Investigations on the Flow Fields of Hypersonic Flow Past the Wall-Mounted Square Cylinder with Various Heights

By Dawen XUE, Zhihua CHEN and Xiaohai JIANG

National Key Laboratory of Transient Physics, Nanjing University of Science & Technology, Nanjing, China

(Received July 4th, 2012)

Combined with the large eddy simulation (LES), a 3rd-order WENO scheme and the adaptive mesh refinement (AMR) technique, the hypersonic compressible flow ($Ma = 5$) past wall-mounted square cylinders with different heights is simulated numerically. Our results reveal that with the variation of cylinder heights, the pressure distributions of the flat plane in front of the cylinder are similar and have multiple peaks. The different flow patterns of the flow field around the cylinders are shown and discussed in detail. Our calculated results are in good agreement with corresponding experimental data which can provide important data for related research.

Key Words: Hypersonic Flow, Square Cylinder, Shock Wave, Flow Separation, Vortex

1. Introduction

Rectangular cylinders have been used as one kind of typical convex shape for the design of supersonic and hypersonic space vehicles. Thus, the characteristics of the hypersonic flow past rectangular cylinders and their induced three-dimensional separation phenomena are of importance in determining both the aerodynamic loads and heating in these interactive regions.

During the last five decades, extensive investigations have been conducted on the supersonic and hypersonic flow past bluff bodies and their induced separations. In 1968, Young et al.\(^1\) carried out the experiments of supersonic flow past a blunt fin to investigate the interaction between the shock wave and the boundary layer. They summarized the double peaks of pressure distribution on the front flat of the body based on their measured data. Similar pressure distribution has also been found in the survey of the supersonic flow past the blunt fin with a semi-circular cylinder leading edge by Dolling and Bogdonoff.\(^2\) Later, Asó et al.\(^3\) investigated the supersonic flow characteristics past sharp/blunt fins. Their experimental pressure distribution and the oil flow pattern revealed high unsteadiness in the interaction region, and they succeeded in visualizing the secondary separation in the interaction region using the oil paint method. Li\(^4\) carried out a series of wind tunnel experiments on the supersonic and hypersonic flow past both circular and rectangular cylinders. The experimental results obtained by their schlieren photographs, oil flow patterns and measured pressure distributions showed that the flow field features depend on the cylinder heights. Some recent research even focuses on the flow control at hypersonic speeds.\(^5\)

Although many wind tunnel experiments and flight tests have been carried out, detailed flow structures and separation characteristics have not yet been fully understood. With the development of computational techniques, numerical simulation has become one of the important ways to investigate the flow characteristics. Based on the Reynolds-averaged Navier-Stokes (RANS) equations, Hung et al.\(^6\) simulated supersonic flow past circular and rectangular cylinders using the MacCormack scheme, and their calculated results are in good agreement with corresponding experimental data, but the scheme is inadequate for capturing shock. In the same way, based on RANS, Chaussee\(^7\) simulated the viscous supersonic/hypersonic flow over blunt cones, space shuttle orbiters and a wing-fuselage-nacelle successively. The calculated pressure distribution and lift coefficient are in good agreement with corresponding experimental data. Ma et al.\(^8\) also computed the RANS equations using the FVM method and Roe scheme to investigate the shock/boundary layer interaction induced by circular cylinders with various heights. The pressure distribution of the flat plane in front of the cylinder and the shock wave structures are in good agreement with experimental data. Jagadeesh et al.\(^9\) investigated the flow fields of a large-angle, spiked blunt cone at hypersonic Mach numbers. They found that for different attack angles, different nose spike configurations are fitted as drag reducing devices for blunt cones, and the flow characteristics they computed agree well with their experimental data.

However, with the application of the RANS method, the obtained parameters are averaged. To get the transient flow information of the flow field, large eddy simulation (LES) has been applied recently. Rodi\(^10\) compared the LES and RANS calculations of the three-dimensional (3D) flow past a surface-mounted cube. Their numerical drag coefficient and vortex shedding frequency are in good agreement with corresponding experimental measurements, and the computations showed superior results to those used in RANS. Schmidt et al.\(^11\) simulated the turbulent flow over a square cylinder with RANS, LES and DES. They demonstrated that...
DES and LES are able to capture the most dominant flow patterns, while RANS gave only a poor representation of the unsteady flow phenomena. Krajnovic et al.\(^{12}\) also computed the flow around a sharp-edged surface-mounted cube with the use of LES, and different techniques were used to visualize the flow characteristics. Streamlines around the cube revealed the flow separation and re-attachments clearly. Kim et al.\(^{13}\) simulated the turbulent flow past a square cylinder confined in a channel at \(Re = 3000\). The computational results are in good agreement with the experimental data. The well-known Karman vortex shedding was observed, but the vortices shed from the cylinder are affected by the plates, and the mean drag and fluctuation of lift force are increased too. LES was performed to simulate the flow over a square cylinder with different Reynolds numbers by Sohankar.\(^{14}\) He discussed the influence of the Reynolds number on the flow field, and found that the mean and large scale instantaneous flow structures were not very complicated at higher Reynolds numbers while the small scale ones were more complex and chaotic. Based on LES, Rodi\(^{15}\) simulated the flow passing around surface-mounted finite-height circular cylinders. In comparison with experiments, the flow feature details were well predicted. The iso-surface of the wake behind the cylinder showed the vortex developing and shedding clearly. Liang et al.\(^{16}\) simulated the pulsating cross-flow over a single circular cylinder with LES. The effect of the external flow pulsation on the wake characteristics was quantified by their numerical results.

However, there are only a few investigations concerning the hypersonic flow past a square cylinder with various heights. The purpose of the present study is to investigate the effects of cylinder height variation on the flow patterns of the hypersonic flow past a wall-mounted square cylinder. The WENO scheme combined with the adaptive mesh refinement (AMR) technique is employed to solve the LES equations, and various flow characteristics of different cylinder height are shown and discussed.

2. Numerical Approach and Model

2.1. Numerical approach

The 3D compressible Navier-Stokes equations after Faver filtering have the following form.

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho \bar{u}_j)}{\partial x_j} = 0
\]

\[
\frac{\partial \rho \bar{u}_i}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{i j}}{\partial x_j}
\]

\[
\frac{\partial \rho \bar{E}}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{E} + \rho \bar{u}_i \bar{u}_j \bar{u}_j)}{\partial x_j} = -\frac{\partial q_j}{\partial x_j} + \frac{\partial (\bar{u}_i \sigma_{i j})}{\partial x_j}
\]

Here, \(\sigma_{i j}\) is viscous stress tensor,

\[
\sigma_{i j} = \mu_{\text{eff}} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} - \frac{2}{3} \delta_{i j} \frac{\partial \bar{u}_k}{\partial x_k} \right),
\]

\[
\mu_{\text{eff}} = \mu_{\text{lam}} + \mu_{\text{sgs}},
\]

\(\mu_{\text{eff}}\) refers to the turbulent viscous coefficient, \(\mu_{\text{lam}}\) is the dynamic viscous coefficient and \(\mu_{\text{sgs}}\) denotes the Smagorinsky subgrid viscous coefficient. The heat flux is defined as

\[
q_j = -\lambda_{\text{eff}} \frac{\partial T}{\partial x_j},
\]

and

\[
\lambda_{\text{eff}} = \lambda_{\text{lam}} + \lambda_{\text{sgs}} = \frac{\mu_{\text{lam}} c_p}{Pr_{\text{lam}}} + \frac{\mu_{\text{sgs}} c_p}{Pr_{\text{sgs}}},
\]

\(\lambda_{\text{eff}}\) and \(\lambda_{\text{sgs}}\) are the turbulent and the subgrid heat exchange coefficients, respectively. \(Pr_{\text{lam}}\) and \(Pr_{\text{sgs}}\) are chosen as the constants. Assuming that the gas is the perfect gas, the equation of state is denoted as \(\tilde{p} = \rho R T\). According to the Smagorinsky subgrid model, \(\mu_{\text{sgs}} = 2C_s^2 \Delta^2 |\tilde{S}|\). Here, \(C_s = 0.1\) is the model constant, \(\Delta\) is the filter width, and \(|\tilde{S}| = \sqrt{2S_j S_j}\).

The governing equations are solved using the finite volume method (FVM) with 3D Cartesian grids. A third-order WENO scheme is used to discretize the convective term and the second-order central difference scheme used for the viscous term. The time is approximated with an explicit third-order Runge-Kutta scheme. A block structured adaptive mesh refinement technique is utilized to supply the required temporal and spatial resolution locally on the basis of the hydrodynamic refinement criteria.\(^{17}\) This adaptive method uses a hierarchy of spatially refined sub-grids which are integrated recursively with reduced time steps.

2.2. Computational model

The computational model and domain for the hypersonic flow past a square cylinder is shown in Fig. 1. The computational area is \(13.2 \times 9.6 \times 4\) in the unit of the cylinder width (\(D\)). The cylinder is located 6.72 downstream from the inlet and its cross-section is \(1 \times 1\). The height of the cylinder is chosen as 0.4, 1.2 and 2.0 respectively, the incoming flow Mach number is 5 and the inflow is laminar flow. The computational boundaries are all outflow conditions except for the bottom boundary, and the wall is adiabatic. A typical adaptive grid system during the numerical simulation at \(t = 0.001\) s is shown in Fig. 2.
3. Results and Discussions

Figure 3 shows the comparison of the experimental schlieren picture 4) with our numerical schlieren picture where \( H/D = 1.2 \). It is clear that our simulation is in agreement with the experimental picture, and the shock structure induced by the bow shock and the separated shock is shown clearly.

Figure 4 shows the comparison of our calculated pressure distribution along the symmetric line of the flat plane in front of the cylinder with the experimental results of Ref. 4) for \( H/D = 1.2 \). To normalize the coordinate, the parameter \( x/D \) (\( D \) is the cylinder width which is shown in Fig. 1) is used. The leading edge of the cylinder is taken as the origin of \( x \)-coordinates, and the parameter \( p/p_{\infty} \) is used for \( y \)-coordinates.

From Fig. 4, in front of the leading edge, the pressure distributions have the characteristics of multi-peak structures. This is caused by the occurrence of flow separations resulting from the adverse pressure gradient. It shows both experimentally and numerically that there are multiple separation zones in front of the cylinder, and within the separation zone, the pressure fluctuates due to the unsteady flow. Therefore, our numerical result at \( t = 0.0005 \) s within the separation zone is somewhat different to the corresponding experimental result.

Three-dimensional pressure distributions (the flat and symmetric planes) around the cylinders are shown in Fig. 5. It is clear that the pressure distribution tendencies of these three cylinders are similar to each other, and the highest pressure region (red area) exists between the bow shock and the cylinder, and also on the flat plane just in front of the cylinder. The bow shock structures are almost the same for \( H/D = 1.2 \) and \( H/D = 2 \), and the leading edge shock is much more straight for \( H/D = 0.4 \).

Figure 6 shows the pressure distributions along the symmetry axis of the flat with different cylinder heights. It is clear that the pressure fluctuating area becomes larger with the increase of \( H/D \), which means that the separation area increases also. After the flow reattachment near the front of the cylinder, the nearer the location with the cylinder windward, the larger the pressure, and the interactions of flow with the cylinder become strong. It can be seen that the second peak value also rises as \( H/D \) increases. However, the increased value with the variation of \( H/D \) from 1.2 to 2 is larger than that of from 0.4 to 1.2, which agrees well with the results of Ref. 18). Between the neighboring peaks, there is a valley which stands for the region of high-speed and low-pressure zone of the vortex core. It can be seen that the pressure fluctuating area becomes larger as \( H/D \) increases, which means the separation zone enlarges with the increase of \( H/D \).
Our results also show that if $H/D > 2$, the flow field features including highest pressure near the cylinder etc. are independent. However, for $H/D$ less than 1.5, the flow field features strongly depend on $H/D$. For $H/D$ between 1.5 to 2, the flow field will change slowly.

To discuss the variation of flow characteristics in detail, we take the cylinder of $H/D \approx 1.2$ as an example. The surface streamlines on the flat ($xoy$) and the symmetry ($xoz$) are shown in Fig. 7. Here S stands for separation point and A represents the reattachment point, while S2, A2, S3 and S4 stand for the secondary, third and fourth separation and reattachment points, respectively, and each separation line corresponds to a reattachment line. The surface friction lines converge to the separation lines and separate from the attached lines.

From Fig. 7(a), we know, at the early stage, when the supersonic flow passes around the cylinder, oblique shocks form, and an adverse pressure gradient appears in front of the cylinder windward, which causes the first separation of the flow, and the vortex is large. Later, due to the interaction of main vortex with the boundary layer and the unsteadiness of the flow, a new adverse pressure gradient zone forms, which causes the secondary separation, resulting in the appearance of two counter-rotating vortex pairs (Fig. 7(b)). At the same time, due to the corner vortex in front of the cylinder and counter-clockwise vortex, the two clockwise vortices are forced to lift up, forcing the flow to reseparate, and ultimately, forming the multi-vortices structure shown in Fig. 7(c), and lastly in Fig. 7(d). However, based on previous experiments and our numerical results, due to the unsteadiness of the flow, the vortex structures and separation zones in front of the cylinder will vary with time.

Figure 8 displays the surface streamlines for different cylinders. Figure 8(a) shows one separated line and one attached line and is a typical two-vortex structure. Figure 8(b) and (c) show typical four-vortex structures. On the other hand, with the increase of parameter $H/D$, the topology becomes more and more complex, and the separation region increases too. Corresponding to Fig. 8, the streamlines of different planes at $t = 0.0005$ s are presented in Fig. 9. The vortex structures in the separation regions can also be observed clearly, and the vortex number and vortex structure in the separation region reveal the difference and similarity as parameter $H/D$ increases.

The 3D wake of different cylinders is shown in Fig. 10 and the corresponding surface streamlines of the flat plane behind the cylinders are shown in Fig. 11. From Fig. 10, it can be seen that the streamlines are intertwined behind the cylinder. For a short cylinder, the interaction between the flow and the cylinder is weak and the cylinder obstruction effect is small. Therefore, the vortices behind the cylinder are large and their location is low. As a result, large vortices appear on the flat plane behind the cylinder (Fig. 11(a)). With the increase of cylinder height (Figs. 10(b), 11(b), 10(c) and 11(c)), their interaction becomes strong, and the vortex pair location behind the cylinder rises, which makes the vortex footprints appear small on the flat plane. At the same time, the separation point becomes closer to the cylinder. It also reveals that the streamlines are long and narrow with a short cylinder and the streamlines are outspread for a higher cylinder.

4. Conclusions

Based on the LES, combined with the WENO scheme and the AMR technique, the hypersonic compressible flow past rectangular cylinders with different heights was simulated. Our calculated results were in agreement with corresponding experimental data. The numerical results revealed that the flowfield is highly unsteady and the pressure in the separation area fluctuates with the time. For all three cases, the
pressure distribution in front of the cylinder was similar, which has multi-peak structures.

On the other hand, with the increase of $H/D$, the interaction between the shock and the cylinder became strong, the highest pressure increased, the size of the separation region in front of the cylinder increased, the vortex structure became complex and the streamlines were outspread for a high cylinder. The flow characteristics depend on the cylin-
der height \((H/D)\). However, for \(H/D > 2\), the highest pressure of the flow field is independent.

**Acknowledgments**

The authors gratefully acknowledge financial support from the Specialized Research Fund for the Doctoral Program of Higher Education of China (No 20103219110037).

**References**

1) Young, F. L., Kaufman, L. G. and Robert, H. K.: Experimental Investigation of Interactions between Blunt Fin Shock Waves and Adjacent Boundary Layers at Mach Numbers 3 and 5, Aerospace Research Labs Wright-Patterson, Ohio, AD0684537, 1968.

2) Dolling, D. S. and Bogdonoff, S. M.: Blunt Fin-induced Shock Wave/turbulent Boundary-layer Interaction, *AIAA J.*, 20 (1982), pp. 1674–1680.

3) Aso, S., Maekawa, S. and Okuyama, S.: Intensive Studies on Unsteady Secondary Separations and Oscillating Shock Waves in Three-dimensional Shock Waves/turbulent Boundary Layer Interaction Region Induced by Sharp/blunt Fins, AIAA Paper 93-2939, 1993.

4) Li, S.: Complex Flow Caused by Shock and Boundary Layer, Science Press, Beijing, 2007, pp. 68–80 (in Chinese).

5) Kontis, K.: Jet Control Effectiveness Studies on a Flat-plate Body at Hypersonic Speeds, *Trans. Ipn. Soc. Aeronaut. Space Sci.*, 47 (2004), pp. 131–137.

6) Hung, C. M. and Buning, D. G.: Simulation of Blunt-fin-induced Shock Wave and Turbulent Boundary-layer Interaction, *J. Fluid Mech.*, 154 (1985), pp. 163–185.

7) Chaussee, D. S.: High-Speed Flow Calculations Past 3-D Configurations Based on the Reynolds Averaged Navier-Stokes Equations, NASA Technical Memorandum-100082, Moffett Field, 1988.

8) Ma, H., Li, S. and Wu, L.: A Study on Characters of Hypersonic Plate/cylinder Juncture Flow, *J. Aeronaut.*, 21 (2000), pp. 1–5 (in Chinese).

9) Gopalan, J., Menezes, V., Reddy, K. P. J., Hashimoto, T., Sun, M., Saito, T. and Takayama, K.: Flow Fields of a Large-Angle, Spiked Blunt Cone at Hypersonic Mach Numbers, *Trans. Ipn. Soc. Aeronaut. Space Sci.*, 48 (2005), pp. 110–116.

10) Rodi, W.: Comparison of LES and RANS Calculations of the Flow around Bluff Bodies, *J. Wind Eng. Ind. Aerodynam.*, 69 (1997), pp. 55–75.

11) Schmidt, S. and Thiele, F.: Comparison of Numerical Methods Applied to the Flow over Wall-mounted Cubes, *Int. J. Heat Fluid Flow*, 23 (2002), pp. 330–339.

12) Krajnovic, S. and Davidson, L.: Flow around a Three-dimensional Bluff Body, The 9th International Symposium on Flow Visualization, Edinburgh, 2000, pp. 177.1–177.10.

13) Kim, D. H., Yang, K. S. and Senda, M.: Large Eddy Simulation of Turbulent Flow Past a Square Cylinder Confined in a Channel, *Comput. Fluids*, 33 (2004), pp. 81–96.

14) Sohankar, A.: Flow over a Bluff Body from Moderate to High Reynolds Numbers Using Large Eddy Simulation, *Comput. Fluids*, 35 (2006), pp. 1154–1168.

15) Rodi, W.: DNS and LES of Some Engineering Flows, *Fluid Dynam. Res.*, 38 (2006), pp. 145–173.

16) Liang, C. and Papadakis, G.: Large Eddy Simulation of Pulsating Flow over a Circular Cylinder at Subcritical Reynolds Number, *Comput. Fluids*, 36 (2007), pp. 299–312.

17) Berger, M. and Colella, P.: Local Adaptive Mesh Refinement for Shock Hydrodynamics, *J. Comput. Phys.*, 82 (1988), pp. 64–84.

18) Li, S., Chen, Y. and Ni, Z.: Flowfield Features on Hypersonic Flow over Rectangular Obstacles, *J. Visual.*, 4 (2001), pp. 73–79.