LEADING EDGE SKEWED-SWEPT DIAGONAL COMPRESSOR ROTOR AND NUMERICAL ANALYSIS ON ITS INTERNAL FLOW MECHANISM

BO ZHOU, BIN YOU, WEI WU AND KE-QI WU

College Energy and Power Engineering,
Huazhong University of Science and Technology,
Wuhan 430074, China
choubo8@163.com

(Received 13 November 2007; revised manuscript received 12 December 2008)

Abstract: The structure of each part of a diagonal compressor directly affects its overall performance and internal flow. We introduce the Reynolds-averaged Navier-Stokes flow simulation for unit calculation on the whole system including a diagonal impeller, a vaneless diffuser and a volute. By analyzing different flow chromatograms of specific sections, we can compare the configuration of three types of diffusers and volutes and the meridian flow status of the corresponding diagonal compressors which serves as a basis for the impeller flow path as well as for its matching designs. Considering the interference between the rotor and the upstream and downstream stillness body, this thesis analyzes how the vaneless diffuser meridian flow path, the volute flow path and its section secondary flow affect the upstream rotor flow. Both the calculation and experimental data on the rotor outlet are compared, as well as the calculated numerical value of the meridian plane streamline distribution and the diffuser velocity distribution, upstream and downstream, coincides with the designed numerical value. Without changing the conventional quasi-three-dimensional design system, the thesis applies the annulus wall boundary layer theory and the velocity distribution diagram to sweep and skew the leading edge of the airfoil. A performance test shows that the leading edge skewed-swept diagonal rotor can better improve the stall characteristic in a low flow rate area and expand the surge margin, compared with conventional diagonal rotor. It can also efficiently restrain the low-momentum fluid conglomeration near the wall region and reduce the secondary flow loss by sweeping and skewing the blade properly. The purpose of the thesis is to make a contribution to optimizing the overall structure design of diagonal compressors and to study further the complex internal flow between the leading edge skewed-swept diagonal rotor and the cover.

Keywords: skewed-swept diagonal compressor, vaneless diffuser, performance test

1. Introduction

Taking advantage of the increasing speed of the circumferential blade between the rotor inlet and outlet, diagonal compressors reach an appropriate compromise in terms of the high pressure ratio and mass flow, on the one hand, and a favorable performance and efficiency on the other hand. Mixed-flow compressors have been investigated both experimentally and theoretically for many years; most of the investigations have focused on the outlet way of the impeller axial [1–3], while the flow characteristic about the whole system including the diagonal impeller, diffuser and volute has been studied rarely. Wallace *et al.* [4] have predicted streamline, velocity and pressure distributions in a variety of vaneless diffusers. Whitfield *et al.* [5] have presented total pressure and angular momentum gradients between the hub and shroud surfaces at the diffuser inlet, as produced by the impeller, causing the flow separation in a vaneless diffuser. Abir *et al.* [6] have studied the development of the flow profiles throughout the diffuser with some particular reference to the onset of unstable conditions. Niizeki *et al.* [7] have developed a design method of a radially curved mixed-flow diffuser whose inlet flow has a total enthalpy gradient. The above mentioned research works have been focused mainly on the flow phenomena in each part of the compressors, and they have not launched any further studies on the influence which the downstream component has on the upstream flow, namely the interference between the rotor and stillness body in the whole flow passage.

The structure of each part of a diagonal compressor directly affects its overall performance and internal flow. This paper uses the CFD software to carry out three-dimensional viscosity numerical calculations to the inner fluxion of the whole system including the diagonal impeller, vaneless diffuser and volute. By analyzing different flow chromatograms of specific sections, we compare the configuration of three types of diffusers and volutes and the meridian flow status of the corresponding diagonal compressors. Afterwards, the best of the three is subjected to a detailed internal flow calculation, also the interferences of each part are analyzed.

One of the more significant design trends in recent years has been the use of an aerodynamic sweep to improve the performance and stability of compressor blades [8]. Compared to the conventionally stacked radial rotors, the forward swept blades have demonstrated improvements in terms of the stall margin, efficiency and clearance sensitivity. A forward sweep causes a spanwise redistribution of the flow toward the blade tip and reduces the tip loading in terms of the static pressure coefficient. This results in a reduced tip-clearance flow blockage, a shallower (more axial) vortex trajectory and a smaller region of the reversed flow in the clearance gap.

2. Integrated design of diagonal compressors

The motionless inlet, vaneless diffuser, volute and diagonal impeller comprise the diagonal compressors (see Figure 1). The superiority of the overall function of diagonal compressors is restricted by two factors: one is the aerodynamic performance of impellers; the other is the matching of the overall flow passage (the matching of impellers and fixed components) which mainly refers to the interference of the downstream stillness body and the upstream rotor. The effects of every component on each other are serious. Abstracted from the perspective of mathematics, the flow transmission of a motional body and a stillness body could be finished by the mixed surface between them. The research of this paper focusing on the matching between the diagonal impeller, the diffuser and the volute indicates that the outlet flow non-stationarity of the diagonal impeller always leads to non-stationarity of fluxion in the downstream fixed through-flow components; moreover, the change of the structure of the diffuser and the volute also has some influence on the inner flow of the upstream impeller. The diagonal compressors influx runner is of a complex three-dimensional viscosity and of a non-stationarity fluxion, which not only stresses a certain direction particularly but also influences the whole area.

This paper adopts three different volute distributions as follow-ups to do the influx calculation: a symmetrical circular volute with a radial entry, asymmetric circular volute with a radial entry and an asymmetric circular volute with a diagonal entry. The inwall molded line of circular volutes is designed according to the free fluxion trace. The biggest cross-section radius is 120 mm. The gas flow in the volute follows the momentum moment invariant law. As a convergent channel at the entrance of volute, the vaneless diffuser uses two connection ways of radial and diagonal inflows.

We apply a conventional non-skewed-swept diagonal rotor which bases the on quasi-three-dimensional theory and the NACA65 series of original airfoil for calculation. According to the „imagination equivalent speed triangle", the airfoil type of the rotor is modified considering the influence of the inclined stream surface. And Schlichting's singular method is used to correct the flexivity and original profile, setting the angle according to the potential flow theory. This rotor type has already been applied on a widespread scale. The impeller parameter is as follows: the outlet radius of the impeller is 400 mm, the hub case ratio is 0.75, the inclined angle and outlet setting angles of the blade tip/root are 25°/45°, 57.8°/34.4°, respectively, and the number of blades is 6.

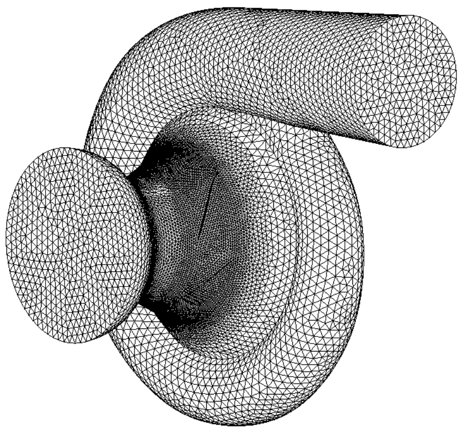

Figure 1. Configuration and calculation domain

3. Impeller design

The low-energy fluid body of the boundary layer can be seen near the hub and the case when the internal flow of a normal unswept impeller is measured. One aim of designing the skewed-swept blade is to activate the low-energy fluid in this part. In the low-flow region where the flow is smaller than the designed flow, we observe the existence of a separation near the housing case which makes the stalling region arise from the blade tip in the experiment. The point of designing the skewed-swept blade

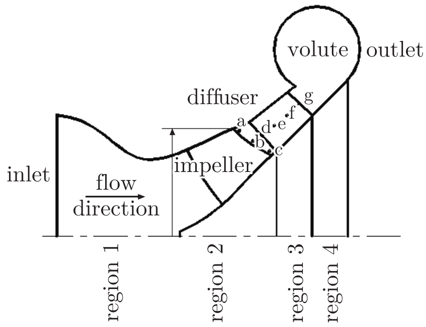

Figure 2. The meridian plane of the calculation domain and distribution of the monitoring points

is to restrain or delay the separation. In this paper, the annulus wall boundary layer theory and velocity distributions are applied to sweep and skew the leading edge of the airfoil, without changing the conventional quasi-three-dimensional design system. The designed skewed-swept diagonal rotor has the following distinctive features: The leading edge is skewed and swept near the hub and the casing wall region. The blade's camber line is the same as the conventional rotor; that is to say, the rotor's outlet flow angle is the same as the conventional rotor. Hence, the leading edge skewed-swept diagonal rotor can be designed under the conventional design system. Figure 3 gives a comparison of the three-dimensional structures, and Table 1 gives the distribution characteristics of a skewed-swept blade.

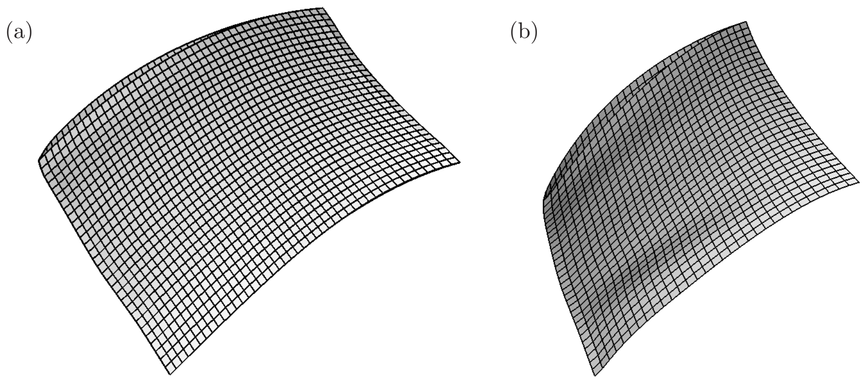

Figure 3. Two different blades: (a) conventional blade; (b) leading edge skewed-swept blade

Table 1. A survey of distribution characteristics of a skewed-swept blade

Blade height ratio	0%	18.8%	33.0%	54.9%	79.8%	93.6 %	100%
Relative thickness	0.010	0.094	0.088	0.076	0.058	0.046	0.040
Cascade solidity	1.481	1.387	1.327	1.242	1.130	1.074	1.054
Bending angle	12.22°	7.041°	0.167°	0°	2.380°	5.110°	6.450°

4. Test facility

The aerodynamic characteristic curves of both the leading edge skewed-swept diagonal rotor and the normal diagonal rotor are measured by automatic testing

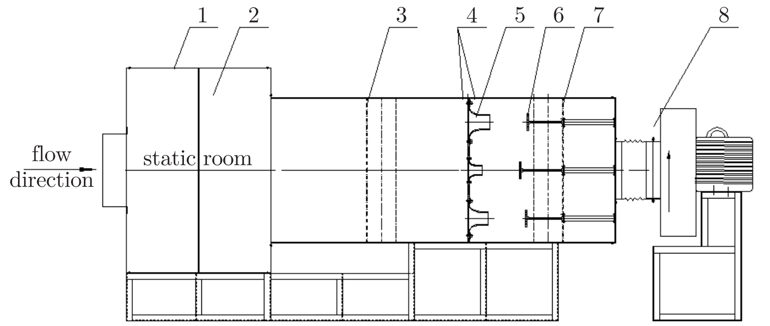

Figure 4. Test system scheme: 1. static opening, 2. humidity and temperature sensor, 3. fore rectifier, 4. differential manometer, 5. nozzle, 6. back rectifier, 7. air-powered choke plug, 8. variable-frequency fan

equipment (see Figure 4) which is developed by us. It meets the AMCA210-85 and GB1236-2000 standards.

5. Results

The flow calculation domain from the upstream rotor is 242.5 mm (pressure inlet surface) to the downstream volute outlet 400 mm (pressure outlet surface), as shown in Figure 2. This region is also the calculation region for the diagonal rotor quasi-three-dimensional design. The whole region is divided into the following four parts: inlet domain, rotating domain, vaneless diffuser through-flow domain and volute stillness domain. In order to control the quality of meshes, each domain is divided into meshes separated for calculation purposes, and the adjacent bodies use the same surface with a node number.

The time averaged three-dimensional Reynolds-averaged Navier-Stokes equations are used in our calculation. The Spalart-Allmaras mode is selected as the viscous mode. And the standard wall function is applied for the domains near the walls. We use the segregate implicit method for calculation. And we adopt first order discrete method for the turbulence kinetic energy, turbulence dissipation and momentum equation. We also use the SIMPLE method for the couple of pressure and velocity.

We calculate three different matching values, using the same impeller with the same boundary condition. Figure 5 gives the distribution of meridional streamline for comparing the effects of three different matching cases. It follows from the figure that the flowage in Figure 5a is better, and it is in coincidence with the designed values. While there is either circumfluence phenomenon as shown in Figure 5b or pressure grads which extrude the fluid to the middle part of blades as shown in Figure 5c in the other two matching cases. The comparison shows that any impropriety of the matching design can easily lead to a secondary flow loss. Therefore, inclined entrance volute matching designs are selected for further calculation.

Figure 6 gives the changing trend of the impeller's radial outlet meridional velocity. It can be seen in Figure 6 that the calculated data is in coincidence with the experimental data of the article [9]. The velocity decreases when the radius increases, and the calculated value is a little higher than the experimental value. As the fluid passes the inclined outlet diffuser and then goes steadily into the volute, it does not

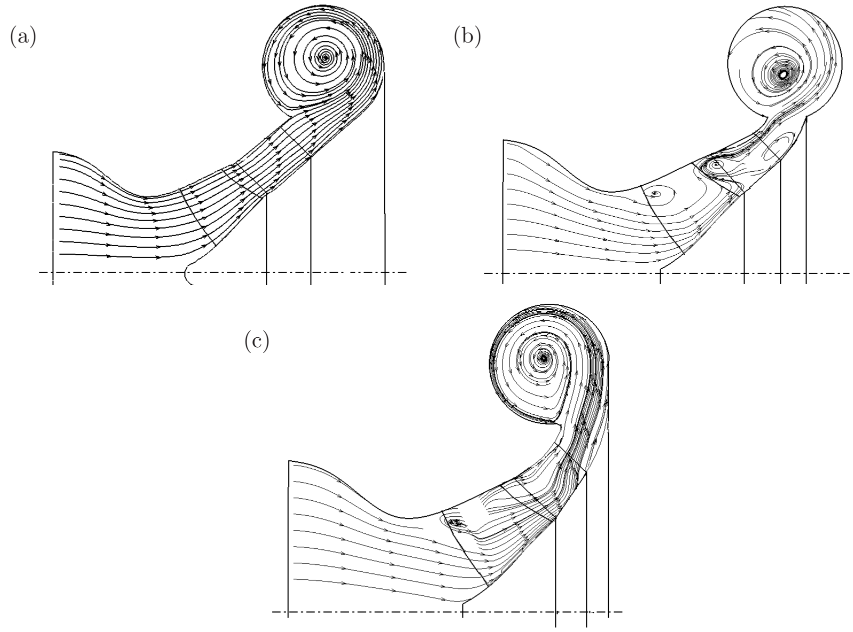

Figure 5. Meridian streamline of three different matching designs: (a) asymmetric circular volute with diagonal entry; (b) symmetrical circular volute with radial entry; (c) asymmetric circular volute with radial entry

need to change the flow direction steeply, so the flux is bigger and the meridional velocity C_{m_2} is bigger than that in the experiment. The calculated value near the blade tip is lower than that in the experiment, as a fraction loss of the inclined outlet diffuser is heavier which makes the velocity decrease rapidly under the boundary layer effect.

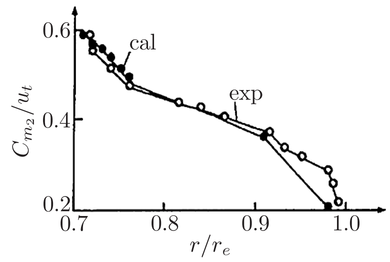

Figure 6. Changing the trend of radial exit meridional velocity

In order to understand the rotor wake and its interrelationship with the volute and the diffuser, we calculate the meridional velocity circumferential distribution of different positions in the rotor outlet and in the diffuser. The wake monitoring points are shown in Figure 2. The meridional velocity radial distribution on three different positions, 5 mm downstream the rotor can be seen in Figure 7. It follows from Figure 7, the meridional velocity at the monitoring point **b** along the volute circumference gradually increases from the inlet to the outlet, which shows that the flow is nonaxisymmetrical. While the velocity at the monitoring point **a** and **c**,

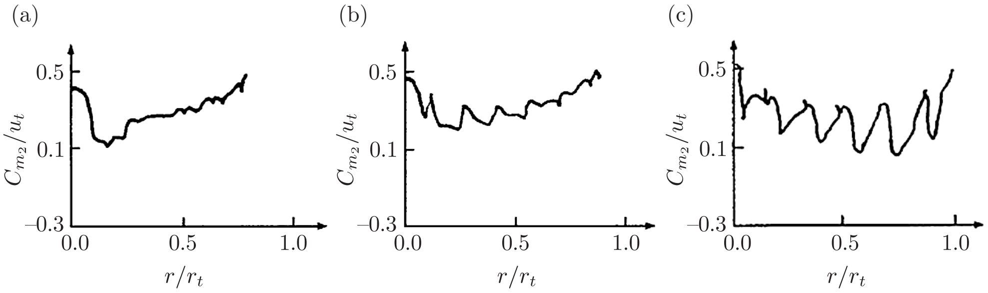

Figure 7. C_{m_2} circumferential distribution 5 mm downstream the rotor: (a) monitoring point **a**; (b) monitoring point **b**; (c) monitoring point **c**

increases generally, but the change scope is not great, which shows that the hub and the case have an effect on the boundary layer.

Figure 8 shows the meridional velocity distributing curves along the diffuser meridional direction at different points, such as diffuser inlet point **d**, 33 mm downstream point **e**, 66 mm downstream point **f** and diffuser outlet point **g**, *etc.* It follows from the figure that the meridional velocity gradually decreases along the flow direction in the diffuser. The meridional velocity In the diffuser inlet section grows circumferentially flatter and flatter. As is shown in Figure 8a, the velocity from the rotor is well-proportioned. At the section 33 mm downstream diffuser, the velocity decreases relatively as shown in Figure 8b. At this time, the diffuser makes the velocity lower, and the kinetic energy is transformed into pressure energy. As the diffuser is vaneless, the velocity does not change greatly along the flow direction after going into the diffuser. At the outlet of the diffuser, the volute has some effect on the upstream flow. The closer it is to the volute inlet, the more obviously the meridional velocity decreases, which shows that the volute has a greater effect on the upstream flow, as shown in Figure 8d.

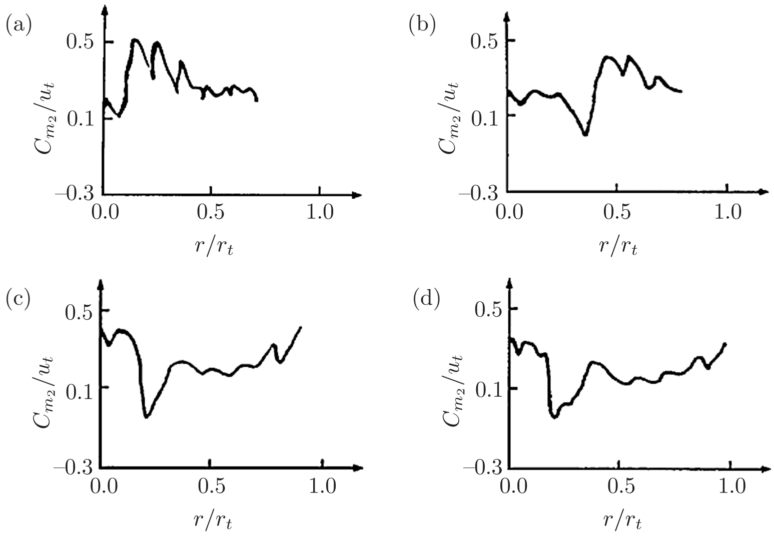

Figure 8. Distributing curves of C_{m_2} along diffuser meridional direction: (a) diffuser inlet point **d**; (b) downstream 33 mm point **e**; (c) downstream 66 mm point **f**; (d) diffuser exit point **g**

Figure 9 shows the secondary flow vector quantities at different sections. It follows from the figure that as the 360° section is near to the outlet and it has a big flow area, the secondary flow is comparatively weak and there is a vortex adjacent to the inner walls. There are clear eddies at the 270° and 180° sections due to the volute asymmetry. As the flux area is the smallest at the 90° section near the inlet, clear eddies exist and a part of the fluid flows back into the diffuser. Hence, when designing the volute molded lines, with a precondition that the flux is ensured, it is necessary to enlarge the cross-section area of the volute, especially in the small sections. We should suitably adopt an arc connection to enlarge the flux area in which we can decrease the loss and improve the diffuser function.

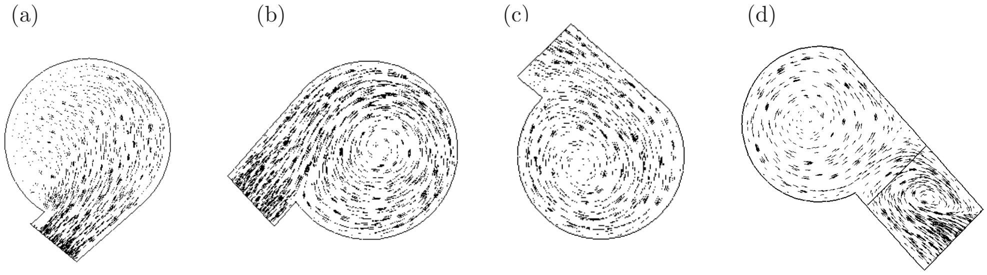

Figure 9. Secondary flow vector at different sections: (a) 360° section; (b) 270° section; (c) 180° section; (d) 90° section

It can be seen in Figure 10 and Figure 11 that the tested results show that the skewed-swept diagonal rotor has diminished the stall region (see Figure 12) in the lower flow rate of the normal diagonal rotor, and it increases the operating range.

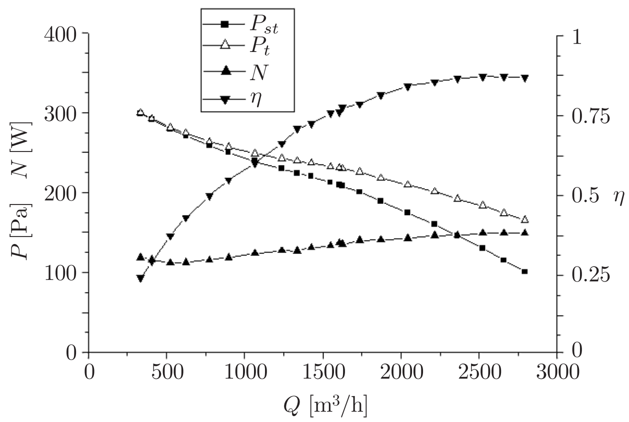

Figure 10. Aerodynamic characteristic curves of leading edge skewed-swept rotor ($n = 1150$ rpm)

It follows from Figure 11 that the calculated dimensionless flux-pressure characteristic curve of the skewed-swept diagonal rotor is in good coincidence with the tested data. They have the same changing trend. The calculated data are a little bigger than the tested ones as we do not take into consideration the influence of the tip clearance during calculation, while there is a two to three millimeter tip clearance in the experiment (considering the machining error of the shell). One cause for that

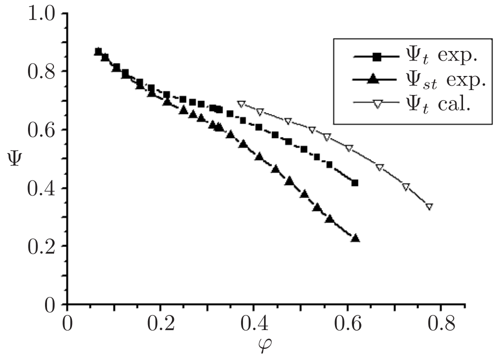

Figure 11. Performance of leading edge skewed-swept rotor

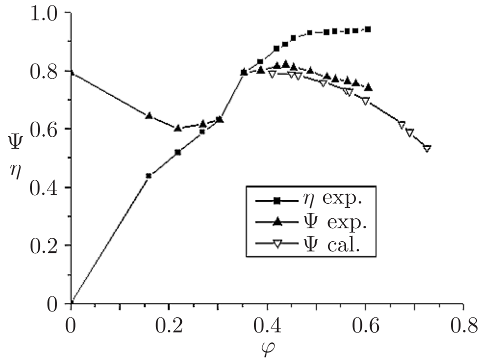

Figure 12. Aerodynamic characteristic curves of conventional diagonal rotor

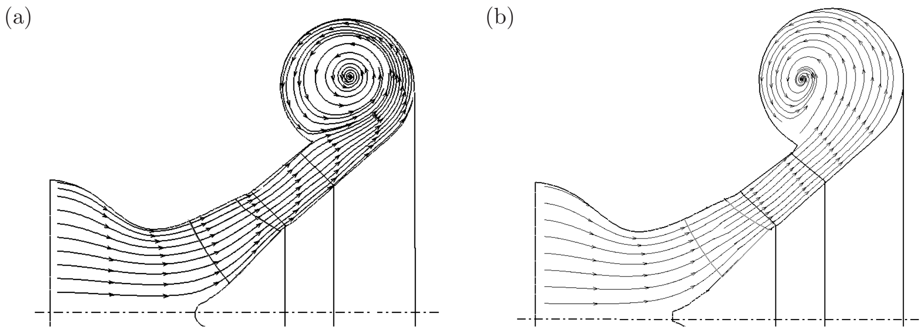

Figure 13. Meridional streamline distribution of two different compressors: (a) conventional diagonal compressor; (b) leading edge skewed-swept diagonal compressor

is that the pressure value in the zero clearance is bigger than that with some clearance. What is more, the local and flow losses in the experiment do not exist in the calculation. The discrepancies between the experiment and the calculation in case of a skewed-swept compressor are not so small: namely, between $\Delta\Psi = 0.05$ to 0.1, and it is much better in the case of a conventional compressor, where the discrepancies are smaller (see Figure 12), as there is a one millimeter tip clearance in the experiment with a conventional compressor.

It follows from the meridional streamline distribution shown in Figure 13 that the meridional flow of the leading edge skewed-swept diagonal rotor is improved; the secondary flow and vortex influence range of the volute outlet section is reduced and deflects from the center to the inner section.

If we magnify the vaneless diffuser meridional streamline of the two diagonal compressors, we will have the circumfluence phenomena (see Figure 14a) in s normal diagonal compressor under the designed working condition. And the circumfluence area disappears while the compressor uses a leading edge skewed-swept diagonal rotor (see Figure 14b). As there is little change of the air flow angle in the vaneless diffuser inlet and the outlet, while the backing pressure increases with the reducing flux engender circumfluence, therefore, there is a vortex in a normal diagonal compressor. A clockwise separating vortex exists for a diagonal impeller, while an anti-clockwise separating vortex exists for a centrifugal compressor because of the radial outlet. That separation eddy may lead to a great pitot loss, so we must pay special attention to the diffuser design.

(a) (b)

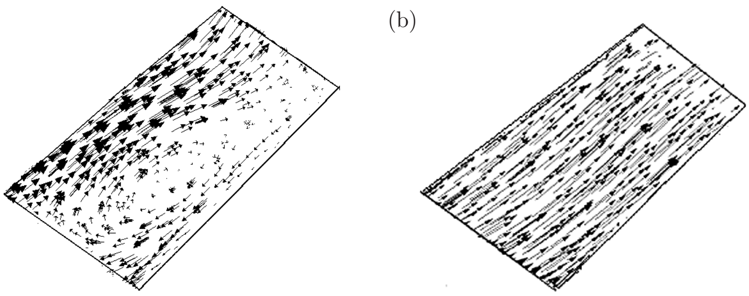

Figure 14. Velocity vector diagram in different compressor diffusers: (a) conventional diagonal compressor; (b) leading edge skewed-swept diagonal compressor

6. Conclusions

1. When comparing the inner flow calculation results of three the matching precepts of a diffuser and a volute, the matching of a diagonal rotor and a asymmetric circular volute with an inclined entry can get a better meridional streamline. The calculation results of the rotor outlet meridional velocity distribution are in coincidence with the experimental data except for the tip near the blade.
2. The quick decay soon after the rotor outlet wakes enter the vaneless diffuser shows an interrelationship between the stillness bodies and the rotor. The matching design should take care of the circumfluence phenomenon arising at the 90° volute section near the inlet.
3. A leading edge skewed-swept diagonal rotor can better improve the stall characteristic in a low flow rate area and enlarge the surge margin compared with a normal diagonal rotor. It can efficiently restrain the development of a secondary flow and reduce the conglomeration of the low-momentum fluid near the wall region by sweeping and skewing the blade properly.

References

[1] Elmendorf W, Kurz H and Gallus H E 1995 *Proc. Int. Gas Turbine and Aeroengine Congress and Exposition*, June 1995, Houston, TX, USA, pp. 5–8

[2] Wei B and Wu K 2000 *J. Engng Thermophysics* **21** (6) 716

[3] Karamanis N, Martinez-Botas R F and Su C C 2001 *J. Turbomachinery* **123** (2) 359

[4] Wallace F J A and Whitfield A 1975 *J. Mech. Engng Sci.* **17** (6) 348

[5] Whitfield A and Roberts D V 1981 *Effect of Impeller Tip Design on the Performance of a Mixed Flow Turbocharger Compressor* **81-GT-7** 7

[6] Abir A and Whitfield A 1987 *Stability of Conical and Curved Annular Diffusers for Mixed-Flow Compressors*, Papers Presented at the ASME Gas Turbine Conference and Exhibition, Anaheim, CA, USA

[7] Niizeki Y S and Toshimichi S 1995 *Nippon Kikai Gakkai Ronbunshu* **61** (583) 990

[8] Decker J J, Beacher B F, McNulty S G and Arif Khalid S 2003 *ASME Turbo Expo*, June 2003, Atlanta, GA, United States, pp. 16–19

[9] Inoue M and Wu K 1984 *Proc. of China-Japan Joint Conference on Hydraulic Machinery and Equipment*, Hangzhou, pp. 21–25