INTERACTIVE EFFECT BETWEEN ROTATING DRUM ORIFICE AND TUBED VORTEX REDUCER ON FLOW RESISTANCE CHARACTERISTICS OF SECONDARY AIR SUB-SYSTEM

Yabo Xue
AECC Commercial Aircraft Engine Co., Ltd.
yaboxue@163.com
Shanghai, China

Xiao Chen
AECC Commercial Aircraft Engine Co., Ltd.
chenx@acae.com.cn
Shanghai, China

Lijun Wu
AECC Commercial Aircraft Engine Co., Ltd.
wulj@acae.com.cn
Shanghai, China

Xiuxiu Chen
AECC Commercial Aircraft Engine Co., Ltd.
944318818@qq.com
Shanghai, China

ABSTRACT
The secondary air system (SAS) extracts the pressured air from the aero engine for sealing and cooling itself. The SAS flow resistance characteristics should be reduced for improving the performance of the aero engine. According to the theory of turbo machinery, the main pressure loss of cavity is caused by free vortex. Thus, vortex reducers are widely employed in SAS of modern aero engine. To reduce the pressure loss and improve the energy transfer efficiency, the tubed vortex reducer is designed in the cavity of the aero engine. However, it is seldom reported that the interactive effect rotating drum orifice and tubed vortex reducer on flow characteristics of Secondary Air Sub-system. This paper mainly focused on this work. First, the 3D geometrical model of the secondary air sub-system with tubed vortex reducer is modelled. Second, SST k-ω turbulence model is employed to solve the N-S equations. Third, the flow resistance of the vortex reducer is analysed in the secondary air sub-system. The results indicate that the interactive effect between rotating drum orifice and tubed vortex reducer has influence on the flow behaviour of SAS. The swirling ratio and extra vortex at rotating drum orifice exit bring extra loss of SAS. Besides, the rotating drum orifices have unbalanced mass flow rate around the drum circumference so they can be replaced by less oblong orifices. Numerical results also show that there is angular phase difference between main rotating drum orifice and Vortex reducer. Based on the numerical results, orifice number and angular phase should be optimized during SAS design.

INTRODUCTION
The secondary air system (SAS) extracts the pressured air from the aero engine for sealing and cooling itself. The SAS flow resistance characteristics should be reduced for improving the performance of the aero engine. According to the theory of turbo machinery, the main pressure loss of cavity is caused by free vortex. Vortex reducers are widely employed in SAS of modern aero engine. It plays a significant role and has remarkable influence on the SAS flow resistance and energy transfer efficiency. To reduce the pressure loss and improve the flow behaviour, nozzle and tubed vortex reducer are designed in the cavity of the aero engine.

Facing this topic, so many researchers were engaged in it. At the very beginning, researchers noticed that there was complex flow behaviour in the cavity. Hide (Hide, 1968) proposed the source-sink model and argued that there was critical radius for the angular velocity. As the angular momentum was converted, the angular velocity was less than disk when the radius was larger than the critical radius and it was larger when the radius is smaller than the critical radius. In his work, it was found that the rotating core mainly caused pressure loss. Following this work, core region was studied by many researchers. Owen and Rogers (Owen and Rogers, 1989, Owen and Rogers, 1995) found the thermal gradients between the shroud and inner shaft induced thermal stress. They conducted subsequent investigations of the flow and heat transfer in the inflow cavity. Besides, Owen and Rogers (Owen and Rogers, 1985) used von Karman integral method.
to found a linear Ekman-layer solutions and numerical solutions of full non-linear integral equations. The predictions for the rotational velocity in the core region were in good agreement with the experimental data. Later on, Firouzian (Firouzian et. al., 1985; Firouzian et. al., 1986) measured the rotational velocity in the core region by using LDA. They studied the shroud geometry effect on the inlet swirl and heat transfer within the cavity. Then, Chew and Snell (Chew and Snell, 1988) improved the momentum integral solution by extending Owen’s work and presented the predictions of the pressure loss through the cavity where the fluid entered the cavity with the same tangential velocity as the disc. Till now, the previous works mainly focused on the flow behaviour in the cavity.

After comprehensive understanding the cavity flow, reducing pressure loss began to be concerned about by some researchers. Chew (Chew, 1988) studied the radial fins on the disc surface in the rotating cavity effect on reducing pressure losses. Du (Du et. al., 2012) studied the intersection angle and curvature of the baffles on the disc effect on reducing pressure losses. Based on this work, Vortex reducer was concerned by several researchers. Farthing (Farthing, 1989) used the solutions of momentum integral equations to predict pressure losses in a rotating cavity with de-swirl nozzles and confirmed the flow structure by using flow visualization. Peitsch (Peitsch et. al., 2002) analysed the pressure and temperature distribution using compressor disc model with and without LP shaft through numerical method in tubed vortex reducer. Günther (Günther et. al., 2008) studied the reducing pressure losses effect of four different types of tubed vortex reducer and found them perform well. Following Günther’s work, Wu (Wu et. al., 2014) mainly discussed the geometrical parameters effect on the pressure drop of tubed vortex reducer. Besides, Liang (Liang et. al., 2015) studied the pressure loss in a co-rotating cavity with radial inflow employing tubed vortex reducer with varied nozzles and found that the performance of oblong nozzles was better than the circular nozzles. Meanwhile, Young also studied the influence of nozzles through CFD analysis. Compared with tubed vortex reducer, tubeless vortex reducer achieves lighter weight and easier attachment. However, the tubeless vortex reducer caused unstable operation and led to difficult system design.

Among these works, it is seldom reported that the interactive effect rotating drum orifice and tubed vortex reducer on flow characteristics of Secondary Air Sub-system. This paper mainly focused on the tubed vortex reducer and rotating drum orifice interactive effect on the flow resistance characteristics of the secondary air sub-system. Firstly, SST k-ω turbulence model is employed to solve the N-S equations. Secondly, the 3D geometrical model of the secondary air sub-system with tubed vortex reducer is modelled. Thirdly, the flow resistance of the vortex reducer is analysed in the secondary air sub-system. The results indicate that the interactive effect between rotating drum orifice and tubed vortex reducer has influence on the flow behaviour of SAS. Finally, based on the numerical results discussions, some advices are proposed for improving the vortex reducer.

**MATHEMATICAL MODEL**

**Numerical simulation controlling equations**

Navies-Stocks (N-S) equations, as shown in the equations (1)-(2), are employed as flow control equations in this numerical simulation.

\[ \nabla \left( \bar{\rho} \bar{u} \right) = 0 \]  
\[ \bar{\rho} \frac{\partial \bar{u}}{\partial t} = -\nabla p + \mu \nabla^2 \bar{u} + \bar{f} \]  

\[ \text{SST} \quad k - \omega \]  
Model is employed to solve the Reynolds stress averaged N-S equations, as shown in equations (3)-(4).

\[ \frac{\partial}{\partial t} \left( \rho k \right) + \frac{\partial}{\partial x_i} \left( \rho u_i k \right) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \]  
\[ \frac{\partial}{\partial t} \left( \rho \omega \right) + \frac{\partial}{\partial x_i} (\rho u_i \omega) = \frac{\partial}{\partial x_j} (\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + D_\omega + S_\omega \]

Where \( k \) is the kinetic energy; \( \omega \) is the energy dissipative rate; the term \( G_k \) represents the production of turbulence kinetic energy, and is defined in the same manner as in the standard \( k-\omega \) model. \( G_\omega \) represents the generation of \( \omega \), calculated as described for the standard \( k-\omega \) model in modelling the turbulence production. \( Y_k \) and \( Y_\omega \) represent the effective diffusivity of \( k \) and \( \omega \), respectively. \( Y_k \) and \( Y_\omega \) represent the dissipation of \( k \) and \( \omega \) due to turbulence, calculated as described in modelling the turbulence dissipation. \( D_\omega \) represents the cross-diffusion term. \( S_k \) and \( S_\omega \) are user-defined source terms. The modelling of these parameters refers to Ansys Workbench CFX HELP (Ansys workbench CFX help, 15.0).

**Data processing method**

To evaluate the total pressure loss and total temperature rise, total pressure loss coefficient and total temperature rise coefficient are defined.

Total pressure loss coefficient, which defined that the total pressure ratio between total pressure drop and inlet total pressure, as shown in equation (5). DPO represents total pressure loss coefficient in stationary frame, and DPOR represents total pressure in rotating frame loss coefficient.

\[ \text{DPO} = \frac{p_{\text{out}} - p_{\text{in}}}{p_{\text{in}}} \]  
\[ \text{Total temperature rise coefficient, which is defined that the total temperature ratio between total temperature rise and inlet total temperature, as shown in equation (6). DTO represents total temperature rise coefficient in stationary frame, and DTOE represents total temperature rise coefficient in rotating frame.} \]

\[ \text{DTO} = \frac{T_{\text{out}} - T_{\text{in}}}{T_{\text{in}}} \]  
\[ \text{The total physics quantities are averaged by using mass flow average method.} \]

**GEOMETRICAL AND MESH MODEL**

**3D Geometrical model**

The air system mainly provides the pressurized air for the sealing and cooling the aero engine system. To complete these functions economically, flow resistance should be...
reduced in the flow channel. Vortex reducers are widely used in the aero engines to reduce the pressure loss in the cavity. A kind of vortex reducer with tube has its advantages, such as easily assembled. As shown in figure 1, this kind of vortex reducer consists of drum, cavity, orifice, plate, tube. The pressured air flows from the orifice to the cavity, and then into the tube rotating with the plate. The pressure drop mainly happens in the cavity, rotating drum orifice, inlet and outlet of the tube. According to the structure of vortex reducer, the 3D geometrical flow channel is modelled. To reduce the inlet and outlet effect on the flow resistance of the vortex reducer, the flow domain is enlarged, as shown in figure 1.

3D mesh model
The model is meshed by unstructured elements. According to mesh sensitive analysis, when the elements amount reaches 3 million, the simulation results will not sensitive to the elements amount. To improve the accuracy of the numerical simulation, at least 15 boundary layers mesh was added by using inflation method, the first layer height is 0.04mm, and the inflation ratio is 1.25, as shown in figure 2. During simulation, the elements amount reaches 5 million.

Boundary conditions
The flow domain is set as stationary domain and rotating domain. In this part, the boundary conditions of the numerical model are set as shown in figure 3. The detail boundary conditions are listed below.
1) Inlet domain: Stationary domain;
2) Chamber domain: Rotating domain;
3) Main stream inlet: total pressure and total temperature;
4) Main stream outlet: static pressure and total temperature;
5) Opening outlet: static pressure and total temperature;
6) Periodic boundary: rotating periodic boundary;
7) Chamber outlet: mass flow;
8) Interface: stage method;
9) Chamber stationary wall: counter rotating;
10) Chamber rotating wall: stationary wall.

NUMERICAL RESULTS AND ANALYSIS
According to the previous results, the shape of orifice, mass flow of the system, relative phase angle between orifice and tube, and the length of tube affects the flow behaviour of the system. To investigate these parameters, 10 cases were conducted during this numerical simulation, as shown in table 1. Case 1 represents the circular orifice situation. Compared to case 1, the circular orifice is replaced by oblong orifice positioned in the middle of the domain in case 2. Compared to case 2, the oblong orifice is moved to the right of domain in case 3 and to the left in the case 5. Then, based on case 5, the mass flow characteristics of the system is analysed with mass flow rate varying from 0.775kg/s to 4.20kg/s in case 4, case 5, case 6, case 7, and case 8 respectively. Compared to case 5, the tube length is reduced to 120mm and 90mm in case 9 and case 10 respectively.

Table 1 case study list

<table>
<thead>
<tr>
<th>Flow rate (kg/s)</th>
<th>0.775</th>
<th>1.55</th>
<th>2.30</th>
<th>3.10</th>
<th>4.20</th>
</tr>
</thead>
<tbody>
<tr>
<td>Circular orifice</td>
<td></td>
<td>Case 1</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Middle</td>
<td>-</td>
<td>Case 2</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Oblong Orifice</td>
<td></td>
<td></td>
<td>Case 3</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Right</td>
<td>-</td>
<td>Case 4</td>
<td>Case 5</td>
<td>Case 6</td>
<td>Case 7</td>
</tr>
<tr>
<td>Left</td>
<td>150</td>
<td>Case 9</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>120</td>
<td></td>
<td>Case 10</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td></td>
<td>90</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Comparison of Flow behaviour

As figure 4 shows, case 1 shows the streamline of the sub-system with circular orifice and case 2 shows the streamline of the sub-system with oblong orifice. At the very beginning, 80 circular orifices are employed in the SAS design. However, numerical simulation results indicate that there is distinguishing mass flow rating difference among the orifices around the tangential direction. The air flows into the circular orifice with a very small angle. So the air is accelerated to a very high tangential speed by the circular orifices. In this situation, every circular orifice plays a vortex producer at the outlet of circular orifice. These will enlarges the pressure drop and the total temperature rise of the system. According to analysis, oblong orifices are employed into the system. Compared case 1 and case 2, the oblong orifices improve the flow behaviour orifice’s outlet. It reduces the extra vortex production at the outlet of the orifice. So the following part, we will mainly concern about the Vortex reducer system with oblong orifice.

As figure 5 shows, based on case 2, case 3 and case 5 are conducted to analyse the phase angle effect on the system’s flow behaviour. Case 3 shows the streamline with oblong orifice positioned in upstream and case 5 shows the streamline with oblong orifice positioned in downstream. Compared case 2, case 3 and case 5, it improves the flow behaviour of orifice outlet and suppresses the vortex when the oblong orifice is positioned in the upstream of the tube inlet. The best situation is that the air comes out of the orifice and directly flows into the inlet of the tube. This will reduce the extra vortex produced between the orifice outlet and tube inlet.

Figure 4 Flow behaviour comparison of circular-orifice and oblong-orifice

Figure 5 Flow behaviour comparison of oblong-orifice with different orifice position

As figure 6 shows, based on case 5, case 9 and case 10 are conducted to analyse the tube length effect on the sub-system flow behaviour. Case 9 shows the streamline with tube length 120mm and case 10 shows the streamline with tube length 90mm. Compared case 5, case 9 and case 10, if the tube length is reduced largely, the air coming out of the orifice outlet will not flows into the inlet of the tube. So the extra vortex will be produced, which brings extra the pressure drop between the orifice outlet and tube inlet. According to the analysis, the phase angle and tube length should be coordinated during system design.
Figure 6 Flow behaviour comparison of oblong-orifice system with different tube length

Mass flow characteristic

As figure 7 shows, based on case 5, case 4, case 6, case 7 and case 8 are conducted to analyse the flow characteristic of the system with mass flow rate varying from 0.775 kg/s to 4.2 kg/s. Compared to the temperature rise, pressure drop is sensitive to the mass flow rate of the system. At low mass flow rate, the pressure drop increases slower when the mass flow rate increases. However, the pressure drop increases remarkably in high mass flow rate region. This means the system’s mass flow rate is determined by the critical element. So the element resistance should be designed balancedly.

Figure 7 System pressure drop and temperature rise characteristics with different mass flow rate

Phase angle between oblong orifice and tube effect on Vortex reducer system

As figure 8 shows, relative phase angle effect on pressure drop and temperature rise of sub-system is discussed below. Based on case 2, case 3 and case 5 are conducted to analyse the flow characteristic of the system with oblong orifice position varying from tube’s downstream to tube’s upstream. When the oblong orifice position varies from tube’s downstream to tube’s upstream, both the system’s total pressure drop and temperature rise in stationary frame decrease slightly. This means phase angle affects the flow behaviour of orifice outlet. There must be a best phase angle between oblong orifice and tube.

Figure 8 System pressure drop characteristics with different phase angle
As figure 9 shows, tubed length effect on pressure drop and temperature rise of sub-system is discussed below. Based on case 5, case 9 and case 10 are conducted to analyse the flow characteristic of the system with tube length reduced varying from 150mm to 90mm. Compared to the temperature rise, the system’s total pressure drop in stationary frame increase remarkably. This means that the tube length reducing destroyed the best phase angle between oblong orifice and tube. The oblong orifice outlet produce large vortex and bring extra pressure loss.

Figure 9

Interactive effect on pressure drop and temperature between oblong orifice and tube

To verify the analysis of system behaviour above, the local flow behaviour of oblong orifice and tube are analysed below. As figure 10 shows, Based on case 2, case 3, and case 5 are conducted to analyse the flow characteristic of the system with oblong orifice position varying from tube’s downstream to tube’s upstream. When the oblong orifice position varies from tube’s downstream to tube’s upstream, the pressure loss of oblong orifice decrease and the pressure loss of tube keeps same, so the phase angle makes contribution to improve the system pressure drop.

As figure 11 shows, based on case 5, case 9, and case 10 are conducted to analyse the flow characteristic of the system with tube length reduced varying from 150mm to 90mm. The tube’s total pressure drop in stationary frame decrease remarkably and the pressure loss of the oblong orifices keeps same, however, the system pressure increase remarkably. This means that the tube length reducing destroyed the best phase angle between oblong orifice and tube. The oblong orifice outlet produce large vortex and bring extra pressure loss.
DISCUSSION AND CONCLUSION

In this paper, the flow resistance of vortex reducer is analysed based on numerical simulation. During the simulation, the shape of the orifice, the relative position of the orifice and tube, the length of the tube and the mass flow characteristics are mainly concerned about. After discussion, the conclusion and proposals are drawn in the paper.

First, the circular orifice is not the best design of the rotating drum orifice in the SAS sub system. There is a distinguishing mass flow rating difference among the orifice around the tangential direction. The circular orifice accelerates the air to a very high rotating speed. This will enlarge the pressure drop and raise the total temperature of the air. In terms of the oblong orifice, it improves the flow characteristic of rotating drum orifice. Instead, oblong orifice is employed in the SAS design.

Second, according to the mass flow characteristics of the system, the pressure drop increases remarkably in high mass flow rate region. This means the system’s mass flow rate is determined by some element. So the element resistance should be designed balancedly.

Third, the relative phase angle between orifice and tube will affects the pressure drop of the sub-system. Based on the numerical simulation results, if the oblong orifice position is in the upstream of the tube, the pressure drop will decrease in the same boundary condition. There must be a best phase angle between the oblong orifice and the tube. The best phase angle should be determined by more detail work.

Finally, if the tube length is changed, the pressure drop and temperature rise characteristic will be changed obviously. According to the numerical simulation results, the tube length reducing destroyed the best phase angle between oblong orifice and tube. The oblong orifice outlet produces large vortex and bring extra pressure loss. So the phase angle and tube length should be coordinated during system design.

ACKNOWLEDGMENTS

The author appreciates Peng Wu and Xiaojie Guo for discussion the vortex reducer simulation results.

REFERENCES

[16] Ansys workbench CFX help, 15.0.