Gas–liquid flow field analysis of the compressor vorticity and rotational speed based on NASA Stage 36

Liwen Wang¹, Jianwei Hu¹, Qiang Liu¹, Jie Tang¹, Xudong Shi²

¹Aviation Engineering Institute, Civil Aviation University of China, Tianjin, People’s Republic of China
²College of Electronic Information and Automation, Civil Aviation University of China, Tianjin, People’s Republic of China

Abstract: The aero-engine to be cleaned online can significantly improve compressor performance and increase maintenance intervals, but the cleaning online cannot achieve the anticipated effects due to the lack of cognisance of the internal multiphase flow field. In order to explore the characteristics of the compressor's internal flow field, a compressor and nozzle's flow channel model was established based on the data of the Rotor36 compressor blades published by NASA. The fluid simulation software was used to analyse the effects of circumferential vortex and axial vortex on the wall surface at different jet parameters and characteristics of the gas–liquid flow field formed by the compressor blades at different rotation speeds. It is concluded that the flow field formed at a jet pressure of 5 atm and a rotational speed of 2880 rpm is more favourable to the removal of fouling.

1 Introduction

Aircraft engine in the work process inevitably inhales haze, dust, salt, and other suspended particles. These pollutants will gradually accumulate on the surfaces of the compressor blades, especially in the root and tip clearance areas. The airflow scales of compressor increase the thickness and roughness of the wall surface, which leads to a reduction of intake airflow rate, change of aerodynamic flow field, separation of inlet air flow rate, and so on, so that the engine surge [1] to endanger flight safety can be caused in casual situations. The fouling of gas path components can increase the fuel consumption and the maintenance frequency of the engine overhaul interval, which leads to an increase of operating costs.

The online cleaning of aero-engine can effectively remove the fouling of the gas path components [2], enhance the aerodynamic performance of the compressor, improve the efficiency of the compressor, significantly increase the engine's exhaust gas temperature margin (EGTM), extend the illy time of the engine, and save the maintenance costs.

Based on this, the Civil Aviation University of China, Harbin Institute of Technology, and Nanjing University of Aeronautics and Astronautics carried out the research on the problem of the online cleaning of aero-engines, initially explored the basic process of scales in gas path components, and designed the basic equipment of online cleaning tanker, as shown in Figs. 1 and 2. The engine performance data were provided by Beijing Ameco, and the effectiveness of online cleaning was measured by the EGTM, N1, and N2 engine speeds through the experiments of the aero-engine online before and after cleaning. After washing, the EGTM, N1, and N2 have all been improved, but they did not meet expectations. This is mainly due to the following insufficiencies: (i) the mechanical model of the scale of the compressor blades and the wall surface are not perfect, so it cannot provide accurate flow and pressure for the online cleaning jet parameters; (ii) there are few researches on the flow field characteristics of gas–liquid two-phase flow in the compressor, which results in a difficult understanding of the cleaning flow field interactions with the wall surface.

This paper is based on the above two points, and the compressor rotor model was established according to the NASA published Rotor36 compressor blade data [3]. Based on this model, the compressor and nozzle’s flow channel models were established. Fluent software was used to simulate the gas–liquid two-phase flow. The properties of the internal flow field and the influence of the rotational speed on the flow field were studied.

2 Numerical simulation of the full channel compressor

2.1 Governing equations

The fluid flow always meets three basic conservation laws, but for the incompressible gas–liquid two-phase flow studied in this paper, the energy conservation equation is not considered for the moment due to the small amount of heat exchange. From the micro-perspective, the conserved and non-conserved forms of the governing equation are equivalent, and they are all mathematical expressions of the basic conservation law. However, these two forms of governing equations are different based on the finite volume method for the thought of computational fluid dynamics. The non-conservative governing equations facilitate the theoretical
Table 1 Specific form of each symbol of the universal control equation

| Symbol equation | $\phi$ | $\Gamma$ | $S$ |
|-----------------|--------|--------|-----|
| continuous equation | $1$ | $0$ | $0$ |
| momentum equation | $u_i$ | $\mu$ | $-\frac{\partial p}{\partial x_i}+S_i$ |

Fig. 3 Flow channel model

analysis of discrete phases, while the conservative governing equations can better ensure the conservation of physical quantities to solve the non-linear problem of convection terms, which is conducive to use the non-rectangular grids. Therefore, using the conservative governing equations, the generalised discrete equations based on the finite volume method are established in this paper. Although the dependent variables of the continuous equations and the momentum equations are different, they all reflect the conservation of physical quantities in microscopic volumes per unit time. The general control equations can be expressed as follows [4]:

$$\frac{\partial(\rho \phi)}{\partial t} + \text{div}(\rho u \phi) = \text{div}(\Gamma \text{grad}\phi) + S$$

(1)

Its expansion form is as follows:

$$\frac{\partial(\rho \phi)}{\partial t} + \frac{\partial (\rho u \phi)}{\partial x} + \frac{\partial (\rho v \phi)}{\partial y} + \frac{\partial (\rho w \phi)}{\partial z} = \frac{\partial}{\partial x}\left(\Gamma \frac{\partial \phi}{\partial x}\right) + \frac{\partial}{\partial y}\left(\Gamma \frac{\partial \phi}{\partial y}\right) + \frac{\partial}{\partial z}\left(\Gamma \frac{\partial \phi}{\partial z}\right) + S$$

(2)

In above formula, $\phi$ is a universal variable, which represents $u$, $v$, $w$ solution variable; $\Gamma$ is a generalised diffusion coefficient; and $S$ is a generalised source term. For a specific equation, $\phi$, $\Gamma$, $S$ have specific forms. The correspondence between the three symbols and the specific equations is shown in Table 1.

2.2 Turbulence model

Gas–liquid two-phase flow in the dry run conditions of the engine must be accompanied by turbulent flow. Turbulent flow is a highly complex non-linear flow, but people have been able to numerical simulate turbulent flow through some methods and achieved good results. For specific problems, choosing the correct turbulence model will greatly help improve the reliability of numerical simulation results.

In the full-channel simulation of the compressor in this paper, the density of the gas-phase fluid in the actual flow will change slightly. Bradshaw et al. [5] pointed out that the subtle density changes did not have a significant impact on the fluid flow, so the gas phase is calculated as an incompressible fluid.

Although the $k-\varepsilon$ standard model shows the strong robustness and stability in its calculations, for some flows, such as flow separation, flow reattachment, and so on, the standard model may get poor results and is not suitable for gas–liquid two-phase flow simulation in rotating field. The $k-\omega$ model can better describe the boundary layer flow near the wall, and performs better under the condition of reverse pressure gradient, but the parameter $\omega$ is very sensitive to the incoming flow. Menter proposed to combine the standard $k-\varepsilon$ model and the $k-\omega$ model into SST $k-\omega$ model [6, 7] by introducing a blending function, which preserves the near-wall characteristics of the $k-\omega$ model, and gradually integrates the $k-\varepsilon$ model to eliminate the sensitive problem of incoming flow away from the wall. Considering comprehensively, this paper chooses the SST $k-\omega$ model.

2.3 Initial boundary conditions

According to the online washing parameters specified in the aero-engine washing manual, the actual parameters of the online washing experiments are used as the initial boundary conditions. The type of the nozzle is 5030 model and the injection pressure is 4 atm. The inlet pressure of the compressor is 0.3 atm and its speed is 20 m/s. The compressor blade rotation speed is 2160 rpm. The outlet pressure of the compressor is 0 atm, which is adopted radial equilibrium pressure distribution. In addition, operating pressure is 1 atm, and operating density is based on air. Based on the finite volume method, mixture multiphase flow model is chosen, and the second-order upwind discrete form of the control equations is solved. The coupled algorithm flow field is used to solve variables separation such as velocity, pressure, and so on.

3 Simulation and analysis of the full channel compressor

3.1 Model and grid

Compressor blade data is based on NASA published Stage 36 blade profiles data and flow channel data. The original data coordinates are converted and saved as ‘.curve’ format, which are imported BladeGen to better fit the original cross-section data and meanwhile get more cross-section data. The purpose is to eliminate the bulges and dimples on the surface of the blades and make the data closer to the real compressor blades. After fitting the data of the compressor blades, it is derived and then modelled in SolidWorks and Boolean operation to obtain the compressor blades flow channel model. Based on experimental nozzle, the nozzle flow channel model was established. The compressor blades flow channel model and nozzle flow channel model were assembled and saved as ‘.x_t’ format imported into DesignModeler. The assembled model is shown in Fig. 3.

The division of the grid plays a crucial role in the simulation results. The size of the nozzle flow channel model is much smaller than the size of the compressor blades flow channel model, so it needs to be partitioned. The nozzle exit needs to set in local dimension to make mesh face close to geometry to meet jet requirements. Mesh generation is adopted by the patch conforming algorithm based on the TGRID algorithm in compressor fluid domain. The inlet and outlet dimensions are set, and the element size is set to 2 mm. The upper and lower walls of the compressor fluid domain are set to the boundary layer, whose parameters are set. The inflation option is set to total thickness, number of layers is set to 5, and maximum thickness is set to 5 mm, whose sets are used to capture the flow characteristics of the gas–liquid two-phase flow at the wall surface. Its well-defined overall grid is shown in Fig. 4.

3.2 Effect of vortex on the wall

When aircraft engines are cleaned online, the internal flow field of the compressor exists in various complex vortices. The vortex plays an important role in the removal of fouling. The compressor cleaning flow field is analysed from the perspective of vortex, which has a certain guiding significance that the flow field will be studied to clear fouling from the microscopic perspective. Using the vortex moment theorem proposed by Wu and Zhanyuan [8], the force acting on the wall surface can be expressed:
In this formula, \( r \) stands for the coordinates, \( \omega \) stands for the vortex, \( v \) stands for the velocity, \( s_b \) stands for the entire flow field area, and \( s_d \) stands for the required flow field area, as shown in Fig. 5.

The direct mathematical physical relationship between the compressor inlet flow and the circumferential vortex was established by using the derivative moment transformation method. The vortex value of the observation points was obtained through simulation to optimise the jet parameters. For the case where the outlet of the compressor is nearly straight out, the mathematics expression of flow and circumferential vortex can be derived from integration by parts in the viscous conditions [9]:

\[
F = -\frac{d}{dt} \int \int_{s_b} r \times \omega ds + \frac{d}{dt} \int \int_{s_d} \nu ds
\]  

(3)

In this formula, \( r \) stands for the coordinates, \( \omega \) stands for the vortex, \( v \) stands for the velocity, \( s_b \) stands for the entire flow field area, and \( s_d \) stands for the required flow field area, as shown in Fig. 5.

The direct mathematical physical relationship between the compressor inlet flow and the circumferential vortex was established by using the derivative moment transformation method. The vortex value of the observation points was obtained through simulation to optimise the jet parameters. For the case where the outlet of the compressor is nearly straight out, the mathematics expression of flow and circumferential vortex can be derived from integration by parts in the viscous conditions [9]:

\[
\bar{Q} = \int_{c} \bar{\pi} V_s ds = \int_{c} 2\pi r \bar{\pi} V_s dr
\]

\[= \bar{\pi} \rho V r^2 |_{hub} + \frac{1}{2} \int \rho r o_d dA = \frac{1}{2} \int \rho o_d dA\]

(4)

In this formula, \( \bar{Q} \) stands for the mixed flow, \( \bar{\pi} \) stands for the equivalent density, \( r \) stands for the radial coordinate, \( \omega \) stands for the circumferential vortex, the subscript \( c \) represents the axial cross-section, \( S \) represents the axial cross-sectional area, and the hub and shroud, respectively, stand for upper and lower walls of compressor. The formula \( \omega = (\partial V_r / \partial z) - (\partial V_z / \partial x) \) is utilised in the process of flow formula derivation because the wall's speed of the hub and shroud is zero in the viscous non-slip conditions.

The effect of circumferential vortex distribution on the wall surface is illustrated with Fig. 6 in the compressor blades flow channel model. The positive circumferential vortex is generated at the upper boundary layer of the flow channel, and the negative circumferential vortex is generated at the lower boundary layer. By the formula (2), the positive circumferential vortex space moments accelerate fluid through, and the negative circumferential vortex space moments impede fluid through. By the formula (1), the negative circumferential vortex exerts greater force on the wall and it is easier to remove the fouling. Therefore, the influence of jet parameters on the vortex of the compressor wall is studied through simulation to determine the optimal jet parameters.

A cross-section is established along the 20% chord length of the compressor blades and three equally spaced surfaces are formed along the circumference. So the compressor hub and shroud wall have, respectively, formed six points with a serial number. The compressor blades turn counterclockwise from the view of incidence, as shown in Fig. 7. The effect of the wall circumferential vortex is quantitatively analysed through the vortex value at the point, which represents the vortex value of the micro-area at the compressor wall. As shown in Fig. 8, the vortex value of the shroud wall first increases and then decreases. This is mainly because the water injected into the compressor rotates with the compressor blades, and the liquid phase mainly concentrates on the right side, resulting in the right vortex larger than the left vortex. At the fifth measurement point, the vortex value is greatest and then decreases. When the nozzle injection pressure is 5 atm, the formed circumferential vortex value is the best. When the nozzle injection pressure is 6 atm, the injection velocity of the liquid has higher speed, and it is easier to flow through the compressor blades, so the vortex value of the shroud wall is low. At the fourth and fifth measurement points, the vortex value of the 4 atm injection pressure is higher than its 6 atm. This is because the lower speed of liquid is more likely to be thrown towards the wall by the rotating airflow. As shown in Fig. 9, the vortex value of the hub wall also first increases and then decreases, but the vortex value of the hub wall is much smaller than the shroud wall vortex value. When the nozzle injection pressure is 6 atm, the circumferential vortex of the hub wall is slightly dominant, but the effect is not significant.

Based on the cross-section established at the 20% chord length of the compressor blades, five cross-sections were established in...
succession with gaps of 20 mm, intersecting the axial plane of the
nozzle, and five measurement points were formed in the shroud
wall, as shown in Fig. 10. The effect of axial direction vortex on
the wall is also quantitatively analysed by using vortex values. As
shown in Fig. 11, the vortex values increase first and then decrease
in the axial direction. This is because the gas–liquid two-phase
flow accelerates further in the rotating field and reaches the
maximum value at the point 4. After that, the compressor
decelerates and pressurises, which cause the decrease of the flow
fluid kinetic energy and the increase of the pressure energy, so the
vortex values decreases. The nozzle injection pressure has little
effect on the axial direction vortex, but also it can be seen that the
axial direction vortex value at 4 atm injection pressure is slightly
higher than its 6 atm injection pressure before the fourth
measurement point and later the result is the opposite trend. This is
because when the injection pressure is high, more liquid passes
through the compressor blades and is thrown towards the shroud
wall under the impact of the centrifugal force. In comprehensive
analysis, the axial direction vortex value is superior when the
injection pressure is at 5 atm.

3.3 Flow field effected by rotational speed

The axial section created in Fig. 10 is the observation surface. The
pressure nephogram with different rotational speeds is shown in
Fig. 12. It can be seen from the figure that after the water sprayed
from the nozzle, the pressure of the gas–liquid two-phase flow in
the compressor channel gradually first increases, reaches the
maximum at the compressor blades, and then passes through the
compressor blades with the pressure decreasing in the axial
direction. As the rotational speed of the compressor blades
increases, the pressure around the compressor blades gradually
increases. This is because the pressure ratio gradually increases
with the rotational speed increases. However, the pressure on the
hub wall of the compressor outlet gradually decreases, and this is
because the Bernoulli equation shows that the pressure energy can
be reduced with the kinetic energy of gas–liquid two-phase flow
increased [9].

The velocity nephogram of the compressor axial section is
shown in Fig. 13. From the nephogram, it can be seen that in the
axial direction, the velocity of the gas–liquid two-phase flow
increases first and then decreases. This is because the compressor
blades work on the cleaning flow field. In unit time, with the
higher the rotational speed of the compressor blades, the more
work is done on the flow field, and the average axial flow velocity
gradually increases. As the flow velocity of the hub wall of the
compressor outlet gradually increases, the pressure gradually
decreases, which coincides with the results of the pressure
nephogram of the axial section in Fig. 12. The existence of the
eddies and secondary flow in the wall and blades wall accelerates the
momentum exchange between the fluid and the wall, which is
beneficial to remove the fouling to some extent.

Through the analysis of the simulation nephogram, when the
2880 rpm is selected in the dry run speed, the distribution of the
pressure and velocity on the compressor wall is more appropriate.

4 Conclusion

Through the comparison of pre-experiment and simulation results,
it is verified that the whole channel simulation of gas–liquid two-
phase flow is closer to the actual flow inside the compressor.

By analysing the circumferential vortex and the axial direction
vortex of the wall, it is concluded that a better cleaning flow field
can be formed on the compressor wall at the 5 atm jet parameter.
The rotational speed of the engine is appropriately increased, which is favourable for the generation of eddy and secondary flow to accelerate the fouling removal on the compressor walls during online washing.

5 Acknowledgments

Thanks for teacher's guidance and modifications to make this manuscript more perfect. This thesis was supported by the National Natural Science Foundation (U1733201), the Science and Technology Project of Civil Aviation (20150218), and the Fundamental Research Funds for the Central Universities (3122016D017).

6 References

[1] Seddigh, F., Saravanamuttoo, H.I.H.: ‘A proposed method for assessing the susceptibility of axial compressors to fouling’, J. Eng. Gas Turbines Power, 1991, 113, (4), pp. 595–601
[2] Mund, F.C., Pilidis, P.: ‘A review of gas turbine online washing systems’. ASME Turbo Expo 2004: Power for Land, Sea, and Air, 2004, pp. 519–528
[3] Reid, L., Moore, R.D.: ‘Design and overall performance of four highly loaded, high speed inlet stages for an advanced high-pressure-ratio core compressor’, (National Aeronautics and Space Administration (NASA), TP-1337, USA, 1978)
[4] Wang, F.J.: ‘The analysis of computational fluid dynamics-the principle and application of the CFD software’ (Tsinghua University Press, Beijing, 2004) (in Chinese)
[5] Bradshaw, P., Cebeci, T., Whitelaw, J.H.: ‘Engineering calculation methods for turbulent flow’ (Academic Press, Salt Lake City, 1981)
[6] Menter, F.R.: ‘Two-equation eddy-viscosity turbulence models for engineering applications’, AIAA J., 1994, 32, (8), pp. 1598–1605
[7] Wilcox, D.C.: ‘Reassessment of the scale-determining equation for advanced turbulence models’, AIAA J., 1988, 26, (11), pp. 1299–1310
[8] Wu, J.C., Zhenyuan, W.: ‘Elements of vortex aerodynamics’ (Shanghai Jiaotong University Press, Shanghai, 2014) (in Chinese)
[9] Daogan, Y.: ‘Total flow Bernoulli equation in fluid dynamics’, Phys. Eng., 2014, 24, (4), pp. 47–53 (In Chinese)