Vortex Structure Analysis of Grooved Surface Flow Loss

Mei Liu1,2*, Shuguang Li1, Kaishun Zhang1, Hongyue Yang1 and Songling Wang1

1School of Energy, Power and Mechanical Engineering, North China Electric Power University, Baoding, Hebei, 071003, China
2Department of Economic Management, North China Electric Power University, Baoding, Hebei, 071003, China
*Corresponding author’s e-mail: liumei_w@163.com

Abstract. Large eddy simulations were used to study the effect of groove structures on rotational motion of fluid in wall turbulence, from the perspective of coherent structures. The results show that groove structures reduce the per unit normal height internal velocity gradient in the near-wall region and decrease shear stress and viscous resistance. Concurrently, owing to the decrease in the velocity gradient, shear action is weakened, and the vortices’ level of stretch is reduced in the near-wall region of the grooved surface. The groove structures change the distribution of the shear layers in the boundary layer and increase the stability of the strips. The reduction of spanwise vorticities delayed the formation of hairpin vortices caused by the instability of strips and so on; and the energy dissipation is effectively reduced in the flow process.

1. Introduction

Turbulence is a ubiquitous state of flow in nature; unordered turbulent motion has stronger momentum transport capacity, but flow loss in turbulent conditions is larger than laminar. Thus, turbulent drag reduction is an important research topic in not only the aspect of fluid mechanics but also the field of engineering. In a series of related studies, flow boundary layer drag reduction in bionic-based grooved surfaces has received particular attention. In the 1980s, Walsh et al. [1,2] found that symmetric V-shaped groove surface can achieve drag reduction through experimental research under turbulent conditions, which indicates directions for future research.

In order to obtain optimal drag reduction effect, Walsh [3], Bechert [4,5], and Debisschop et al. [6] focused their research on the optimization design of the groove topography. Rene et al. [7] conducted a direct shear stress measurement on a test plate with rectangular riblet grooves in a fully turbulent flow, and achieved a good drag reduction effect. The direct numerical simulation of fully developed turbulent channel flow with a sinusoidal riblet surface has been carried out, and Sasamori et al. [8] found that the maximum total drag reduction rate is approximately 9.8% at a friction Reynolds number of 110.

The excellent drag reduction effect of groove structures also trigger scholars to study the mechanism of drag reduction. After the coherent structure was discovered in shear turbulence, people gradually realized that not only irregular small-scale vortices but also periodic large-scale vortices were involved in turbulent flow [9]. Xu Chunxiao [10] reviewed the research on coherent structure, specifically expounded the influence of vortex structures on energy dissipation in turbulence, and gave a variety of methods for controlling the evolution process of vortices in turbulent flow. Yang Shaoqiong et al. [11] observed this vortex structure through experiments. On the other hand, El-
Samnbi et al. [12] through direct numerical simulations found that large non-dimensional spacing makes streamwise coherent structures rooted in the bottom of grooves, increasing the resistance.

In conclusion, present studies have mainly focused on the drag reduction effects of groove structures, and analyzed groove structures’ impact on flow field characteristics from the perspective of flow characteristics. However, there are few studies on the mechanism of flow loss that are based on the evolution of coherent structures in groove structures’ flow fields. Therefore, from the perspective of using V-shaped groove structures to passively control the coherent structures of wall turbulence fields, this study analyzed how groove structures affect the wall shear stress, and examined how V-shaped groove structures change the vortex structure evolution process; it then analyzed the influence of passive control for the turbulent flow field.

2. Calculation Model and Numerical Calculation Method

2.1 Model setup

Based on the Fluent numerical simulation software, large eddy simulations were used to analyze the mechanism of flow loss that are based on the evolution of coherent structures in groove structures’ flow fields. The arrangement, geometry, and size are shown in figure 1.

![Figure 1. Calculation model](image)

2.2 Numerical method

Structured grids were used in the computational domain, and an H-type grid split form was applied, and the total number of grids was approximately three million. In order to distinguish the turbulence structures in the near-wall region, nonuniform grids were used in the normal direction; the grids near the wall were more serried, which eventually made the smallest wall grid size $y^+$ less than 1. Concurrently, grid independence was verified; the grids of the computational domain are shown in figure 2.

![Figure 2 Mesh schematic of partial computation domain](image)

Realizable $k$ - epsilon turbulence model with a rotation correction was used, and then steady simulations was conducted until the results were convergent. The transient field was then used as the initial calculation of the large eddy simulation model based on the Smagorinsky–Lilly subgrid. The time step $\Delta t = 1.0 \times 10^{-6}$ s. The streamwise and spanwise directions used periodic boundary conditions, and the mass flow rate are fixed. The normal direction used adiabatic non-slip wall conditions. When the flow
field was convergent, the simulation continued for 10000 time steps to obtain the statistical average results of the flow field.

2.3 Governing equation
The Navier–Stokes equations were decomposed, and the complete transient state was decomposed into two parts — one part is the component that is less than subgrid scale, and the other part is the large-scale average component. The equations are as follows:

\[
\rho \left( \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \mu \nabla^2 \mathbf{u} - \rho \mathbf{u} \cdot \nabla \mathbf{u} - \rho \mathbf{g}
\]

(1)

\[
\tau_{ij} = -\frac{1}{2} \left( \nabla \mathbf{u} + \nabla \mathbf{u}^T \right)
\]

(2)

Formula (1) is the revised Navier–Stokes equation for solving the large-scale component. In formula (2), \( \tau_{ij} \) is the subgrid-scale stress tensor expression. This formula even combined with a filtered continuity equation cannot work out \( \mathbf{u} \), \( \nabla \mathbf{u} \), and \( \mathbf{u} \cdot \mathbf{u}^T \) at the same time; thus, \( \tau_{ij} \) is transformed using a Smagorinsky subgrid model, and is expressed by the eddy viscosity coefficient.

\[
\nu_s = \frac{1}{2} \left( \tau_{ij} + \frac{1}{3} \delta_{ij} \right) \mathbf{S}_i \mathbf{S}_j = (C_s \Delta)(\mathbf{S}_i \mathbf{S}_j)^{\frac{1}{2}}
\]

(3)

In formula (3), \( S_{ij} \) denotes the shear deformation velocity tensor, the value of \( C_s \) is set to 0.1, and \( \Delta \) denotes a filter dimension determined by grid size.

3. Accuracy verification of numerical calculation
Comparing the theoretical value and simulation value of the frictional resistance coefficient and turbulent boundary layer average velocity distribution can verify the reliability of the numerical model. The empirical formula for wall frictional resistance coefficient \( C_f \) is:

\[
C_f = 0.074 \text{Re}^{-\frac{1}{2}}
\]

(4)

Table 1. Comparison of wall friction coefficients

| Velocity (m/s) | Theoretical value | Simulation value | Absolute error |
|---------------|-------------------|------------------|----------------|
| 12            | 0.0122            | 0.0123           | 1.152%         |
| 20            | 0.0110            | 0.0109           | 0.816%         |
| 24            | 0.0106            | 0.0104           | 1.602%         |
| 30            | 0.0102            | 0.0098           | 3.350%         |
| 40            | 0.0096            | 0.0094           | 1.877%         |

A comparison of the velocity distributions of the numerical calculation model and an experiment [13] in the normal direction is shown in figure 3. The result shows that the error rate of the numerical calculation model is very small, which proves the correctness of the model.

![Figure 3. Distribution of mean velocity in turbulent boundary layer](image-url)
4. Results analysis

4.1 Distribution of wall shear stress

The fluids within groove structures increase the viscous sublayer thickness of the boundary layer, and the velocity gradient of a grooved surface is always less than that of a flat plate surface when in the same location. Further, the decrease in the partial velocity gradient will reduce wall shear stress accordingly. Figure 4 illustrates the shear stress distributions of a flat plate surface and a grooved surface.

Figure 4(a) illustrates the wall shear stress distribution of a single groove structure. The interaction of the wall and vortex inside the groove structure makes the wall shear stress distribution appear as two hump-like curves. Therefore, the energy of the vortices is consumed by wall shear stress, and in order to maintain the development of vortices inside the grooves, the external energy will enter these vortices to maintain their intensity. In this process, the energy of the external flow field is consumed. By integrating the two curves in figure 4(a) within the same distance, we can see the effects of shear stress on the groove structure surface and flat plate surface. In the same flow distance, we can see that the energy reduction caused by the groove structures is far greater than the power required to maintain the intensity of the vortices. As shown in figure 4(b), on the surface of y = 0, the shear stresses of the groove structure surface are collectively less than those of the flat plate surface. However, the vortices do not completely fill the grooves in the groove structure; thus, compared with the flat plate surface, the speed changes sharply and the shear stress of the grooved surface increases. The change in the velocity of the per unit normal height is reduced in the near-wall region, and the shear stress is also decreased, therefore, flow loss caused by viscosity is demonstrably decreased.

![Figure 4](image)

Figure 4. (a) Distribution of the wall shear stress at the bottom at 24 m/s. (b) Distribution of shear stress of y=0 at 24 m/s.

4.2 Analysis of strip instability

Strip structures can be characterized by the average velocity contours of the spanwise section in the near-wall region. Figures 5(a) and figures 5(b) show the velocity contours and strip structures seen when the flat plate surface and groove structure surface are the same at location y+ = 5. Warm colors represent high-speed strips, while cool colors represent low-speed strips. By comparing the strip structures of two different surfaces, we can see that the strip structures are chaotic and irregular on the flat plate surface; this reflects the randomness of the turbulent flow field. Compared with the flat plate surface, the strip structures of the grooved surface are more stable; this suggests that in the groove structures area, the instability and fragmentation of the strip structures have been effectively suppressed. There are a large number of spanwise vortices in the strips, and the shearing action formed in different velocity layers causes tension in vortex tubes that are formed by the spanwise vortices. The conservation of vortex tube intensity shows that when the sectional area of a vortex tube is reduced gradually, the vorticity of this sectional area is bound to increase, such that the stretched vortex tubes will convert to streamwise vortex structures that are mainly in the form of hairpin vortices.
Because of the existence of the groove structures, in the near-wall region the velocity distribution of normal height \( y^+ \) between 0 and 5 is changed. Therefore, the velocity gradient decreases demonstrably, which weakens the shearing action formed in different velocity layers; it also reduces the possibility of uplift and stability for the strips, and eventually reduces the corresponding energy loss and flow loss.

4.3 Evolution process of vortex structures based on \( Q \) criterion

In the near-wall region, the general method of showing the vortex structures is to use the \( Q \) criterion. Under conditions with the same flow velocities and \( Q \) values, Fig. 6 presents an evolutionary schematic of vortex structures passing a flat plate surface and a grooved surface. Figure 6(a) illustrates that, on the flat plate surface, a large number of rotating vortex structures will form; in these vast and complex vortex structures, we can observe familiar coherent structures such as hairpin vortices and crutch vortices. Coherent structures also exist on the grooved surface, but the densities of hairpin vortices and crutch vortices decrease significantly, as shown in figure 6(b). When coherent structures reach the grooved area, small reductions in the normal velocity gradient in the near-wall region (as compared with the flat plate area) suppress the rotation of spanwise vortex structures in the normal direction; this weakens the rotation intensity of spanwise vortices. As fluid flows over an increasing number of groove structures, the effects of inhibition become stronger; ultimately, a balance is achieved when the reduction in rotation intensity coincides with the velocity gradient of the grooved surface. The reduction of spanwise vorticities effectively delays the formation of hairpin vortices caused by the instability of strips and so on; therefore, the energy dissipation is effectively reduced in the flow process.

Figure 6. (a) Vortex core isosurface contour of flat plate with a velocity. (b) Vortex core isosurface contour of grooves with a velocity.
5. Conclusions
In this study, the flow characteristics of the boundary layer of a grooved surface were analyzed through a numerical simulation. The groove structures’ influence on shear stress, strip instability, vortex structures was analyzed. These analyses provide a reference for exploring groove structure mechanisms that can reduce flow loss. The main conclusions are as follows:

(1) In the analysis of the groove structure, the existence of groove structures reduces the per unit normal height internal velocity gradient in the near-wall region and decreases shear stress. This results in a significant reduction in flow resistance caused by viscosity.

(2) The decrease of velocity gradient weakens the shearing action formed in different velocity layers; it also reduces the possibility of uplift and stability for the strips.

(3) The smaller normal velocity gradient inhibits the normal rotation of spanwise vortices and decreases the spanwise vorticity, which effectively delays the formation of streamwise vortices caused by the instability of strips.

Acknowledgments
This work has been supported by Hebei Province Natural Science Foundation (Grant No. E2016502088), by Fundamental Research Funds for the Central Universities (Project No. 2015MS105).

References
[1] Walsh M.J. (1984) Optimization and application of riblets for turbulent drag reduction. In: American Institute of Aeronautics and Astronautics, Aerospace Sciences Meeting, 22nd, Reno.
[2] Howard F.G., Goodman W.L., Walsh M.J. (1983) Axisymmetric bluffbody drag reduction using circumferential grooves. In: American Institute of Aeronautics and Astronautics, Applied Aerodynamics Conference, Danvers.
[3] Walsh M. (1982) Turbulent boundary layer drag reduction using riblets. In: 20th Aerospace Sciences Meeting. Orlando, U.S.A. pp. 769-787.
[4] Bechert D.W., Bruse M., Hage W. (1997) Experiments on drag reducing surfaces and their optimization with an adjustable geometry. Fluid Mechanics, 338: 59-87.
[5] Bechert D.W., Bartenwerfer. (1989) The viscous flow on surfaces with longitudinally riblets. Fluid Mechanics, 206: 105-129.
[6] Debisschop J.R., Nieuwstadt T.M. (1996) Turbulent boundary layer in an adverse pressure gradient-effectiveness of riblets. AIAA journal, 34(5): 932-937.
[7] Grüneberger R., Wassen E., Hage W., Meyer R., Thiele F. (2013) Experimental Investigation of Oscillating Riblets for Turbulent Drag Reduction. Notes on Numerical Fluid Mechanics & Multidisciplinary Design, 121(1): 193-200.
[8] Sasamori M., Iihama O., Mamori H., Iwamoto K., Murata A. (2017) Parametric Study on a Sinusoidal Riblet for Drag Reduction by Direct Numerical Simulation. Flow Turbulence & Combustion, 99(2): 1-23.
[9] Jiménez J. (2013) Near-wall turbulence. Physics of Fluids, 25(10): 97-120.
[10] XU Chunxiao. (2015) Coherent structures and drag reduction mechanism in wall turbulence. Advances in Mechanics, 45(1): 111-140
[11] Yang S.Q., KwingSO C., Jiang N. (2015) Flow Visualizations on Kelvin-Helmholtz-like Roller Structures in Turbulent Boundary Layer Over Riblets. Chinese Journal of Theoretical and Applied Mechanics, 47(3): 529-533
[12] Ei-Samni O.A., Yoon H.S., Chun H.H. (2005) Turbulent flow over thin rectangular riblets. Journal of Mechanical Science & Technology, 19(9): 1801-1810.
[13] Eckelmann, H. (2006) The structure of the viscous sublayer and the adjacent wall region in a turbulent channel flow. Journal of Fluid Mechanics, 65(3): 439-459.