Comparision of Turbulence Models in Two-dimensional V-gutter Flameholder Flow Field

Ronghao Cui, Weijun Fan, Rongchun Zhang\textsuperscript{1}, Hongwei Lei and Hanqing Xu

National Key Laboratory of Science and Technology on Aero-Engines Aero-Thermodynamics, School of Energy and Power Engineering, Beijing University of Aeronautics and Astronautics, Beijing 100191, China

E-mail: zhangrongchun@buaa.edu.cn

Abstract. In this paper, nine common turbulence models are used to numerically simulate the flow field before and after the flame stabilizer. Comparing the flow velocity, normal velocity, vorticity and recirculation zone length between experiments and simulations, it is found that the SA model and the k-kl-ω model have good prediction performance for different parts of the simulation results. At the same time, it is found that the flow in the numerical simulation is more sensitive to the flow field distortion and the separation occurs before the experimental results. The velocity profile obtained by numerical simulation recovers more slowly as the flow develops. Compared with the normal velocity, the simulation results of the streamwise velocity distribution agree well with the experimental results.

1. Introduction

Flameholders are an indispensable part of an afterburner. In order to ensure the efficiency of the engine, the flameholder should have as little flow loss as possible when the afterburner is not working. Therefore, research on the flow field of the flameholder is critical in the design of the flameholder. The V-gutter flameholder is widely used in afterburners because of its simple structure. A large part of the flameholder used in modern aircraft engine is an improved modification of the V-gutter flameholder. Therefore, this paper chooses the V-gutter flameholder as the basic flameholder and studies its flow field.

With the continuous improvement of computer performance and numerical models, numerical simulation has been widely adopted as a method for efficiently solving flow problems. For most engineering problems, RANS is the best way to combine the calculation amount and the accuracy of the results. For the turbulence problem, when using the RANS method for simulation, due to the non-closed nature of the Reynolds equation, it must be closed by means of a turbulence model. So far, a variety of turbulence models have been developed for different flow problems. The flow near the V-gutter flameholder is a typical bluff body flow, and Bush et al.\cite{1} conducted detailed experimental measurements on the flow after the V-gutter flameholder and provided abundant flow field data. In this paper, the flow of the V-gutter flameholder is numerically simulated using the nine most common turbulence models. The nine turbulence models are the Spalart-Allmaras model \cite{2}(SA), the standard k-ε model\cite{3}(SKE), the realizable k-ε model\cite{4}(RKE), the standard k-ω model \cite{5}(SKW), the SST k-ω model \cite{6,7}(SST KW), the Reynolds stress model \cite{8,9}(RSM), the RNG k-ε model \cite{10}(RNG KE), the transition SST model (T SST) and the transition k-kl-ω model (K-KL-W). By comparing the simulation results with the experimental results, we intend to find the most suitable turbulence model to provide reference and basis for the numerical simulation of the flameholder flow field in the future.
future.

2. Numerical Method

2.1 Geometry Model
Numerical simulation simplifies the flame stabilizer from three dimensions to two dimensions, i.e., the flameholder span width is ignored. The flameholder dimensions are shown in the Figure 1. The trailing edge of the flameholder is 254 mm from the inlet boundary, and the total length of the calculation domain is 600 mm; the normal dimension is 101.6 mm; the axis of the flameholder coincides with the central axis of the flow field.

![Figure 1. Flameholder Size](image)

2.2 Computational Mesh
In this paper, according to the geometric size, the node distribution is adjusted and 6 sets of grids are selected for independence test. It is carried out under the condition of $V_b=53.3 \text{ m/s}$ and normal pressure. After calculation, the velocity distribution from the trailing edge of the $Y=0$ cross-section stabilizer to the downstream of 185 mm is selected for analysis. The curve of the first set of grids does not coincide with the other five sets of grids. At the same time, the length of the recirculation zone is calculated as shown in the table 1. The calculation results of the first set of grids and the other five sets of grids also have a certain gap. After excluding the first set of grids, the relative error of the other five sets of grid simulation results is less than 1%, reaching the grid independence standard. Therefore, this paper selects the third set of grids (the number of grids is 78772) for the next numerical calculation.

| Grid Number | 1st  | 2nd  | 3rd  | 4th  | 5th  | 6th  |
|-------------|------|------|------|------|------|------|
| Length of recirculation zone (X/D) | 1.804 | 1.857 | 1.875 | 1.879 | 1.898 | 1.879 |

2.3 Computational Method
In order to reduce the influence of swirl, pressure with direction gradient and separation calculation on accuracy, a standard wall model is used on the wall of the near stabilizer, and the wall $y^+$ of the outer near stabilizer is greater than 30. At the same time, the coupling of flow and pressure adopts SIMPLE algorithm. In order to improve the calculation accuracy, the convection term is discrete using the second-order upwind style. The convergence accuracy standard is $10^{-6}$.

2.4 Boundary Conditions
In the experimental case, the inlet boundary of the flow field was set at 254 mm upstream of the flame stabilizer. The inlet velocity was $V_b=53.3 \text{ m/s}$, the static pressure at the inlet and outlet were 1 atm and
the inlet temperature was 644.3K. The exit boundary condition is set to the pressure outlet boundary, and the pressure and temperature are set to normal temperature and pressure.

3. Computation Results and Discussion

The parameters of the simulated two-dimensional flameholder and the data of experiment are from the experiments of Bush et al. The X and Y axis coordinate values are dimensionlessly processed with the width D of stabilizer. The speeds are all nondimensionalized by the streamwise velocity \( V_b \).

It can be seen from Figure 2(b) and Figure 2(c) that the simulation results of the SA model on the streamwise velocity component are more accurate in the recirculation region downstream of the stabilizer. Since the SA has a smaller variable gradient near the wall, the position of other turbulence models has a flow distortion at the trailing edge of the stabilizer which is closer to the wall. In the range of \( Y/D = -0.5\sim0.5 \), the SA model's streamwise velocity recovery is faster and more consistent with experimental data. From the Figure 2 (a), the SA model is not sensitive to flow distortion at the trailing edge of the stabilizer. The simulation results of the flow velocity are quite different from the experimental results. The velocity distribution is also the largest among the turbulence models.

It can be seen from Figure 2(d) and Figure 2(e) that the velocity recovery of the K-KL-W model is more consistent with the experimental data downstream of the recirculation zone. At the same time, it is found that except for the SA model and the K-KL-W model, the numerical simulation results of the other turbulence models have a larger streamwise velocity on the inner side of the wall, which also leads to a lower axial velocity near the axial line.

Figure 2. Axial Velocity Distribution

From the overall view of the Figure 2, the normal velocity simulated by the SA model and the K-KL-W model forms peaks on both sides of the trailing edge of the stabilizer and the simulation of the flow field distortion is more accurate. However, the speed recovery in the simulation results are all slower. In the outside of the recirculation zone, the simulation results of each model are in good agreement with the experimental data. From the structure of the recirculation zone, only SA model matches the experimental data in the structure and position of the recirculation zone. Although the absolute values differ greatly, considering the important effect of the length and shape of the recirculation zone on the flame stability after the flame stabilizer, the predicted results of the SA model are more appropriate.
The Figure 4 shows the streamwise velocity distribution on the axis during the downstream development of the flow. Similar to the results shown in the Figure 4, the SA model predicts the flow velocity more accurately in the recirculation zone after the flame stabilizer. Although the prediction results of the K-KL-W model are generally high, the simulation results are more consistent with the experimental data downstream from the recirculation zone. Combined with the Figure 2 and Figure 3, the velocity simulation results of the SA model are more in line with the experimental results.

In order to compare and analyze the difference between the simulated results and the experimental data more accurately, this paper further studies the flow vorticity distribution along the trailing edge of the stabilizer Y/D=0.5. As shown in the figure 5, in both of the experimental and numerical simulations, the maximum value of the vorticity appears near the trailing edge of the stabilizer.
However, the maximum value of the vorticity in the numerical simulation is three times higher than the experimental data, and the vorticity is also decayed very quickly. Especially in the recirculation zone, the simulation results of the vorticity are larger than the experimental results and the error is large. It indicates that the turbulence model does not predict the flow in the recirculation zone well. From the absolute value of the vorticity downstream of the recirculation zone, the simulation results of the SA model are still more accurate.

4. Conclusion
We analyzed the simulated performance of the turbulence model in the velocity and vorticity and obtained the following conclusions:

1) Compared with the normal velocity, the simulation results of the streamwise velocity distribution agree well with the experimental results.

2) The result of the simulation of separation vortex on the trailing edge of the flameholder by each turbulence model is larger and the attenuation is also quite rapid. Overall, the SA model is also more accurate for the simulation of separation.

In summary, the existing turbulence model cannot accurately predict the flow field after the flame stabilizer. In comparison, the Spalart-Allmaras model has a relatively accurate prediction of the length of the recirculation zone in the downstream of the stabilizer, and has a good simulation of the distribution of the streamwise velocity component along the centerline while the rest of the models are more different to the test results.

Acknowledgments
This work was supported by National Natural Science Foundation of China (No. 51506003).

Reference
[1] Bush, Scott M., and E. J. Gutmark. "Reacting and Nonreacting Flowfields of a V-Gutter Stabilized Flame." Aiaa Journal 45.3(2006):662-672.
[2] Menter, F. R. "Two-equation eddy-viscosity turbulence models for engineering applications." Aiaa Journal 32.8(1994):1598-1605.
[3] "Fluent 6.1.22 user's guide." www.fluentusers.com/fluent61/doc/ori/html/ug/main_pre.htm.
[4] Spalding, and B. D. Lectures in mathematical models of turbulence. Academic Press, 1972.
[5] А.А. Авраменко, et al. "RENORMALIZATION GROUP ANALYSIS OF TURBULENCE." (2013).
[6] B. E. Launder, G. J. Reece, and W. Rodi. "Progress in the development of a Reynolds-stress turbulence closure." Journal of Fluid Mechanics 68.3(2006):537-566.
[7] Launder, B. E., and N. Shima. "Second-moment closure for the near-wall sublayer - Development and application." Aiaa Journal 27.10(1987):1319-1325.
[8] Wilcox, David C. Turbulence modeling for CFD. Turbulence modeling for CFD /. DCW Industries, 2006:363-367.
[9] Gibson, M. M, and B. E. Launder. "Ground effects on pressure fluctuations in the atmospheric boundary layer." Journal of Fluid Mechanics 86.3(2006):491-511.
[10] Shih, Tsan Hsing, et al. "A new k-ε eddy viscosity model for high reynolds number turbulent flows." Computers Fluids 24.3(1995):227-238.