as

$$\frac{\partial}{\partial \xi} JUu_i = \delta_x (JUu_i^{\xi}) + \alpha_{ND} J|U| \left[ u_{i-2} - 4u_{i-1} + 6u_i - 4u_{i+1} + u_{i+2} \right]$$

$$\bar{u_i}^{\xi} = \frac{-u_{i-3/2} + 9u_{i-1/2} + 9u_{i+1/2} - u_{i+3/2}}{16}$$

$$\delta_x = \frac{f_{i-3/2} - 27f_{i-1/2} + 27f_{i+1/2} - f_{i+3/2}}{24}$$

$$U = \frac{\partial \xi}{\partial x_j} u_j$$

where coefficients $\xi$ and $J$ denote the metrics and Jacobian of the transformation.

The dynamic Smagorinsky model and the dynamic mixed model are used to study the effect of the SGS model.
3 COMPUTATIONAL MODEL

The over-set grid system is employed to distribute the grid points over the whole domain around a circular cylinder with a balanced size. Figure 1(1),(2) show the computational domain and over-set grid system near the cylinder in the present computations. It is composed of three kinds of meshes. In order to investigate the influence of the grid resolution, computations are carried out using four types of grids. The grid points in each grid are shown in Table 1. Grid 2 and Grid 3 have finer grid resolutions in the azimuth direction and the span-wise direction, respectively. Grid 4 has much finer grid resolution in the span-wise direction, although the length in the span-wise direction is shorter than in the other cases. The smallest grid size near the cylinder surface is $0.1/(\text{Re})^{0.5}$. Standard inflow conditions, $u=1$, $v=w=0$ are imposed at the upstream boundary, and the convective condition is imposed at the downstream boundary. The no-slip condition is used at the cylinder surface. The Reynolds number is equal to 600,000.

The present computations were carried out using a supercomputer SX8R made by NEC.

Figure 1  Overset grid system
Table 1 Computational grid points

<table>
<thead>
<tr>
<th>Grid</th>
<th>Dir</th>
<th>Mesh1</th>
<th>Mesh2</th>
<th>Mesh3</th>
<th>length in span-wise direction</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid1</td>
<td>x(ξ)</td>
<td>101</td>
<td>311</td>
<td>501</td>
<td>2D</td>
</tr>
<tr>
<td></td>
<td>y(η)</td>
<td>91</td>
<td>181</td>
<td>61</td>
<td></td>
</tr>
<tr>
<td></td>
<td>z(ζ)</td>
<td>31</td>
<td>51</td>
<td>51</td>
<td></td>
</tr>
<tr>
<td>Grid2</td>
<td>x(ξ)</td>
<td>101</td>
<td>311</td>
<td>801</td>
<td></td>
</tr>
<tr>
<td></td>
<td>y(η)</td>
<td>91</td>
<td>181</td>
<td>61</td>
<td></td>
</tr>
<tr>
<td></td>
<td>z(ζ)</td>
<td>31</td>
<td>51</td>
<td>51</td>
<td></td>
</tr>
<tr>
<td>Grid3</td>
<td>x(ξ)</td>
<td>101</td>
<td>311</td>
<td>501</td>
<td></td>
</tr>
<tr>
<td></td>
<td>y(η)</td>
<td>91</td>
<td>181</td>
<td>61</td>
<td></td>
</tr>
<tr>
<td></td>
<td>z(ζ)</td>
<td>31</td>
<td>151</td>
<td>151</td>
<td></td>
</tr>
<tr>
<td>Grid4</td>
<td>x(ξ)</td>
<td>101</td>
<td>311</td>
<td>501</td>
<td>D</td>
</tr>
<tr>
<td></td>
<td>y(η)</td>
<td>91</td>
<td>181</td>
<td>61</td>
<td></td>
</tr>
<tr>
<td></td>
<td>z(ζ)</td>
<td>31</td>
<td>151</td>
<td>151</td>
<td></td>
</tr>
</tbody>
</table>

3. COMPUTATIONAL RESULTS

3.1 Influence of numerical conditions

The computed aerodynamic quantities such as the time-averaged drag coefficient ($C_{D_{ave}}$), the r.m.s. value of the fluctuation lift coefficients ($C_{L_{rms}}$) and the Strouhal numbers ($St$) are summarized in Table 2. Because of the effects of the numerical dissipation, calculations including large numerical dissipation ($α_{ND} = 0.2$) produce higher values of time-averaged drag coefficient than the other cases. When using $α_{ND}$ smaller than 0.05, the computations became unstable and overflowed. In the present numerical conditions, the SGS models and the grid resolution in the azimuth direction have a little effect on the aerodynamic characteristics. Furthermore, none of the cases using Grid 1 and Grid 2 accurately predict an increase in the Strouhal number in the critical region [2]-[4]. These discrepancies are similar to those of the previous computations. However, the calculations using finer grid resolution in the span-wise direction (Grid 3 and Grid 4) succeed in simulating an increase in the Strouhal number and a decrease in the time-averaged drag coefficient, observed in the previous experiments [2]-[4]. It is thus clarified that a very fine grid in the span-wise direction is required to capture the flow around a circular cylinder in the critical Reynolds number region.

Decreases in the drag coefficients and increases in the Strouhal number are recognized when using Grid 4, which has much finer grid resolution in the span-wise direction, and in those when using the Dynamic mixed model.

In the next section, the details of the flow and pressure characteristics of a cylinder surface in the critical region are investigated on the basis of the computations using Grid 3 and Grid 4.
Table 2: Computed aerodynamic coefficients

<table>
<thead>
<tr>
<th>Numerical condition</th>
<th>SGS model</th>
<th>(\alpha_{ND})</th>
<th>(C_{D,ave})</th>
<th>(C_{L,rms})</th>
<th>(St)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grids1 DS</td>
<td>0.2</td>
<td>0.46</td>
<td>0.13</td>
<td>0.34</td>
<td></td>
</tr>
<tr>
<td>Grids1 DS</td>
<td>0.1</td>
<td>0.36</td>
<td>0.13</td>
<td>0.30</td>
<td></td>
</tr>
<tr>
<td>Grids1 DS</td>
<td>0.05</td>
<td>0.36</td>
<td>0.13</td>
<td>0.31</td>
<td></td>
</tr>
<tr>
<td>Grids1 DM</td>
<td>0.1</td>
<td>0.33</td>
<td>0.12</td>
<td>0.30</td>
<td></td>
</tr>
<tr>
<td>Grids1 DM</td>
<td>0.05</td>
<td>0.34</td>
<td>0.17</td>
<td>0.29</td>
<td></td>
</tr>
<tr>
<td>Grids2 DS</td>
<td>0.1</td>
<td>0.32</td>
<td>0.12</td>
<td>0.33</td>
<td></td>
</tr>
<tr>
<td>Grids2 DM</td>
<td>0.1</td>
<td>0.33</td>
<td>0.115</td>
<td>0.3</td>
<td></td>
</tr>
<tr>
<td>Grids3 DS</td>
<td>0.1</td>
<td>0.27</td>
<td>0.13</td>
<td>0.4</td>
<td></td>
</tr>
<tr>
<td>Grids3 DM</td>
<td>0.1</td>
<td>0.23</td>
<td>0.07</td>
<td>0.43</td>
<td></td>
</tr>
<tr>
<td>Grids 4 DS</td>
<td>0.1</td>
<td>0.26</td>
<td>0.13</td>
<td>0.43</td>
<td></td>
</tr>
<tr>
<td>Grids 4 DM</td>
<td>0.1</td>
<td>0.21</td>
<td>0.08</td>
<td>0.48</td>
<td></td>
</tr>
<tr>
<td>EXP [2]-[4]</td>
<td>0.2-0.4</td>
<td>0.02-0.13</td>
<td>0.42-0.48</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

3.2 Study of averaged flow

Figure 2 shows the distributions of the time-averaged pressure coefficient on the cylinder. Flat distributions are recognized at around 100 degrees and on the rear side at 140 degrees associated with flow separation and flow reattachment or re-separation. Looking at the upstream flat region, the computed length of the flat region associated with the time-averaged size of separation bubbles shows agreement with the experiments by Tani [4] and Flachsbart [5]. There is little effect of the SGS model and the grid resolution in the spanwise direction on the pressure coefficient on the upstream flat region, although the computations using the dynamic mixed model tend to show slight recoveries of base pressure coefficients.

Figure 3 shows the distributions of skin friction. The computed skin frictions in all cases show negative values at around 90 degrees, and then they slightly increase to around 0. After that, the skin frictions show negative peaks at around 110 degrees and then increase associated with flow reattachment. Looking at the reattachment region at around 115 degrees, the present computations, except when using the dynamic mixed model and Grid 4, can’t accurately capture the characteristics of the separation bubble. This is because the positive values on the rear side of the separation points, which mean complete flow reattachment, are not clearly recognized. Only the case using the dynamic mixed model and Grid 4 shows positive values. In this case, the flow velocity rapidly changes from negative to positive around the reattachment point. It should be noted that the acceleration of the flow velocity leads to a slight increase in the level of negative pressure, as shown in Figure 2.

Comparing the computed skin frictions and the experimental data by Achenbach [6], the peak value in the computation is much lower than the experimental value. Furthermore, the experimental results don’t show negative values associated with formation of separation bubbles. These characteristics of skin friction should be discussed from experimental and computational approaches in the future.
3.3 Effect of grid resolution on instantaneous reattachment

(1) Grid 3 (2D/150points)                  (2) Grid4 (1D/150points)

Figure 4 Velocity vectors at first grid points on cylinder surface
(Red vector shows stream-wise velocity is positive, blue shows negative)
Figure 4(1), (2) show velocity vectors on the cylinder surface when using Grid 3 and Grid 4, respectively. As discussed in 3.2, positive velocities are clearly recognized when using Grid 4. It can also be seen that flow reattachment occurs apart from the flow separation points when using Grid 4. A dead water region is recognized on the rear side of the separation points. On the other hand, in the computations using Grid 3, the reattachment points are close to the flow separation points. The interaction of flow separation and reattachment is observed. It is presumed that this interaction causes the levels of velocities around the separation bubble to be weak and couldn’t lead to positive values of skin friction.

4. Study of instantaneous flow structure in the critical Reynolds number region

Figure 5 shows the contours of stream-wise velocities around a circular cylinder. The very narrow near wake, which results in a decrease in $C_D$ and an increase in $St$, is recognized. The computed velocity vectors near the cylinder are also shown in Figure 6. It can be seen that the flow separates from the cylinder surface and then reattaches. These flow characteristics cause the separation bubble to be instantaneously formed. Looking at the details of the separation bubble, a secondary vortex that has positive velocities is recognized. It should be noted that this secondary vortex causes the skin friction to slightly increase after the flow separation shown in Figure 3.

Figure 6(1) (2) show iso-surfaces of $\omega_z$ pressure around a cylinder and contours of vortices $\omega_z$ close to a cylinder. It can be seen that separation bubbles are randomly distorted in the span-wise direction. That is, the disruption and fragmentation of the separation bubbles are observed in the present computations. According to previous research, 2D separation bubbles are stably formed after separation in the critical Reynolds number region. However, Bearman [2] noted that the two-dimensionality is sensitively disrupted by small protrusions on the cylinder surface such as a pressure tapping hole or dust particles. Furthermore, some flow visualizations showed that separation bubbles tend to be broken as Reynolds number increases in the critical region [4]. The present computation simulates these characteristics of the separation bubble in the critical region. Furthermore, Figure 7 compares the computed streamlines with the oil flow patterns in the experiments by Loiseau et al. [4]. It is confirmed that the computed streamlines show distortion in the span-wise direction as do the oil flow patterns in the previous experiments.

In order to investigate the mechanism of this distortion of the separation bubbles in detail, Figure 8 shows the time averaged and instantaneous velocity vectors near the cylinder. In the time-averaged flow, the reattachment line is parallel to the separation line. That is, a two-dimensional separation bubble is formed. However, looking at the instantaneous flow around the reattachment points, some stagnation points are recognized instead of a re-separation line. These stagnation points become the sources of the flow, and several divergences are recognized. These divergences cause the velocities in steam-wise and span-wise direction to rapidly increase and results in distortion of the separation bubbles.

Figure 9 shows the iso-surface of stream-wise velocity ($U=0.0$) near the separation points. The development of boundary-layer turbulence along the cylinder surface in the rear of the reattachment region, and turbulent separation on the cylinder, are captured in the present computations.
4 CONCLUSIONS

The LES method is applied to the flow around a circular cylinder at high Reynolds number. In order to introduce a sophisticated numerical model, the influences of the numerical dissipation, the SGS model and grid resolution on the computed results are investigated. The complicated flow structure near the flow separation region is also investigated using the LES method.

The following conclusions are obtained.

1. Using very fine grid resolution in the span-wise direction, the LES model succeeded in accurately simulating the aerodynamic and time-averaged pressure coefficient of a circular cylinder. In particular, rapid change in the Strouhal number can be captured by the present numerical model.

2. Based on the comparison of the computations for distribution of skin friction, most computations cannot simulate the positive value associated with flow reattachment. Only a computation using a dynamic mixed model with very fine resolution in the span-wise direction can accurately simulate the characteristics of flow reattachment.

3. In the present computation, it can be seen that the separation bubbles include secondary vortices, which results in a slight increase in skin friction after flow separation.

4. The present computation succeeds in simulating the fragmentation of separation bubbles in the upper critical region observed in previous experiments. The time-averaged reattachment line is parallel to the axis of the cylinder. However, instantaneous reattachment occurs at several stagnation points, where divergent type of the flow can be simultaneously recognized. These flow characteristics cause the velocity to rapidly increase near the reattachment region.

5. In the rear of the reattachment region, development of boundary-layer turbulence along the cylinder surface and turbulent separation on the cylinder are simulated.

REFERENCES


Figure 5 Contours of instantaneous velocity

Figure 6 Velocity vectors near cylinder (XY section)
(Red vectors indicate positive stream-wise velocity, blue indicate negative)

Figure 7 Computed flow near cylinder in critical region
Computations

Oil flow patterns [4]

Figure 8  Stream lines

(enlargement of Figure 4(2))

time-averaged velocities  instantaneous velocities

Figure 9  Velocity vectors on the cilinder

Figure 10 Iso-surface of the stream-wise velocity(u=0)