Effects of residual riblets of impeller’s hub surface on aerodynamic performance of centrifugal compressors

Dong Lei* and Cao Lixin

School of Mechanical Engineering, Dalian University of Technology, Dalian 116023, China
(Received 3 March 2014; final version received 15 August 2014)

In this article, the effects of residual riblets of impeller’s hub surface on the aerodynamic performance of a compressor are studied in detail. Several accurate geometrical models of impellers after their hub surface are machined, with various tool passes are built. The number of tool passes for the hub surface finish of a single passage of the impeller is denoted by N. Meanwhile, a dimensionless parameter $B_{\text{max}}$ is introduced to characterize the size of the ribs of the arbitrary impeller, which is the percentage ratio rate of the maximum cusp height of the hub surface to the exit blade width. The geometrical models are numerically studied by solving the Reynolds averaged Navier–Stokes equations using the SST turbulent model. The predictions show that a group of secondary cross-stream vortices are generated by the interaction of streamwise vortices above the hub surface with riblets’ tips. For the impeller researched in this article, with the decline of $B_{\text{max}}$, the average viscous shear stress on the hub surface decreases gradually at first and then starts increasing and leveling off; accordingly, the polytropic efficiency of the compressor, on the contrary, increases gradually and then starts leveling off later. When $B_{\text{max}}$ comes to 0.58, the average viscous shear stress of the fluids on the hub surface reaches its minimum, and it is found that the polytropic efficiency of the compressor reaches a peak and stable value. The simulation results are validated against experimental data at the design Mu, and the maximum discrepancies are less than 3.8%.

Keywords: residual riblets; numerical study; average viscous shear stress; polytropic efficiency; experiment

1. Introduction

The centrifugal compressor has been widely used in several areas, such as metallurgy, chemical and petroleum, natural-gas transporting, and refrigeration. It has features of simple structure, high reliability, efficiency, pressure ratio of a single stage, and a wide range of working conditions. The impeller is one of the most important components of the centrifugal compressor. The machining precision of the impeller has a great impact on the compressor’s aerodynamic performance; thus, the designing and manufacturing of impellers plays a key role in centrifugal compressors. The aerodynamic performance of the compressor mainly relies on the flowing status of transported fluids. In general, the flow in the centrifugal compressor is turbulent and more complicated near solid walls. Several factors such as fluid viscosity, appearance, and surface roughness of solid walls affect the fluids’ flowing status. The turbulent viscosity is the most important factor. Previous research had been proposed by Crowe, Elger, and Roberson (2005), Shereena, Vengadesan, Idichandy, and Bhattacharyya (2005). The turbulent viscosity generated not only the skin friction drag but also the pressure drag. Since the boundary layer separation is unlikely to occur at a design point for the centrifugal impeller, skin friction drag is more pronounced than pressure drag. Therefore, it is more important to reduce skin friction drag in the turbulent boundary layer in the research of wall drag reduction for centrifugal impellers.

During the last decades, fine surface finish has been proved not to always get more drag reduction effectiveness, and riblets had shown potential for the reduction of skin friction drag. From the research presented by Oehlert and Seume (2006), tiny riblets aligned in the streamwise direction had been proved to be able to reduce skin friction and wall shear stress in turbulent flows compared with smooth surfaces. Chu and Karniadakis (1993) numerically simulated a channel flow over riblets using a spectral element Fourier method and obtained approximately 6% drag reduction for the riblet-mounted wall compared with the smooth wall. For the principle of riblets’ surface drag reduction, two important mechanisms were proposed in the literature. Bechert and Bartenwerfer (1989) introduced a so-called “protrusion height” mechanism for different types of riblets and related it to the ability of riblets to impede the cross-flow. Bacher and Smith (1985) put forward the generating mechanism of secondary cross-stream vortex, which suggested that the effect of the riblets is to modify and effectively reduce the momentum exchange properties of the streamwise vortices due to a consequent reduction in surface shear stress. The mechanism

*Corresponding author. Email: donglei2003599@126.com

© 2015 The Author(s). Published by Taylor & Francis. This is an Open Access article distributed under the terms of the Creative Commons Attribution License (http://creativecommons.org/licenses/by/4.0/), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.
of secondary cross-stream vortex had been validated by Lee (2001) and Klumpp, Meinke, and Schröder (2010). Through an extended analysis of the two mechanisms, it has been found that the appearance and size of riblets is the key to determining the drag reduction effectiveness from various studies, such as Walsh and Lindmann (1984), Walsh (1982), Walsh (1983), Lazos and Wilkinson (1988), El-Samni, Yoon, and Chun (2005), Choi, Moin, and Kim (1993), Qamar and Sanghi (2012), and Boese and Fottner (2002). Drag reduction effectiveness will be improved if the riblet structure is immersed into the lower part of the viscous sublayer and transition layer; otherwise, it will be weakened if the sharp corner of the riblet is completely beyond the turbulent boundary layer. However, most research on riblets’ surface drag reduction is focused on plates. If the riblets’ surface drag reduction is applied on complex surfaces, it will be very difficult to solve the problem.

For semi-open three-dimensional impellers, solid walls are composed of hub surfaces and blade surfaces. The blade surfaces of most three-dimensional impellers are ruled surfaces and can be machined by the flank milling method using a cylindrical cutter or a conical cutter. This method can obtain high machining precision and low surface roughness from Gong, Cao, and Liu (2005). As mentioned earlier, the fine surface finish would not always produce higher drag reduction effectiveness. Klumpp, Meinke, Schröder, Feldhaus, and Klocke (2009), Büttner and Schulz (2010) developed the rolling technique and electrodeposition technique, respectively, to manufacture riblets on turbine blades; total drag reduction of 4.7% and 4.9% were obtained, respectively. Furthermore, Lietmeyer, Oehlert, and Seume (2013) had clarified the best choices where riblets should be applied on the blade surface. The results showed a higher potential of the profile loss reduction by riblets on the suction side. Another conclusion of this research is that more drag reduction effectiveness would be got to adopt segmented riblets compared with a constant riblet geometry along the compressor blade surfaces.

The hub surfaces of the impeller are usually machined by ball-end mill or filleted end-mill and the surface roughness is high, whereas some tiny riblets are left on the hub surfaces as shown in Figure 1. Currently, there are no related works on this kind of tiny riblets, which would affect the boundary flow and the aerodynamic performance of compressors.

As shown in Figure 2, Kang and Hirsch (2001) conducted an in-depth study on a number of centrifugal impellers and observed different vortices in the passage of the centrifugal impeller. Among them, PVH and CV are streamwise vortices near the hub surface. These streamwise vortices are able to concentrate low-speed fluid in the spanwise sense and trigger an outward eruption of low-momentum fluid that has unquestionably adverse effects on the flow in the centrifugal impeller. However, for impellers with riblets on their hub surface, the results may be different. According to the mechanism that the secondary cross-stream vortex is generated by the interaction of the riblet tip and the flow above, it can be speculated that a group of secondary cross-stream vortices will be generated at the peaks of the riblets, which can inhibit the spanwise concentration of low-speed fluid into streak formations, weaken PVH and CV to a certain degree, reduce the wall shear stress of turbulent boundary layer flow, as well as restrain the development of the boundary layer so as to realize the drag reduction effectiveness. One of the key points that is used to determine the drag reduction effectiveness is the sizes of riblets. Drag reduction effectiveness will be weakened with a large cusp height left after hub surface finish; moreover, the aerodynamic performance of the compressor will deteriorate due to a larger friction area, so $B_{\text{max}}$ should not be too large. On the other hand, drag reduction effectiveness will also be weakened even if the machining precision and surface roughness were improved; moreover, the processing costs will also increase correspondingly. In order to obtain the optimal drag reduction effectiveness of the residual riblets on the hub surface and avoid the increase in processing costs from the unnecessary decrease in the $B_{\text{max}}$ value, drag reduction effectiveness of different cusp height and its effect on the aerodynamic performance of a compressor are investigated in this article, which aims at obtaining the optimal $B_{\text{max}}$ value.

There are several difficulties in our research. First, due to the complexity of the impeller’s hub surface, it is difficult to distribute the geometric modeling and mesh generation for riblets. To accurately predict the flow condition of fluids near the residual riblets, a finer grid scheme in the boundary layer should be used. Furthermore, since the size and density of the riblets of different positions are variable, these bring big troubles to flow field analysis. Meshes are divided by manual operation in this article. Every rib element is set as an independent block to guarantee the density and quality of the grid. The flow field
Figure 2. Sketch of secondary vortices in centrifugal impeller (Extracted from Kang and Hirsch (2001)).

analysis primarily focuses on the analysis of wall shear stress; in-depth analyses are conducted on the wall shear stress of the rib surface from the whole to the part. The content of this article is managed as follows:

In this article, the 3D steady viscous flow field of a small centrifugal impeller is numerically studied. Impellers with different values of $B_{\text{max}}$ after the hub surface finish are modeled and meshed in §2.2 and §2.3, followed in §3.1 by a grid independence test. After the single-flow passage numerical simulation presented in §3.2, the generation of secondary cross-stream vortices is confirmed by an analysis of velocity vector distribution over the cross-section at 90% of the flow passage from inlet to outlet in §3.3.1; meanwhile, the hub surface wall shear stress is analyzed from the whole to the part in §3.3.2. By comparing the polytropic efficiency of compressors that apply different values of $B_{\text{max}}$ after hub surface finish, drag reduction characteristics of residual riblets on the hub surface are also reported in §3.3.2. A performance experiment is carried out and described in §4; the result is well matched with simulation results, followed by a brief conclusion in section 5.

2. Numerical simulations

2.1. Research object

As shown in Figure 3, a small-sized centrifugal compressor is selected as the research object, which consists of a semi-open impeller, a vaneless diffuser, and a volute as its major components. The primary structure dimensions and flow conditions are summed up as follows.

Geometric parameters of the impeller: the diameter of the impeller is 240 mm, and the number of blades is 16. The exit blade width is 15.96 mm, and average blade tip clearance is 0.5 mm. The flow conditions of the design flow coefficient points for four $\phi$ are listed in Table 1.

2.2. Generation of computational domain

Generally, the hub surface of the impeller is machined by ball-end milling cutter and the circular arc riblets will be left on the hub surface, the distribution of which will be different based on different methods of tool-path planning. In this article, the method of iso-parametric tool-path planning is applied to the hub surface finish. Finally, the residual riblets on the hub surface are the enveloping surface that is formed by the motion of the ball-end milling cutter along the iso-parametric tool path. In this study, a ball-end milling cutter whose radius is 3 mm and half cone angle is 3 degrees is applied to the hub surface finish. Five computational domains where $N$ is 7, 13, 17, 23 and infinite are modeled, respectively, and the maximum cusp height of the hub surface is 0.75 mm, 0.167 mm, 0.093 mm, 0.0489 mm, and 0 mm, respectively. Obviously, the optimization in terms of the number of riblets ($N$) cannot be generalized, as it depends on the impeller size. Thus, a non-dimensional parameter $B_{\text{max}}$ is introduced, which is the percentage ratio rate of the maximum cusp height of the hub surface to the exit blade width. For the impeller researched in this article, when $N$ is 7, 13, 17, 23 and infinite, $B_{\text{max}}$ is equal to 4.70, 1.06, 0.58, 0.31, and 0, respectively. The computational domain where $B_{\text{max}}$ is equal to 1.06 is shown in Figure 4.

2.3. Numerical research method

The steady numerical simulations mentioned earlier are performed for the computational models of the centrifugal impeller using the commercial software CFX. Its computational method is based on the finite volume method. The Shear Stress Transport (SST) turbulence model (1994) is employed, which is a two-equation eddy-viscosity model. The use of a $k-\omega$ formulation in the inner parts of the boundary layer makes the model directly usable all the
way down to the wall through the viscous sub-layer. The SST formulation also switches to a $k-\varepsilon$ behavior in the free stream, thereby avoiding the common $k-\omega$ problem that the model is too sensitive to the inlet free stream turbulence properties. So, it can accurately predict the flow condition of fluids near the residual riblets on the hub surface and obtain precise numerical solutions of separated flows by solving the transport problems of turbulent shear stress. The transport equations for the SST model are listed next:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + \tilde{G}_k - Y_k + S_k$$  \hspace{1cm} (1)

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}(\rho \omega u_j) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega,$$  \hspace{1cm} (2)

where $\tilde{G}_k$ represents the generation of turbulence kinetic energy due to mean velocity gradients; $G_\omega$ represents the generation of $\omega$; $\Gamma_k$ and $\Gamma_\omega$ represent the effective diffusivity of $k$ and $\omega$, respectively; $Y_k$ and $Y_\omega$ represent the dissipation of $k$ and $\omega$ due to turbulence; $D_\omega$ represents the cross-diffusion term; and $S_k$ and $S_\omega$ are user-defined source terms. Furthermore, the high resolution scheme put forward by Barth and Jesperson (1989) is applied to discretize the turbulence equation in order to obtain the flow information of a structured grid within the boundary layer more accurately. The impeller is solved on the rotating coordinate system, and all stators are solved on the absolute reference frame. The type of interfaces between stators and rotors is set to be frozen rotor.

The all-hexahedral meshes for the computational domain of the impeller, vaneless diffuser, and volute are created using commercial software ICEM. Meshes near all solid walls are refined geometrically in the normal direction of the solid walls and $y+$ of all the solid walls are maintained below 2. A meshed computational domain of single-flow passage of the impeller is shown in Figure 5, where $B_{\text{max}}$ is equal to 1.06.

Boundary conditions are treated as follows: Total temperature and total pressure are specified at the inlet, and mass flow rate is specified at the outlet; the adiabatic and non-slip condition is applied at all the solid walls; the blades and hub surfaces are set as rotating surfaces; and the other solid walls are set as stationary surfaces.

### 3. Results and analysis

#### 3.1. Grid independent test

In order to obtain the grid independent results, a grid sensitivity study should be conducted for all models. Mesh1, 2, and 3 are, respectively, divided by manual operation for the five computational domains, which are mentioned earlier. For mesh1 to mesh3, the grid is refined by scaling up the number of nodes on each edge, so the grid quantity of mesh1, 2, and 3 increased successively, which is shown in Figure 6. Numerical simulations of a single-flow passage are performed for all computational models at the design flow coefficient point of $\text{Mu} = 0.95$; the volute is not included in the numerical simulations, and the outlet boundary of the computational domain is on the exit plane of the vaneless diffuser. The predictions are shown in Figure 7. Three points in each line are results of mesh1,
mesh2 of all models can accurately predict the flow conditions of fluids near the residual riblets on the hub surface, the quantity of grid nodes in the residual riblets of all models’ mesh2 is further scaled up, and numerical simulations are performed again using methods mentioned earlier for all computational models. Error for inlet-to-outlet total pressure ratio and compressor polytropic efficiency are less than 0.06%, based on which the credibility of all models’ mesh2 is confirmed. The results imply that more accurate results can be obtained to perform the numerical simulation by using mesh2 from each model, and the quantity of grid nodes is 1.02 million, 1.16 million, 1.62 million, 1.79 million, and 0.8 million, respectively, when N is 7,13,17,23 and 0. Furthermore, as shown in Figure 6, the grid quantity of mesh2 from all models increases with an increase in N value. Thus, it can be seen that it will need a larger grid quantity to obtain more accurate results with the increase of the density of riblets aligned on the hub surface except that N is infinite.

3.2. Simulation results of a single-flow passage
Comparing the aerodynamic performance of impellers with different sizes of riblets aligned on the hub surface, the numerical simulations of a single-flow passage for five computational domains are performed at the design flow coefficient points of four Mu using mesh2 from each model, the predicted results are depicted in Figures 8–10, which show that the inlet-to-outlet total pressure ratio and polytropic efficiency of the compressor are the lowest when $B_{\text{max}}$ is equal to 4.7. With the decline of the $B_{\text{max}}$ value, the two parameters gradually increase and reach a maximum when $B_{\text{max}}$ is equal to 0.31. However, the discrepancy of the two parameters is very small when $B_{\text{max}}$ is equal to 0.31, 0.58, and 0, based on which it is considered that the two parameters reach a peak and stable value when $B_{\text{max}}$ is reduced to 0.58. It can be seen from Figure 10 that the average shear stress on the hub surface reaches a maximum when $B_{\text{max}}$ is equal to 4.7, then gradually decreases with

mesh2, and mesh3, respectively. It is obvious that the predicted results tend to be stable along with the gradually refined mesh; error for inlet-to-outlet total pressure ratio and compressor polytropic efficiency between mesh2 and mesh3 are less than 0.21%. In order to confirm whether

Figure 6. Grid quantity of all models in grid independence test.

Figure 7. Results of grid independence test.

Figure 8. Comparative results of total pressure ratio.
the decline of the $B_{\text{max}}$ value, and reaches a minimum when $B_{\text{max}}$ is equal to 0.58, which means that the flow resistance is the smallest at this moment. After this minimum, it starts increasing again with the decline of the $B_{\text{max}}$ value. When $B_{\text{max}}$ comes to 0, the average shear stress and the aerodynamic performance of the compressor are very close to the ones when $B_{\text{max}}$ is equal to 0.31. So, it can be considered that the aerodynamic performance of the compressor reaches a peak and stable value when $B_{\text{max}}$ is reduced to 0.58, and it is the optimum $B_{\text{max}}$ value. The varying tendencies of the three parameters are the same for the four $\mu$ mentioned earlier, which implies that the effect of residual riblets of the impeller’s hub surface on the aerodynamic performance of the compressor is closely related to the size of the riblets; constantly, there should be an optimum value for the cusp height, which corresponds to an optimum $B_{\text{max}}$ value and $N$ value. Under this number of tool passes, the average shear stress on the hub surface reaches a minimum and the drag reduction effectiveness is the best; the total pressure ratio and polytropic efficiency of the compressor also reach a peak and stable value.

3.3. Flow field analysis

3.3.1. Flow structure analysis

To compare and analyze the near-wall flow structures of the turbulent boundary layer over the residual riblets’ surface, a cross-section of the flow passage at 90% of the flow passage from the inlet to the outlet is taken as shown in Figure 11(a), and velocity vector distribution over the cross-section of the flow passage of smooth hub surface and riblets’ hub surface is indicated in Figure 11(b) and Figure 11(c), respectively. Based on Figure 11(c), an interaction between streamwise vortices and the riblets’ structure enable flow direction of the fluid to deflect at
the tips of residual riblets so that a group of secondary cross-stream vortices are generated and accelerated. The streamwise vortices that spawn the secondary cross-stream vortices are weakened and retain lots of low-speed fluids within the riblets, which are restrained from rising in order to avoid the generation of large vortices and retard the development of the turbulent boundary layer. This is the root cause of riblets’ surface drag reduction. A schematic
will be generated. And riblets' surface, and secondary cross-stream vortices will be deflected due to an interaction between the fluids shown in Figure 14, it is obvious that velocity of the fluids tips of the riblets. From the limiting streamline, which is more secondary cross-stream vortices are distributed on the riblets, and with an increase in the depth of the cusp height, secondary cross-stream vortices are generated at the tips of the impellers. From which it is found that a group of secondary cross-stream vortices above the residual riblets on the hub surface, where Bmax is equal to 0 and Bmax = 0.58, respectively, as shown in Figure 15(a). The residual riblets on the hub surface. It will be certain that the “jet-wake” structure have the characteristic of high-entropy. Cross section a, b and c are taken at the same position when Bmax = 0 and Bmax = 0.58 compared with that when Bmax = 0. As can be seen, the “jet-wake” structure could be improved to a certain extent by the residual riblets on the hub surface.

3.3.2. Analysis of wall viscous shear stress

Figures 16 and 17 show the shear stress contour on the impellers’ hub surface, where Bmax is equal to 0 and 0.58, respectively.

The internal flow field of the centrifugal impeller near the hub surface is more complicated, compared with plate flow. Shear stress on the smooth hub surface and hub surface aligned with residual riblets is not uniformly distributed as shown in Figure 16 and Figure 17. It seems that they are similar as a whole, shear stress on the hub surface gradually increases from the inlet to the outlet of the impeller. Shear stress on the smooth hub surface is more evenly distributed, analyzed from the local region; however, when it comes to shear stress on the hub surface with residual riblets, some large differences between the tips and valleys of riblets will emerge. The inlet of flow passage is narrower than the outlet, which results in the gradual increase of cusp height from the inlet to the outlet. The height of riblets is lower at the upper and middle part of the hub surface, where the wall shear stress is smaller as well compared with that at the middle and lower part, so that fluids are less likely affected by the residual riblets and shear stress is more evenly distributed in this region. However, the distribution of wall shear stress is obviously

Figure 14. Limiting streamline diagram of hub surface.
affected by the residual riblets and differs a lot between the tips and valleys of riblets at the middle and lower part of the hub surface. In order to further analyze the drag reduction effectiveness of the residual riblets, local wall shear stress distribution in the same place of the hub surface when $B_{\text{max}}$ is equal to $4.7, 0.58, 0.31, \text{and } 0$ is comparatively analyzed. A part of the intersecting line of the cross-section with the hub surface between cartesian coordinates of $(-0.007, 0.12, Z_{1B_{\text{max}}})$ and $(-0.013, 0.12, Z_{2B_{\text{max}}})$ is taken, which is shown in Figure 11(a), where $Z_{1B_{\text{max}}}$ and $Z_{2B_{\text{max}}}$ are the coordinates at the axis direction of the impeller. Due to different sizes of residual riblets, $Z_{1_{4.7}}, Z_{1_{0.58}}, Z_{1_{0.31}}, \text{and } Z_{1_{0}}$ are different, and $Z_{2_{4.7}}, Z_{2_{0.58}}, Z_{2_{0.31}}, \text{and } Z_{2_{0}}$ are also different. Then, points are taken uniformly from the part of the intersecting line. Horizontal ordinate 'A' represents the position of a point on the intersecting line that is also shown in Figure 11(a). The shear stress of every point is taken to analyze the distribution rule of the shear stress on the hub surface with different $B_{\text{max}}$ values, and the results are shown in Figure 18.

For the smooth hub surface, the shear stress distribution is more uniform, but the nonuniform distribution for shear...
stress on the riblets surface is obvious. Regardless of what the $B_{\text{max}}$ value is, the minimum points of the shear stress are at the bottom of the riblets and significant upward shifts are obtained at both sides of them, which reach a maximum at the tip points of the riblets. Through the comparison of different residual riblets on the hub surface where $B_{\text{max}}$ is equal to 4.7, 0.58, 0.31, and 0, it is seen that shear stress is the smallest around the riblets’ valley but is the largest around the riblets’ tips when $B_{\text{max}}$ is equal to 4.7, on comparing with the ones when $B_{\text{max}}$ is equal to 0.58 and 0.31. With the decline of $B_{\text{max}}$, the height and width of residual riblets decrease accordingly. When $B_{\text{max}}$ is equal to 0.58, shear stress around riblets’ valley is larger than that when $B_{\text{max}}$ is equal to 4.7, but the shear stress around the
riblets’ tips is much smaller than that when $B_{\text{max}}$ is equal to 4.7. Shear stress around the riblets’ valley continues to increase with $B_{\text{max}}$’s decline, and shear stress around the riblets’ tips also increases. So, when $B_{\text{max}}$ reaches 0.31, the drag reduction effectiveness becomes very small. By integration, results are explained as follows: Total shear stress on the intersecting line when $B_{\text{max}}$ is equal to 0.58 is much smaller than that when $B_{\text{max}}$ is equal to 4.7, 0.31, and 0, and it is consistent with the results of average shear stress of the whole hub surface shown in Figure 10.

4. Numerical simulation of full flow passages and experimental verification

4.1. Experimental rig

Three impellers are machined in order to verify the accuracy of the earlier results, which apply 13 ($B_{\text{max}}$ is equal to 1.06), 17 ($B_{\text{max}}$ is equal to 0.58), and 23 ($B_{\text{max}}$ is equal to 0.31) as the number of tool passes in a single-flow passage in hub surface finish, respectively. The ball-end milling cutter whose radius is 3 mm and half cone angle is 3 degrees is applied. The residual riblets left on the hub surface are shown in Figure 19. It is well known that the tip clearance has a great impact on the performance of the compressor. The tip clearance contains two parts, the radial clearance and the axial clearance, as shown in Figure 20(a). The radial clearance is secured by machining precision. The machining precision of processing equipment in our factory is so high that the machining error could be constrained below 0.01 mm. So, the differences of the radial clearances among the tested impellers could be neglected. The axial clearance is secured during the assembling process, which could have a greater impact than the radial clearance. In order to reduce the influence of the axial clearance, some coating was added to the internal surface of the rings before the performance experiments as shown in Figure 20(b). The impeller was pushed along the axial direction during the assembling process to cling to the internal surface of the rings. During the experiment, the rotating impeller will shave off some of the coating. In fact, the axial clearance is almost 0 for all the tested impellers. So, the differences of the axial clearances of different impellers are almost negligible. In conclusion, the efficiency change due to different tip clearances could be neglected.

Measurements are performed in a single-stage centrifugal compressor on a high-speed test bed of Shenyang Blower Works Group Co, LTD, China. Figure 21 is a photograph of experimental arrangements.

4.2. Experimental method

The overall performance of the experimental rig is measured for the first three $\mu$ mentioned earlier, and three measurements near the design flow coefficient points are
Figure 19. Residual riblets left on hub surface.

Figure 20. Tip clearance.

Figure 21. Photograph of experimental arrangements.

taken for each Mu. Four thermocouple probes and four total-pressure probes are arrayed along the circumference of the intake pipe to measure the total temperature and total pressure of the inlet of the rig; another four thermocouple probes and four total-pressure probes are arrayed along the circumference of the outtake pipe to measure the total temperature and total pressure of the outlet of the rig. The overall performance is calculated from the arithmetic mean values of the total pressure and total temperature of the inlet and outlet measured earlier.

4.3. Numerical simulation of full flow passages

The volute is not included in the numerical simulations of a single-flow passage; in order to be consistent with the experimental conditions, a numerical simulation of full flow passages is also performed, because numerical simulation of full flow passages is time consuming. Therefore, it is conducted only for Mu = 0.95, and the calculation domain is shown in Figure 3. Mesh 2 of the three computational domains when B_{max} is equal to 1.06, 0.58, and 0.31 and mesh of volutes are applied. The boundary conditions (T_{inlet}, P_{inlet}, and Q_m) are set to ones measured in the experiment, and boundary conditions are maintained the same as the ones in the numerical simulations of a single-flow passage.

4.4. Results and discussion

In Figure 22(b), Figure 22(e), a comparison of polytropic efficiency and total pressure ratio is made between performance experiments and numerical simulations of full flow passages when Mu = 0.95 for three impellers when B_{max} is equal to 1.06, 0.58, and 0.31, which indicate that the experimental trend is in reasonable accord with the numerical trend of the full flow passage and the discrepancies between them at each test point are less than 4.2%. In Figure 22(a), Figure 22(c), Figure 22(d), and Figure 22(f), experiment results of Mu = 0.85, 1.05 are shown, respectively, which indicate that the experimental trend is also
in reasonable accord with the numerical trend of a single-flow passage. From the comparison, it can be found that the influence of the size of the residual riblets on the performance of impellers in the experiments is more evident than that in the numerical simulations, especially when $B_{\text{max}}$ is equal to 1.06. According to the lab record, there is a problem that the radial clearance is about 0.4 mm when $B_{\text{max}}$ is equal to 0.58 and 0.31, but it is 0.5 mm when $B_{\text{max}}$ is equal to 1.06; it is the operator’s mistake in causing it, which is the most important reason that the performance curves when $B_{\text{max}}$ is equal to 1.06 are lower in the experiments. While taking into account the presence of value differences between results of performance experiments and numerical simulations, analyses are made as follows. Four thermocouple probes and four total-pressure probes are arrayed along the circumference of the intake pipe and out-take pipe, respectively, in the performance experiments. First, measurement errors of probes are inevitable; furthermore, value differences coming from different data processing methods should be taken into consideration,
because the arithmetic mean values of measured values are taken as the results to calculate the overall performance of the compressor. Due to limited number of the probes, it is difficult to obtain accurate values of total temperature and total pressure, and the results in numerical simulation are calculated by the mass flow average method, so there are value differences between results of experiments and simulations, but there is no influence on the judgment of the overall trend of the performance curves.

Comprehensive analysis results of experiments and simulations are given as follows. Take the results when $\text{Mu} = 0.85$, the average wall shear stress on the hub surface is, respectively, 75.21 Pa, 66.87 Pa, 61.73 Pa, 63.65 Pa, and 65.35 Pa when $B_{\text{max}}$ is equal to 4.7, 1.06, 0.58, 0.31, and 0 in the single-flow passage numerical simulation, which indicate that the drag reduction effectiveness of riblets improves when the $B_{\text{max}}$ value declines from 4.7 to 0.58, and reaches almost the best when $B_{\text{max}}$ is around 0.58, and the polytropic efficiency of the compressor also comes to a peak value at this moment. Furthermore, the drag reduction effectiveness of riblets declines slightly, and the polytropic efficiency of the compressor changes little when the $B_{\text{max}}$ value declines from 0.58 to 0.31 in the simulations. The polytropic efficiency of the compressor when $B_{\text{max}}$ is equal to 0.58 and 0.31 is almost the same obtained in the performance experiments. However, the polytropic efficiency of the compressor when $B_{\text{max}}$ is equal to 0.58 and 0.31 is significantly better than that when $B_{\text{max}}$ is equal to 1.06 in both simulations and experiments. In conclusion, in the hub surface finish of the impeller researched in this article, the optimum $B_{\text{max}}$ value is around 0.58 in all the working conditions; thus, it is unnecessary to apply a smaller $B_{\text{max}}$.

5. Conclusion

(1) During the working process of the three-dimensional impeller, streamwise vortices above residual riblets on the hub surface interact with the tips of the riblets, which makes the velocity deflexion of fluids happen in the rotating direction of the impeller. The secondary cross-stream vortex forms above the sharp corner of the riblet. All secondary cross-stream vortices above the riblets constitute the structure of the secondary cross-stream vortices group, which cause lots of low-speed fluids to be retained inside the riblets. Since the low-speed fluids are restrained from rising, the generation of large vortices is avoided, so as to produce the drag reduction effectiveness.

(2) The reasonable size and structure of residual riblets can reduce the average shear stress on the hub surface and the viscous drag of the boundary layer on the hub surface, so as to increase the efficiency of the whole compressor.

(3) For the impeller studied in this article, drag reduction effectiveness and aerodynamic performance of the compressor will reach the optimum value when $B_{\text{max}}$ is around 0.58 and the maximum of the cusp height is about 0.093 mm. However, $B_{\text{max}}$ is only one factor to affect the drag reduction effectiveness of the residual riblets. So, even for the impellers with similar size, there are many other factors to influence the drag reduction effectiveness of the residual riblets, such as specific speed, meridional channel shape and so on. But the research methods in this paper offer an idea on confirming the optimal $B_{\text{max}}$ of impellers. A lot more work would need to be done through further research and experiments in the future.

(4) Designers should consider the influence of hub surface residual riblets after finish on the aerodynamic performance of compressors during the arrangement of processing accuracy. If the $B_{\text{max}}$ value of the residual riblets on the hub surface is carefully chosen, it can result in drag reduction effectiveness and improve the compressor efficiency by reducing the flow loss.

Acknowledgments
This research was supported by National Natural Science Foundation of China (51175065); and Shenyang Blower Works Group Co, LTD, China, under the research project NO: KYG2012-14.

Nomenclature

$N$ number of tool passes in a single-flow passage in hub surface finish

$B_{\text{max}}$ percentage ratio rate of the maximum cusp height of hub surface after finish to the exit blade width

$n$ rotating speed of impeller

$D_2$ diameter of impeller

$u_2$ linear velocity of impeller outlet

$c$ local speed of sound

$k$ adiabatic exponent of air, 1.4

$\rho$ air density at room temperature and atmosphere pressure, 1.1691Kg/m$^3$

$T_{\text{inlet}}$ total temperature of compressor inlet

$T_{\text{outlet}}$ total temperature of compressor outlet

$P_{\text{inlet}}$ total pressure of compressor inlet

$P_{\text{outlet}}$ total pressure of compressor outlet

$Q_v$ volumetric flow rate of impeller inlet

$Q_m$ mass flow rate of the whole machine, $Q_v/\rho$

$\varphi$ flow coefficient, $Q_v/(\pi \cdot D_2^2 \cdot u_2/4)$

$\text{Mu}$ machine Mach number, $u_2/c$

$\eta_{\text{pol}}$ polytropic efficiency of compressor, $(k-1)\lg(P_{\text{outlet}}/P_{\text{inlet}})/k \cdot \lg(T_{\text{outlet}}/T_{\text{inlet}})$

References

Bacher, E. V., & Smith, C. R. (1985). A combined visualization-anemometry study of the turbulent drag reducing mechanisms of triangular micro-groove surface modifications. AIAA-85-0548.
Barth, T. J., & Jesperson, D. C. (1989). *The design and application of upwind schemes on unstructured meshes*. AIAA-89-0366.

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

Boese, M., & Fottner, L. (2002, June 3–6). *Effect of riblets on the loss behavior of a highly loaded compressor cascade*. Proceedings of ASME Turbo Expo, Amsterdam, The Netherlands. ASME-GT-2002-30438.

Büttner, C. C., & Schulz, U. (2010, September 28–October 1). Fabrication, oxidation and characterization of shark skin inspired. Proceedings of the ASME 2010 Conference on Smart Materials (SMASIS2010). Philadelphia, PA, USA. ASME-SMASIS2010-3626.

Choi, H. C., Moin, P., & Kim, J. (1993). Direct numerical simulation of turbulent flow over riblets. *Journal of Fluid Mechanics*, 255, 503–539.

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.

Crowe, C. T., Elger, D. F., & Roberson, J. A. (2005). *Engineering fluid mechanics* (8th ed.). New York: John Wiley & Sons.

El-Samni, O. A., Yoon, H. S., & Chun, H. H. (2005). Turbulent flow over thin rectangular riblets. *Journal of KSME International*, 19(9), 1801–1810.

Gong, H., Cao, L. X., & Liu, J. (2005). Improved positioning of cylindrical cutter for flank milling ruled surfaces. *Journal of Computer-Aided Design*, 37(12), 1205–1213.

Kang, S., & Hirsch, C. (2001). Numerical simulation and theoretical analysis of the 3D viscous flow in centrifugal impellers. *Journal of TASK Quarterly*, 5(4), 433–458.

Klumpp, S., Meinke, M., & Schröder, W. (2010). Numerical simulation of riblet controlled spatial transition. *Journal of High Performance Computing in Science and Engineering*, 9, 311–324.

Klumpp, S., Meinke, M., Schröder, W., Feldhaus, B., & Klocke, F. (2009, June 8–12). *Riblets in turbulent flow regimes of 2-D compressor blades*. Proceedings of ASME Turbo Expo, Orlando, FL, USA. ASME-GT2009-59352.

Lazos, B., & Wilkinson, S. P. (1988). Turbulent viscous drag reduction with thin-element riblets. *AIAA Journal*, 26(4), 496–498.

Lee, S. J., & Lee, S. H. (2001). Flow field analysis of a turbulent boundary layer over a riblet surface. *Journal of Experiments in Fluids*, 30, 153–166.

Lietmeyer, C., Oehlert, K., & Seume, J. R. (2013). Optimal application of riblets on compressor blades and their contamination behavior. *ASME Journal of Turbomachinery*, 135, 011036.

Oehlert, K., & Seume, J. R. (2006, July 17–20). Exploratory experiments on machined riblets on compressor blades. Proceeding of Fluids Engineering Division Summer Meeting (FEDSM2006). Miami, FL. ASME-FEDSM2006-98093.

Qamar, A., & Sanghi, S. (2012). Aerodynamics characteristic of axisymmetric surface protuberance supersonic regime. *Journal of Engineering Applications of Computational Fluid Mechanics*, 6(3), 316–335.

Shereena, S. G., Vengadesan, S., Idichandy, V. G., & Bhattacharya, S. K. (2005). CFD study of drag reduction in axisymmetric underwater vehicles using air jets. *Engineering Applications of Computational Fluid Mechanics*, 7(2), 193–209.

Walsh, M. J. (1982, January 11–14). Turbulent boundary layer drag reduction using riblets. *AIAA 20th Aerospace Sciences Meeting*, Orlando, FL. AIAA-82-0169.

Walsh, M. J. (1983). Riblets as a viscous drag reduction technique. *AIAA Journal*, 21(4), 485–486.

Walsh, M. J., & Lindmann, A. M. (1984). *Optimization and application of riblet for turbulent drag reduction*. AIAA-84-0347.