Performance analysis of underwater pump for water-air dual-use engine

Jun Xia\textsuperscript{a}, Yun Wang\textsuperscript{b}, Yu Chen\textsuperscript{c}

School of Aircraft Engineering, Nanchang Hangkong University, Nanchang, China.
\textsuperscript{a}tomxj375@163.com  \\textsuperscript{b}wangyun66@126.com  \\textsuperscript{c}745317205@qq.com

Abstract. To make water-air dual-use engine work both in air and underwater, the compressor of the engine should not only meet the requirements of air flight, but also must have the ability to work underwater. To verify the performance of the compressor when the water-air dual-use engine underwater propulsion mode, the underwater pumping water model of the air compressor is simulated by commercial CFD software, and the flow field analysis is carried out. The results show that conventional air compressors have a certain ability to work in the water environment, however, the blade has a great influence on the flow, and the compressor structure also affects the pump performance. Compressor can initially take into account the two modes of water and air. In order to obtain better performance, the structure of the compressor needs further improvement and optimization.

Key words: Water-air dual-use engine; Compressor; Underwater propulsion; Numerical Simulation; Flow field analysis.

1. Introduction

With the development of the war in the future, the all-dimensional weapon is more important \cite{1}, but, now the aircraft can only flight, the enemy region has the ability to survive the air is greatly reduced. If the aircraft can dive into the water after finished the mission, it can avoid tracking and search, then the survival ability can be greatly improved \cite{2, 3}. So, the water-air aircraft are the product of such demands, the aircraft needed to achieve air fly and underwater propulsion, requiring a unique engine.

The water-air dual-use engine has two unique modes to work both in the air and the water. Among them, water-reactive metal materials are applied in the underwater working mode to achieve the underwater propulsion. The engine’s thrust in air propulsion is from jet fuel combustion, which is similar to the current aero-engine \cite{1}. Underwater propulsion is from high temperature gas which is produced by hydro-reactive metal reacting with water \cite{4}.

In order to explore the work condition of traditional aero-engine compressor under the underwater environment and evaluate the feasibility in compressive pump water performance of water-air dual-use engine in the underwater working mode, the CFD software has been applied in this paper to make the preliminary simulation analysis. Taking the example of the NASA STAGE35 compressor, this paper firstly makes analysis on its compression performance under the air environment to make comparison and verification, then making the compression work simulation of the compressive performance of the
compressor under the water environment. Thus to analyse the compressive performance of the compressor under the water environment and provide the useful reference for the design of water-air dual-use engines.

2. Model Calculation And Analysis on the Gas Performance

2.1. Physical model
This paper uses NASA STAGE35 as calculation model. NASA STAGE35 is an axia-flow transonic compressor, which is one of four inlet stage for an advanced core compressor, designed by NASA Glenn research center in 1978. The rotor of NASA STAGE35 has 36 blades, the tip solidity is 1.3, the rotor tip speed is 455 meters per second, and the tip clearance is 0.2% of blade span. The stator has 46 blades, the tip solidity is 1.3, and the root gap is 0.5% of blade span [5, 6].

2.2. Model mesh generation and calculation
As the flow solver the well-known commercial software Numeral/ Fine turbo is used in present study. Auto grid/ IGG module is used to produce the stage three-dimension meshh (Han et al., 2010). The first layer of the wall grid is 0.003mm and the y+ is 1. After grid generation, the grid number of rotor is about 1.03 million, stator is about 1.18 million, and the total number of single channels is about 221 million [7] .As show in Fig1. FINE module is applied for flow field simulation, and Spalart-Allmaras model is applied for the turbulence model calculation, and the working fluid is ideal gas. The CFL number is 3. The boundary conditions as flows: the flow direction is axial, the compressor speed is 17188.7RPM / min, the total inlet pressure is 101325 Pa, the total inlet temperature is 288.2K, and the inlet turbulent viscosity is 0.00005m2/s, also the average static pressure of the outlet is 130,000Pa. Meanwhile, considering the flow’s periodicity and calculation, here only the single-channel zone is calculated and the rotation periodic boundary condition is applied.

![Figure 1. Grid of NASA Stage 35](image)

2.3. Analysis on the calculation results
As the calculated results show, the flow rate is 20.59Kg / s, the pressure ratio is 1.79, the calculation error is within the allowable range with a small difference from the design condition, indicating the model and the meshing generation are correct.

3. Work simulation of compressors in underwater environment
When the compressor is in the water, the fluid is changed from air to water, to make the mesh more adaptable to fluid changes, the first layer of the wall grid modified to 0.00019mm. The rest conditions are not modified. Then, meshing again, as the compressor enters the water environment, the compressor operation should be readjusted. In the underwater environment, the compressor is similar
to a pump, the differential pressure and the efficiency have become significant underwater parameters for the compressor. The underwater calculation mode is the same as the air mode. However, the compressor speed is 1000 RPM / min. The performance curve of this ‘water pump’ could be obtained by the change of the outlet mass flow. Preliminary consider the shallow water area, the boundary conditions as follows: the total inlet pressure is 110000 Pa, the total inlet temperature is 293 K, and the static pressure of the outlet is 130400 Pa.

3.1. Analysis of compressors’ operating point performance & parameters of the rated water pressure in underwater working mode

According to the above boundary conditions, the compressor simulation & calculation are made in the underwater environment and the total pressure distribution map (as shown in Fig. 2) of the compressor flow path in the underwater working is drawn from the calculation results.

![Figure 2. Total pressure distribution](image)

The total pressure difference between the inlet and the outlet could be obtained from the total pressure distribution map, and this compressor operating point performance & parameter in underwater working mode is also obtained from this calculation result, respectively as: speed is 1000 RPM/min, pressure differential is $1.72 \times 10^4$ N, efficiency is 87%, and mass flow is 920 kg / s. In the rated working state, its underwater working mode capability has initially met the underwater performance requirements, and further analysis is still needed.

3.2. Characteristic analysis of compressors’ underwater operating mode

In order to understand compressors’ underwater performance, the characteristic curve should be obtained. The compressor works similarly to the axial flow pump in the underwater, and external features of the pressure and efficiency need to be worked out [8, 9].

The efficiency of the pump can be calculated by the following equation:

$$\eta = \frac{gQ(P_{\text{out}} - P_{\text{in}})}{M \omega}$$  \hspace{1cm} (1)

Where $Q$ and $M$ are mass flow and torque.
In order to calculate the characteristic curve for the underwater operating of the compressor, the outlet boundary condition should be changed as the mass flow outlet, flows of 840kg/s, 880kg/s, 920kg/s, 960kg/s, 1000kg/s, 1040kg/s should be separately selected as the new series of outlet boundary conditions near the above-mentioned rated working point. The simulation results are plotted into mass flow-pressure differential curves (as show in Fig. 3) and flow-efficiency curve (as show in Fig. 4).

![Flow-pressure differential curve](image.png)

**Figure 3. Flow-pressure differential curve**

![Flow-efficiency curve](image.png)

**Figure 4. Flow-efficiency curve**

The flow-efficiency curve graph shows that the underwater efficiency of the compressor has the tendency to rise first and then descend, with the maximum efficiency of 87% at 960kg/s. Similarly, there’s small changes of the efficiency near the flow. The compressor has the maximum underwater efficiency of 87%, has certain underwater work capabilities and meets the underwater work requirements for water-air dual-use engines. However, as the compressor is designed to compress the air, its underwater performance would be affected. Compared with the traditional pump, its efficiency is still relatively low, the working capacity is far less than the pump, and therefore the further analysis and optimization are needed.

### 3.3. Flow field analysis of compressors' underwater operating mode

In order to further improve the compressing efficiency, detailed analysis on the flow field should be made to provide reference for the subsequent impeller optimization. The underwater internal flow of the compressor mainly depends on its velocity distribution and pressure field. The working condition
with the maximum efficiency of 960kg/s is selected to verify the underwater working flow field of the compressor. The pressure surface and suction surface static pressure diagram of its rotor and stator are respectively given as below (the right sides of Fig. 5 and Fig. 8 are the leading edge of the blades, and the left sides of Fig. 6 and Fig. 7 are the leading edge of the blades). Meanwhile, surface S1 is made in the midpoint of the blade height to display its speed cloud picture (as show in Fig. 9). The speed cloud picture of the rotor is also provided (as show in Fig. 10).

![Figure 5. Rotor pressure surface](image1)

![Figure 6. Rotor suction surface](image2)

![Figure 7. Stator pressure surface](image3)

![Figure 8. Stator suction surface](image4)
As the pressure surface of the rotor indicates, a small range of high pressure zones is formed in the leading edge of the blades due to the impact of the water flow, meanwhile the pressure changes on its pressure surface are not uniform. On the suction surface, it can be seen that a small range of low pressure zones forms in its leading edge with even changes backwards, and no big changes in the water flow occur.

Low pressure zones form in the central region of the suction surface of the stator, as S1 section shows, the blade diversion has increased in the flow rate and decreased the pressure in this region, forming the low pressure zone in the central of the blade. Meanwhile, high pressure zones form in the upper end of the blade’s leading edge as shown in Figure 8. Through the observation of the moving blade velocity distribution Figure 10, it could be seen that the water flow is high close to the tip of the moving blade, making more water flows strike on the tip of the stationary blade, thus to impede the
water flow and increase the pressure, which also reduces the efficiency of the compressor. As the compressor channel is convergent, the reduced flow path would surely make the flow velocity increase in the case of constant flows, therefore a small range of low pressure zones from in the front edge of the stationary blade root as the flow velocity increases.

4. Conclusion

Through numerical simulation of water model of compressor underwater pump, we can conclude that:

(1) While the compressor operating underwater, the compressor speed is 1000RPM / min, and the outlet flow varies from 840kg / s to 1040kg / s, the compressor efficiency varies from 82.8% to 87%. Under this condition, the compressor has the ability to work under water.

(2) The air compressor model is used to simulate the underwater conditions, as the rotational speed of the compressor without any blade optimization is reduced to a certain extent, the performance of the compressor in the water has been reduced in a certain degree as shown in the cloud picture, the flow fluidity in the channel is not as smooth as in the air, especially the stator would impede the flow of the water and the compressor’s work, while even failing to take into account of the operational capability in the underwater environment, the compressor to compress the air still have a certain degree of underwater work ability without modification and has the underwater work abilities for compressors. Meanwhile, to better meet the underwater working conditions, the compressor blades and even the layout still need to be improved. This paper also provides certain theoretical basis for the future optimization and improvement.

Acknowledgments

This work was supported by the National Natural Science Foundation of China NO.51266012

References

[1] W. Zhang, Aerodynamic design of impeller of centrifugal compressor for water-air engine. Nanchang, 2013.

[2] X.L.Zhong, the conceptual design and research on multi-bypass counter-rotating air-water engine. Nanchang, 2016.

[3] S.Zhu, The design and research of power system in Water-Air UAV. Nanchang, 2012.

[4] X. GAN, G. Q. HE and J. WANG, 2008, Experimental investigation on concept of water ram jet, Journal of Solid Rocket Technology, (01): 4-7.

[5] Z.H. Li, Y.M. Liu, Blade-end treatment for axial compressors based on optimization method. Energy, 2017, 126:

[6] W. J. Han, S. T. Wang and X. F. Wang, Design optimization of transonic compressor stage using CFD and response surface model. Journal of Harbin Institute of Technology, (01): 112-118. 2010.

[7] Information on https://wenku.baidu.com/view/9eb781e780eb6294dd886cdf.html.

[8] Q. Zhang, 2008, Improved Lifting Design Method and Internal Flow Field Calculation of Axial Flow Pump Impeller, Xihua University.

[9] F. Feng, X. Han and Y. Shi, 2009, Influence of impeller parameters on the performance of 020Q84 water jet axial flow pump, Ship Engineering, (01): 11-14.