Numerical Analysis of Transpiration Cooling in Annular Slinger Combustor

Tao Zhang\textsuperscript{1,a}, Qin Feng\textsuperscript{2,b}, Wenxiang Cai\textsuperscript{2,c, *} and Wu He\textsuperscript{3,d}

\textsuperscript{1}Beihang University, Beijing
\textsuperscript{2}Nanjing University of science and Technology, Nanjing
\textsuperscript{3}Beijing Aerospace Propulsion Institute, Beijing
\textsuperscript{a}3835680@qq.com, \textsuperscript{b}651477045@qq.com, \textsuperscript{c}14486370@qq.com, \textsuperscript{d}714028720@qq.com

Abstract. Wilcox \textit{k-ω} turbulence model, EBU - Arrhenius turbulence combustion model and six-flux radiation model were applied to simulate turbulence combustion flowfield in a real annular slinger combustor. Structured grids of the annular slinger combustor were generated through TTM method and the anchored grid method. For considering the transpiration holes’ cooling effect, reaction flow field in whole annular slinger combustor and in the near-wall region of flame tube were analyzed by numerical simulation, respectively. A new method for data exchanging had been developed to transfer whole flow field numerical result to near-wall region by the inverse distance method, which can solve the contradiction between the requirement of vast computing resources and the computer’s processing ability. Wall temperature and cooling effect of flame tube were researched by numerical simulation method, which provides a technological support for the majorization of flame tube’s structure.

1. Introduction

Slinger combustor can utilize the centrifugal force caused by the high-speed rotation nozzle to make the oil atomized, which matches to centrifugal compressors. And such compressors have been widely used in small and medium size aeroengines for its excellent altitude combustion performance. However, narrow range of fuel-air ratio and synchronous revolution between oil hole and the turbine blades make it easy to generate some fixed hot spots on the turbine blades. Besides, complex geometric structure and complicated combustion flow field in the combustor make it difficult to perform experiment research about such a kind of combustor.

To this day, some scientists put attention on the slinger combustor and have gained some useful conclusions. A kind of preliminary design method has been verified by experiment, which is based on aerodynamic properties and thermodynamic properties of the combustion and the flow field\cite{1}. Fuel slinger rotating speed’s influence on combustion performance is analyzed through experimental methods\cite{2}. Cavity slinger combustor’s flow field has been studied by Fluent and high reflow intensity can reduce the total press loss\cite{3}. Experimental results indicate that the ignition performance of slinger combustor chamber is mainly affected by the rotation speed and air flow rate, and temperature and the constituent of the exhaust gas are also influenced by the rotation speed \cite{4}. Huebner points out numerical simulation can be applied to improve slinger combustor performance \cite{5}. Yan Ying-wen pays a attation on numerical research of combustion flow field in a slinger combustor and finds that the EBU-Arrhenius combustion model is more suitable for engineering research \cite{6}. It can be found that research about slinger combustor is very limited and less work is utilized to understand the complex reaction flow field and the flame tube’s cooling effect.
Reaction flow field in the whole real slinger combustor was numerically researched for evaluating engine’s combustion performance with coarse grid. And fine grid was applied to analyze flame tube wall’s temperature distribution and cooling effect of the divergence hole. In the coarse grid, divergence holes in the flame tube wall were simplified as film. Control equations were discretized by the mixed difference scheme and solved by the PISO algorithm. Turbulence combustion and radiation heat transfer play an important role on the combustion performance and the temperature distribution of flame tube wall in the combustor, which had been respectively considered by using Wilcox k-ω model [7], EBU-Arrhenius model [8] and six flux model. Based on the reciprocal distance method, numerical results gained through the simulation of the whole combustor were used as the boundary condition to analyse the flame tube temperature distribution and the flow field near the wall.

2. Research Object, Grid Generation and Boundary Condition
Annular slinger combustor consists of diffuser, flame tube, inner ring and out ring, as shown in Fig.1. Based on the periodicity hypothesis, structured mesh for one eighteen of the whole combustor was generated by homemade program in order to improve the computational efficiency. As shown in Fig.2 and Fig.3, coarse grid was adopted to consider whole combustor reaction flow field. And whole simulation flow field near inner flame tube wall and outer flame tube wall was divided into 15 subzones. Fig.4 and Fig.5 shows fine mesh for Subzone1 and flame tube wall, respectively. In this kind of combustor, divergence holes are used to cool the flame tube wall down, which can be defined by inclination angle(α=25.), deflection angle(β=30.), hole diameter( d=0.68mm) and wall thickness(h), as shown in Fig.6.

Figure 1. Structure sketch of annular slinger combustor

Figure 2. coarse grids of slinger combustor
Mass flow rate and temperature of air is 7.3kg/s and 742K, respectively. And that of fuel is 0.13kg/s.
3. Numerical Models and Methods

3.1. Wilcox k-ω Model
Wilcox k-ω model has high precision without complex nonlinear oscillation function, which has strong applicability for the prediction precision of near wall region’s flow. In this model, transport equations of turbulent kinetic energy $k$ and turbulent flow frequency $\omega$ can be considered through solving through Eq.1 and Eq.2.

$$\frac{\partial \rho k}{\partial t} + \nabla(\rho U k) = \nabla \left( \frac{\mu}{\sigma_k} \nabla k \right) + P_k - \beta \rho \omega$$  \hspace{1cm} (1)

$$\frac{\partial \rho \omega}{\partial t} + \nabla(\rho U \omega) = \nabla \left( \frac{\mu}{\sigma_\omega} \nabla \omega \right) + \frac{\alpha}{k} P_k - \beta \rho \omega^2$$ \hspace{1cm} (2)

In above formula, $P_k$ is turbulence kinetic energy production rate, and $\beta = 0.09$, $\alpha = 5/9$, $\beta = 0.075$, $\sigma_k = 2$, $\sigma_\omega = 2$.

3.2. Heat Conduction Through Flame Tube Wall
The flame tube wall was divided into n-layer as shown in Fig.7, and the temperature of each layer can be calculated by following formula.

$$T_{w1} = T_{w} - \frac{Q}{G_{th}C_p} \hspace{1cm} T_{w1} = T_{w1} - \frac{Q_0 \sigma_{t-1}}{\lambda} (t = 2 \cdots n)$$  \hspace{1cm} (3)

Figure 7. sketch of n-layer grids for wall temperature computation

Figure 8. Overlap of coarse grids and fine grids
Where $T_H$ is combustion temperature near the inside surface of flame tube, $T_C$ is the temperature of the compressed air near flame tube’s outside surface, $\alpha_r$ is radiation heat transfer coefficient, and $Q$ is heat flux. And $\lambda(T)$ is the thermal conductivity of the flame tube materia, GH3044, which can be expressed as follows formula:

$$\lambda(T) = 11.6 + \frac{14}{840}(T - 373)$$  \hspace{0.5cm} (4)

In this paper, reaction flow field of unabridged slinger combustor was considered by the way of 5-layer solution, and 30-layer solution was applied in simulation of cooling effect on the flame tube.

### 3.3. Data Exchange

The black part represents of the grids used in the simulation of the whole combustor flow field and the red lines indicated the zones used for computing the temperature of the flame tube wall, as shown in Fig. 8. As an example, grids detail for exchanging data near outer flame tube wall’s dilution hole (Subzone 1) is shown in Fig. 9.

![Figure 9. Grids near dilution hole of out flame tube](image)

**Figure 9.** Grids near dilution hole of out flame tube

![Figure 10. sketch for data transfer from coarse grid to fine grid](image)

**Figure 10.** Sketch for data transfer from coarse grid to fine grid

Scale of fine grid for numerical analysis of the flame tube wall’s temperature is about 0.1mm. Vast of computer resources are needed to perform real combustor’s three-dimensional reaction flow field based on this scale, which can’t be satisfied to current researchers for now. In order to research the temperature distribution of flame tube’s wall, the boundaries are set to 5mm up of the wall’s outer surface and 5mm down of the inner surface, which establish the research area. Boundary condition required in the simulation can be obtained by interpolating using the Reciprocal Distance Method (RDM). RDM has the following advantages:

1) Fine grid does not need to be consistent with the profile of coarse grid, which can reduce the amount of grid.

2) Physical quantities distribution gained by interpolation of reciprocal distance method is smooth, which keep the boundary condition’s continuity.

Fig.10 illustrates data transfer scheme of the boundary node from coarse grid to fine grid in two-dimensional mesh. Initially, node E is chosen from fine grid the boundary, comparative query is performed in fine grid to determine grid index in coarse grid according to note E’s geometric coordinates. In this example, node E is located in the control volume ABCD, then the variable $\Phi_E$ in the fine grid can be calculated by the following way:
In the above formula, \( \Phi_A, \Phi_B, \Phi_C \) and \( \Phi_D \) are the values of the corresponding variables of the coarse grid, and \( \overline{EA}, \overline{EB}, \overline{EC} \) and \( \overline{ED} \) are the distances between the fine grid nodes E and the coarse grid A, B, C and D.

4. Discussion of Simulation Results

Numerical simulation program is compiled by Intel Visual Fortran 2013 in Microsoft Visual Studio 2013 platform. All simulation work are finished in a workstation with the SuperMicro X10DAi motherboard, which is equipped with two Xeon E5-2695 V3, 128G DDR4 2133 memory configuration. Calculation for whole combustors’ reaction flow field takes 48 hours, and 1000 hours is expended to finish the simulation for the near wall region’s flow field and flame tube wall temperature.

4.1. Combustion Flow Field of Whole Combustor

For Fig.11 and Fig.12 show the velocity vector distribution and temperature contour at the center of primary hole(K=17) respectively. Fresh air enter flame tube through primary holes, secondary holes and dilution holes, which divides flame tube zone into main combustion zone, secondary combustion zone and dilution zone.

Fresh air enter the flame tube through the left wall at the same direction with the main flow in the flame tube and through the right side at the opposite direction, which result in interleaved air flows between primary holes. Effect by fresh air at the left side is significantly stronger than that at the right side. So, flame temperature on the left is lower than that of the right, which increases flame tube wall’s heat load on the right.

![Figure 11. Vector at K=17 center of primary hole](image1)

![Figure 12. Temperature contour at K=17 center of primary hole](image2)
Fig. 13 and Fig. 14 show velocity vectors and temperature contours at the circumferential center of the flame tube (K=32) respectively. Comparing with Fig. 11 and Fig. 13, it can be found that massive fresh air through the primary hole cuts off the recirculation zone in the primary zone. This phenomenon at the cross section of K=32 is stronger than that at the section of K=17. Comparing Fig. 12 and Fig. 14, it can be concluded that the heat capacity intensity in the primary zone is not symmetrical, and the right side is significantly higher than the left side at both sections.

![Figure 13. vector at K=32 circumference center ( \( \Delta I = 3, \ \Delta J = 3 \) )](image)

![Figure 14. temperature contour at K=32 circumference center](image)

4.2. Wall Temperature and Near Wall Combustion Flow Field
For considering flame tube’s wall temperature and near-wall zone field, numerical results of whole combustor’s reaction flow were used as the boundary conditions by use of RDM. Part of simulation results are shown in Fig. 15~Fig. 21.

Fig. 15 indicates temperature distribution of the flame tube outer wall. It can be found that there is a large temperature gradient on the wall of the flame tube. The fresh air entering the flame tube through the dilution hole forms a good cooling effect on the wall surface and results in an obvious temperature gradient, which not only takes part in the combustion, but also reduces the temperature of the flame tube.
Fig. 16 shows the wall temperature cloud map and velocity vectors near the dilution hole in the Subzone 1. The dilution hole on outer flame tube is close to the diffuser’s outlet. Higher pressure in the diffuser leads to more efficient mixing and cooling effectiveness, which decreases flame tube wall temperature immediately. So, the amount or the diameter of the divergence hole behind the dilution hole can be appropriately reduced in order to improve fresh air’s utilization efficiency.

In Subzone 2, fresh air through divergent holes flows at the same direction with combustion gas and cools the flame tube at a long distance as shown in Fig. 17. As shown in Fig18, wall temperature of flame tube is sufficiently cooled down.

![Figure 15. Temperature contour at out flame tube’s wall](image1)

**Figure 15.** Temperature contour at out flame tube’s wall

![Figure 16. wall temperature&vector near dilution hole in Subzone 1](image2)

**Figure 16.** wall temperature&vector near dilution hole in Subzone 1

![Figure 17. vector of effusion hole center in Subzone 2](image3)

**Figure 17.** vector of effusion hole center in Subzone 2
Fig. 18. temperature & vector of dilution hole inlet in Subzone 2

Fig. 19 shows the wall temperature cloud map and velocity vectors near the dilution hole in the Subzone 3. Comparing with Fig. 20, it can be found that fresh air flowing through the divergent hole into the flame tube is less because the direction of the divergent hole is opposite to the direction of the outer ring fresh air. The diversion effect of the divergence hole makes the cooling air moving downward to the right.

Fig. 19. Temperature & vector of dilution hole out in Subzone 3

Fig. 20. Temperature & vector of dilution hole in Subzone 3

5. Conclusion
In this paper, reaction flow field in whole slinger combustor with fuel slinger have been solved by a homemade program to understand slinger combustor’s wall temperature and near-wall region flow field. The result of the coarse grid was transferred to the fine grid as the boundary conditions for the fine grid through using of RDM. And following conclusions can be obtained:

(1) The number or the diameter of the divergent orifices in the Subzone1 can be reduced, which can improve utilization efficiency of the fresh air.
(2) The wall cooling effect between the flame tube regions 3 and 4 is relatively poor. And the structure in this position need to be further optimized.

(3) One-way data transfer between grids with different scales is realized by the reciprocal distance method, which gains continuous and reasonable boundary conditions for fine grid. This kind of one-way data transfer method effectively reduce the cost of the simulation, ensure the precision and provide an effective solution for the further optimization of flame tube cooling structure.

6. Acknowledgments
The authors gratefully acknowledge the financial support of Project 51306092 supported by National Natural Science Foundation of China, “Mechanism research of pulse combustion and its pollution emission”. The authors would also like to thank the reviewers for their valuable comments and suggestions.

7. References
[1] ZENG Chuan, WANG Hong-ming, SHAN Peng. The design and study of the annular combustion chamber with centrifugal fuel injection. Journal of Aerospace Power, 2011, 18(1):93-96.
[2] SONG Shuang-wen, JIANG Rong-wei, CHEN Jian, etc. Performance experiment of a reflection flow annular combustor with slinger. Journal of Aerospace Power, 2007, 22(9): 1401-1404.
[3] YU Zong-ming, HUANG Yong, WANG Fang. A study on the flowfield of a z-flowpath combustor[J]. Journal of Propulsion Technology, 2010, 31(5): 533-538.
[4] Choi S, Lee D, Park J. Ignition and combustion characteristics of the gas turbines slinger combustor[J]. Journal of Mechanical Science and Technology, 2008, 22(3):538-544.
[5] Huebner S, Exley T. Numerical analysis of the flows in annular slinger combustors[R]. AIAA 90-2164, 1990.
[6] YAN Ying-wen, SONG Shuang-wen, HU Hao-sheng, etc. Numerical investigations of two-phase spray combustion flow field in slinger annular combustor. Journal of Aerospace Power, 2011, 26(5):1003-1010.
[7] Wilcox, D C. Multiscale model for turbulent flows[R]. AIAA 24th Aerospace Sciences Meeting. 1986.
[8] Guo Shang-qun, Zhao Jian-xing, Numerical Simulation of Three-Dimensional Spray Combustion Flow Field by RNG k-E Turbulent Model, Journal of Aerospace Power, 2005, 20(5):807-811.
[9] CAI Wen-xiang, ZHAO Jian-xing, HU Hao-sheng, etc. Numerical methods in simulation of annular combustor cold flow field. Journal of Aerospace Power, 2011, 24(4):748-753.
[10] CAI Wen-xiang, ZHAO Jian-xing, ZHANG Jing-zhou. Calculations of gas combustion and two-phase flows in annual combustors. Journal of Engineering Thermophysics, 2006, 27(Z2):179-182.