International Journal of Fluid Machinery and Systems
Vol. 6, No. 3, July-September 2013

DOI: http://dx.doi.org/10.5293/IJFMS.2013.6.3.160
ISSN (Online): 1882-9554

Review Paper (Invited)

Study of Mechanism of Counter-rotating Turbine Increasing Two-Stage Turbine System Efficiency

Yanbin Liu[1,3], Weilin Zhuge[1], Xinqian Zheng[1], Yangjun Zhang[1], Shuyong Zhang[2], Junyue Zhang[2]

[1]State Key Laboratory of Automotive Safety and Energy, Tsinghua University
Beijing, 100084, China, liu-yb10@mails.tsinghua.edu.cn, zhugewl@tsinghua.edu.cn, zhengxq@tsinghua.edu.cn, yjzhang@tsinghua.edu.cn
[2]National Key Laboratory of Diesel Engine Turbocharging Technology
Datong Shanxi, 037036, China, 13935298796@163.com, dt70zjy@163.com
[3] Department of Mechanical engineering, Academy of Armed Force Engineering
Beijing, 100072, China

Abstract

Two-stage turbocharging is an important way to raise engine power density, to realize energy saving and emission reducing. At present, turbine matching of two- stage turbocharger is based on MAP of turbine. The matching method does not take the effect of turbines' interaction into consideration, assuming that flow at high pressure turbine outlet and low pressure turbine inlet is uniform. Actually, there is swirl flow at outlet of high pressure turbine, and the swirl flow will influence performance of low pressure turbine which influencing performance of engine further. Three-dimension models of turbines with two-stage turbocharger were built in this paper. Based on the turbine models, mechanism of swirl flow at high pressure turbine outlet influencing low pressure turbine performance was studied and a two-stage radial counter-rotation turbine system was raised. Mechanisms of the influence of counter-rotation turbine system acting on low-pressure turbine were studied using simulation method. The research result proved that in condition of small turbine flow rate corresponding to engine low-speed working condition, counter-rotation turbine system can effectively decrease the influence of swirl flow at high pressure turbine outlet imposing on low pressure turbine and increases efficiency of the low-pressure turbine, furthermore increases the low-speed performance of the engine.

Keywords: Two-stage turbocharger, Swirl flow, Interaction, Radial counter-rotation turbine, Turbine efficiency, Three-dimension simulation

1. Introduction

Turbocharging is an effective way to raise engine power density, save energy and control level of emission. Nowadays, engine must has higher power density to solve the problem of energy and make use of higher EGR rate to reduce emission, which makes engine raising intake pressure ratio up to 5. To meet the engine demands, turbocharging for vehicle was developed from single stage turbocharging, variable-area turbocharging to two-stage turbocharging.

One advantage of turbocharging is the energy of exhaust gas being recovered by turbine. As the driver parts of turbocharger, the outputting power of turbine decides the performance of the turbocharging system, and influences engine performance further. So turbine designing and matching is the key issue in the design of the whole turbocharging system.

Josef Božek Research Center studied the matching method for the two-stage turbocharger of a diesel engine, and raised a new method for turbine matching which is aiming to improve the engine performance by raising MAP, a decisive factor of turbine[1]. Voith Turbo Aufladungssysteme GmbH & Co KG made some researches on the performance changing law of turbines in two-stage turbocharger under high inlet pressure and temperature and put forward a method to raise 1D simulation precision by revising the turbine MAP based on its working conditions[2]. In fact, this is a turbine matching method based on turbine MAP. Relatively appropriate turbines for a particular engine can be selected by this method quickly and conveniently, and performance of the engine can basically meet the demands. While, in this method it is assumed that the flow field between two stages is uniform, ignoring the interaction of the two turbines. Actually, there are swirl flows at the outlet of the high pressure turbine, then these swirl flows spread to

Received August 31 2012; revised February 28 2013; accepted for publication March 4 2013: Review conducted by Prof. Hyung-Hee Cho (Paper number R13007S).
Corresponding author: Yangjun Zhang, Professor, yjzhang@tsinghua.edu.cn
This manuscript was presented at the 26th IAHR Symposium on Hydraulic Machinery and Systems, August 19-23, 2012, Tsinghua University, Beijing, China.

low pressure turbine and change the flow conditions in the turbine which will also influence the performance of the low pressure turbine. Saab Automobile Powertrain AB made comparative tests to analyze the performance of one stage turbine and the low pressure turbine in a two-stage turbocharger. The researchers found that the efficiency of the low pressure turbine decreased greatly, and they owe the phenomenon to the interaction between the two turbines in a two-stage turbocharger[3]. The Royal Institute of Technology studied the action law and mechanism of the secondary flow at turbine inlet acting on the turbine. The result proved that there is difference between the performance in the condition of the secondary flow at the turbine inlet and uniform flow at turbine inlet which reflected that the outlet cortex in the high-pressure turbine would influence the performance of the low-pressure turbine[4].

Tsinghua University studied the influence of two-stage turbines' efficiency on the performance of the whole engine. The research result proved that output torque and fuel consumption of the engine under low-speed and part load working conditions can be raised greatly by improving the efficiency of the low- pressure turbine [5]. So it is necessary to study the interaction law and mechanisms of the flow fields between the two turbines and the method to control the flow field for the purpose of improving the performance of two-stage turbocharger and the engine further.

2. Three-dimension flow simulation model for two-stage turbocharger

Two-stage turbocharger simulation model includes high pressure turbine, low pressure turbine and the connection pipe. The layout of the turbine system is shown in figure 1. The overall arrangement is decreased the effect of connection pipe acting on the flow field between the two turbines [6]. The rotation axes of the two turbines are perpendicular to each other, and the pipe is made together with the volute of low pressure turbine. The outlet flow of high pressure turbine flows into the volute of low pressure turbine along the rotation axis. The arrangement reduced the absolute pressure loss which will help raise the efficiency of two-stage turbine system.

The flow field interaction between the two turbines mainly refers to the action that the flow field at high-pressure turbine outlet imposes on that of the low-pressure turbine, which will influence the performance of low pressure turbine. Two-stage turbine system model for numerical calculation consist of three parts: high-pressure turbine, volute of the low-pressure turbine and the rotor of the low pressure turbine for the purpose of reducing calculation error and ensure the model are able to reflect the interaction between the two turbines.

N-S equation including mass conversation equation, momentum conservation equation and energy conservation equation is used as the flow equation in our numerical calculation.

The basic expression is shown as follows:

$$\frac{\partial \vec{U}}{\partial t} + \nabla \vec{F} = \nabla \vec{Q} \tag{1}$$

In the equation, the first item reflects influence time imposing on flow state; the second item reflects influence brought in by contra-flow; the third item refers to source item.

A time matching method was used to solve it. Spatial discretization of equation uses Jameson centered difference scheme and four-level Runge-Kutta method were applied to calculate the spatial discretization and time discretization solutions. Turbulence model uses S-A[7] model was used as the turbulence model to ensure that the flow equation is closed.

The boundary conditions of numerical calculation model can be obtained from two-stage turbocharger test. Turbocharger test bench was built with flow sensor, total temperature sensor , total pressure sensor and static pressure sensor fixed in front of the high pressure turbine, and total temperature sensor, total pressure sensor and static pressure sensor fixed between the two turbines and behind the low pressure turbine. The detailed layout of the test bench is shown below in Figure 2.

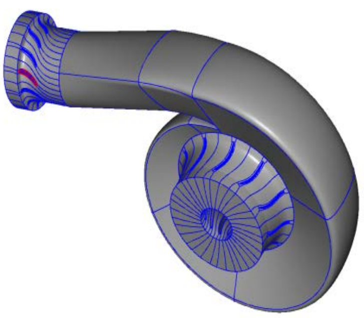


Fig. 1 Arrangement of two stage turbocharger

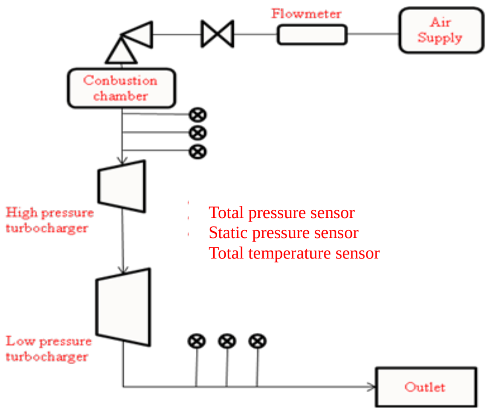


Fig. 2 Test bench for turbine system

In test, the different working states of two turbines were got by adjusting air supply. Flow mass, rotation speed of each turbine, total temperature, total pressure and static pressure in different working condition were recorded which can be used as boundary condition when turbine flow simulation.

Mesh generation for the calculation model referred to reference [8]. Because the paper mainly focuses on influence high pressure turbine imposing on low pressure turbine, calculation mesh is divided into three parts. One is high pressure turbine rotor, the second is connection pipe and low pressure turbine volute and the third is low pressure turbine rotor which can decrease calculation time consumption.

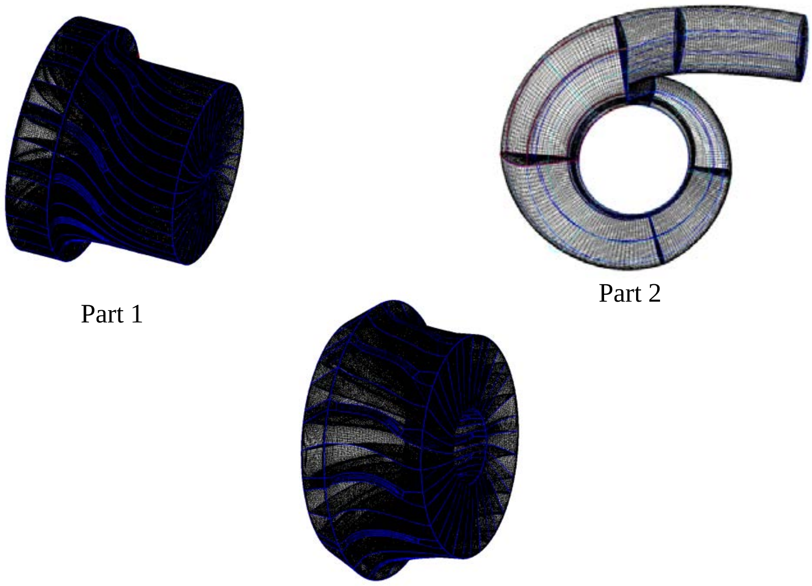


Fig. 3 Mesh of turbine system of two-stage turbocharger

Mesh generation technology was discussed in detail and verified reference [8], so it can be regarded credible to use the mesh generation method mentioned in the paper for turbines in two-stage turbocharger.

3. Flow field of high pressure turbine outlet

In the design of turbines, the outlet flow of turbine should be along rotating axis to achieve a higher efficiency. So the angle between rotor outlet blade and rotor rotating direction exceeds 90°. The shape of turbine rotor is shown in Figure 4.

As the tangential velocity of outlet blade differs at different radial positions of the blade, the declination angle of outlet blade differs. With the rotor blade declines along the passageway, the fluid flow in passageway will decline. The declination angle is determined by the outlet blade angle. So the absolute outlet velocity is mainly determined by blade tangential outlet velocity and the relative velocity to the rotor. The velocity triangle is shown in figure 5.

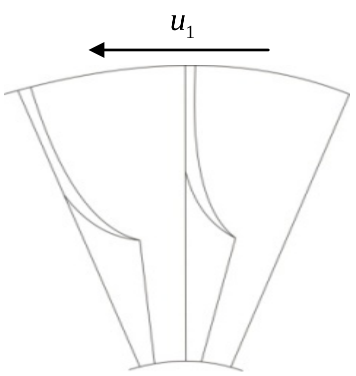


Fig. 4 Shape of turbine rotor

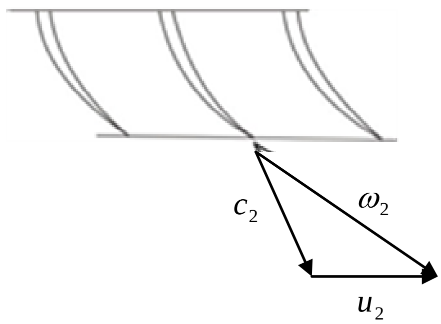
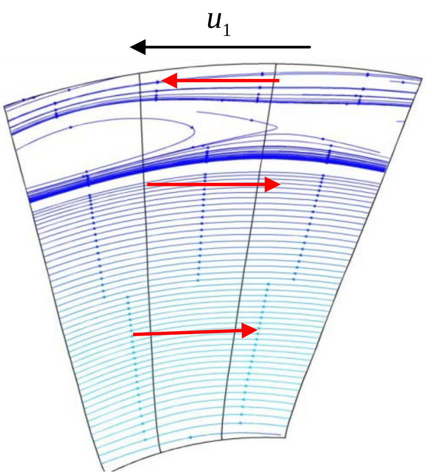


Fig. 5 Triangle of velocity at rotor outlet

In figure 5, ω_2 refers to relative velocity in rotor passage, u_2 refers to rotor rotation tangential velocity, c_2 refers to flow absolute velocity.

Under different working conditions, the tangential velocity at rotor outlet is different, so is the relative velocity which results in flow at turbine rotor outlet declining towards or against the rotating direction [9].

The flow state of high pressure turbine was simulated under low-speed and part-load conditions. Mass and total temperature is given as the inlet boundary conditions with the magnitude of 0.558kg/s and 970K separately. Total pressure of 190000 Pa is given as the outlet boundary condition, which is obtained by two-stage turbocharger test results. The detailed flow field distribution is shown in Figure 6.



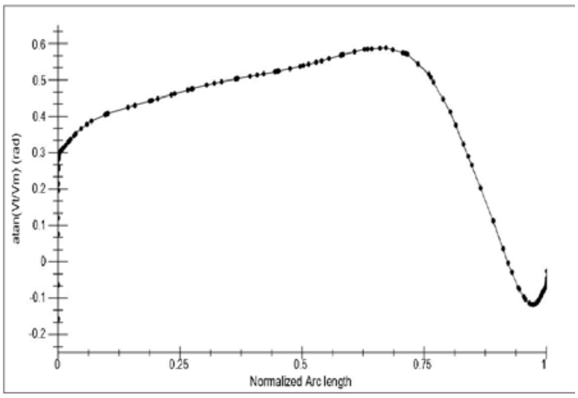


Fig. 6 Flow streamline diagram of high pressure turbine

Fig. 7 Velocity distribution of different blade height

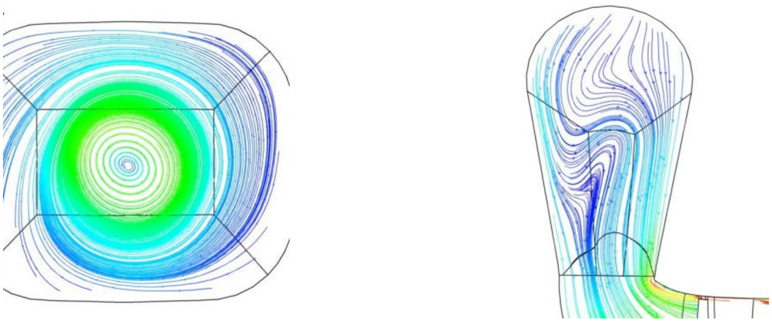
From figure 6, it is indicated that absolute outlet velocity of high pressure turbine has rotational component, which is in the opposite direction to the rotating direction at different position of the blade. However, the direction of the rotational component is the same to the rotating direction of the rotor at large height of blade. This is mainly because the tangential velocity of the blade at the positions of large radial distance is so high that makes the direction of the rotational component decline to the rotating direction. Figure 7 reflects the ratio tangential velocity to axial velocity changing with blade height. At blade height 0 to about 90%, the ratio is above zero, which shows that direction of tangential velocity is opposite to rotating direction. So the rotational component is mainly in the opposite direction to the rotating direction.

4. Influence of the nonuniform flow at high pressure turbine outlet on the performance of low pressure turbine

4.1 Flow field analysis of low pressure turbine

The function of a volute is to guide the flow declining from the direction along the volute inlet to the radial direction, and then entering into the turbine rotor with a certain angle to push the rotor to make power. As is shown in figure 6, the flow field at the volute inlet is nonuniformly distributed. The nonuniformity refers to the velocity along the radial direction.

Flow field in the volute with nonuniform inlet flow at the volute entrance was simulated in this paper based on 3D simulation model for low pressure turbine. Mass, static temperature and velocity distribution was given as the inlet boundary conditions. Mass is 0.558kg/s and the outlet velocity and temperature of high pressure turbine is made as the inlet boundary conditions of low pressure turbine. Total pressure of 190000 Pa is given as the outlet boundary condition, which is obtained by the flow field calculation for low pressure turbine mentioned above. The flow streamline diagrams in different cross sections are shown in figure 8. It is shown that rotating component spreads along the volute and the velocity is nonuniformly distributed at the volute outlet.



(a) Section 1 (b) Section 2

Fig. 8 Streamline diagram in different cross sections

The reason why volute outlet velocity distribution is nonuniform is analyzed as follows.

In the paper, the flow in volute is regarded as ideal flow, and the flow field at the volute outlet is supposed uniform along the circumferential direction.

A simple ideal model is based on the constant angular momentum:

$$rC_\theta = \text{constant} \tag{2}$$

and the continuity equation for the mass flow through any cross-section plane at azimuth angle θ:

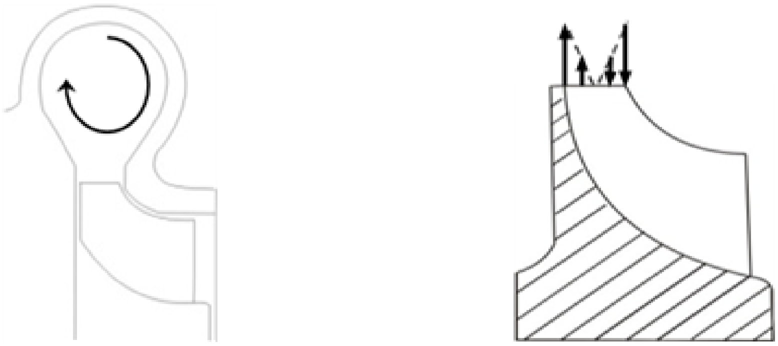
$$m_\theta = \rho_\theta A_\theta C_\theta \tag{3}$$

The angle if flow at volute exit α is given by

$$\tan\alpha = C_\theta / C_m = \frac{\rho_1 (A_1/r_1)}{\rho_0 (A_0/r_0)} \tag{4}$$

The equation implies that the exit flow angle is determined mainly by the choice of inlet area and radius. And if flow velocity at volute inlet is determined, it can be drawn that flow tangential velocity C_θ and radial velocity C_m at volute exit are determined.

In the paper the rotating component of the velocity at the volute inlet of low pressure turbine of two-stage turbocharger is perpendicular to the flow direction. The rotational component at the volute outlet can be regarded as the radial velocity C_{m2} linearly changing along the radial direction of the blade which is shown in figure 9. So the rotational component brings in the effect that makes the radial velocity increase or decrease as is shown in figure 8.



(a) (b)

Fig. 9 Radial velocity of main flow at rotor inlet caused by rotation component

At volute exit, tangential velocity is constant and radial velocity C'_m is:

$$C'_m = C_m + C_{m2} \tag{5}$$

In the condition of a constant tangential velocity at the rotor inlet, radial velocity change will cause the incidence angle of flow at rotor inlet deviates from the optimal angle, which will result in flow field deterioration in rotor and will influence the turbine performance. Velocity triangle change with changing radial velocity is shown in figure 10.

(a) (b)

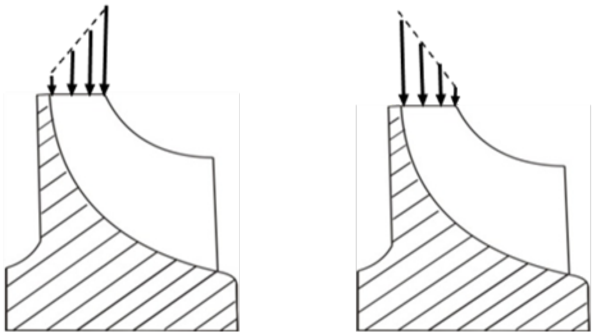
Fig. 10 Velocity triangle change of when radial velocity increases and decreases

Change of the velocity triangle will influence the flow within the rotor and influence the turbine performance further.

Nonuniform pressure field and temperature field will become uniform gradually in the volute. Besides, at the volute outlet, the pressure and temperature will be uniformly distributed. So the nonuniformity of the total pressure temperature has little effect on the performance of the whole turbine.

4.2 Changing mechanism of low pressure turbine

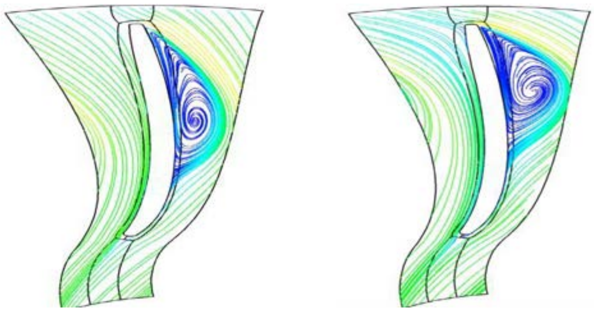
Mechanisms that nonuniform inlet velocity of the rotor at different positions acts on turbine performance were studied in this paper. As is shown in figure 9, the volute inlet circulation caused nonuniform distribution at the rotor inlet. The clockwise circulation is defined as positive swirl flow, and the anticlockwise circulation is defined as the negative one. The velocity ratio between rotational component and the main flow is defined as swirl flow intensity. Positive swirl flow makes rotor inlet flow flat at shroud and violent at hub; negative swirl flow makes rotor inlet flow violent at shroud and flat at hub. Flow incidence angle at rotor inlet changes with the velocity changes. The nonuniform velocity at rotor inlet was simplified to study the mechanisms of action that the inlet circulation acting on turbine performance. The paper imposed linear radial velocity at the rotor inlet, as is shown in figure 11.



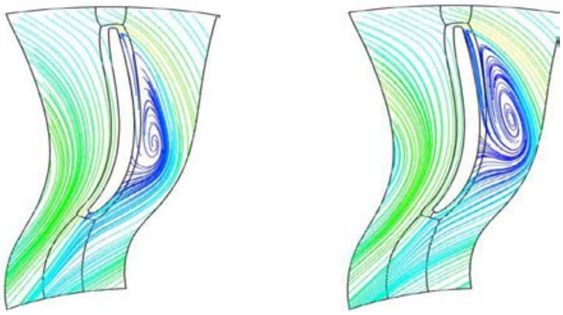
(a) Positive swirl flow (b) Negative swirl flow

Fig. 11 Radial velocity at rotor inlet in different inlet swirl flow

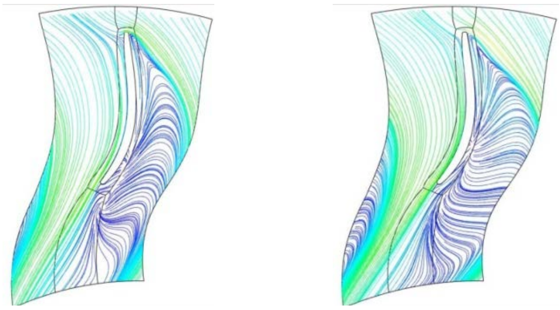
In two-stage turbocharger, low pressure turbine has higher negotiability. When high pressure turbine is under the designed condition, the velocity ratio of low pressure turbine is relatively low and the flow incidence angle is a small negative angle or even a positive angle. In low-pressure turbine rotor model, turbine rotational speed is set 50000r/min. The mass is 0.558kg/s and the inlet total temperature is 800K. The flow incidence angle at rotor inlet is positive. The calculated flow streamline diagram of the blade to blade section at 10%, 50%, 90%blade length in different inlet swirl flow conditions is shown in figure 12.



Negative swirl flow-10% Positive swirl flow-10%



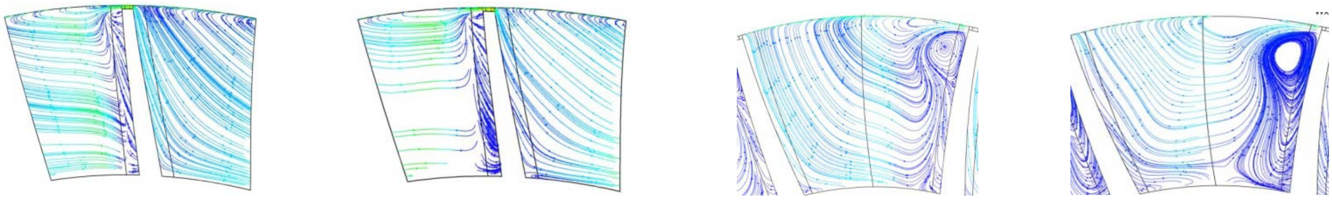
Negative swirl flow-50% Positive swirl flow-50%



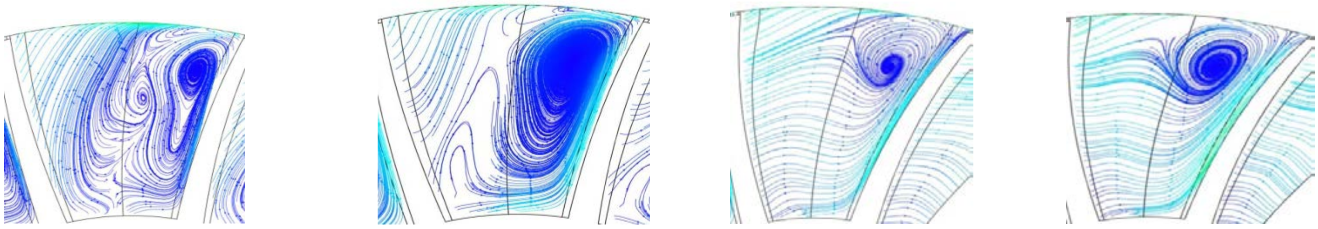
Negative swirl flow-90% Positive swirl flow-90%

Fig. 12 streamline diagram of the blade to blade section at 10%,50%,90% blade height

As is shown in figure 12, as the inlet incidence angle is positive, there is separated flow at blade suction surfaces which results in passage cortex. The cortex developed along the passage which caused the cortex intensity becoming greater gradually. From the contrast of negative and positive swirl flow, it is shown that the cortex intensity in positive swirl flow is higher than that in negative swirl flow because the inlet incidence angle is smaller at the shroud and larger at the hub. So, positive swirl flow will cause greater separated flow at rotor hub and the separated flow at hub plays more important role in the generation and development of passage vortex [10]. It can be seen from the flow streamline diagrams of cross stream sections. The secondary flow in rotor passage through four streamline diagrams of cross stream sections from rotor inlet to outlet is shown in Figure13.



Negative swirl flow-section1 Positive swirl flow-section1 Negative swirl flow-section2 Positive swirl flow-section2

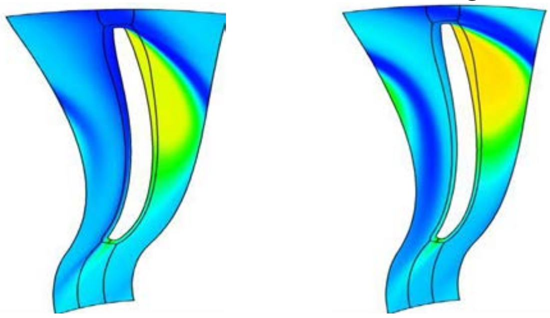


Negative swirl flow-section3 Positive swirl flow-section3 Negative swirl flow-section4 Positive swirl flow-section4

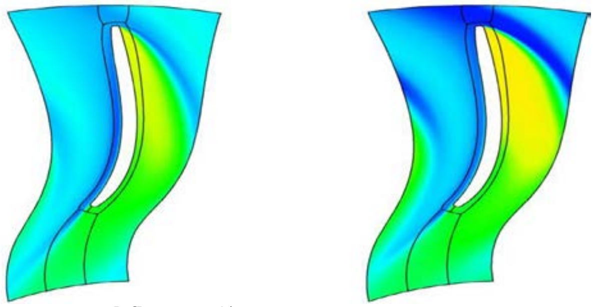
Fig. 13 streamline diagram of cross stream section

From section1 to section4, it is indicated that passage cortex forms under the action of separated flow at the suction surface as well as the leakage flow. Passage cortex developed from zones close to the shroud and expanded in density and scale along the flow downwards. In section 3, the cortex colonized about half of the passage. At the rotor outlet, cortex dissipated gradually and moved to the upper positions of the passage. By comparing the streamline diagrams under Negative swirl flow condition and Positive swirl flow condition, it is shown that separated flow is much more violent at the hub, which helps the cortex expand and cause greater cortex scale. So separated flow at the hub have great influence on the development of passage cortex.

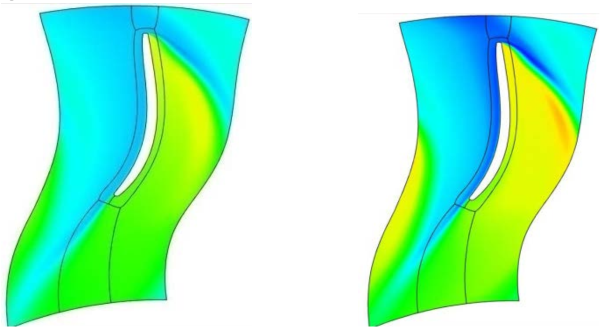
Entropy increasing diagram of blade to blade section at 10％, 50%, 90％blade length is shown in figure 14.



Negative swirl flow-10％ Positive swirl flow-10％



Negative swirl flow-10％ Positive swirl flow-10％



Negative swirl flow-10％ Positive swirl flow-10％

Fig. 14 Entropy increasing diagram of blade to blade section at 10％,50%,90％ blade height

As is shown in figure 14, separated flow at blade suction surface caused entropy increasing. The separated flow intensity under the condition of negative swirl flow is higher than that under the positive swirl flow, so is the entropy which will cause higher efficiency loss under the condition of positive swirl flow.

5. Mechanism that counter-rotation turbine increasing efficiency of low pressure turbine

From the analysis in section 3, it can be drawn that rotating flow at low pressure turbine inlet has great influence on the performance of the turbine. In two-stage turbocharger, inlet swirl flow flow is inevitable, whose intensity and rotating direction are determined by flow in high pressure turbine. If the structure and the working condition of turbocharger system are determined, the flow swirl flow at low pressure turbine is determined too.

As is analyzed before, the rotating direction of inlet rotation flow will influence flow state in turbine and cause turbine performance change further. Turbine efficiency in condition of inlet negative swirl flow is higher than positive swirl flow. So the concept of counter-rotation turbine was brought in considering the inlet flow of low pressure turbine in two-stage turbocharger. Using counter-rotation turbine, the flow field at rotor inlet can be changed from that in condition of positive swirl flow to that in condition of negative swirl flow, which will help to improve the turbine efficiency and then the engine performance. The structure and principle of counter-rotation turbine is shown in figure 15. In this figure, red arrows refer to outlet circulation direction of high pressure turbine and inlet circulation direction of low pressure turbine.

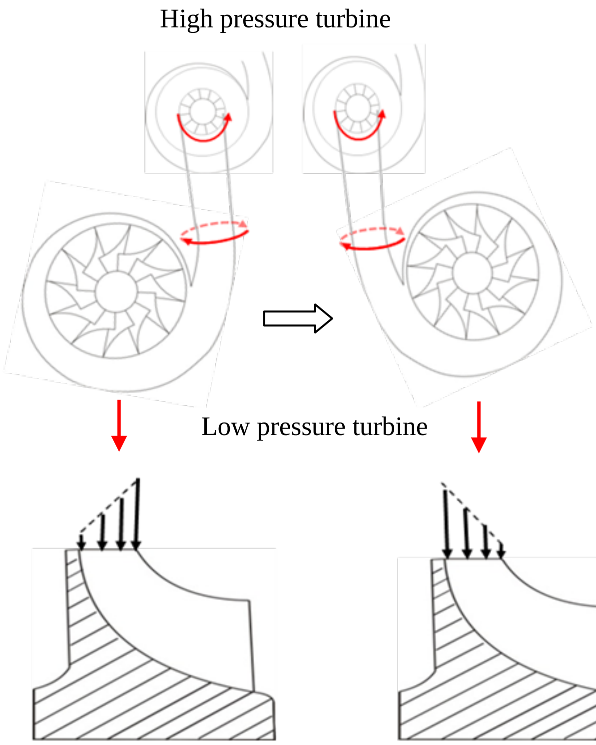


Fig. 15 Structure and principle of counter-rotation turbine

The concept of counter-rotation turbine is extended from aviation field in this paper. A main characteristic of counter-rotation turbine in aviation engine is the rotating directions of two rows of turbine rotors are opposite. As to the two-stage turbocharger in vehicle engine, the rotating directions of two turbine rotors are the same. If the rotating direction of high pressure turbine is clockwise, outlet flow has an anticlockwise circulation in the working condition under low-speed or part-load conditions. As is analyzed before, the inlet swirl flow made the radial velocity high at shroud and low at hub which cause flow incidence angle at rotor inlet nonuniform along blade height, influence flow state in rotor and turbine performance further. In counter-rotation turbine, the rotating directions of two turbine rotors are opposite. Even if inlet circulation at low pressure turbine inlet is clockwise, the swirl flow makes the flow radial velocity high at hub and low at shroud, as is the same to flow state in condition of negative swirl flow shown in figure 15.

Based on simulation model, the paper calculated efficiency of low pressure turbine in two-stage turbocharger in condition of non counter-rotating and counter-rotating. Flow mass is 0.5 kg/s corresponding to engine low speed working condition. If interaction of two turbines is ignored, efficiency of low pressure turbine is 0.77 based on simulation result. However, if flow interaction is considered, efficiency decreases greatly and in different inlet condition, efficiency raised from 0.56 to 0.644 . By simulation, turbine efficiency can be raised 10% at least using counter-rotation turbine system in the specific working condition.

6. Conclusion

The efficiency of low pressure turbine has an important influence on two-stage turbocharger, and is the key factor to the performance of the engine. At present, the flow field between the two turbines is considered uniform, neglecting the interaction between the two turbine's flow fields.

1) Characteristics of flow field at high pressure turbine outlet was studied using 3D simulation model. It is indicated that outlet velocity, pressure and temperature are nonuniform along the blade height. And under working conditions, turbine outlet flow has circulation whose direction is opposite to turbine rotating direction.

2) The interaction mechanism between the two turbines is studied using simulation method. Swirl flow at high pressure turbine caused nonuniform velocity distribution along the blade height at low pressure turbine rotor inlet. The flow states in turbines in condition of inlet positive swirl flow and negative swirl flow is researched. It is shown that inlet positive incidence angle at rotor inlet is small at hub and big at shroud with positive inlet swirl flow while the situation is just opposite with negative swirl flow. The different incidence angle results in different passage cortex in intensity and scale in turbine rotor, which caused the turbine efficiency with negative swirl flow is much higher than that with positive swirl flow.

3) Counter-rotation turbine is able to change the flow field by changing positive swirl flow to negative swirl flow, which will help to increase turbine efficiency and then the engine performance.

Acknowledgments

This work was supported by the National Basic Research Program of China ("973" Program) (Grant No. 2011CB707204)

Nomenclature

u_1	rotor rotation tangential velocity at blade inlet	m	flow mass
u_2	rotor rotation tangential velocity at blade exit	θ	azimuth angle of volute
ω_1	relative velocity in rotor passage at blade inlet	C_θ	flow tangential velocity at volute exit
ω_2	relative velocity in rotor passage at blade exit	C_m	flow radial velocity at volute exit
c_1	flow absolute velocity at blade inlet	C_{m2}	radial velocity caused by rotational component
c_2	flow absolute velocity at blade exit	$C_m^{'}$	actual radial velocity at volute exit
r	radius of volute		

References

[1] Ricardo, "2-Stage Turbocharger Matching for a Light Duty Diesel Engine," http://vladimir.fsik.cvut.cz/bozekdb/filestore.

[2] Mathias Vogt, Florian Frese, Holger Mai, Roland Baar, 2011, "Improving the Application of Turbine Maps in 1D Engine Process Modelling," Proceedings of ASME Turbo Expo 2011, Vancouver, British Columbia, Canada, GT2011-45.

[3] Fredrik Westin, Ragnar Burenius, 2010, "Measurement of interstage losses of a two-stage turbocharger system in a turbocharger test rig," SAE Technical Paper 2010-01-1221.

[4] Fredrik Hellstrom, Laszlo Fuchs, 2008, "Effects of inlet conditions on the turbine performance of a radial turbine," Proceedings of ASME Turbo Expo 2008, Berlin, Germany, GT2008-51088.

[5] Xie Lei, 2010, "The Study of Matching and Design for Two-Stage Turbocharged Internal Combustion Engine," Master Thesis, Department of automotive engineering, Tsinghua University.

[6] "New series of two-stage turbochargers," http://www.marinelog.com.

[7] Ashford G A., 1996, "Unstructured grid generation and adaptative solution technique for high-reynolds number compressible flow," VKI(Von Karman Institute) Lecture Series.

[8] LiChen, Weilin Zhuge, Yangjun Zhang Shuyong Zhang, Jizhong Zhang, 2008, "Investigation of the Secondary Flow Structure in the Mixed Flow Turbine for a High Pressure Ratio Turbocharger," Proceedings of ASME Turbo Expo 2008, Berlin, Germany, GT2008-50941

[9] Karamanis N, 2000, "Inlet and Exit Flow Characteristics of Mixed Flow Turbines in Advanced Automotive Turbocharging," Ph.D. Thesis, Imperial College of Science, Technology and Medicine, London, England.

[10] Palfreyman D., Martinez-botas R. F.,2002, "Numerical Study of the Internal Flow Field Characteristics in Mixed Flow Turbines," Proceedings of the ASME Turbo Expo 2002, Amsterdam, The Netherlands, GT2002-30372.

Liu Yanbin He obtained his BS and MS from academy of armored force engineering in 2003 and 2006. At present, he is a doctoral candidate of Tsinghua university. His research mainly focuses on engine turbocharging.

Zhang Yangjun* He obtained his BS, MS and PhD degrees from the Department of Propulsion at Beijing University of Aeronautics and Astronautics in1989, 1992 and 1995. Since 2003, he has been a professor in the Department of Automotive Engineering at Tsinghua University. His research mainly focuses on engine turbocharging and micro turbo generating.