

文章编号: 100028055(2010) 062135208

Numerical simulation of the three2dimensional flow2path in a ram2rotor

HAN J2ang¹, ZHONG Jing2jun¹, YAN Hong2ming²

(1. Marine Engineering College, Dalian Maritime University, Dalian 116026, China;
2. School of Energy Science and Engineering, Harbin institute of Technology, Harbin 150001, China)

Abstract: A kind of three2dimensional flow2path of the ram2rotor was designed in this paper according to the design method of the typical supersonic intake of aircraft and missiles, especially the design method of the com2pression ramp, the throat and the subsonic diffuser. The design Mach number of the flow2path was based on the averaged relative Mach number, regardless of the change of incoming Mach number along radial direction of the flow2path. Then, the flowfield of the designed flow2path was simulated numerically with computational fluid dynamics software at design point. The purpose of simulation was to obtain the details of the flowfield, including the structure of the shock wave system, position of flow separation zone, and so on. The perform2ance of the ram2rotor was also evaluated. The numerical results show the structure of the shock wave system in the flow2path is similar to that in the supersonic intake. To improve the overall performance of the ram2rotor, the reasonable compromise between the total pressure ratio and the isentropic adiabatic efficiency should be reached. The structure optimization of the flow2path and the configuration of the shock wave system have great influence on the performance of the ram2rotor.

Key words: ram2rotor; flow2path; supersonic intake; numerical simulation; design point

CLC number: V231.1 Document code: A

Nomenclature

R_y	Radius of points on compression ramp	l_t	Length of throat
R_l	Rim radius of ram2rotor	h_t	Height of throat
R_t	Radius of points on throat	D	Angle of ramp
R_k	Radius of points on bottom wall in subsonic di2fuser	A	Divergent angle of subsonic diffuser
$\$H$	Central angle difference between two points on compression ramp	U	Stagger angle of strake wall
$\$H$	Central angle difference between two points on bottom wall in subsonic diffuser	z_y	z coordinate of the points on compression ramp
H	Central angle corresponding to the length of throat	z_t	z coordinate of the points on throat
$\$H$	Central angle difference between two points on throat	z_k	z coordinate of the points on bottom wall in subsonic diffuser
		Ma	Mach number
		p	Pressure
		T	Temperature
		n	Rotational speed

Introduction

With the development of design technique of compressor, some new aerodynamic concepts and methods begin to be applied to the design of compressor, such as three dimensional profiled

Received: 200920209; revision received: 20100420
Foundation item: Supported by National High2tech R&D Program (2007AA05Z255).
E2mail: hit_hja@hotmail.com

technique of blade, splitter blade, high tip speed, lower aspect ratio, higher solidity and unsteady flow, so the stage pressure ratio and the per2formance of compressor increase gradually. At the same time, some new structure is also applied to the design of compressor to obtain high pres2sure ratio, such as aspirated compressor^[1], counter2rotation compressor^[2] and Rampressor compressor^[325]. Rampressor compressor is a new kind of compression system based on shock wave compression technology, and combines the de2sign technique of axial2 and centrifugal2flow compressor. Comparing to the traditional com2pressor, Rampressor compressor has higher pres2sure ratio (up to 15^[6] or higher) and simpler structure, and also has smaller volume and light weight. Because Rampressor compressor has many merits, the thrust2weight ratio of aero2engine will be significantly increased if we use Rampressor compressor to displace the tradition2al compressor in aeroengine^[7].

Ram2rotor is the core component of Ram2pressor compressor. Its performance has decisive effect on the characteristics of Rampressor com2pressor. So it is very important to study the de2sign method of the ram2rotor, especially the de2sign method of the flow2path in ram2rotor. In this paper, the three2dimensional flow2path is de2signed, including the compression ramp, the throat and the subsonic diffuser. The flowfield of the designed flow2path is simulated numerically at design point so as to obtain the details of the flowfield and evaluate the performance of the ram2rotor.

1 Design of 32D flow2path

The compression performance and efficiency of the ram2rotor(Fig. 1^[3]) are directly decided by the configuration of the shock wave system in the flow2path. However, the structure of the shock wave system is mostly influenced by the compression ramp mounted in the flow2path. Therefore, the design of the compression ramp is one of the key technologies. Because the flow2path can be treated as the bending deformation

of a supersonic intake (Fig. 2^[4]), the three2di2mensional flow2path of the ram2rotor can be de2signed according to the design technologies of a three dimensional supersonic intake.

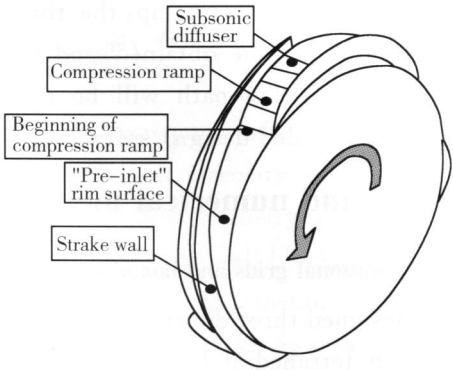


Fig. 1 Schematic diagram of the ram-rotor^[3]

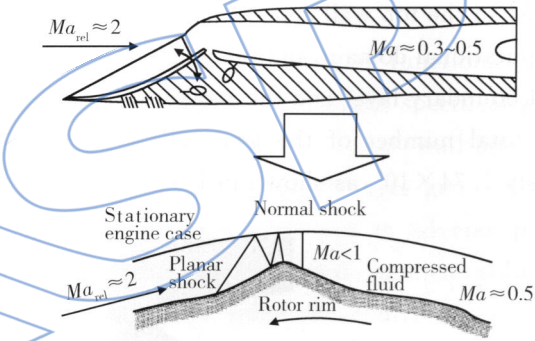


Fig. 2 Flow-path change of the ram-rotor^[4]

The radius of the points on the compression ramp can be calculated by using the following formula

$$R_y = R_l \# e^{SH_1 \# \tan D}$$

The ratio of the length and height of the throat in the designed flow2path is 1, therefore

$$\frac{l_t}{h_t} = 1$$

The central angle corresponding to the length of the throat is

$$H = \frac{l_t}{R_t}$$

The radius of the points on the bottom wall in the subsonic diffuser can be given as

$$R_k = R_t \# e^{SH_2 \# \tan A}$$

The z coordinate of the points on the compression ramp, the throat and the bottom wall are

$$z_y = R_l \# \tan < \# \$ H$$

$$z_t = R_l \# \tan < \# (\$ H + \$ H)$$

$$z_k = R_l \# \tan < \# (\$ H + \$ H + \$ H)$$

Based on the expression above, the genera2 trices of the compression ramp, the throat and the bottom wall can be obtained, and then the three dimensional flow2path will be drawn by CAD (computer aided design) software.

2 Model and numerical method

2.1 Computational grids and boundary conditions

The designed three dimensional flow2path is meshed with tetrahedral/hybrid grids. In order to improve the resolution of solution near the wall of the flow2path, the boundary layers are at2 tached to the top wall and the bottom wall of the computational domain, and the height ratio of the total boundary layer and the strake wall is 0.25. The total number of the grid cells is approx2 mately 1.74 @10⁶, as shown in Fig. 3.

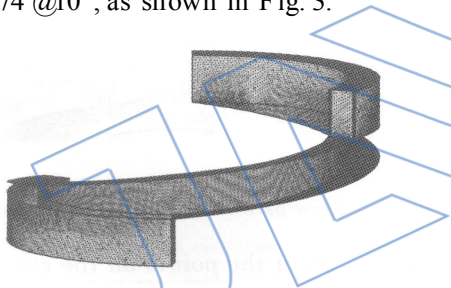


Fig. 3 Computational grids

The boundary conditions used in the numer2 ical simulation include inlet boundary condition, outlet boundary condition, periodic boundary condition and wall boundary condition, as shown in Fig. 4. At the inlet of the flow2path, total pressure, static pressure, total temperature and the direction of incoming flow should be given. As the relative velocity is less than velocity of sound at the exit, static pressure is only needed at the exit of the three dimensional flow2path. The bottom and the side walls of the flow2path are set as adiabatic walls which have the same rotational speed with the working fluid, and the top wall is set as adiabatic wall that is absolutely stationary. The initial conditions of numerical simulation are listed in Table 1.

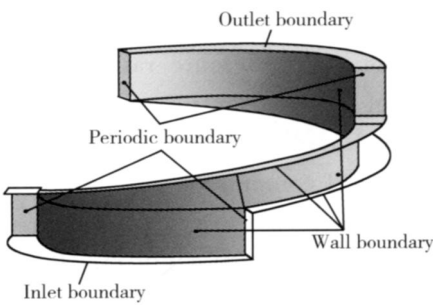


Fig. 4 Computational domain and boundary conditions

Table 1 Initial conditions of numerical simulation

Ma ₀	p ₀ /MPa	T ₀ /K	n/(r/min)
0.348	0.1	288.15	29362

2.2 Numerical method

FLUENT software package is adopted to simulate numerically the flowfield of the de2 signed flow2path in this paper. FLUENT soft2 ware is one kind of popular computational fluid dynamics software^[89], and it can be used to study and analyze flow and heat transfer in com2 plex geometries. As FLUENT software has man2 y advantages, the solution of high2speed com2 pressible flow can be obtained satisfied results, for example, the simulation of the supersonic in2 take^[1011]. In this paper, the flowfield of the flow2 path is simulated numerically with three dimen2 sional steady Reynolds averaged Navier2Stokes (N2S) equations and Spalart2Allmaras (S2A) turbulence model. Without taking account of gravity, radiation and chemical reaction, the three dimensional steady Reynolds averaged N2S equa2 tions in vector form is

$$\mathbf{R}(\mathbf{F} - \mathbf{G}) \# \mathbf{ds} = 0$$

where

$$\mathbf{F} = F_x \mathbf{i} + F_y \mathbf{j} + F_z \mathbf{k}$$

$$\mathbf{G} = G_x \mathbf{i} + G_y \mathbf{j} + G_z \mathbf{k}$$

$$F_x = \begin{bmatrix} Q_t \\ Q_t^2 + p \\ Q_{tv} \\ Q_{tw} \\ (\mathbf{Q} + p)u \end{bmatrix}, F_y = \begin{bmatrix} Q_v \\ Q_v u \\ Q_v^2 + p \\ Q_{vw} \\ (\mathbf{Q} + p)v \end{bmatrix}$$

$$F_z = \begin{bmatrix} Q_w \\ Q_{wu} \\ Q_{wv} \\ Q_w^2 + p \\ (Q + p)w \end{bmatrix}$$

$$G_x = \begin{bmatrix} 0 \\ S_x \\ S_y \\ S_{xz} \\ uS_{xx} + vS_{xy} + wS_{xz} + q_x \end{bmatrix}$$

$$G_y = \begin{bmatrix} 0 \\ S_{yx} \\ S_y \\ S_{yz} \\ uS_{yx} + vS_{yy} + wS_{yz} + q_y \end{bmatrix}$$

$$G_z = \begin{bmatrix} 0 \\ S_x \\ S_y \\ S_z \\ uS_{zx} + vS_{zy} + wS_{zz} + q_z \end{bmatrix}$$

$$S_{xx} = -\frac{2}{3}L \left\{ \frac{5u}{5x} + \frac{5v}{5y} + \frac{5w}{5z} \right\} + 2L \frac{5u}{5x}$$

$$S_{yy} = -\frac{2}{3}L \left\{ \frac{5u}{5x} + \frac{5v}{5y} + \frac{5w}{5z} \right\} + 2L \frac{5v}{5y}$$

$$S_{zz} = -\frac{2}{3}L \left\{ \frac{5u}{5x} + \frac{5v}{5y} + \frac{5w}{5z} \right\} + 2L \frac{5w}{5z}$$

$$S_{xy} = S_{yx} = L \left\{ \frac{5u}{5y} + \frac{5v}{5x} \right\}$$

$$S_{xz} = S_{zx} = L \left\{ \frac{5u}{5z} + \frac{5w}{5x} \right\}$$

$$S_{yz} = S_{zy} = L \left\{ \frac{5v}{5z} + \frac{5w}{5y} \right\}$$

$$q_x = k_{eff} \frac{5T}{5x} \quad q_y = k_{eff} \frac{5T}{5y} \quad q_z = k_{eff} \frac{5T}{5z}$$

u, v and w are component velocity in x, y and z direction respectively. Q, p, E, T, S, q, L and k_{eff} are density, pressure, total energy per unit volume, temperature, viscous stress tensor, heat flux, viscosity factor and effective heat transfer coefficient respectively.

The S2A model is a relatively simple one2equation model that solves a modeled transport equation for the kinematic eddy viscosity. This embodies a relative new class of one2equation model in which it is not necessary to calculate a

length scale related to the local shear layer thickness. The transport equation used in the S2A model is

$$\frac{5}{5t}(QV) + \frac{5}{5x_i}(QV_{u_i}) = G_v +$$

$$\frac{1}{R} \left\{ \frac{5}{5x_j} \left[(L + Q) \frac{5V}{5x_j} \right] + C_{\Omega} Q \left\{ \frac{5V}{5x_j} \right\}^2 \right\} - Y_v + S_v$$

where V is the transport variable. G_v is the production of turbulent viscosity and Y_v is the destruction of turbulent viscosity that occurs in the near2wall region due to wall blocking and viscous damping. R and C_{Ω} are constants and V is the molecular kinetic viscosity. S_v is a user2defined source term. So, the turbulent viscosity, L is computed from

$$L = QVf_{\pi}$$

where f_{π} is the viscous damping function.

The S2A model was designed specifically for aerospace applications involving wall2bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. The coupled implicit solution method is used to solve the control equations, and the second order upwind scheme is used to the discretization of the convection term in the control equations.

3 Validation of numerical method

To validate the feasibility of the numerical method used in this paper, the flowfield of a supersonic intake is studied firstly. The initial conditions, the setting of boundary conditions, the geometrical parameters of the intake and the experimental results are all consistent with those in Reference [12]. In Fig. 5, the wall pressure distribution of numerical simulation and experimental results show a good agreement of the shock wave system. In Fig. 6, the flow separation induced by the reflected shock waves from the lip side occurs at the throat, and matched as well as those of the experimental results (Fig. 7). Because it gives good results for the resolution of the shock wave system and flow separation, the numerical method adopted in this paper is feasible.

ble to solve supersonic ompressible flow.

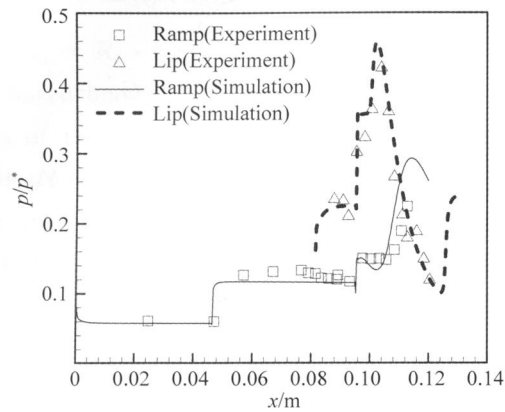


Fig. 5 Comparison of wall pressure distribution

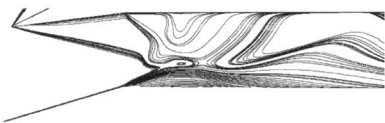


Fig. 6 Local Mach number contour of numerical simulation

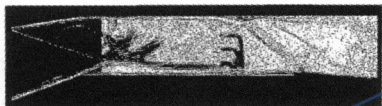


Fig. 7 Local Mach number contour of experimental result^[12]

4 Results and discuss

In order to facilitate the analysis, the con2cept of S₁ and S₂ flow surface of turbomachinery is used to analyze the computational results of the flow2path. The width of the flow2path, is de2fined as pitch (t), and the direction along the width of the flow2path is defined as pitch wise. The detailed definition is shown in Fig. 8.

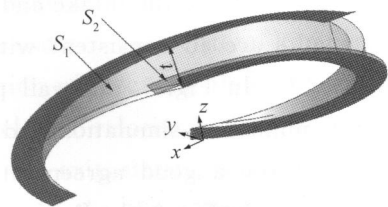


Fig. 8 Definition of flowfield representation method

4.1 Flowfield analysis of the flow2path

In Fig. 9, we can see that there are two main curved shock waves in front of the throat en2trance of the flow2path. One is incident shock

wave, and the other is reflected shock wave. The incident line of the reflected shock wave on the bottom wall of the flow2path just lies in the re2gion near the throat entrance. In Fig. 10, the flow separation zone near the top wall of the subsonic diffuser is small. As the rotation of the fluid and the effect of the shock wave strain, the stream2lines turn toward the bottom wall of the subson2ic diffuser.



Fig. 9 Relative Mach number contour of S₂ flow surface at 20% pitch of the flow-path

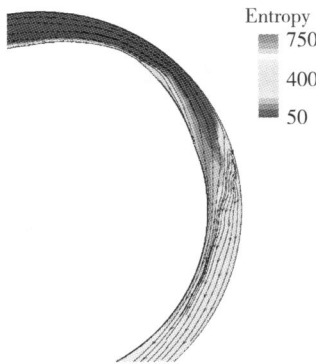


Fig. 10 Entropy and streamlines of S₂ flow surface at 20% pitch of the flow-path

In the subsonic diffuser, the shock wave strain is stably located at the exit of the throat (Fig. 9, Fig. 11, and Fig. 13), and the major part of the shock wave strain is attached to the top wall of the subsonic diffuser. The main flow sep2aration zone is also attached to the top wall, and it is above the shock wave strain.

As the smaller influence of both side walls on flow, the shock wave system of S₂ flow surface

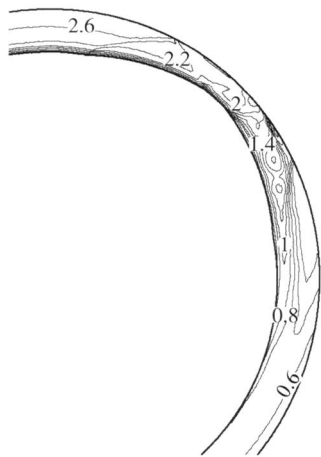


Fig. 11 Relative Mach number contour of S_2 flow surface at 50% pitch of the flow-path

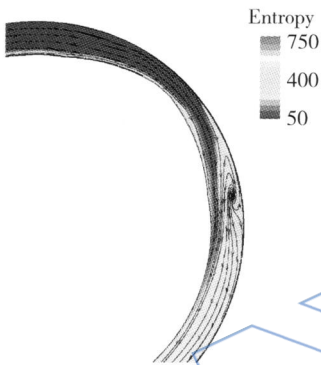


Fig. 12 Entropy and streamlines of S_2 flow surface at 50% pitch of the flow-path

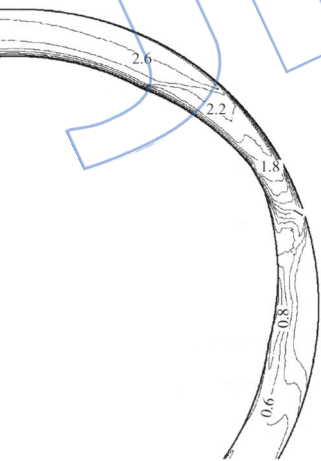


Fig. 13 Relative Mach number contour of S_2 flow surface at 80% pitch of the flow-path

at 50% pitch is clearer than that at 20% pitch. With flow separation zone enlarging (Fig. 12) and forward movement, the shock wave strain moves from the top wall to the bottom wall in the subsonic diffuser. At 80% pitch, the flow separation zone is dominant in the subsonic diffuser (Fig.

14), and the shock wave strain is vague and attached to the bottom wall.

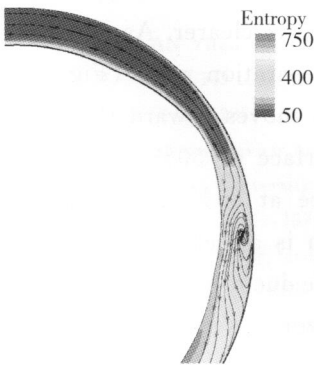


Fig. 14 Entropy and streamlines of S_2 flow surface at 80% pitch of the flow-path

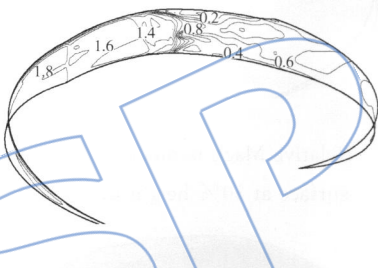


Fig. 15 Relative Mach number contour of S_1 flow surface at 10% height of throat

As shown in Fig. 15, there are two obvious flow slowdown zones in front of throat. This demonstrates that there are two shock waves which intersect with the S_1 flow surface. In the subsonic diffuser, the shock wave strain approximately locates on the middle of S_1 flow surface at 10% throat height. At beginning of the shock wave strain, there are two flow separation zones which locate on both sides of the shock wave strain, as shown in Fig. 16. As the existence of flow separation zones, the width of the shock wave strain is narrow, and then the shock wave strain occupies the whole S_1 flow surface due to the disappearance of flow separation zones.

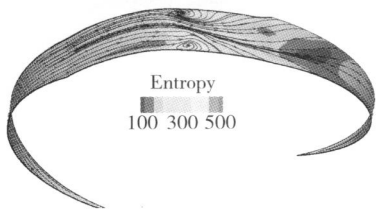


Fig. 16 Entropy and streamlines of S_1 flow surface at 10% height of throat

Fig. 17 shows the relative Mach number contour of S_1 flow surface at 50% throat height. Comparing to the 10% throat height, the shock wave system is clearer. As the enlarging of the left flow separation zone (Fig. 18), the shock wave strain moves toward the right side of the S_1 flow surface at 50% throat height. For S_1 flow surface at 80% throat height, the shock wave strain is attached to the right side of S_1 flow surface due to the further expanding of flow separation zone, as shown in Fig. 19 and Fig. 20.

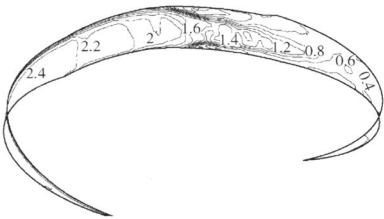


Fig. 17 Relative Mach number contour of S_1 flow surface at 50% height of throat

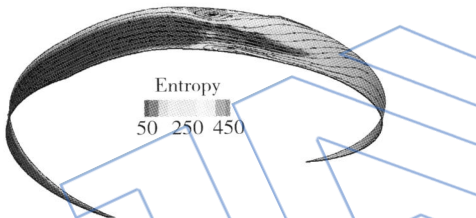


Fig. 18 Entropy and streamlines of S_1 flow surface at 50% height of throat

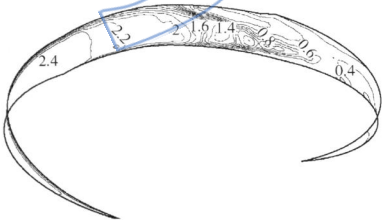


Fig. 19 Relative Mach number contour of S_1 flow surface at 80% height of throat

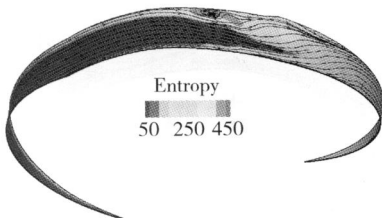


Fig. 20 Entropy and streamlines of S_1 flow surface at 80% height of throat

Some performance parameters of ram2rotor are listed in Table 2. Comparing to a traditional compressor, the total pressure ratio of the ram2 rotor is higher, but the adiabatic efficiency is somewhat lower. As the ram2rotor is mainly based on the shock wave compression, the existence of the shock waves consequentially induces shock wave loss. In addition, the strong flow separation in the subsonic diffuser also results in the increasing of flow loss. So, it is necessary to compromisingly choose the total pressure ratio and efficiency except the structure optimization of the flow2path and the reasonable configuration of the shock wave system in the design process in order to improve the performance of the ram2 rotor.

Table 2 Performance of the ran2rotor		
Total pressure ratio	Isentropic adiabatic efficiency	Averaged absolute Mach number at the exit
13	0.694	1.073

5 Conclusion

The following conclusions can be acquired from the numerical simulation results:

- 1) The shock wave system in the three dimensional flow2path is similar to that in the three dimensional supersonic intake. But the shock waves in the three2dimensional flow2path are curved shock waves, and they are different from linear shock waves in the three dimensional supersonic intake.
- 2) The position and size of the shock wave strain and the flow separation zone are different along pitch wise and radial direction. This shows flow has strong three dimensional character in the subsonic diffuser, and it will affect flow parameters distribution at the exit of the flow2 path.
- 3) To improve overall performance of the ran2rotor, the reasonable compromise choice between total pressure ratio and adiabatic efficiency is needed. At the same time, it is also necessary to optimize the structure of the flow2path and

properly configure the shock wave system.

参考文献:

- [1] LIU Bo, NAN Xiangyi, WANG Yangang, et al. Development and prospect of aspirated fan/compressor[J]. Journal of Aerospace Power, 2007, 22(6): 942-954. (in Chinese)
- [2] JI Lucheng. Analysis of technical challenges in vaneless counter-rotating turbomachinery[R]. ASME Paper GT2007-27617, 2007.
- [3] Lawlor S P, Hinkey J B, Machin S G, et al. Supersonic compression stage design and test results[R]. IMEC2004-59914, 2004.
- [4] Grosvenor A D, Brown P M, Lawlor S P. Design methodology and predicted performance for a supersonic compressor stage[R]. ASME Paper GT2006-290409, 2006.
- [5] Broichhausen K D, Ziegler K U. Supersonic and Transonic compressors: past, status and technology trends [R]. ASME Paper GT2005-69067, 2005.
- [6] De Biasi V. 482kW turbine rated 35% simple cycle at 1700 F firing temperature[J]. Gas Turbine World, 2002, 32(5): 13-16.
- [7] Steel R, Baldwin P, Kesseli J. Insertion of shock wave compression technology into micro turbines for increased efficiency and reduced costs[R]. ASME Paper GT2005-68203, 2005.
- [8] LI Yong, LIU Zhiyou, AN Yiran. A brief introduction to Fluent) a general purpose CFD code[J]. Journal of Hydrodynamics, 2001, 16(2): 252-258.
- [9] YAO Zheng, CHEN Kangmin. Review on the commercial CFD softwares[J]. Journal of University of Shanghai for Science and Technology, 2002, 24(2): 137-144.
- [10] SUN Bo, ZHANG Kunyuan. Experimental investigation and numerical simulation of hypersonic busemann inlet [J]. Journal of Propulsion Technology, 2006, 27(1): 58-60, 96.
- [11] JIN Zhiguang, ZHANG Kunyuan. Experimental and numerical investigation of a forward sweep sidewall compression scramjet inlet with forebody compression[J]. Journal of Propulsion Technology, 2005, 26(6): 502-512. (in Chinese)
- [12] Schneider A, Koschel W W. Detailed analysis of a mixed compression hypersonic intake [R]. ISABE Paper 99-7036, 1999.