Numerical research on the turbulent drag reduction mechanism of a transverse
groove structure on an airfoil blade

Zhengren Wu\textsuperscript{a}, Shuguang Li\textsuperscript{a}, Mei Liu\textsuperscript{a, b}, Songling Wang\textsuperscript{a}, Hongyue Yang\textsuperscript{a} and Xiujun Liang\textsuperscript{a}

\textsuperscript{a}School of Energy Power and Mechanical Engineering, North China Electric Power University, Baoding, People’s Republic of China; \textsuperscript{b}Department of economic management, North China Electric Power University, Baoding, People’s Republic of China

ABSTRACT
The optimization of the airfoil has a significant impact on the reduction of energy consumption for a rotary machine. In this paper, ANSYS Fluent is used to study the influence of different groove structure sizes on the turbulence drag of NACA0012 airfoil blades, and the drag reduction mechanism is analyzed. The results show that the groove structure can significantly reduce the drag during the working speed of the fan. The optimal groove size is $s = 0.1$ mm and the drag is reduced by 9.65%. The secondary vortices reduce the normal velocity gradient at the top of the groove structure, resulting in a reduction in viscous drag. However, as the groove size increases, the drag reduction effect decreases, and even the drag increases. The overall shear stress of the airfoil surface with the transverse groove structure is smaller than the original airfoil, and the velocity gradient of the airfoil surface is reduced. The two sides work together to reduce the turbulence drag of the airfoil. Besides, the spacing between the grooves increases the shear stress in some areas, but reduces the mutual interference of the vortices, so there is an optimum value for the groove spacing.

1. Introduction

At present, the drag reduction of a bionic groove structure is a research hotspot in turbulent drag reduction. As a passive control technique, groove drag reduction is a relatively inexpensive and effective method. The method does not need to provide additional energy. In the early stage, Walsh (1982) tested the drag reduction effect of different groove shapes, and found that the symmetrical V-shaped groove structure worked best. In 1989, Choi proposed the concept of virtual origin for the groove surface. Bechert and Bartenwerfer (1989) found a solution for calculating the protrusion height of the groove surface. Luchini, Manzo, and Pozzi (1991) used the boundary-element method to calculate the viscous vortex on the grooved surface, which was used to analyze the vortex structure in the wall region and explore the drag reduction mechanism. Bechert, Bruse, Hage, Van Der Hoeven, and Hoppe (1997) studied the turbulence characteristics of the blade-groove, resulting in a 9.9% reduction in drag. Ei-Samni, Yoon, and Chun (2005) studied the rectangular groove structure and found that the structure had a drag reduction rate of up to 15%. Sasamori, Iihama, Mamori, Iwamoto, and Murata (2017) focused on a sinusoidal riblet surface and found that this structure also has a 9.8% drag reduction effect.

Although various micro-groove structures have a certain drag reducing effect, V-shaped grooves are most likely to be widely used for their simple structure and good robustness. Djenidi and Antonia (1996) measured the drag-reduction effect of the groove through experiments and found that the drag reduction effect was related to the peak-to-peak distance. Debisschop and Nieuwstadt (1996) explored the effect of the groove by the experiments and found that the groove had a drag reduction rate of 13% when it was a reverse pressure gradient. Duan and Choudhari (2013) found that the drag reduction mechanism of the groove is still effective for the incompressible high-speed boundary layer by direct numerical simulation. Garcia-Mayoral and Jimenez (2011) analyzed the influence of the groove on the flow state, and its size had a certain relationship with the drag reduction effect. Dean and Bhushan (2010) experimented with a replica of the shark skin and found that it did not have a better effect. Chu and Karniadakis (1993) studied the drag reduction effect of lateral grooves under different Reynolds numbers and
found that the groove reduction effect was better when the Reynolds number is larger.

Zhang and Che (2011) found that the turbulence intensity in the near-wall region is significantly lower than that on the smooth surface, which is believed to be due to the weakening of the number and intensity of the flow vortices. Ahmadi-Baloutaki, Carriveau, and Ting (2013) studied the effect of the groove structure on turbulence by experimental measurements and found that the groove effect would lead to an increase in the thickness of the boundary layer. Wang, Wang, Zhou and Chen (2014) conducted a comparative study of the lateral grooves on the simulation and experiment and found that there was a vortex structure inside the lateral grooves. Sutardi and Wawan Aries (2016) explored the influence of the groove structure on the Laminar Sub-Layer Region through experiments and found that the turbulent energy spectrum of the grooved plate was significantly lower than that of the smooth surface.

For fans, many researchers focus on rotating stall or entropy generation characteristics (Zhang & Engeda, 2018). According to Akbarian et al. (2018), for fluid machinery, the friction caused by the viscosity of the fluid has a significant effect on the performance and energy consumption of the machine, especially in the turbulent state. By modifying the airfoil, the flow condition of the turbulent vortex structure in the near-wall region is improved, and the energy dissipation in the turbulent boundary layer is reduced. The flow field around the airfoil blade also has a large impact on its performance (Yi, Wang, Sun, Huang, & Zheng, 2017 and Milas, Vučina, & Marinić-Kragić, 2014), and the magnitude of the drag directly affects its work efficiency and economic efficiency.

In 2000, Gad-el-Hak placed the groove structure on the upper side of the airfoil to explore its drag reduction mechanism. However, there were many unknown aspects of the problem remained unresolved. Boese and Fottner (2002) placed the groove structure on the airfoil surface and found that it can effectively reduce the drag. Song, Liu, Hu, Huang, and Wu (2010) found that the V-shaped structure arranged on the airfoil surface can significantly reduce the turbulence intensity and turbulent kinetic energy in the boundary layer of the airfoil. Chamorro, Arndt, and Sotiropoulos (2013) studied the drag reduction of wind turbine airfoil with full coverage and partial coverage of the groove surface in the range of 0°—10° attack angle through wind tunnel experiment. The maximum drag reduction is 6% for full coverage and 4% for partial coverage. Besides, Chamorro et al. proposed a formula for the optimal ridge size for the airfoil to achieve the best results. Harun, Abbas, and Dheyaa (2016) conducted a wind tunnel experiment on the obliquely staggered groove structure of the airfoil surface and found that the fluid-structure through the convergence and divergence of the groove structure can significantly change the turbulence characteristics and reduce the airfoil drag. Domel et al. (2018) arranged a row of bionic shark skin structures on the upper surface of the naca0012 airfoil by parametric modeling and found that it can simultaneously reduce drag and increase lift on the airfoil. Zhang and Bijay (2018) carried out a simulation study on the drag reduction effect of micro-textures and found that the micro-texture structure can be arranged in the flow separation region to achieve the drag reduction effect.

However, some researchers believe that the drag reduction effect of the trench structure is not outstanding. Chen, Tang, and Chen (1990) argued that arranging the groove structure on the upper surface may increase the possibility of flow separation, resulting in increased drag. A study by Nagao and Breugelmanns (1999) shows that the arrangement of the groove structure on the upper surface of the airfoil increases the airfoil drag. Lee and Jang (2005) used PIV experiments to investigate the effect of grooves on the airfoil. At a speed of 3 m/s, a drag reduction effect of 6.6% was obtained. However, as the speed increases, the airfoil drag gradually increases. Lietmeyer, Oehlert, and Seume (2013) selected groove structure parameters with drag reduction effect and placed them on the upper surface of the compressor blade, which only obtained a drag reduction effect of 1.01%. Shi (2016) numerically simulated the airfoil with grooved V-groove structure by the SST k-w turbulence model. It is considered that the V-shaped groove at any position on the upper surface can not produce the ideal effect.

The researchers have different views on the drag reduction effect of the grooves arranged on the airfoil. The effect of the grooved airfoil surface on the aerodynamic performance of the airfoil requires careful study. Only by thoroughly exploring its influence mechanism and understanding the drag reduction mechanism of the groove can this problem be effectively solved. It is well known that when gas flows through the airfoil, vortex shedding occurs, so we arrange the groove structure in the middle-back regions of the airfoil that is most prone to this phenomenon, which covers 40% of the airfoil, effectively covering the separation point. In this paper, ANSYS Fluent is used to simulate the turbulent drag reduction effect of the groove structure of the NACA0012 blade to explore the drag reduction effect of the groove structure and improve the aerodynamic performance of the blade.
2. Numerical methods

2.1. The geometric model and mesh

As shown in Figure 1, this paper takes the airfoil section for calculation. Besides, the upper section (pressure side) is selected to be calculated.

The length of the airfoil chord is 20 cm, and the length of the transverse groove region is approximately 8 cm. The left boundary is 10C (chord length) from the leading edge of the airfoil, the right boundary is 14C from the trailing edge of the airfoil, and the distance from the upper boundary to the airfoil is 10C. This ensures that the flow has entered a fully developed state of turbulence as it flows through the airfoil. For boundary conditions, both sides of the airfoil are set to symmetrical boundary conditions. The inlet and outlet boundary conditions are the velocity inlet and out-flow boundary conditions. The upper part of the computational region is the symmetric boundary condition. The transverse groove structure is arranged in the middle of the airfoil. In the paper, four transverse groove structures with $s = 0.1 \text{ mm}$, $s = 0.2 \text{ mm}$, $s = 0.5 \text{ mm}$ and $s = 1 \text{ mm}$ are selected for simulation. The angle of the groove is 53.14°, and $d = h = s$. In this paper, the triangular grid is used to fill the computational domain, and the size function is used to refine the grid of the airfoil and groove regions.

The first grid height is 0.005 mm and the growth factor is 1.04. The partial detailed grid diagram and Y-Plus figure of the wall are shown below (Figures 23–4).

According to the differences in the models, several sets of different meshing schemes are established. After verification of the grid independence (in Figure 5), the number of grid cells in each model is determined. The final grid number of the smooth surface model is 650,000. The model of the transverse groove structure with a...
2.2. The governing equations and numerical methods

The Reynolds-averaged Navier-Stokes equations can be written as:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) = 0
\]

\[
\frac{\partial}{\partial t} (\rho u_i) + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} (\sigma_{ij} - \rho u'_i u'_j)
\]

In the formula:
- \( u_i \) denotes the average Reynolds velocity component of the average symbol omitted,
- \( \rho \) is density,
- \( p \) is pressure,
- \( u'_i \) is pulsation velocity and \( \sigma_{ij} \) is stress tensor component.

The turbulence model used is the RNG k-\( \varepsilon \) two-equation model. The equations are as follows:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] + S_\varepsilon \\
+ C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}
\]

In the formula:
- \( G_k \) is the turbulent energy generation caused by the velocity gradients, \( G_b = Y_M = 0 \), \( C_{1\varepsilon} = 1.42 \), \( C_{2\varepsilon} = 1.68 \) and \( C_{3\varepsilon} = 0 \). \( \sigma_k = \sigma_\varepsilon = 1.39 \). \( S_k \) and \( S_\varepsilon \) are the source term related to \( k \) and \( \varepsilon \).

For the RNG k-\( \varepsilon \) equation, an enhanced wall function was used. Adjusting the grid \( y^+ < 5 \) ensures that the \( y^+ \) condition of the wall function meets the requirements. To obtain detailed flow field characteristics, the finite volume method was used to discretize the governing equations. The governing equations were all handled by the ‘second-order upwind’ option. The SIMPLEC method was applied to the pressure-velocity coupling. The convergence condition was that the calculated residual was less than 10–5. By calculating the equation, the data can achieve good convergence, and compared with the experimental data, this part has been given in Section 2.3 of the paper.

2.3. Evaluation and verification through experimental testing

To investigate the turbulent drag reduction mechanism of a transverse groove structure on an airfoil blade, an
Figure 6. Experimental verification.

Accuracy verification of the RNG k-ε model is first performed. This paper compares the experimental data of Gregory and O’Reilly (1970). In the experiment, the Reynolds number (Re = uL/ν, u is the flow velocity; L is the chord length; and ν is the kinematic viscosity) was $2.88 \times 10^6$. This paper model was used to simulate the experimental condition and compared with the experimental data published by Gregory. The results are shown in the following Figure 6. It can be seen that the simulated data of the RNG k-ε model is almost the same as the experimentally obtained pressure drag coefficient $C_p$, which can prove the accuracy of the results.

3. Results and discussion

3.1. Analysis of the drag reduction characteristics and the mechanism of the transverse groove structure

Compared to an airfoil with a smooth surface, the frictional drag caused by the viscosity of air and the effects of transversely grooved surfaces are significant.

Figure 7. Contours of static pressure on the transverse groove surface. (a) $s = 0.1$ mm, $U = 10$ m/s; (b) $s = 0.1$ mm, $U = 22$ m/s; (c) $s = 0.1$ mm, $U = 40$ m/s; (d) $s = 1$ mm, $U = 10$ m/s; (e) $s = 1$ mm, $U = 22$ m/s; (f) $s = 1$ mm, $U = 40$ m/s.
of the transverse groove structure changes the pressure distribution on the blade surface. As a result, a lower pressure area on the leeward side is formed, as shown in Figure 7. Then, the pressure difference is formed between the windward and leeward, thereby generating a backward pressure drag.

Figure 8 shows the relationship between the drag coefficient and the size of the transverse groove structure at velocities of 22 and 40 m/s. At the same velocity, the pressure drag coefficient gradually increases as the size of the transverse groove structure increases. This result is caused by the pressure difference generated between the windward surfaces and the leeward surfaces of the transverse grooves. With the increased size of the transverse groove structure, the area affected by the pressure difference increases, resulting in the increasing pressure drag coefficient. At the same velocity, the viscous drag coefficient of each of the transverse groove surfaces is all significantly reduced. The magnitude of the reduction differs slightly because of the difference in the sizes of the transverse groove structures. Therefore, the total drag coefficient exhibits a complex trend of changes because of the influence of the viscous drag coefficient and pressure drag coefficient.

Figure 8 (a) and Figure 8 (b) show that for increasing flow velocity, The three parameters in Figure 8 are all significantly increased. This result is explained by the fact that with increasing flow velocity, both the turbulent kinetic energy and the turbulent intensity increase, so the flow instability increases, and the viscous drag caused by the friction of the working medium increases, resulting in an increase in energy dissipation and an increase in loss. The uneven degree of pressure distribution becomes aggravated, and the pressure drag coefficient increases, eventually resulting in a total drag coefficient increase.

3.2. Analysis of the drag reduction mechanism and the flow parameters

3.2.1. Flow analysis of the fluid near the transverse groove structure

Figure 9 shows the velocity vector near the transverse groove surface. It is obvious in Figure 9 that the direction of the velocity vector changes and some stable secondary vortices are formed in the grooves. The reflux caused by the vortices in the transverse groove structure causes the direction of the viscous drag in the transverse groove structure to be opposite to the direction of the viscous drag in the upper side flow, acting as a 'rolling bearing' in the machinery, which reduces the viscous drag. However, as mentioned earlier, the secondary vortices also cause the pressures on the left and right sides of the groove structure to be unequal, generating pressure drag. When the groove is s = 0.1 mm or s = 0.2 mm, the decrease in the viscous drag is greater than the increase in the pressure drag, resulting in a decrease in the total drag. But in the case of s = 1 mm, the opposite is true, and the drag is increased.

Figure 9. Flow velocity vector near the groove structure (U = 22 m/s, h = s = d = 1mm).

Figure 8. The relationship between the total (pressure, viscous) drag coefficient and size of a transverse groove structure for different velocities. (a) U = 22 m/s (b) U = 40 m/s.
3.2.2. The distribution of the vortex velocity in the transverse grooves

Figure 10 shows the distribution of the velocity in the vortex of a transverse groove structure for different flow velocities. ‘y’ represents the vertical distance from the bottom of the V-type transverse groove structure groove. ‘u’ represents the velocity at this point in the groove structure. For the transverse groove surface, there is a velocity slippage at the center of the microgrooves since a micro vortex could be found in the grooves. The velocity is found to exhibit a linear distribution at the bottom. At the position of the vortex, the distribution becomes quadratic. With the growth of the transverse groove size, the quadratic feature is more obvious, the velocity of the vortex is larger, and the ‘rolling bearing’ effect produced by the same flow velocity becomes stronger. However, simultaneously, in the transverse groove structure with a larger size, the turbulent dissipation of the vortex is larger. As the size of the groove increases, both the turbulent kinetic energy and the turbulent intensity increase, and the flow instability increases, which will increase the viscous drag caused by the friction of air. The combined action of the ‘rolling bearing’ effect and the groove size led to changes in the viscous drag of the airfoil surface.

3.2.3. The distribution of the mean velocity on the transverse groove surface

Figure 11 shows the average wall velocity profile. Except for the size of the s = 0.5 mm, the position of points is at the bottom of the groove, the points of other groove sizes are located on the outer surface of the grooved structure, corresponding to the corresponding positions of the surface of the plate. In the near-wall region, as $y^+ = y(\tau_w/\rho)^{0.5}/\nu$ (y is the normal distance; v is the kinematic viscosity; $\tau_w$ is the wall shear stress and $\rho$ is the air density) increases, the viscous shear stress and turbulent shear stress also change. Figure 11 shows that the dimensionless velocity $u^+ = u/(\tau_w/\rho)^{0.5}$ of the airfoil surface with the groove structure is significantly greater than the smooth surface. The velocity of the near-wall region of the 0.5 mm groove surface is significantly greater than the smooth surface, which can be considered as a manifestation of the reduction of viscous drag. From the curve of size 0.5 mm, some low-speed flow is found to be produced in the transverse grooves due to the viscosity, resulting in a reduction in the average gradient of the speed on the wall and a decrease of the shear stress. This also reflects the drag reduction effect of the transverse groove structure.

![Figure 10](image1.png)

**Figure 10.** Distribution of velocity in the vortex of transverse groove structure for different flow velocities. (a) $U = 10$ m/s; (b) $U = 22$ m/s; (c) $U = 40$ m/s
Corresponding to the velocity distribution curves of the vortex in Figure 10, when the fluid reaches the top of the secondary vortex, the flow velocity is observed to change according to the response shown in Figure 11. All of the curves are gradually consistent in the far flow field. This behavior indicates that the vortex has a great influence on the velocity in the transverse groove structure but has no influence on the far flow field.

3.2.4. The distribution of the vorticity in the normal direction

Vorticity, which is the origin of the velocity gradient in the flow field, is the measurement of the rotation of velocity. The addition of vorticity results in the formation of a vortex. From the definition of vorticity and the vorticity transport equation, the velocity gradient (or pressure gradient), the viscous stress and other factors are all found to cause changes in the vortex. The magnitude of the vorticity increases with increasing velocity gradient. A large number of experiments and numerical simulation studies have already proven that the energy dissipation is accomplished through the vortex. Therefore, in the boundary layer, the magnitude of the vorticity can reflect the situation of the energy dissipation loss. A greater vorticity indicates that additional smaller vortexes would be produced, resulting in greater energy dissipation.

Figure 12 shows the vorticity distribution on a section along the Y-direction for different velocities in the smooth model and the transverse groove structure model. From the Figure 12, as the flow velocity increases, the magnitude of the vorticity also gradually increases. The vorticity of the transverse groove surface is found to be smaller than the smooth surface at $U = 22$ m/s. If the velocity is too large or too small, the vorticity of the transverse groove surface would increase much more compared to the smooth surface. Besides, there is a significant difference between the increases in the amplitudes caused by the different transverse groove structures for different velocities. Given an appropriate velocity and transverse groove size, the transverse groove surface...
has a better effect on drag reduction. The order of the boundary layer thickness is approximately $2 \times 10^{-4}$ m, and thus, the region where the vorticity of the transverse groove surface is larger than the smooth surface is limited only within the boundary layer. The vorticity of the lateral groove surface is greater in some of the boundary layer. The effect of the ‘rolling bearing’ still exists but is relatively weak, which agrees with the preceding analysis.

### 3.2.5. The distribution of the wall shear stress

Figure 13 shows the distribution of wall shear stresses for 1 and 0.1 mm groove surface models and smooth airfoil surfaces at different speeds. Wall shear stress and wall friction are closely related. For the transverse groove surface, the shear stress of most areas in the grooves is smaller, and as the flow velocity increases, the area increases. This result indicates that as the flow rate increases, the wall shear stress of the model increases, and for the same transverse groove structure, the reduction effect of the transverse groove surface for the shear stress is gradually enhanced. From the contrast of shear stress between different models under the same flow velocity, as the transverse groove structure decreases, a relatively large region can be found whereby the shear stress in the grooves is smaller than the smooth surface. For the transverse groove structure with a small size, the range

![Figure 13](image-url)
of higher shear stress is relatively small compared to the smooth surface model. As a result, the small-sized transverse groove has a better effect. However, the shear stress could only reflect the frictional force on the airfoil surface; it cannot reflect the total drag. Therefore, it is necessary to combine the influence of the flow rate and the transverse groove structure on the pressure drag when performing analyses and studies.

Figure 13 shows that, at the bottom of the grooves, the shear stress is very small and close to zero. This low value occurs because the flow velocity near the bottom is so low that the fluid itself had no effect on the wall. Also, the shear stress at the front of the interval between grooves is the maximum, being significantly higher than the smooth surface at the same position, and along the flow direction, it gradually decreases. This behavior shows that the interval between grooves causes the shear stress at some regions to increase, especially in the vicinity of the windward side of the transverse groove structure. However, the existence of the interval between grooves can mitigate the mutual interference between vortices within the groove at the same time, as shown in Figure 9. Therefore, it is assumed that there would be an optimal interval that could mitigate mutual interference and minimize the shear stress as well.

3.2.6. The influence of the position of the transverse groove structure on the drag reduction characteristics

The above analysis indicates that a transverse groove structure with a size of $s = 0.1$ mm can achieve the drag reduction effect under all simulated velocities. To study the influence of the relative position of the transverse groove structure arranged on the airfoil on the drag reduction characteristics, we arranged the same transverse groove structure at the posterior part of the airfoil and studied its drag reduction characteristics. The relative position is shown in Figure 14(a).

Figure 14(b) shows the distribution of the total drag coefficient of the transverse groove structures as the Reynolds number increases. Compared with the arranged transverse groove structure of the middle part, the pressure drag coefficient of the posterior part of the arranged transverse groove structure is slightly reduced, but the viscous drag coefficient is significantly increased. The amount of decrease in the former is always less than the increase in the latter, resulting in the total drag coefficient of the transverse grooves disposed at the rear of the structure being always greater than the total drag coefficient of the transverse groove structure disposed at the intermediate portion.

Figure 14(c) shows the drag reduction rate curves of the two differently arranged transverse groove structures. The formula for calculating the drag reduction rate is as follows:

$$\eta = \frac{C_{ds} - C_{dg}}{C_{ds}} \times 100\%$$

Where: $C_{ds}$ is the total drag coefficient to the smooth surface; $C_{dg}$ is the total drag coefficient to the groove surface.

In the Figure 14, the drag reduction rate of the transverse groove structure arranged in the posterior part is always smaller than the transverse groove structure arranged in the middle part, and its maximum drag reduction rate is only 7.5%. Given large flow velocities, the drag would increase rather than decrease. This result is possibly due to the contracted flow line of the posterior part of the airfoil. When the transverse groove structure is arranged, a relatively large attack angle occurs between the flow and the transverse groove surface, causing the
secondary vortices formed in the transverse grooves to be unstable. The vortices diffuse outward and impact each other, thus increasing the turbulent dissipation. Also, the pressure drag increases rapidly as the speed increases; if the velocity is sufficiently high, the surface of the transverse groove will have a greater drag than the smooth surface.

Therefore, it is unsuitable to arrange the transverse groove structure at the posterior part of the airfoil. When the flow velocity is large, the transverse groove structure at the posterior part would increase the drag rather than achieve the drag reduction effect.

4. Conclusions

Through numerical simulation, we studied the drag reduction characteristics of a transverse groove structure of an airfoil blade.

1) The transverse groove structure arranged at the middle part achieves a significant effect for the airfoil blade. The transverse groove with a size of 0.1 mm has the best effect and the drag reduction rate is 9.65%.

2) An airfoil surface having a transverse groove structure can reduce the viscous drag, resulting in a decrease in total drag. The stable secondary vortex formed within the groove of the transverse groove structure can significantly reduce viscous drag. The vorticity increases as the flow rate increases and its magnitude reflects energy dissipation loss in the boundary layer. A larger vorticity results in greater energy dissipation. Only when the scale of the vortices formed by the airflow and transverse groove structure matches the size of the grooves is the drag reduction effect of the viscous drag the most significant.

3) The total shear stress of the airfoil surface to which the lateral grooves are added is significantly reduced, reflecting the viscous drag reduction effect of the transverse groove structure on the airfoil surface. However, the interval between grooves increases the shear stress of certain regions. Considering that the interval between the grooves could mitigate the mutual interference between vortices within the grooves at the same time, it is speculated that there would be an optimal interval that could mitigate the mutual interference and minimize the shear stress.

4) The relative position of the transverse groove structure along with the airfoil influences the drag reduction effect. Compared with the transverse groove structure arranged in the intermediate portion, as the Reynolds number increases, the groove structure disposed at the rear of the airfoil causes a significant increase in the viscosity drag coefficient, resulting in a significant decrease in the drag reduction rate. When the flow velocity is slightly higher, the drag increased rather than decreased. Therefore, it is unsuitable to arrange the transverse groove structure at the posterior part of the airfoil.

5) There are also some imperfections in the paper. In the paper, the drag reduction analysis is carried out for the common triangular groove structure. However, it is well known that stress concentration may occur at the sharp corners and some bad effects are formed. The model can be optimized later. Also, among the factors affecting the efficiency of the fan, the drag of the airfoil blade only accounts for a part of it, and other interactions such as the blade also have a great influence. In future work, it is also desirable to combine several factors to explore the overall effect.

Disclosure statement

No potential conflict of interest was reported by the authors.

Funding

This work was supported by the Natural Science Foundation of Hebei Province; under Grant E2016502088; The Fundamental Research Funds for the Central Universities under Grant 12QN39, 2015MS111.

References

Ahmadi-Baloutaki, M., Carriveau, R., & Ting, D. S.-K. (2013). Effect of free-stream turbulence on flow characteristics over a transversely-grooved surface. Experimental Thermal and Fluid Science, 51, 56–70.

Akkarian, E., Najafi, B., Jafari, M., Ardabili, S. F., Shamshirband, S., & Chau, K. W. (2018). Experimental and CFD-based numerical simulation of using natural gas in a dual-fuelled diesel engine. Engineering Applications of Computational Fluid Mechanics, 12(1), 517–534.

Bechert, D. W., & Bartenwerfer, M. (1989). The viscous flow on surfaces with longitudinal ribs. Journal of Fluid Mechanics, 206, 105–129.

Bechert, D. W., Bruse, M., Hage, W., Van Der Hoveen, I. G. T., & Hoppe, G. (1997). Experiments on drag-reducing surfaces and their optimization with an adjustable geometry. Journal of Fluid Mechanics, 338, 59–87.

Boese, M., & Fottner, L. (2002). Effects of riblets on the loss behavior of a highly loaded compressor cascade[C]/ASME Turbo Expo 2002: Power for Land, Sea, and Air. [S. I.]:American Society of Mechanical Engineers, 5, 743–750.

Chamorro, L. P., Arndt, R. E. A., & Sotiropoulos, F. (2013). Drag reduction of large wind turbine blades through riblets: Evaluation of riblet geometry and application strategies. Renewable Energy, 50(3), 1095–1105.

Chen, F., Tang, Y. P., & Chen, M. Z. (1990). An experimental investigation of loss reduction with riblets on cascade blade surfaces and isolated airfoils[C]/ ASME 1990 International
gas turbine and aeroengine congress and exposition. [S. l.]: American Society of Mechanical Engineers, 1, 11–14.

Choi, K. S. (1989). Near-wall structure of a turbulent boundary layer with riblets. Journal of Fluid Mechanics, 208, 417–458.

Chu, D. C., & Karniadakis, G. E. (1993). A direct numerical simulation of laminar and turbulent flow over riblet-mounted surfaces. Journal of Fluid Mechanics, 250, 1–42.

Dean, B., & Bhushan, B. (2010). Shark-skin surfaces for fluid-drag reduction in turbulent flow: A review. Philosophical Transactions, 368, 4775–4806.

Debisschop, J. R., & Nieuwstadt, F. T. M. (1996). Turbulent boundary layer in an adverse pressure gradient: Effectiveness of riblets. AIAA, 34, 932–937.

Djenidi, L., & Antonia, R. A. (1996). Laser Doppler anemometer measurements of turbulent boundary layer over a riblet surface. Aiaa Journal, 34, 1007–1012.

Domel, A. G., Domel, G., Weaver, J. C., Saadat, M., Bertoldi, K., & Lauder, G. V. (2018). Shark skin-inspired designs that improve aerodynamic performance. Journal of the Royal Society Interface, 15(139), 20170828–1–20170828–9.

Duan, L., & Choudhari, M. (2013). Effects of riblets on skin friction and Heat Transfer in high-speed turbulent boundary Layers. Aiaa Aerospace Sciences Meeting Including the New Horizons Forum and Aerospace Exposition.

Ei-Samni, O. A., Yoon, H. S., & Chun, H. H. (2005). Turbulent flow over thin rectangular riblets. Journal of Mechanical Science and Technology, 19, 1801–1810.

Gad-el-Hak. (2000). Mohamed. Flow control[M]. Cambridge: Cambridge University Press.

Garcia-Mayoral, R., & Jimenez, J. (2011). Drag reduction by riblets. Philosophical Transactions. Series A, Mathematical, Physical, and Engineering Sciences, 369, 1412–1427.

Gregory, N., & O’Reilly, C. L. (1970). Low-speed aerodynamics of NACA0012 aerofoil section, including the effects of upper surface roughness simulation hoarfrost[R]. National Physical Laboratory, NPL Aero Report 1308.

Harun, Z., Abbas, A. A., & Dheyaa, R. M. (2016). Ordered roughness effects on NACA 0026 airfoil[C]// AEROTECH VI.

Lee, S. J., & Jang, Y. G. (2005). Control of flow around a NACA0012 airfoil with a micro-riblet film. Journal of Fluids and Structures, 20(5), 569–672.

Leonardo, P. C., Arndt, R. E. A., & Sotiropoulos, F. (2013). Drag reduction of large wind turbine blades through riblets: Evaluation of riblet geometry and application strategies. Renewable Energy, 50, 1095–1105.

Lietmeyr, C., Oehlert, K., & Seumej, R. (2013). Optimal application of riblets on compressor blades and their contamination behavior. Journal of Turbomachinery, 135(1), 443–455.

Luchini, P., Manzo, F., & Pozzi, A. (1991). Resistance of a grooved surface to parallel flow and cross-flow. Journal of Fluid Mechanics, 228(228), 87–109.

Milas, Z., Vučina, D., & Marinić-Kragić, I. (2014). Multi-regime shape optimization of fan vanes for energy conversion efficiency using CFD, 3D optical scanning and parameterization. Engineering Applications of Computational Fluid Mechanics, 8(3), 407–421.

Nagao, S., & Breugelmans, F. A. E. (1999). Investigation of riblets in a CDB, DCA and 65-S compressor cascade[C]. Proceedings of International Gas Turbines Congress, Kobe, Japan:[s. n.], 14–19.

Sasamori, M., Iihama, O., Mamori, H., Iwamoto, K., & Murata, A. (2017). Parametric study on a sinusoidal riblet for drag reduction by direct numerical simulationp. Flow Turbulence & Combustion, 2, 1–23.

Shi, Y. L. (2016). Study on drag reduction characteristics of surface structure of airfoil Special airfoil [D]. Yangzhou: Yangzhou University.

Song, B. W., Liu, Z. Y., Hu, H. B., Huang, Q. G., & Wu, W. H. (2010). Numerical simulation study on drag reduction characteristics of airfoil surface ridge structure. Chinese Journal of Computational Mechanics, 27(5), 913–918.

Sutardi, S., & Wawan Aries, W. (2016). Analysis of turbulence characteristics in the laminar sub-layer region of a Perturbed turbulent boundary layer. Applied Mechanics and Materials, 836, 115–120.

Walsh, M. (1982). Turbulent boundary layer drag reduction using riblets. AIAA, Aerospace Sciences Meeting. AIAA, Aerospace Sciences Meeting, 1982, 769–787.

Wang, B., Wang, J., Zhou, G., & Chen, D. (2014). Drag reduction by microvortexes in transverse microgrooves. Advances in Mechanical Engineering, 6, 734012-1–734012-7.

Yi, P. H., Wang, Y., Sun, X. J., Huang, D. G., & Zheng, Z. Q. (2017). The effect of variations in first- and second-order derivatives on airfoil aerodynamic performance. Engineering Applications of Computational Fluid Mechanics, 11(1), 54–68.

Zhang, C., & Bijay, K. S. (2018). Investigation on drag reduction performance of aero engine blade with micro-texture. Aerospace Science & Technology, 72, 380–396.

Zhang, Y. B., & Che, D. F. (2011). Turbulence statistics in a rectangular channel flow with one groove roughened wall[C]. AIP Conference Proceedings, 1376, 90–93.

Zhang, L., & Engeda, A. (2018). Numerical simulation of rotating stall in a two-stage axial fan. Thermal Science, 22(S2), 565–5663.