Study on Aerodynamic Performance and Optimization Design of a Turbine in Liquid Rocket Engine

Wenhao Zhang
Xi'an Jiaotong University
zhangwenhao@stu.xjtu.edu.cn
Xi'an, Shaanxi, China

Kaifu Xu
Xi'an Aerospace Propulsion Institute
xukaifu067@163.com
Xi'an, Shaanxi, China

Zhiduo Wang
Air Force Engineering University
wangzhiduo@foxmail.com
Xi'an, Shaanxi, China

Pengfei Zhang
Xi'an Aerospace Propulsion Institute
z126feifei@126.com
Xi'an, Shaanxi, China

Zhong Ren
Xi'an Aerospace Propulsion Institute
renzhong519@gmail.com
Xi'an, Shaanxi, China

Liming Song
Xi'an Jiaotong University
songlm@mail.xjtu.edu.cn
Xi'an, Shaanxi, China

Zhenping Feng*
Xi'an Jiaotong University
zpfeng@mail.xjtu.edu.cn
Xi'an, Shaanxi, China

ABSTRACT

In liquid rocket engine, turbines are used to drive the pumps that deliver fuel and oxygen with high-pressure into combustion chamber. The turbine performance affects rocket engine specific impulse, thrust-to-weight ratio, and reliability in a rocket propulsion system. To improve the turbine efficiency is of great significance for enhancing the performance of rocket engine. In this paper, steady and unsteady aerodynamic characteristics were investigated firstly for an original transonic turbine of a large thrust liquid rocket engine, taking the consideration of upstream admission chamber, shroud leakage flow, disk cavity, and downstream exhaust duct in the numerical simulation. The unsteady flow field of the original turbine was analysed to evaluate its aerodynamic performance. Then, the optimization of the single passage of turbine stage was conducted on the in-house optimization platform for the maximization of turbine efficiency with constraints on mass flow rate. Finally, the performance of the optimal turbine with a full-annular turbine stage was studied at the same conditions of the original turbine. The aerodynamic performances and internal flow structures of original and optimal turbines were compared. Results indicate that the flow structures of the optimal turbine are improved significantly and the turbine efficiency is increased by 3.61%, from the original case of 79.02% to the optimal case of 82.63%.

INTRODUCTION

Liquid rockets are widely applied to transport space stations and space probes to the space. To meet the needs of future advanced launch mission, rocket engine components

should be designed with more compact structure, higher performance and better reliability. These requirements push turbine of turbopump working under more complicated and extreme operating conditions. Usually, the turbine in rocket engine is high work design, transonic, supersonic or uses a dense drive gas. Therefore, the internal flow structures in the turbine stage are totally three dimensional and the stage flow field is particularly unsteady. The flow unsteadiness is a major factor affecting turbine efficiency, and will directly influence the rocket performance and reliability. The present work aims at the steady and unsteady numerical simulations of the turbine aerodynamic characteristics and the improvement of turbine efficiency through optimization design.

As mentioned above, to get knowledge of unsteady flow phenomena in turbine is very critical to the successful optimization design of rocket engine turbines. Studies have concentrated on the numerical predictions and experimental measurements of turbine unsteady aerodynamic performance in rocket engine. The effects of the axial gap between the stator and rotor (Griffin and Dorney, 1999), the rotor tip clearance (Dorney et al., 2000), and the stator vane size and count on the transient and time-averaged flow field in a supersonic or transonic turbine (Jöcker et al., 2002) were simulated and investigated. Both experimental and numerical studies were performed to reveal the unsteady pressure distribution on rotor blade surface (Zoladz et al., 2003; Hudson et al., 2003; Snellgrove et al., 2003). These unsteady phenomena and performance gave some recommendations for designers to optimize the turbine in rocket engine. However, these studies simulated the turbine without the consideration of upstream and downstream structures in

rocket engine. Therefore, an improved understanding of the influences of full integrated configurations will be beneficial to optimization design of rocket engine turbines.

With the accurate analysis of flow field in turbine, the improvement of turbine efficiency could be obtained based on reliable and efficient optimization methodology. The optimization algorithm for the aerodynamic optimization of turbine airfoil geometry can be divided into gradient-based methods and gradient-free methods. By using the gradient-based optimization methods, the optimum staking line of stator was found in an axial compressor (Lee and Kim, 2000), the optimum two-dimensional blade shape was obtained in an axial turbine (Janus and Newman III, 2000), and the optimum three-dimensional blade shape was acquired in an axial compressor (Samad et al., 2008). As for gradient-free optimization methods, the evolutionary algorithms especially combined with surrogate models, such as response surface model and Kriging model, have been widely used in aerodynamic design in turbomachinery (Mack et al., 2006; Siller et al., 2009; Jin et al., 2007; Paniagua et al., 2009; Tani et al., 2008; Kawatsu et al., 2011). Besides, in order to overcome the large computational consumption of the traditional evolutionary algorithms, the Adaptive Range Differential Evolution (ARDE) algorithm was used in the optimization design of a low aspect ratio transonic turbine stage (Song et al., 2005).

Based on the aforementioned studies, it can be concluded that unsteady numerical simulation is essential for the aerodynamic performance analysis of a rocket engine turbine, and the evolutionary algorithms have been successfully used in optimization design of blade shape. It needs to be emphasized that most of the optimization designs mentioned above were based on a steady Reynolds-averaged Navier-Stokes (RANS) analysis, except for the study (Kawatsu et al., 2011) in which unsteady RANS simulation was carried out in each optimization population to estimate turbine performance more clearly. However, using the unsteady RANS simulation in each population requires a large number of computational resources. Therefore, in this study, the unsteady simulation results of the original turbine with the consideration of upstream admission chamber, shroud leakage flow, disk cavity, and downstream exhaust duct were fully considered before the optimization process. The present optimization work was completed on the in-house optimization platform with the ARDE algorithm combined with steady RANS simulation technique. Finally, unsteady flow field were compared between original and optimal designs. It is believed that this work has some values for the optimization design of a turbine in liquid rocket engine with the complicated upstream and downstream configurations.

NUMERICAL METHOD

In this paper, steady and unsteady RANS equations were solved to analyse the turbine aerodynamic performances by using the commercial CFD software ANSYS CFX, with a SST $k-\omega$ turbulence model for turbulence enclosure. High-resolution second-order central difference scheme for space

and second order backward Euler scheme for time were adopted to discretize the equations.

Computational Model and Grids

The geometry used in this study was a one stage transonic full-annular turbine of a large thrust liquid rocket engine. The upstream admission chamber, downstream exhaust duct, rotor leakage flow and disk cavity were considered in numerical simulation, as shown in Figure 1. The admission chamber comprises of radial inlet duct and axial spherical shell. The exhaust duct consists of two symmetrical outlets and one small outlet in the middle. There are labyrinth seal structures in the shroud tip clearance and between the rotational disk and the stator. The computational grids of the upstream admission chamber and the downstream exhaust duct were generated by using the commercial software Gambit. The hybrid unstructured mesh counted about 7.42×10^6 elements. For rotor shroud leakage flow and disk cavity, the structure hexahedral mesh containing 4.53×10^6 elements was generated in ANSYS ICEM. And the multi-block structure grids of the stator and rotor domain were generated in Numeca Autogrid 5, as shown in Figure 2. The elements number of the grids for stator and rotor domain was 2.74×10^6 and 4.66×10^6 , respectively. Total number of the computational grid elements is about 19.35×10^6 .

Figure 1 Computational Domain

Figure 2 Computational Grids: Surface of Vane, Blade and Disk Cavity

Numerical Conditions and Interface Models

The inlet total pressure and temperature as well as the outlet static pressure were specified based on the operation results. A turbulence intensity of 5% and eddy length scale of 10 mm were specified at the inlet. Besides, solid surfaces were treated as adiabatic walls with a no-slip condition. Ideal oxygen gas was used as the working fluid and the effects of temperature on specific heat capacity, viscosity coefficient and heat conductivity were considered by referencing the NIST Standard Reference Database (2007).

There are three interface models (frozen rotor, stage and transient rotor-stator) which are available in ANSYS CFX.

Frozen rotor model produces a steady-state solution to the multiple frame of reference problems, with some consideration of the interaction between different frames. However, the losses caused by the mixing between stationary and rotational domain are not modelled. Stage model, another steady-state model, considers the mixing losses by averaging the flow parameters, while the circumferential variation of flow field is neglected in rotational domain. The two steady-state models were both used in this paper. Then the simulation results with the frozen-rotor model were set as the initial flows of the unsteady simulation. There were 210 time steps for one period in unsteady simulation, which means the rotor passing through a full circle.

Moreover, the position of interface between the stator and rotor influences the frame of domain for rotor shroud leakage flow and the disk cavity. The effects of interface positions were investigated and the results indicated that the interface near the stationary domain predicts the invasion of main flow correctly (Li et al., 2016). Thus, the interface between stator and rotor was set near the trailing edge of stator in this study, and then the shroud leakage flow and the disk cavity were set as rotational domain with the counter rotating walls for stationary solid surfaces.

Simulation Convergence

Due to the non-axial inflow at the admission chamber inlet and the special design of exhaust duct, large scale of vortices were generated in the upstream and downstream components, which influence the stability of simulation convergence. When root mean square residual errors of energy, mass and momentum equations were less than 0.0001 and imbalances for components were within the range of $\pm 1.5\%$, and parameters of monitor points fluctuated smoothly, steady results were considered to be converged.

Unsteady pressure and temperature of the monitor point at rotor leading edge are plotted in Figure 3. The high frequency fluctuations of the parameters are mainly caused by the interaction of the flow through a stator to a rotor passage. Furthermore, five typical periods can be observed in Figure 3 and good periodicity has been obtained. The overall performance like efficiency and mass flow rate were calculated from the average parameter of the last 2 periods.

Figure 3 Monitored Parameters at Blade Leading Edge

AERODYNAMIC PERFORMANCE OF ORIGINAL TURBINE

Steady and unsteady numerical simulations were conducted for an original turbine in a liquid rocket engine including the upstream admission chamber, downstream

exhaust duct, rotor shroud leakage flow and disk cavity.

After the flow entering the upstream chamber, it gradually turns from radial to axial direction with large and complex vortices generated in the admission chamber. Therefore, the entire flow field is stirred to be completely three dimensional. As shown in Figure 4, there are high and low total pressure zone distributing locally at the admission chamber outlet. As mentioned in related reports (Hermanson and Thole, 2002), when the inlet total pressure presents a pattern with low value in end walls, it will strengthen the development of passage vortex; when the inlet total pressure presents a pattern with low value in mid-span, it will induce counter rotating vortex. The streamlines on pressure side of vane in Figure 5 show radial expansion (enhanced passage vortex) or compression (counter rotating vortex) at different circumferential positions. Not only the non-uniform total pressure but also the variations of incidence angle are generated at stator inlet. Negative or positive incidence angle is clear to be observed in Figure 5, and the time-averaged limited streamlines show flow separations near the leading edge hub regions of some vanes with the negative incidence angle. The area of flow separation in Figure 5 is magnified and displayed in the yellow box.

Figure 4 Time-Averaged Total Pressure at the Admission Chamber Outlet

Figure 5 Time-Averaged Limited Streamlines on the Surfaces of Original Stator Vanes

In order to observe the separation clearly, streamlines on 10% span section of the stator is plotted in Figure 6. From the $-x$ view, the saddle points are almost near the suction side and there is obvious flow separation near the pressure side due to the large negative incidence. Correspondingly, the saddle points tend to the pressure side viewing from the $+x$ axis. The positive incidence arouses velocity reduction and pressure augment near the pressure side and then the secondary flow will be strengthened. Not only negative but also positive incidence angle increases the aerodynamic losses and thereby affects the turbine efficiency.

(a) Viewing from -x axis (b) Viewing from +x axis
Figure 6 Time-Averaged 2D Streamlines on 10% Span of the Original Stator

Negative incidence angle is obvious at the rotor blade leading edge as shown in Figure 7. It should be noted that each blade rotates across the full annular in the unsteady computation, so the time-averaged aerodynamic parameters are nearly same in each rotor passage. Most of the stagnation lines appear on the suction side, which induces the local flow acceleration and pressure decrease near the leading edge on pressure side. Hence, the pressure load of the rotor blade changes badly near the leading edge. Similar to stator, the negative incidence angle at the inlet of the rotor also reduces the turbine efficiency.

Figure 7 Time-Averaged Limited Streamlines on the Surfaces of Original Rotor Blades

As mentioned before, the internal flow structure of the turbine is affected by the interface models. Figure 8 shows the temperature distributions on the surfaces of the rotor blade and disk. The temperature distributions predicted by the three interface models are quite different. There are obvious temperature variations on the different blades for the steady results of the case with the frozen rotor model and for the unsteady instantaneous results of the case with transient rotor-stator model. However, the differences in rotor passages are flattened during the simulation by using the stage model and the data processing of the time-averaged method. Compared to the frozen rotor model, the maximum temperature on blades is reduced by 3% for both stage and the unsteady time-averaged results. Besides, the area-averaged temperature on disk surface in Figure 8(a) and Figure 8(c) for time-averaged results is predicted 16% higher than in Figure 8(b). The reason is that the stage model considers less interaction between two different frames and the invasion of upstream hot flow are weakened. The circumferential variation is obviously shown both on the

surfaces of the rotor blade and the disk for the transient solution of the unsteady simulation. Therefore, unsteady simulation is more reliable, but requires larger computational time as compared to steady simulation. The overall turbine performance is listed in Table 1. It can be observed that the efficiency of unsteady time-averaged results is the highest among three cases. Here, efficiency of steady simulation is derived based on the shaft power and the isentropic enthalpy drop between the turbine inlet and outlet planes. And the efficiency of the unsteady simulation is calculated from the average efficiency of the last 2 periods. The flow rate and the shaft power keep nearly same for the different cases.

Figure 8 Temperatures on the Surfaces of Blades and Rotational Disk

Table 1 Overall Performance of the Original Turbine

Performances	Frozen Rotor	Stage	Transient Rotor-Stator
Efficiency (%)	78.39	77.57	79.02
Normalized flow rate	1.00	1.01	1.00
Normalized shaft power	1.00	0.99	1.00

OPTIMIZATION PLATFORM

On the basis of analysis for flow field in the original turbine, turbine stage optimization was performed based on the optimization design platform presented in Figure 9 (Song et al., 2011). The platform is combined with a modified

adaptive range differential evolution (ARDE) algorithm, three-dimensional parameterization method based on the technique of non-uniform B-Spline and RANS Solver technique (Song et al., 2005). There are four main modules in this platform including optimization algorithm module, design variable resolver module, performance analysis module and constraint handling module. Except for the performance analysis module showed in NUMERICAL METHOD, the other three modules will be introduced following.

Figure 9 Flowchart of the Optimization Platform
Global Optimization Method ARDE

The search performance of optimization method plays a significant role on the optimization design. The ARDE algorithm is used as the optimization algorithm in this work, which is developed based on the self-adaptive mechanism from evolution strategies. ARDE uses the master-server parallel strategy and adapts to reduce the search domain in response to the change of population entropy. In early searching, it reduces search domain of population by faster speed and accelerates convergence. Then, the speed of reducing the search domain is slow done in order to ensure global convergence. The ARDE algorithm has shown to have excellent and efficient global search ability through testing on typical non-smooth, multi-modal and ill-conditioned mathematical problems (Song et al., 2005).

Parameterization Method and Design Variables

The parameterization method determines the design space of optimization process, which is crucial for the optimization process. Based on the non-uniform B-spline (NUBS), a 3D blade profile parameterization method is proposed, which parameterize the blade profiles and the radial stacking line of turbine stage sequentially.

Firstly, one or several section contours at different spans will be selected. The initial section contour is generated by using 2D cascade design method with some geometrical parameters including the number of blades, axial chord length, mounting angle, inlet and outlet blade angle, leading edge and trailing edge radius, etc. For each initial section contour, the blade profile of both pressure and suction side will be fitted by NUBS. Then, 4-6 active control points will be selected to move in normal direction of profile, thereby each active point can be described by one design variable. In addition, to avoid profile distortion, the controlling points of

NUBS near the leading and trailing edge are fixed. After finishing the rebuilding of section contour, the radial stacking line will be fitted by the NUBS, where, the section contours of vane and rotor blades will be stacked by the trailing edge point and the center of gravity, respectively. Then, the points at different spans such as 10%, 50%, 90% spans and etc. will be selected as active control points to form lean/sweep for the new cascade. After finishing the rebuilding of the cascade profile, they are imported into the performance evaluation module for the analysis.

Objective Function

In this paper, the maximizing of the efficiency is worked as the design objective with constraints on mass flow rate. The constraint-handling approach based on penalty method is integrated into the ARDE algorithm to make the individual designed candidates to be evaluated as the fitness by the following formulation.

$$\begin{aligned} \text{Max} \quad & F_{obj} = F(B_{stator}, B_{rotor}) \\ \text{s.t.} \quad & 0.985 \cdot m_{ori} \leq m \leq 1.015 \cdot m_{ori} \end{aligned} \quad (1)$$

where F_{obj} denotes the objective function, B_{stator} denotes the design variables for stator, B_{rotor} denotes the design variables for rotor, m denotes the mass flow rate, and the subscript ori denotes the original design.

OPTIMAL RESULTS AND DISCUSSION

Based on the in-house optimization platform, the optimization design of the single passage of turbine stage has been performed. In this paper, 90 design variables were selected to control both the 3D shape of the stator vane and the rotor blade and 6,000 CFD evaluations with the stage interface model were made as part of the optimization process. The influences of stator inlet non-uniformities and incidence angle conditions were considered to improve the optimization of the profiles. While, all stator vanes are the same for a given set of design parameters. The aerodynamic performance and internal flow structures of the original and optimal turbine are compared to explain the better performance through the successful optimization.

Turbine Stage Geometry Shape Comparison

Aerodynamic performance of the turbine can be improved by reducing the internal secondary flow losses with the relative high solidity of the vanes (Jöcker et al., 2002). So, the number of original stator vanes is increased from 17 to 22, while, the number of blades remain unchanged. Figure 10 shows 3D profile comparison of the vane. Comparing with the original design, the optimal vane presents the gradual increase of the length of chord from hub to shroud, which can reduce the profile losses with the optimum relative pitch for the different span sections of the vane. It has been found that wake losses may be decreased along with the decrease of vane trailing edge diameter. The decrease of the diameter is obvious in Figure 11(a). It should be noted that the profiles in Figure 11 are specially treated for confidentiality. Besides, in order to weaken the influences of stator inlet non-uniform condition, the diameter of the vane leading edge and the thickness of the vane are all increased.

Figure 10 Stator Vane 3D Profile Comparison

Figure 11 Profile Comparisons on 50% Span

Inlet blade angles of the rotor are designed and decreased by 23.8 degrees compared to the original, shown in Figure 11(b). After the optimization, the shape of the profile and the stacking line is changed, as shown in Figure 12. Table 2 gives the time-averaged overall performance of original and optimal design. In this paper, the objective function is maximizing efficiency with the constraints on mass flow rate. The results indicate that the turbine time-averaged efficiency is increased from 79.02% to 82.63% and the mass flow rate is increased by 1.24% which is less than 1.5%. The shaft power is increased by 5.68% compared to the original design.

Figure 12 Rotor Blade 3D Profile Comparison

Table 2 Overall Performance Comparison

Performances	Original design	Optimal design
Efficiency (%)	79.02	82.63
Mass flow rate change (%)	--	1.24
Shaft power change (%)	--	5.68

Performance Comparison in Stator Passage

The number of vanes is increased for the optimal design. With the similar inlet and outlet boundary condition, the pressure load on single optimal vane is smaller than the original as shown in Figure 13. Therefore, the secondary flow losses in optimal vane passages will be decreased. The obvious differences in pressure distribution for each vane especially near the leading edge are generated by inlet non-uniform condition. In order to show the pressure distribution clearly, the

average values of the various vanes are presented in Figure 13(c), in which the pressure decrease on the pressure side of optimal vane is continuous except for leading and trailing edge. So, the thickness of boundary layer on the pressure side will be decreased and the profile loss of the pressure side is lower than the original design. The Mach number at the outlet of the optimal vane is bigger than that of the original vane as shown in Figure 14. The maximum Mach number is 1.08 downstream of the throat in the optimal design. Comparing with the original design, the flow separation region near the pressure side where Mach number is small disappears as shown in the red circles.

Figure 13 Time-Averaged Static Pressure Distributions on 50% Span Section of the Stator

Figure 15 gives the limited streamlines on the surface of optimal vane. There is still positive or negative incidence angle at the inlet of the vane. While, some separation bubbles near the vane hub regions are effectively weakened and some of them are completely disappeared (marked by the yellow square). In order to compare the details of flow field near stator hub, Figure 16 presents the streamlines at 10% span section of the optimal vanes. The vortices near the pressure in Figure 6 block the flow in the passages and this status is weakened after the optimization. Furthermore, the large positive incidence on the view of +x is improved and the optimal stator shape is more adaptable to the inlet non-uniformities.

Figure 14 Time-Averaged Mach Numbers on Vane 50% Span

Figure 17 presents the radial plot of total pressure loss coefficient C_p distribution at stator exit. Here, C_p is defined as

$$C_p = \frac{P_{o\_inlet} - P_{o\_local}}{P_{o\_inlet}} \quad (2)$$

where P_{o_inlet} is the mass flow averaged total pressure at stator inlet, and P_{o_local} is the local total pressure at stator exit. From the figure it can be seen that C_p is decreased obviously after the optimization. The averaged C_p of original design is 0.026, while the optimal one is 0.022. The total pressure loss in the stator passages is reduced by 15.4%.

Figure 15 Time-Averaged Limited Streamlines on the Surfaces of Optimal Stator Vanes

(a) Viewing from -x axis (b) Viewing from +x axis

Figure 16 Time-Averaged 2D Streamlines on 10% Span of the Optimal Stator

Figure 17 Circumferential Averaged Total Pressure Loss Coefficients at Vane Exit

Performance Comparison in Rotor Passage

Figure 18 gives the time-averaged limited streamlines on the surfaces of optimal blades. The negative incidence angle at the rotor inlet is improved after the optimization except for the region near the hub where the blade shape is not very sensitive with incidence angle due to the increased leading edge diameter. Figure 19 shows blade surface pressure load at 10%, 50% and 90% span. It is obviously to observe that the negative

pressure differences between pressure sides and suction sides near the leading edge disappear after the optimization. And the pressure load distributions on the blade surfaces are improved especially at the upper regions. For the optimal design, the pressure on suction side is decreased before the length of 70% axial chord and it is increased before the length of 50% axial chord on pressure side in Figure 19(b). On the whole, the blade pressure load increases except for the trailing edge. This is the main reason for the increase of turbine shaft power and stage efficiency, as shown in Table 2.

Figure 18 Time-Averaged Limited Streamlines on the Surfaces of Optimal Rotor Blades

The radial plot of flow angle and Mach number at rotor exit is presented in Figures 20 and 21. Positive flow angle means the tangential velocity component pointing to the rotating direction. The leaving velocity loss is the minimum when the flow direction is close to the rotating axis. After the optimization, flow angle is decreased near 5% and 90% span and increased near 30% span, which will make the flow angle at the rotor exit close to zero except for the hub region. The decrease of leaving velocity can be observed in Figure 21 which gives the radial plot of Mach number at rotor exit. The Mach number is decreased comparing with the original design above 10% span. And the mass flow averaged Mach number is reduced by 5.7% over the whole rotor exit plane. Thus, the energy loss of leaving velocity is reduced after the optimization.

Figure 19 Time-Averaged Static Pressure Distributions on Rotor Blade Surfaces

Figure 20 Circumferential Averaged Flow Angles at Rotor Exit in Stationary Frame

Figure 21 Circumferential Averaged Mach Numbers at Rotor Exit in Stationary Frame

CONCLUSIONS

The present study performed the steady and unsteady numerical simulations for a full annular turbine of a rocket turbo-pump with the consideration of upstream and downstream components and then conducted the aerodynamic optimization design of turbine stage based on the in-house optimization platform with ARDE algorithm. The conclusions of this study are summarized as follows.

(1) For the original design, large vortices caused by the admission chamber, make the non-uniform inlet conditions at the stator entrance. Flow separation is obvious near pressure sides of original vanes and the passage vortex or counter rotating vortex is enhanced in the different stator passages depending on the total pressure distribution. The matching between the original vane and blade is not well designed, thus negative incidence is induced at the blade leading edge.

(2) The in-depth understanding of the influences of upstream non-uniform flow structures on the turbine unsteady flow field is critical to the optimization design of turbine airfoils. In order to achieve better matching between the upstream intake geometry and the turbine through-flow path during the optimization process, the diameter of the stator vane leading edge and rotor blade leading edge near the hub region, and the thickness of the stator vane are all increased.

(3) Comparing with original design, the time-averaged turbine efficiency of the optimal design is increased from 79.02% to 82.63% and the mass flow rate and shaft power are increased by 1.24% and 5.68%, respectively. The optimal turbine airfoils are more adaptable to the non-uniform conditions at admission chamber exit. The flow separation near stator leading edge is reduced significantly and the total pressure loss in stator passages is reduced by 15.4%. The negative incidence angle at rotor inlet is improved. The pressure load on blade surface is enhanced a lot. The absolute

Mach number at the rotor exit is also reduced by 5.7%, thus the leaving velocity loss is decreased.

REFERENCES

- Dorney D J, Griffin L W, and Huber F W (2000). A Study of the Effects of Tip Clearance in a Supersonic Turbine. *ASME Journal of Turbomachinery*, 122(4), pp. 674-683. doi: 10.1115/1.1290400
- Griffin L W, and Dorney D J (1999). Simulations of the Unsteady Flow through the Fastrac Supersonic Turbine. *ASME Journal of Turbomachinery*, 122(2), pp. 225-233. doi: 10.1115/1.555453
- Hermanson K S, and Thole K A (2002). Effect of Inlet Conditions on Endwall Secondary Flows. *AIAA Journal of Propulsion & Power*, 124(2), pp. 286-296. doi: 10.2514/2.5567
- Hudson S T, Zoladz T F, and Dorney D J (2003). Rocket Engine Turbine Blade Surface Pressure Distributions: Experiment and Computations. *AIAA Journal of Propulsion & Power*, 19(3), pp. 364-373. doi: 10.2514/2.6140
- Janus J M, and Newman III J C (2000). Aerodynamic and thermal design optimization for turbine airfoils. *AIAA Paper No. 2000-0840*. doi: 10.2514/6.2000-840
- Jin X, Shen C B, and Wu X Y (2017). Numerical Simulation and Optimization Design for Kerosene Turbo-Pump with Small Flow Rate. *AIAA Paper No. 2017-2116*. doi: 10.2514/6.2017-2116
- Jöcker M, Hillion F X, Fransson T H, and Wåhlén U (2002). Numerical Unsteady Flow Analysis of a Turbine Stage with Extremely Large Blade Loads. *ASME Journal of Turbomachinery*, 124(3), pp. 429-438. doi: 10.1115/1.1458023
- Kawatsu K, Tani N, Shimagaki M, Uchiumi M, Yamanishi N, Mitsuhashi K, and Mizuno T (2011). Multi Objective Optimization of a Supersonic Axial Turbine Blade Row Shape for Rocket Engine Turbopump. *AIAA Paper No. 2011-5784*. doi: 10.2514/6.2011-5784
- Lee S Y, and Kim K Y (2000). Design Optimization of Axial Flow Compressor Blades with Three-Dimensional Navier-Stokes Solver. *ASME Paper No. 2000-GT-0488*. doi: 10.1115/2000-GT-0488
- Li J, Gao Q, Li Z G, and Feng Z P (2016). Numerical Investigations on the Sealing Effectiveness of Turbine Honeycomb Radial Rim Seal. *ASME Journal of Engineering for Gas Turbines and Power*, 138(10), p. 102601. doi: 10.1115/1.4033139
- Mack Y, Haftka R, Griffin L, Snellgrove L, Dorney D, Huber F, and Shyy W (2006). Radial Turbine Preliminary Aerodynamic Design Optimization for Expander Cycle Liquid Rocket Engine. *AIAA Paper No. 2006-5046*. doi: 10.2514/6.2006-5046
- NIST Standard Reference Database 23 (2007). NIST thermodynamic and transport properties of refrigerants and refrigerant mixtures REFPROP, Version 8.0.
- Paniagua G, Lavagnoli S, Verstraete T, Mahmoudi W, and Benamara T (2009). Aerodesign of a Transonic Contrarotating Turbine for a Rocket Engine. *AIAA Paper No. 2009-5233*. doi: 10.2514/6.2009-5233

Samad A, Kim K Y, Goel T, Haftka R T, and Shyy W (2008). Multiple Surrogate Modeling for Axial Compressor Blade Shape Optimization. *AIAA Journal of Propulsion & Power*, 24(2), pp. 302-310. doi: 10.2514/1.28999

Siller U, Voß C, and Nicke E (2009). Automated Multidisciplinary Optimization of a Transonic Axial Compressor. *AIAA Paper No. 2009-863*. doi: 10.2514/6.2009-863

Snellgrove L M, Griffin L W, Sieja J P, and Huber F W (2003). Experimental Performance Evaluation of a Supersonic Turbine for Rocket Engine Applications. *AIAA Paper No. 2003-4918*. doi: 10.2514/6.2003-4918

Song L M, Feng Z P, and Li J (2005). Shape Optimization of Turbine Stage Using Adaptive Range Differential Evolution and Three-Dimensional Navier-Stokes Solver. *ASME Paper No. GT2005-68280*. doi: 10.1115/GT2005-68280

Song L M, Luo C, Li J, and Feng Z P (2011). Aerodynamic Optimization of Axial Turbomachinery Blades Using Parallel Adaptive Range Differential Evolution and Reynolds-averaged Navier–Stokes Solutions. *International Journal for Numerical Methods in Biomedical Engineering*, 27(2), pp. 283-303. doi: 10.1002/cnm.1307

Tani N, Oyama A, Okita K, and Yamanishi N (2008). Feasibility Study of Multi Objective Shape Optimization for Rocket Engine Turbopump Blade Design. *AIAA Paper No. 2008-4659*. doi: 10.2514/6.2008-4659

Zoladz T F, Griffin L W, and Dorney D J (2003). Experimental Blade Surface Pressures of a Supersonic Turbine for Rocket Engine Applications. *AIAA Paper No. 2003-4916*. doi: 10.2514/6.2003-4916