A mathematical model for predicting the pressure drop in a rotating cavity with a tubed vortex reducer

Song Wei, Jiaxi Yan, Junkui Mao, Xingsi Han, and Zecan Tu

ABSTRACT

The flow characteristics of a tubed vortex reducer, which was used to reduce the pressure drop in an aeroengine air bleed system, were studied in this paper. Accurate prediction of the pressure drop in a rotating cavity with a tubed vortex reducer installed is important, but the complex flow structure results in complicated local loss characteristic at the tube entrance, which is difficult to estimate. In this paper, a mathematical model for predicting the pressure drop in a rotating cavity with a tubed vortex reducer was established. For the first time, the effect of the local loss at the tube entrance under rotating conditions was considered, and a method of introducing the local loss factor was established based on a research method applied in traditional rotating orifices. Furthermore, the flow characteristics and the distribution of the static pressure in the tubed vortex reducer cavity were studied by the CFD method and experimental tests, and the pressure information inside the tubed vortex reducer was obtained by detailed experimental testing. This study reveals that the local loss characteristics at the tube entrance are mainly influenced by the incident angle; the closer the incident angle is to 0, the greater the speed coefficient is. The pressure drop at the tube entrance is important and represents approximately 10% of the total pressure drop. The mathematical model shows good accuracy for calculating the pressure drop, with an average error of approximately 2% compared to the experimental data.

1. Introduction

In modern aeroengines, cooling air is usually bled from the compressor and transferred to the hot sections through certain complex channels, which are defined as the secondary air system (SAS). The SAS is applied to cool hot components, such as turbine blades and combustor liners, to seal the gaps between the rotor and stator and to balance axial loads. The air bleed system of the compressor is one source of the SAS and has a great impact on the SAS’s total mass flow rate and cooling potential.

When coolant is bled from the compressor and flows radially inward through the corotating disks to the shafts, strong vortices and centrifugal force are generated, leading to a dramatic pressure drop; accordingly, the flow capability of cooling and sealing is quite small. Various devices may be used to reduce the pressure drop across the cavity, such as fins (Chew, Farthing, Owen, & Stratford, 1989), tubeless vortex reducers (Farthing, Chew, & Owen, 1989) and tubed vortex reducers (Negulescu & Pfitzner, 2001). This study focuses on a tubed vortex reducer, which is composed of a series of radial tubes fixed between the disks of the cavity that can markedly guide the radial inward flow.

For a simple cavity (a rotating cavity without vortex reducer systems), Hide (1968) first applied isothermal theory and divided the rotating flow into four regions: the source region, the Ekman-type boundary layers, the core region, and the sink region. On the basis of Hide’s theory, Owen, Pincombe, and Rogers (1985) obtained linear solutions of the Ekman layer equations using the momentum integral method, and the predicted tangential velocity inside a rotating cavity fit well with the experimental data. Firouzian, Owen, Pincombe, and Rogers (1985, 1986) conducted a flow visualization experiment for a corotating cavity with radial inflow, and the flow structure observed was similar to those identified by Hide (1968). They also built a linear model, which has been experimentally validated, based on the linear solutions of Owen et al. (1985) to calculate the pressure drop in a rotating cavity. All these findings were systematically summarized by Owen and Rogers (1995). From these literatures that mentioned before we can see that, in early years, the theoretical analysis and experimental test are...
the commonly used methods in related studies of rotating cavity. However, due to the complex flow and heat transfer characteristics in the rotating cavity, it is not enough to use the theoretical method only. And the rotation experiments are usually expensive, and there are many restrictions on the measurement inside the rotating machine. In this situation, the CFD method shows its advantages. In recent years, CFD method has been more and more widely used in industrial research field because of the effective and economic characteristics (Akbarian et al., 2018; Ardabili et al., 2018; Mou, He, Zhao, & Chau, 2017; Ramezanizadeh, Nazari, Ahmadi, & Chau, 2019), especially in the research direction of aeroengine (Broatch et al., 2014; Fan et al., 2015; Kumar & Govardhan, 2010; Mutlu & Cakan, 2018). Chew and Hills (2007) summarized the applications of CFD in the rotating cavity before 2007, these applications showed that, CFD contributing a lot to the understanding of complex flow characteristic in the rotating cavity. Kumar, Chew, and Hills (2013) compared the performance of three turbulence models RSM, k-ε and SA in estimating the flow in a rotating cavity. Both of these models can give reasonable results. Sun, Amirante, Chew, and Hills (2015) investigated the instability flow characteristic in the rotating cavity by using large eddy simulation (LES). The LES simulation demonstrates that flow unsteadiness in the cavity is largely suppressed by the radial inflow. Later, Onori, Amirante, Hills, and Chew (2016) compares the difference between the LES and RANS in simulating the flow structure in the rotating cavity, and the results shown that the tangential velocity and the pressure drop across the cavity are very well predicted by both RANS and LES.

For a vortex reducer cavity (a rotating cavity with vortex reducer systems), Chew et al. (1989) investigated the several fins attached on one side of the rotating disk by theoretical analysis and experimental methods. He established a theoretical model to predict the pressure drop in a rotating cavity based on the linear solutions of Owen, and he treated the flow as a small perturbation from solid body rotation. Farthing et al. (1989) studied the working characteristics of a tubeless vortex reducer, which refers to a series of inclined nozzles installed in a rotating cavity. He extended the momentum integral method to make it suitable for a tubeless cavity but ignored the loss of the nozzle itself. In a later paper, Farthing and Owen (1991) used the discharge coefficient \( C_d \) to describe the loss of the nozzle and established a complete theoretical model to calculate the flow velocity and pressure drop within a tubeless vortex reducer cavity. Luo, Feng, Quan, Zhou, and Liao (2016) experimental investigated the work performance of several different configurations of vortex reducers, such as the baffle type, the tubeless type and the tubed type. They found that different vortex reducers have different pressure drop characteristic, and the scope of application is also quite different.

Among all the vortex reducers, the tubed vortex reducer is the most widely used one and had installed in many aeroengines such as LEAP-1B, which is designed by CFM International and successfully applied in Boeing 737 max. The researches about the tubed vortex reducer are mainly focused on two aspects. One aspect is using theoretical analysis to study the pressure drop mechanism in the tubed vortex reducer cavity and establish a theoretical model to calculate the pressure drop. The other aspect is using computational fluid dynamics (CFD) or experimental methods to study the effects of operating conditions and geometric parameters on the flow characteristics.

For the first aspect, Pfitzner and Waschka (2000) performed a theoretical analysis on the flow characteristics in a tubed vortex reducer cavity and pointed out that there are several sources of pressure drop in the cavity: the free vortex region in the cavity, the forced vortex region in the tube and the pressure loss at the inlet and outlet of the tube. Furthermore, from the perspective of energy conservation, they gave the equation for calculating the pressure drop in the tubed vortex reducer cavity and used ‘standard correlations’ to estimate the pressure drop at the inlet and outlet of the tube. However, their research focused on a tubeless vortex reducer and did not give verification results for these equations. Günther, Uffrecht, Kaiser, Odenbach, and Heller (2008) analyzed the thermodynamic process in a tubed vortex reducer cavity and proposed a gas-dynamic mixing zone model to calculate the pressure drop. This calculation model considers the local loss at the shroud nozzle and the friction loss inside the tube; however, it does not take into account the pressure drop at the tube entrance. A detailed study of the pressure drop at the tube entrance has not been found in previous literature, and the established prediction models often do not take into account the effects of rotation on the inlet loss.

It is worth pointing out that the local loss characteristics at the rotating tube entrance are similar to those when air passes through orifices because these two phenomena have almost the same flow structure. In both cases, air enters a smaller space from a larger space. During this process, the velocity at the tube or orifice entrance changes rapidly, which causes strong collisions with the wall and creates flow separation with vortices, resulting in additional energy losses. The flow characteristics of air passing through orifices are the focus of research in SASs, and the discharge coefficient \( C_d \) is usually used to describe the local loss and the circulation capacity. Research on a rotating orifice showed that the effects of rotational speed and preswirled flow on the...
discharge coefficient are important. For example, Wittig, Kim, Jakoby, and Weibert (1996) measured the discharge coefficient of an orifice under different rotational speeds. The results showed that the discharge coefficient decreases with increasing rotational speed, and the value at the maximum rotational speed (1000 rpm) is only 66% of the result at 0 rpm for a basic orifice geometry. Dittmann, Geis, Schramm, Kim, and Wittig (2002) and Sousek, Riedmüller, and Pfitzner (2014) studied the discharge coefficient behavior of a rotating orifice with a preswirled approach flow. These studies show that the preswirled flow and rotational speed of the shaft together affect the relative tangential velocity of the air in front of the orifice, and this effect has a large impact on the discharge coefficient. According to these studies of rotating orifices, the effects of the rotational speed and preswirled flow on the local loss at the tube entrance are also important. However, due to the differences in the geometric structure and the flow space downstream between the tube and orifice, it is hard to use the research conclusion of the rotating orifice on the rotating tube entrance directly. In this situation, it is necessary to conduct a detailed study on the local loss characteristics at the tube entrance and establish a complete mathematical model for calculating the pressure drop in the tubed vortex reducer.

The effects of operating conditions and geometric parameters on the flow characteristics of the tubed vortex reducer are summarized as follows. Negulescu and Pfitzner (2001) compared the flow characteristics of a tubed vortex reducer with those of a tubeless vortex reducer, and the former showed more stable behavior than the latter. Günther et al. (2008) measured the pressure drop in a tubed vortex reducer cavity and found that the pressure drop in the cavity shows a monotonic increase with increasing mass flow rate and rotational speed. Peitsch, Stein, Hein, Niehuis, and Reinnmöller (2002) studied the effect of the axial install position of the tube, and they did not find an obvious impact. Liang, Luo, Feng, and Xu (2015) proposed a long-short matched tubed vortex reducer configuration and experimentally studied its performance, observing that such a configuration was still capable of diminishing pressure losses substantially owing to the decreased length of partial tubes. Chen, Feng, and Wu (2014) studied the effect of the tube length on the pressure drop in a rotating cavity. The research results indicate that the length of the tube is an important effect parameter, and the value of the optimal tube length is related to the working conditions. Sibilli, Cho, Kholi, and Mucci (2018) numerically studied the work performance of angled tubes, which shows a good potential in reducing the pressure drop in rotating cavity. Previous studies have reflected the performance of a tubed vortex reducer; however, few studies have focused on the details of the flow structure, and there is a lack of experimental data on the pressure distribution within the cavity.

Based on previous studies, in this paper, a mathematical model for calculating the pressure drop in the tubed vortex reducer cavity was established, and the main influencing factors of pressure drop were considered in this model, especially the local pressure drop at the tube entrance under rotating conditions was analysed in detail. Furthermore, a study on the details of the flow structure was conducted by CFD simulation, and the pressure data inside the rotating cavity were obtained by experimental testing. Finally, the effects of rotational speed and mass flow rate were studied by a mathematical model, CFD simulation and experimental test.

2. Computational method

2.1. Computational model

In this study, the model of the tubed vortex reducer cavity is simplified based on a real air bleed system of a turbine engine that is under development. To reduce the computational cost, an 18° sector physical model is established, as shown in Figure 1(a), that is composed of a stator, two corotating disks, a shroud, a retaining ring and a tube. The stator is the only static body and is painted blue. An oblong type shroud nozzle is located on the mid-axis plane of the shroud, and a labyrinth structure is located between the shroud nozzle and the upstream rotating disk. A retaining ring is installed between the corotating disks and divides the cavity into two chambers. A tube is attached to the retaining ring and is located in line with the shroud nozzle.

The computational model of the tubed vortex reducer cavity, as shown in Figure 1(b and c), contains only the fluid domain and is extracted from the physical model. Coolant enters the rotor-stator cavity, which is surrounded by the shroud, the disks and the static wall. Through the radial inlet, part of the air flow passes through the labyrinth structure and flows out from the radial outlet. The other part of the air enters the corotating cavity through the shroud nozzle, flows along the tube and exits from the axial outlet. The analysis of the flow characteristics is focused on the corotating cavity, and the purpose of setting the rotor-stator cavity is to simulate the flow structure more accurately before the air enters the corotating cavity. The simple cavity shown in Figure 1(d) is a comparison model, and the overall size is consistent with that of the tubed vortex reducer cavity.

The nondimensional sizes of the computational domain are given in Table 1 and are obtained by dividing
2.2. Calculation settings and boundary conditions

In this study, CFD calculations were performed using the commercial software FLUENT. For all cases, the boundary conditions have the same setting method. The periodic boundary condition was applied to the sector’s side surfaces in the circumferential direction. No-slip and adiabatic conditions were set on solid walls. Air was treated as an ideal gas in the present study, and the Sutherland equation was used to calculate the dynamic viscosity, while the specific heat and thermal conductivity were kept constant.

In this paper, our work focuses on the pressure drop in tubed cavity. The SIMPLEC algorithm is an improved method based on the Semi-Implicit Method for Pressure Linked Equations, and it could improve the solution accuracy and the convergence speed of pressure at the same time. And the second-order upwind scheme was used for advection terms to further improve the calculation accuracy. For the turbulence model, both the studies of Kumar et al. (2013) and Peitsch et al. (2002) showed that the $k$-$\epsilon$ model could give reasonable estimation of the...
flow mechanism in rotating cavity. In addition, according to the Fluent user’s guide (2014), the realizable $k$-$\epsilon$ provide superior behavior over the standard $k$-$\epsilon$ model when the computational model has a single moving reference frame system. So the realizable $k$-$\epsilon$ turbulence model was applied in this numerical simulations. In section 2.4, we experimental validated the accuracy of the selected SIMPLEC algorithm and $k$-$\epsilon$ turbulence model.

Pressure values were specified at the inlet and radial outlet. Since the model has two outlets, the target mass flow rate was applied at the axial outlet to ensure that a sufficient mass flow rate passed through the rotating cavity. The specific values of the boundary conditions are summarized in Table 2. Since the rotational speed and mass flow rates change in different cases, only the corresponding ranges are given.

To analyze the flow characteristics in the tubed vortex reducer cavity, we set up a total of 59 calculation cases in 5 groups. A comparison of the flow characteristics between the simple cavity and tubed vortex reducer cavity was conducted in Group A, which included 2 cases. Group B consisted of 12 cases designed to study the effects of rotational speed and mass flow rate on the pressure drop. Furthermore, Group C, Group D and Group E contained a total of 45 cases and were used to study the effect of the geometric parameters on the local loss at the tube entrance. In addition, the possible interaction effects between the rotational speed and geometric parameters were taken into account. All the cases are summarized in Tables 3–7, and the geometric parameters are given in nondimensional form.

### 2.3. Grid system

The ANSYS ICEM CFD meshing tool was used to generate an unstructured grid, where the grid near the wall and the tube entrance have been densified as shown in Figure 2. The first grid size is 0.25 mm near the wall of labyrinth structure, 1 mm near the wall of rotating discs and inner tube, and 0.5 mm at the tube entrance, while the growth ratio is 1.2. The prism generation method was used to improve the grid quality in the near-wall regions. The first layer height is 0.1 mm and the height ratio is 1.2. The dimensionless distance $y^+$ of the first grid center to

### Table 2. Boundary conditions.

| Boundary Conditions                  |
|-------------------------------------|
| Pressure inlet 2117050 Pa, 780 K    |
| Pressure outlet-1 1869470 Pa, 780 K (Backflow) |
| Target mass flow outlet-2 900 kg/h ~ 7200 kg/h, 780 K (Backflow) |
| Rotational speed 955 rpm ~ 21657 rpm |

### Table 3. Group A: Comparison between the simple cavity and tubed cavity.

| Case | $r_i/b$ | $d/b$ | $n$ | $\Omega$ rpm | m kg/h |
|------|---------|-------|-----|--------------|--------|
| 1    | –       | –     | –   | 14438        | 4140   |
| 2    | 0.848   | 0.054 | 20  | 14438        | 4140   |

### Table 4. Group B: The effects of the rotational speed and mass flow rates.

| Case | $r_i/b$ | $d/b$ | $n$ | $\Omega$ rpm | m kg/h |
|------|---------|-------|-----|--------------|--------|
| 3 ~ 6| 0.756   | 0.054 | 20  | 955, 2387, 4775, 14438 | 900    |
| 7 ~ 10| 0.756  | 0.054 | 20  | 955, 2387, 4775, 14438 | 4140   |
| 11 ~ 14| 0.756  | 0.054 | 20  | 955, 2387, 4775, 14438 | 7200   |

### Table 5. Group C: The effect of the length of the tube.

| Case | $r_i/b$ | $d/b$ | $n$ | $\Omega$ rpm | m kg/h |
|------|---------|-------|-----|--------------|--------|
| 15 ~ 21| 0.618, 0.664, 0.710, 0.756, 0.802, 0.848, 0.917 | 0.054 | 20  | 7219         | 4140   |
| 22 ~ 28| 0.618, 0.664, 0.710, 0.756, 0.802, 0.848, 0.917 | 0.054 | 20  | 14438        | 4140   |
| 29 ~ 35| 0.618, 0.664, 0.710, 0.756, 0.802, 0.848, 0.917 | 0.054 | 20  | 21657        | 4140   |

### Table 6. Group D: The effect of the diameter of the tube.

| Case | $r_i/b$ | $d/b$ | $n$ | $\Omega$ rpm | m kg/h |
|------|---------|-------|-----|--------------|--------|
| 36 ~ 40| 0.756   | 0.038, 0.045, 0.054, 0.063, 0.066 | 20  | 7219         | 4140   |
| 41 ~ 45| 0.756   | 0.038, 0.045, 0.054, 0.063, 0.066 | 20  | 14438        | 4140   |
| 46 ~ 50| 0.756   | 0.038, 0.045, 0.054, 0.063, 0.066 | 20  | 21657        | 4140   |

### Table 7. Group E: The effect of the number of tubes.

| Case | $r_i/b$ | $d/b$ | $n$ | $\Omega$ rpm | m kg/h |
|------|---------|-------|-----|--------------|--------|
| 51 ~ 53| 0.756   | 0.054 | 10, 20, 30 | 7219         | 4140   |
| 54 ~ 56| 0.756   | 0.054 | 10, 20, 30 | 14438        | 4140   |
| 57 ~ 59| 0.756   | 0.054 | 10, 20, 30 | 21657        | 4140   |
the wall was nearly 30 with respect to the criteria require for the scalable wall function treatment.

The total number of grids was selected to change from 885674 to 6432112 for this test, and Figure 3 shows the influence of the grid number on the pressure ratio from b to a. This result shows that the pressure ratio has a smaller decreasing trend with increasing grid number, and the pressure ratio no longer shows an effect when the number of grids is greater than 4,125,617, where the average fluctuation is smaller than 1%. Therefore, a method to generate 4,125,617 grids was applied in the next simulation.

2.4. Validation of the computational method

A test case for a simple cavity previously studied by Firouzian, Owen, Pincombe, and Rogers (1986) was simulated first to ensure the proper computational setup in the present simulation. A comparison between the present numerical results, the experimental data (Firouzian et al., 1986) and the linear model solutions (Firouzian et al., 1986) is shown in Figure 4, where the swirl fraction $\beta = V_{\phi}/\Omega r$, which represents the ratio of the air’s absolute tangential velocity to the local linear velocity of the cavity. The CFD results agree well with the experimental data, and the average relative error was approximately 3%. Considering the difference between the simple cavity and the tubed cavity, a further validation has been completed by using the experimental data which were obtained from our own tests in this study. Where, the detailed introduction for our experiment is given in section 3. The conditions of the cases for validate are shown in Table 8. And Figure 5 shows the comparison between the present numerical results and the experimental data, where $P_o$ and $P_a$ represent the static pressure at the outer and inner radii of the cavity, respectively. We can see that, our CFD results show a high accuracy. The average relative error was approximately 1%, which demonstrates the high accuracy of the present CFD method in this paper.

3. Experimental apparatus and test conditions

3.1. Experimental apparatus

The experiments were conducted on the multifunction rotary test rig at the Jiangsu Province Key Laboratory of Aerospace Power System, College of Energy and Power Engineering, Nanjing University of Aeronautics and Astronautics, China. As shown in Figure 6, the
The experimental system includes a gas supply system, test section and data acquisition system. An 11 kW AC motor provides rotating power, and the maximum rotational speed of the rotating cavity can reach 5000 rpm.

The gas supply system provides dry and stable air to the test section. The compressor pumps the airflow into the air resource, passed the drying device, which separates the moisture in the air to ensure the dryness of the compressed air, through rectified. The flow rates are measured by a vortex-shedding flowmeter and controlled by a valve.

Figure 7 shows a schematic of the test section. The test section is surrounded by two rotating disks, a shroud and a shaft. Twenty oblong-type shroud nozzles are located on the mid-axis plane of the shroud, and 20 tubed vortex reducers are fixed on the retaining ring in the cavity. The rotor and the stator are sealed by a contact-type seal ring. The main dimensions of the rotating cavity are consistent with those of the computational domain presented in Section 2.1, where the outer radius of cavity $b$ is 217 mm, the inner radius of cavity $a$ is 77.5 mm, the distance between the two disks $s$ is 50.1 mm, the radius of the tube entrance $r_t$ is 164 mm, and the diameter of the tube $d$ is 13.7 mm. Since the focus of this paper is on the pressure drop in the corotating cavity, the test section represents the region where air enters the shroud nozzle, and
the static pressure measuring points have been marked in the figure.

The data acquisition system is composed of a vortex-shedding flowmeter, an NI pressure acquisition module, a temperature measuring instrument and a rotating signal acquisition device. The vortex-shedding flowmeter and the temperature measuring instrument are used to obtain the volume flow rate and temperature from the pipelines, respectively. The NI pressure acquisition module is used to acquire the pressure data in the static flow area, and the rotating signal acquisition device is used to acquire the pressure data in the rotating cavity, including $P_b$, $P_t$, $P_m$ and $P_a$, which represent the pressure at the outer radius of the cavity, the tube entrance, the middle position and the outlet of the tube, respectively. Two measuring points are arranged at the same radius but separated by an angle of 180° for each pressure. The rotating signal acquisition device is fixed on the shaft and corotating with the cavity; it can collect and store the pressure data and send it out after stopping the rotation.

3.2. Experimental conditions

The purpose of the experiment is to study the mechanism of pressure drop in the tubed vortex reducer cavity, to explore the variation in the pressure drop with the rotational speed and mass flow rate and to verify the accuracy of the mathematical model. The experimental conditions are determined as shown in Table 9. And there are five rotation tests for each mass flow rate, so there are a total of 25 experimental cases.

3.3. Experimental uncertainties

The experimental error mainly comes from the accuracy of used instruments. In this paper, the volume flow rates are measured with a vortex-shedding flowmeter. The temperature in the pipeline near the flowmeter is measured by a K-type thermocouple. All the pressures were obtained by Kulite XTL-190SM transducers and the rotational speed is measured by a photoelectric laser sensor. The range and precision of these instruments are shown in Table 10.

According to the experimental conditions, the uncertainty of pressure is within 2%, and 0.04% for rotational speed. The mass flow rate is calculated by using the ideal gas law,

$$ m = \frac{Q P}{R_g T} $$

where $Q$ indicates volume flow rate, $R_g$ is the gas constant. Based on the error transfer theory, the uncertainty of mass flow varies from 2.6% to 7%. The uncertainty of pressure ratio is within 2.8%.

3.4. Validation of the experimental method

In this experiment, we mainly measure the pressure in rotating cavity, the flow rate and the rotational speed. In order to verify the accuracy of the experiment, a validation experiment of a simple cavity had been completed at first. The comparison between our experimental data and the linear model of Firouzian et al. (1986), which had been experimental validated, is shown in Figure 8. Our test data fit well with the linear model, with an average error of 4.7%.

4. Mathematical model building and results discussion

4.1. Mathematical model building

4.1.1. A theoretical analysis on the pressure drop mechanism

Before performing the theoretical analysis of the pressure drop, a reasonable flow structure model of the tubed vortex reducer cavity was established first based on Hide’s...
Figure 9. The flow structure model for the simple cavity and tubed vortex reducer cavity: (a) Source-sink model (Hide, 1968); (b) Modified model.

theory. Figure 9(a) shows Hide’s ‘source-sink’ model for a simple cavity, in which the flow structure was divided into four regions, a source region near the shroud, separate Ekman-type boundary layers adjacent to the disks, a rotating core in the center of the cavity, and a sink region near the outlet (Hide, 1968). In the source region of the flow, it is assumed that the angular momentum is conserved along the streamlines outside the boundary layers, resulting in a free vortex flow structure. In the core region, the air velocity is tangential, and the rotational speed is greater than that of the disk. The relative sizes of the source and core regions are affected by the mass flow rate and the rotational speed.

Based on the flow structure of the simple cavity mentioned above, a tube is added to represent a tubed vortex reducer cavity, as shown in Figure 9(b). In this modified model, the tube is placed on the retaining ring, the cavity is divided into two chambers, and the radius of the tube outlet is consistent with the inner radius of the cavity. The inner and outer radii of the cavity are denoted by \(a\) and \(b\), respectively, \(r_i\) refers to the radius of the tube inlet, and \(d\) refers to the diameter of the tube. Due to the existence of the tube, the typical source-sink flow structure is changed. In the area before air enters the tube, the flow structure is similar to the source region of the simple cavity, where the incoming flow occurs outside the boundary layers which can be treated as a free vortex flow, and recirculation appears on both sides of the disks. In this paper, we also call this area as the source region. Entering the tube, the air is restricted by the narrow space, which presents a forced vortex flow characteristic known as solid body rotation. This space is named the tube region in this paper.

In the tubed vortex reducer cavity, the pressure drop can be divided into three categories:

1. the pressure drop caused by centrifugal force (Zone A);
2. the pressure drop caused by speed change and local loss (Zone B); and
3. the pressure drop caused by friction loss (Zone C).

In Figure 9(b), Zone A represents the whole cavity, which is marked by a red wireframe and used to indicate the area where the pressure drop is caused by the centrifugal force; Zone B represents the area near the tube entrance marked by a yellow wireframe and indicates the area where the pressure drop is caused by speed change and local loss; and Zone C represents the tube region, marked by a green wireframe, that indicates the area where the pressure drop is caused by friction loss. In Zone A, the air has a large absolute tangential velocity due to the rotation effect, which results in the generation of centrifugal force and a large pressure gradient. In Zone B, due to the rapid change in the flow velocity and the local loss caused by flow separation and collision, a significant static pressure drop occurs. In Zone C, the high radial velocity in the limited area of the tube results in strong friction loss and additional pressure drop.

Here, we ignore the static pressure change at the tube outlet. Due to the feature of the forced vortex flow in
the tube region, the flow characteristics at the tube outlet under the rotating coordinate system are similar to a static pipe outflow. In the process of the static pipe outflow, when the average section of the tube is much smaller than the cross section of the cavity, most of the dynamic pressure are losing due to the local energy loss at the tube outlet, and the static pressure changes slightly.

4.1.2. The pressure drop caused by centrifugal force

Zone A contains both the source region and the tube region. In the source region, Equation (4.1) can be used to describe the absolute tangential velocity. In the tube region, the absolute tangential velocity of the air is the same as that of the tube, which can be expressed by Equation (4.2).

\[
V_{\phi} = c \cdot \omega b^2 r^{-1}, \quad r_t \leq r \leq b \quad (4.1)
\]

\[
V_{\phi} = \omega r, \quad a \leq r < r_t \quad (4.2)
\]

where \(c = V_{\phi,t1}/\omega b\) is the inlet swirl fraction of the air entering the cavity, which means the ratio of the absolute tangential velocity at the outer radius of the cavity to the local linear velocity, where the subscript \(b\) refers to the conditions at the inlet of the cavity. It should be noted that \(c\) is determined by the flow rate, angular speed and geometric parameters of the nozzle on the shroud, and it is not discussed in this paper.

The balance between the pressure gradient and centrifugal force can be expressed in the equation

\[
vdP = \frac{V_{\phi}^2}{r} dr \quad (4.3)
\]

The flow in the cavity is assumed to be an adiabatic process, so we have the equation

\[
P_b V_b^k = P_t^k \quad (4.4)
\]

The integral form of Equation (4.3) is

\[
\int_{P_a}^{P_b} vdp = \int_{r_t}^{b} \frac{V_{\phi}^2}{r} dr + \int_{r_t}^{b} \frac{V_{\phi}^2}{r} dr \quad (4.5)
\]

Solving Equations (4.5) with (4.4) we obtain the equation to calculate the pressure ratio from \(b\) to \(a\) caused by centrifugal force in the tubed vortex reducer cavity.

\[
\Pi_\omega = \left( \frac{P_b}{P_a} \right)_{\omega}
\]

\[
= \left\{ \frac{1}{2} \omega^2 \left[ c^2 b^2 (b^2 r_t^{-2} - 1) + (b^2 - a^2) \right] \frac{k - 1}{k} \right\}^{1/(1-k)}
\]

(4.6)

Here we use the pressure ratio \(\Pi_\omega\) to represent the pressure drop caused by centrifugal force. From Equation (4.6), we can see that when the geometry of the tubed vortex reducer cavity is determined, the pressure drop caused by the centrifugal force is mainly affected by the angular speed of the cavity \(\omega\) and the boundary conditions at the outer radii of the cavity.

4.1.3. The pressure drop at the tube entrance

4.1.3.1. Analysis of the velocity at the tube entrance. In Zone B, the flow condition and local loss are complex. Considering the similarity of the flow structures of the rotating orifice and the tube entrance, the pressure drop at the tube entrance is analyzed in the way applied in the rotating orifice problem (Idris, Pullen, & Read, 2004). Figure 10 shows the velocity triangle of air at the tube entrance under a rotating coordinate system. The subscripts \(t1\) and \(t2\) refer to the conditions before and after entering the tube, respectively, and those positions have been marked with two red dots in the figure.

Considering that the flow area of the cavity is larger than that of the tube, it is assumed that the direction is tangential before air enters the tube with ignoring the micro radial component. Therefore, \(V_{t1}\) can be calculated by Equation (4.1) as follows:

\[
V_{t1} = V_{\phi,t1} - \omega r_t = (c b^2 r_t^{-2} - 1) \omega r_t \quad (4.7)
\]

The velocity after the air enters the tube is in the radial direction and can be expressed by

\[
V_{t2} = \frac{4m}{n \pi d^2 \rho_{t1}} \quad (4.8)
\]

Here, we can see that \(V_{t2}\) is determined by the mass flow rate \(m\), the air density at the tube entrance \(\rho_{t1}\), the number \(n\) and the diameter \(d\) of the tube.
Furthermore, the incident angle is defined as
\[ i = \tan^{-1}(V_{t1}/V_{t2}) \]  
(4.9)
which represents the angle between the direction of the velocity at the tube entrance and the axis of the tube. The sign of \( i \) represents the deflecting direction; a positive value means that the angle of deflection is the same as the direction of rotation, and a negative value indicates the opposite direction. From the definition of \( i \), we can see that this value is influenced by the initial flow conditions at the inlet of the cavity, the rotational speed, the mass flow rate and the geometric parameters of the tube, and this value reflects the combined effect of these parameters on the flow state at the tube entrance. Theoretically, the larger the incident angle is, the smaller the effective flow area at the tube entrance, and this effect results in an increase in the flow separation and the local loss.

4.1.3.2. Analysis of the pressure drop at the tube entrance. The energy conservation equation at the tube entrance for the actual case can be expressed by Equation (4.10), where \( \delta \) indicates the local energy loss at the tube entrance.
\[ T_{t1} + \frac{V_{t1}^2}{2C_p} = T_{t2} + \frac{V_{t2}^2}{2C_p} + \delta \]  
(4.10)
The energy conservation equation for the ideal case, ignoring the local loss, can be expressed by
\[ T_{t1} + \frac{V_{t1}^2}{2C_p} = T_{t2} + \frac{V_{t2,\text{id}}^2}{2C_p} \]  
(4.11)
Considering this is an adiabatic process, we have
\[ \frac{P_{t1}}{P_{t2}} = \left( \frac{T_{t1}}{T_{t2}} \right)^{k/\gamma-1} \]  
(4.12)
From Equation (4.11), the pressure ratio at the tube entrance can be further expressed as
\[ \Pi_f = \frac{P_{t1}}{P_{t2}} = \left[ 1 - \frac{(k-1)(V_{t2,\text{id}}^2 - V_{t1}^2)}{2kR_gT_{t1}} \right]^{k/\gamma-1} \]  
(4.13)
where
\[ T_{t1} = T_b \left\{ 1 - \frac{1/2\omega^2[c^2b^2(b^2r_t^2 - 1)]}{R_gT_b} \right\} \cdot \frac{k - 1}{k} \]  
(4.14)
In Equation (4.13), \( V_{t2,\text{id}} \) is the only unknown variable when the initial boundary conditions are determined.

To obtain the appropriate value of \( V_{t2,\text{id}} \), we introduce the speed coefficient \( C_v \)
\[ C_v = \frac{V_{t2}}{V_{t2,\text{id}}} \]
\[ = \frac{V_{t2}}{\sqrt{(2k/k-1)R_gT_{t1}(1 - (P_{t1}/P_{t2})^{1-k/k}) + V_{t1}^2}} \]  
(4.15)
Comparing Equations (4.10) and (4.11), it can be seen that \( C_v \) can represent the relative severity of the local loss at the tube entrance. The smaller the local loss is, the closer \( C_v \) is to 1. In Section 4.1.3.1, we qualitatively analyze the relationship between the incident angle \( i \) and the local loss, and in Section 4.2.1.2, we use the CFD method to obtain the specific functional relationship between \( C_v \) and \( i \).
\[ C_v = f(i) \]  
(4.16)
Solving Equations (4.8, 4.13, 4.14, 4.15, and 4.16), we obtain the expression of the pressure ratio at the tube entrance as
\[ \Pi_f = \left( \frac{P_{t2}}{P_{t1}} \right) = \left[ 1 - \frac{(k-1)(V_{t2}/f(i))^2}{2kR_gT_b(1 - (1/2)\omega^2^2/c^2b^2r_t^2 - 1)^2)} \right] \]  
(4.17)
Here, we use the pressure ratio \( \Pi_f \) to represent the pressure drop, and this ratio can be calculated as long as the geometric parameters of the cavity and initial boundary conditions are known.

4.1.4. The pressure drop caused by the friction loss
For the pressure drop caused by the friction loss in Zone C, we treat this case as a traditional pipe flow. The standard equation for the pressure drop along the pipeline is
\[ \frac{\Delta P}{\rho} = \lambda \frac{(r_t - a) V_{t2}^2}{2d} \]  
(4.18)
where \( \lambda \) refers to the friction loss coefficient, which can be obtained by the Moody diagram. Converting this equation to a pressure ratio form, we obtain Equation (4.19) to calculate the pressure ratio caused by the friction loss.
\[ \Pi_f = \left( \frac{P_{t2}}{P_{t1}} \right) = \frac{2dR_gT_b}{2dR_gT_b - \lambda (r_t - a) V_{t2}^2} \]  
(4.19)
Here, we use the pressure ratio \( \Pi_f \) to represent the pressure drop.
Since $\Pi_{\omega}$, $\Pi_{t}$, and $\Pi_{f}$ represent the pressure ratio caused by centrifugal force, the pressure ratio caused by speed change and local loss, and the pressure ratio caused by friction loss, respectively, their product can represent the pressure ratio in the entire rotating cavity with a tubed vortex reducer.

$$\Pi = \Pi_{\omega} \cdot \Pi_{t} \cdot \Pi_{f}$$  \hfill (4.20)

### 4.1.5. Procedure for using the mathematical model to calculate the pressure drop

The mathematical model can be used to calculate the pressure drop between outer radius $b$ and inner radius $a$ of the tubed vortex reducer cavity as long as the geometric size of the cavity and initial boundary conditions are known. The procedure for calculating the pressure drop by using the mathematical model is shown in Figure 11. First, the initial boundary conditions and geometric size of the cavity should be given. Second, Equation (4.6) is used to calculate the pressure ratio caused by centrifugal force, $\Pi_{\omega}$. Third, Equations (4.7, 4.8, 4.9, 4.15, 4.17 and 4.21) are used to calculate the pressure ratio at the tube entrance, $\Pi_{t}$. Fourth, Equation (4.19) is used to calculate the pressure ratio caused by friction loss, $\Pi_{f}$. Finally, $\Pi_{\omega}$, $\Pi_{t}$, and $\Pi_{f}$ are multiplied to obtain the total pressure ratio from the outer radius to the inner radius of the cavity.

### 4.2. Numerical and experimental analysis

#### 4.2.1. The flow structure and the local loss characteristics

##### 4.2.1.1. The effect of the tube on the flow structure in the rotating cavity.

To obtain detailed information about the airflow, two surfaces are defined in Figure 12: Surf-1 denotes the central surface of the computational domain in the axial direction, while Surf-2 denotes the mid-axial plane through the tube axis.

The streamlines and the contours of the radial velocity (m/s) on Surf-1 for case 1 and case 2 are shown in Figure 13, where the negative values indicate radial inflow. As shown in Figure 13(a), in the simple cavity, the source region occupies the whole cavity, and the core region, in which the airflow is completely attached to the disks, is not observed. The core region only appears under high rotational speed and low flow rate (Firouzian et al., 1985), while the current case ($\Omega = 14431 \text{ rpm}$, $m = 1.15 \text{ kg/s}$), which is identical to the aeroengine design condition, does not match these criteria. The result of the linear model (Firouzian et al., 1985) for the current case indicates that the outer radius of the core region is $r/b = 0.26$, which is smaller than the inner radius of the cavity $a/b = 0.356$.

The effect of the tubed vortex reducer on the flow structure is shown in Figure 13(b). The tube compresses the source region and greatly enhances the radial inflow.
This phenomenon is consistent with the flow structure model established in Section 4.1.1.

Figure 14 shows the streamlines and the contours of the swirl fraction $\beta$ on Surf-2. From the streamlines of the simple cavity, we can see that the air flows in negative tangential directions under the rotating coordinate system in Zone 1 and deflects to the positive tangential direction in Zone 2 as the radius decreases. In the tubed vortex reducer cavity, the flow structure in Zone 3 is similar to that of Zone 1. After entering the tube, the air flows radially inward, as shown in Zone 4. By comparing the streamlines in Zone 2 with those in Zone 4, the restriction of the tube on the tangential velocity can be easily observed.

From the contours of the swirl fraction, we can also see the effect of the tube on reducing the tangential velocity of the airflow. In Zone 1 of the simple cavity, $\beta$ changes from 0.59 to 0.92 along the radial direction, and the variation range is basically the same as that of the tubed vortex reducer cavity in Zone 3, for which $\beta$ changes from 0.59 to 0.88. In Zone 2, the swirl fraction increases rapidly from 0.92 to 2.91, while in Zone 4, the swirl fraction remains at 1. This result means that, in the radius above the tube entrance, the air in the simple cavity and the air in the tubed vortex reducer cavity have almost the same tangential velocities. Below the tube entrance in the radial direction, the tangential velocity of the air in the tubed vortex reducer cavity is much smaller than that in the simple cavity.

In Zone 5, which is marked in Figure 14(b), the flow separation and reflux vortex at the tube entrance can be observed. This phenomenon occurs when air enters a small space from a larger space and results in additional energy loss. Theoretically, the local loss is affected by the rotational speed, the mass flow rate, the preswirl velocity before air enters the tube and the flow area of the tube.

4.2.1.2. The local loss characteristics at the tube entrance.

In this paper, we use the speed coefficient $C_v$ to represent the relative severity of the local loss at the tube entrance,
and the incident angle \( i \) represents a comprehensive impact parameter.

\( C_v \) and \( i \) are calculated by using Equations (4.15) and (4.9), respectively. The related parameters such as \( V_{t1} \), \( T_{t1} \), \( P_{t1} \), \( V_{t2} \), and \( P_{t2} \) are extracted from the CFD results, where \( V_{t1} \), \( T_{t1} \), and \( P_{t1} \) are obtained at Surf-3, and \( V_{t2} \) and \( P_{t2} \) are obtained at Surf-4, as shown in Figure 15. Surf-3 and Surf-4 refer to the surfaces before and after the tube entrance, respectively. The radial position of Surf-3 is \( r_t \), and the radial distance between Surf-3 and Surf-4 is \( d \). The distance \( d \) was selected to capture the local loss at the tube entrance, and the pressure drop caused by centrifugal force in this region is eliminated.

Figure 16 shows the streamlines at the tube entrance under six different cases, and the speed coefficient \( C_v \) and incident angle \( i \) are also given. These cases have different geometric parameters and rotational speeds. For example, 0.802-0.054-20 means a tube vortex reducer where the length of the tube \( r_t/b \) is 0.802, the diameter of the tube \( d/b \) is 0.054, and the number of tubes \( n \) is 20.

It can be seen from Figure 16(a–c) that the larger the incident angle is, the larger the separation region at the tube entrance. This phenomenon is consistent with the theoretical analysis. The effect of the incident angle on the speed coefficient can be observed from the CFD results such that when \( i \) ranges from \(-20.4\) deg to \(-32\) deg, \( C_v \) decreases from 0.66 to 0.54. Figure 16(d–f) show that when the incident angle remains the same in different cases, the separation region does not change. The CFD results show that when the incident angle is approximately \(-11\) deg, the speed coefficient is essentially maintained at 0.78.

The effects of the rotational speed and the geometric parameters on the incident angle and speed coefficient are shown in Figure 17, in which the sign of \( i \) only represents the deflecting direction.

Figure 17(a) shows that the absolute value of the incident angle \( i \) first increases and then decreases with increasing rotational speed. The closer the incident angle is to 0, the greater the speed coefficient. The maximum value of \( C_v \), 0.812, is obtained when \( i \) is closest to 0°, where \( i = -4.3^\circ \). When \( i \) reaches the peak, \(-20.1^\circ \), \( C_v \) has a minimum value of 0.74. The larger the incident angle is, the larger the flow separation region and the greater the local energy loss. The effect of the rotational speed on the incident angle is explained as follows: For a
Figure 17. The effects of the rotational speed and geometric parameters on the incident angle and speed coefficient: (a) the effect of the rotational speed; (b) the effect of the length of the tube; (c) the effect of the diameter of the tube; and (d) the effect of the number of tubes.

static cavity, the incident angle is zero, while the rotation of the cavity brings the relative tangential velocity to the air at the tube inlet, and the relative velocity increases at first with the increase in the rotational speed. Under these conditions, the linear velocity of the tube inlet is greater than that of the air, which causes a negative incident angle. As the rotational speed continues to increase, the swirl fraction $\beta$ at the tube entrance gradually increases from less than one to greater than one, which causes the incident angle to deflect toward the positive direction. Theoretically, the incident angle will exceed zero if the rotational speed continues to increase.

Figure 17(b) shows that the incident angle is positive for shorter tubes and negative for longer tubes. When $r_1/b = 0.709$, the incident angle is closest to zero, and the speed coefficient reaches a maximum value of 0.775. Both shorter and longer tubes result in greater incident angles and smaller speed coefficients. The effect of the tube length on the incident angle is related to the distribution of the swirl fraction in the radial direction. The swirl fraction at a higher radius is less than 1, resulting in a negative incident angle for longer tubes, and the swirl fraction changes to greater than 1 at a lower radius, so the incident angle is positive for shorter tubes.

From Figure 17(c and d), the effects of the diameter and number of tubes can be observed. As the diameter and number of tubes increase, the absolute value of the incident angle gradually increases, and the speed
coefficient decreases, where $d$ and $n$ together determine the flow area of the tubes and have the same mechanism for the local loss characteristics. The larger the flow area is, the smaller the radial velocity in the tube and the greater the incident angle.

The results shown in Figure 17(a–d) indicate that $i$ is the most important parameter affecting $C_v$. The effects of $m$, $\Omega$, $r/b$, $d/b$, and $n$ on the speed coefficient $C_v$ are conducted by affecting the incident angle $i$.

Figure 18 shows the variation in $C_v$ with $i$ for all CFD cases. The CFD results are symmetrically distributed on both sides of the dashed line with an incident angle of 0 deg. A higher speed coefficient can be observed when the incident angle moves closer to 0 deg. The larger the incident angle is, the smaller the speed coefficient. As the incident angle increases, the speed coefficient decreases significantly. When the incident angle increases from $-3.1$ deg to $-49.6$ deg, the speed coefficient reaches the maximum and minimum values, respectively, decreasing from 0.83 to 0.55, and the reduction range is approximately 34%. A trend curve that is fitted using the CFD results is also given, and the equation is

$$C_v = f(i) = -0.000119i^2 - 0.00021i + 0.774 \quad (4.21)$$

Equation (4.21) is the specific form of Equation (4.16) that can be used to calculate the pressure drop in the tubed vortex reducer cavity. The application scope of this equation is limited by the research conditions of this paper, in which the incident angle ranges from $-50$ deg to $40$ deg.

**4.2.2. The pressure drop characteristics in the tubed vortex reducer cavity**

**4.2.2.1. The pressure distribution in the rotating cavity.**

The mathematical model, CFD method and experimental method were used to study the pressure drop in the rotating cavity.

Figure 19 shows the distribution of the static pressure along the radial direction in the rotating cavity. For this case, the rotational speed is 3000 rpm, and the mass flow rate is 288 kg/h. The measuring points corresponding to the experimental data have been marked in the figure. The CFD results represent the data obtained by the CFD simulation for the experimental test case.

A comparison of the static pressure distributions of the simple cavity and tubed vortex reducer cavity indicates that the pressure drop in the tubed vortex reducer cavity is much smaller than that in the simple cavity. The CFD results show that the tubed vortex reducer decreases the pressure drop in the rotating cavity by 59% for the current case. In the area before air enters the tube $(0.756 < r/b \leq 1)$, the pressure distribution in these two cavities is basically the same, and the differences occur at the tube entrance and inside the tube. At the tube entrance $(r/b = 0.756)$, due to the rapid change in the velocity and the accompanying local loss, the static pressure in the tubed cavity is suddenly reduced. This phenomenon is captured by the CFD calculations and mathematical model, and the data measured by the pressure transducer are also consistent with the results obtained by other methods. In this case, the static pressure drop generated at the inlet of the tube accounts for approximately 11.7% of the pressure drop in the entire tubed vortex reducer cavity. In the tube region $(0.356 \leq r/b < 0.756)$, the pressure drop in the tubed cavity is much lower than that in the simple cavity. The tube effectively limits the tangential velocity of the air, which in turn reduces the centrifugal force and pressure drop.

It can also be seen from the figure that both the CFD method and the mathematical model have high calculation accuracy and can effectively predict the pressure distribution in the tubed vortex reducer cavity. The linear model (Firouzian et al., 1985) provides a good prediction of the pressure drop in the simple cavity;
however, it cannot be used for the tubed vortex reducer cavity.

4.2.2.2. The effects of rotational speed and mass flow rate. The variation in the pressure drop in the cavity with the rotational speed and mass flow rate is also an important research point. Figure 20 shows the variation in $P_b/P_a$ in the tubed vortex reducer cavity with $\Omega$ under different mass flow rates obtained by the experimental test. The mathematical model is also used to calculate the pressure drop, and the curves of the mathematical model are plotted at $c = 0.59$, as the CFD results indicate that the inlet swirl fractions are between 0.5 and 0.63 under these cases. The pressure ratio increases as the rotational speed increases, and the increase in the mass flow rates also increases the pressure drop. The reason is that the higher the rotational speed is, the greater the centrifugal force of the air. When the mass flow rate increases, the absolute local losses at the tube entrance and the radial velocity of the air in the tube both increase, which leads to a reduction in the static pressure.

Figure 20. The variation in the pressure drop with the rotational speed and mass flow rate under experimental conditions.

Figure 21 shows the CFD results for Group B, where the boundary conditions are much closer to the real aero-engine operating conditions. The effects of the rotational speed and mass flow rate on the pressure drop in the tubed vortex reducer cavity are consistent with the conclusions obtained before, and the mathematical model still gives a high accuracy prediction.

5. Conclusions

A combined theoretical, numerical and experimental study was carried out on the flow characteristics of a rotating cavity with a tubed vortex reducer. A mathematical model for calculating the pressure drop in this cavity was established. The local loss at the tube entrance under rotating conditions was discussed in detail, and the conclusions were used to complement the mathematical model. A comparison of the flow characteristics between the tubed vortex reducer cavity and simple cavity was given. In the end, studies on the pressure drop characteristics in a tubed vortex reducer cavity were conducted that included the pressure distribution feature in the cavity and the effects of rotational speed and mass flow rate on the pressure drop. The main observations can be summarized as follows:

(1) The mathematical model established in this paper, which takes into account the effects of centrifugal force, velocity change and local loss at the tube entrance and friction loss inside the tube, showed good agreement with the experimental data and CFD results.

(2) The local loss at the tube entrance is mainly affected by the incident angle, and the speed coefficient $C_v$ decreases with increasing incident angle $i$. The functional relationship between $C_v$ and $i$ was given, $C_v = -0.000119i^2 - 0.00021i + 0.774$.

(3) The pressure distribution in the tubed vortex reducer cavity was obtained by experimental tests, and the static pressure drop generated at the inlet of the tube accounts for approximately 11.7% of the pressure drop in the entire cavity under current research conditions.

(4) The pressure drop characteristics in the tubed vortex reducer cavity are stable. The experimental tests, CFD simulation results and mathematical model predictions show that the pressure drop in the rotating cavity monotonically increases with increasing rotational speed and mass flow rate.

It should be noticed that, the specific functional relationship between $C_v$ and $i$ is fitted by the CFD results, and
its scope of application is limited by the research conditions in this paper, for which the range of rotational speed is 955 rpm $\sim$ 21657 rpm, the range of mass flow rate is 900 (kg/h) $\sim$ 7200 (kg/h). In fact, this scope of application already includes the real aero-engine operating conditions.

Although the tubed vortex reducer has an effective and stable work performance, due to the large size of the tube, it has the additional weight and vibration problems. In the design of the tubed vortex reducer, the size and number of the tubes are expected to be small to reduce the weight, at the same time, the sufficient pressure drop reduction performance need to be protected with enough tubes. So the balance between the weight and the performance of the tubes need to be investigated. In particular, the sensitivity of the pressure drop and the optimization design method for the geometric size of the tubes are worth to be studied in the future. Additionally, instability flow may occur in the tubed rotating cavity due to the disturbance of the tube, and the effects of the unsteady flow on the flow characteristic and heat transfer also need to be further studied.

**Notation**

| Symbol | Description |
|--------|-------------|
| $a$    | Inner radius of cavity |
| $b$    | Outer radius of cavity |
| $c$    | Inlet swirl fraction, $V_\phi_b/\omega b$ |
| $e$    | Length of shroud nozzle |
| $f$    | Diameter of shroud nozzle |
| $C_d$  | Discharge coefficient |
| $C_v$  | Speed coefficient, $V_{t2}/V_{t2{id}}$ |
| $d$    | Diameter of tube |
| $D$    | Nondimensional diameter of tube, $d/b$ |
| $i$    | Incident angle, $\tan^{-1}(V_{t1}/V_{t2})$ |
| $k$    | Adiabatic index |
| $m$    | Mass flow rate |
| $n$    | Number of tubes |
| $P$    | Static pressure |
| $Q$    | Volume flow rate |
| $r$    | Radial coordinate |
| $r_t$  | Radius of the tube entrance |
| $R_g$  | Gas constant of air |
| $s$    | Axial distance between disks |
| $T$    | Static temperature |
| $V$    | Velocity (in a rotating coordinate system) |
| $V_\phi$ | Tangential component of velocity (in a stationary frame) |
| $v$    | Specific volume |
| $r/b$  | Nondimensional radial coordinate |
| $\beta$ | Swirl fraction, $V_\phi/\omega r$ |
| $\omega$ | Angular speed (rad/s) |
| $\Omega$ | Rotational speed (rpm) |
| $\Pi_{\omega}$ | Pressure ratio caused by centrifugal force, $(P_b/P_a)_\omega$ |
| $\Pi_{t}$ | Pressure ratio at tube entrance, $P_{t1}/P_{t2}$ |
| $\Pi_f$ | Pressure ratio caused by friction loss, $(P_{t2}/P_a)_f$ |
| $\Pi_t$ | Total pressure ratio in tubed vortex reducer cavity, $\Pi_{\omega} \cdot \Pi_{t} \cdot \Pi_f$ |
| $\rho$ | Density |
| $\delta$ | Local energy loss at tube entrance |

**Subscripts**

- $a$ Refers to conditions at $r = a$
- $b$ Refers to conditions at $r = b$
- $f$ Refers to the change caused by friction loss
- $id$ Refers to conditions for ideal case
- $t_1$ Refers to conditions at tube entrance before air enters tube
- $t_2$ Refers to conditions at tube entrance after air enters tube
- $\omega$ Refers to the change caused by centrifugal force
- Surf-2 Refers to conditions at Surf-2

**Disclosure statement**

No potential conflict of interest was reported by the authors.

**Funding**

This work was supported by Jiangsu Provincial Natural Science Foundation [grant number BK20160794], The National Natural Science Foundation of China [grant number 51606095].

**ORCID**

Xingsi Han [http://orcid.org/0000-0003-1925-6447](http://orcid.org/0000-0003-1925-6447)

**References**

Akbarian, E., Najafi, B., Jafari, M., Ardabili, S. F., Shamshirband, S., & Chau, K. (2018). Experimental and computational fluid dynamics-based numerical simulation of using natural gas in a dual-fueled Diesel engine. *Engineering Applications of Computational Fluid Mechanics, 12*(1), 517–534.

Ardabili, S. F., Najafi, B., Shamshirband, S., Minaei Bidgoli, B., Deo, R. C., & Chau, K. W. (2018). Computational intelligence approach for modeling hydrogen production: A review. *Engineering Applications of Computational Fluid Mechanics, 12*(1), 438–458.

Broatch, A., Galindo, J., Navarro, R., García-Tiscar, J., Daglish, A., & Sharma, R. K. (2014). Simulations and measurements of automotive turbocharger compressor whoosh noise. *Engineering Applications of Computational Fluid Mechanics, 9*(1), 12–20.

Chen, X., Feng, Y., & Wu, L. (2014, June). The experimental investigations of centripetal air bleed with tubed vortex reducer for secondary air system in gas turbine. *Asme Turbo Expo: Turbine Technical Conference & Exposition, Düsseldorf, Germany, GT2014-26959.*
Chew, J. W., Farthing, P. R., Owen, J. M., & Stratford, B. (1989). The use of fins to reduce the pressure drop in a rotating cavity with a radial inflow. *Journal of Turbomachinery, 111*(3), 349–356.

Chew, J. W., & Hills, N. J. (2007). Computational fluid dynamics for turbomachinery internal air systems. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences, 365*(1859), 2587–2611.

Dittmann, M., Geis, T., Schramm, V., Kim, S., & Wittig, S. (2002). Discharge coefficients of a pre-swirl system in secondary air systems. *Journal of Turbomachinery, 124*, 119–124.

Fan, B., Pan, J., Yang, W., An, H., Tang, A., Shao, X., & Xue, H. (2015). Effects of different parameters on the flow field of peripheral ported rotary engines. *Engineering Applications of Computational Fluid Mechanics, 9*(1), 445–457.

Farthing, P. R., Chew, J. W., & Owen, J. M. (1989). The use of deswirl nozzles to reduce the pressure drop in a rotating cavity with a radial inflow. *Journal of Turbomachinery, 113*(1), 106–114.

Farthing, P. R., & Owen, J. M. (1991). De-swirled radial inflow in a rotating cavity. *International Journal of Heat and Fluid Flow, 12*(1), 63–70.

Firouzian, M., Owen, J. M., Pincombe, J. R., & Rogers, R. H. (1985). Flow and heat transfer in a rotating cavity with a radial inflow of fluid, Part 1: The flow structure. *International Journal of Heat and Fluid Flow, 6*(4), 228–234.

Firouzian, M., Owen, J. M., Pincombe, J. R., & Rogers, R. H. (1986). Flow and heat transfer in a rotating cylindrical cavity with a radial inflow of fluid, Part 2: Velocity, pressure and heat transfer measurements. *International Journal of Heat and Fluid Flow, 7*(1), 21–27.

FLUENT. (2014). *User’s and theory guide*. Canonsburg, PA: ANSYS, Inc.

Günther, A., Ullrecht, W., Kaiser, E., Odenbach, S., & Heller, L. (2008, June). Experimental analysis of varied vortex reducer configurations for the internal air system of jet engine gas turbines. *Proceedings of ASME Turbo Expo 2008: Power for Land, Sea and Air*, Berlin, Germany. GT2008-50738.

Hide, R. (1968). On source-sink flows in a rotating fluid. *Journal of Fluid Mechanics Digital Archive, 32*(04), 737–764.

Idris, A., Pullen, K. R., & Read, R. (2004, June). The influence of incidence angle on the discharge coefficient for rotating radial orifices. *Asme Turbo Expo: Power for Land, Sea, & Air*, Vienna, Austria. GT2004-53237.

Kumar, B. G. V., Chew, J. W., & Hills, N. J. (2013). Rotating flow and heat transfer in cylindrical cavities with radial inflow. *Journal of Engineering for Gas Turbines & Power, 135*(3), 2047–2060.

Kumar, K. N., & Govardhan, M. (2010). Numerical study of effect of streamwise end wall fences on secondary flow losses in two dimensional turbine rotor cascade. *Engineering Applications of Computational Fluid Mechanics, 4*(4), 580–592.

Liang, Z., Luo, X., Feng, Y., & Xu, G. (2015). Experimental investigation of pressure losses in a co-rotating cavity with radial inflow employing tubed vortex reducers with varied nozzles. *Experimental Thermal and Fluid Science, 66*, 304–315.

Luo, X., Feng, A., Quan, Y., Zhou, Z., & Liao, N. (2016). Experimental analysis of varied vortex reducers in reducing the pressure drop in a rotating cavity with radial inflow. *Experimental Thermal and Fluid Science, 77*, 159–166.

Mou, B., He, B. J., Zhao, D. X., & Chau, K. W. (2017). Numerical simulation of the effects of building dimensional variation on wind pressure distribution. *Engineering Applications of Computational Fluid Mechanics, 11*(1), 293–309.

Mutlu, Y., & Çakan, M. (2018). Evaluation of in-pipe turbine performance for turbo solenoid valve system. *Engineering Applications of Computational Fluid Mechanics, 12*(1), 625–634.

Negulescu, D., & Pfützner, M. (2001, June). Secondary air systems in aeroengines employing vortex reducers. *ASME Turbo Expo: Power for Land, Sea, & Air*, New Orleans, Louisiana, USA. 2001-GT-0198.

Onori, M., Amirante, D., Hills, N. J., & Chew, J. W. (2016, June). Les validation for a rotating cylindrical cavity with radial inflow. *ASME Turbo Expo: Turbomachinery Technical Conference and Exposition*, Seoul, South Korea. GT2016-56393.

Owen, J. M., Pincombe, J. R., & Rogers, R. H. (1985). Source-sink flow inside a rotating cylindrical cavity. *Journal of Fluid Mechanics, 155*(155), 233–265.

Owen, J. M., & Rogers, R. H. (1995). Flow and heat transfer in rotating-disc systems, Volume 2: Rotating cavities. Taunton, England: Research Studies Press.

Pfeitsch, D., Stein, M., Hein, S., Niehuis, R., & Reinmoller, U. (2002, June). Numerical investigation of vortex reducer flows in the high pressure compressor of modern aeroengines. *Asme Turbo Expo: Power for Land, Sea, & Air*, Amsterdam, The Netherlands. 2002-GT-30674.

Pfützner, M., & Waschka, W. (2000, August). Development of an aero engine secondary air system employing vortex reducers. 22nd ICAS Congress, Harrogate, U.K.

Ramezanizadeh, M., Nazari, M. A., Ahmadi, M. H., & Chau, K. (2019). Experimental and numerical analysis of a nanofluidic thermosyphon heat exchanger. *Engineering Applications of Computational Fluid Mechanics, 13*(1), 40–47.

Sibilli, T., Cho, G., Kholi, F., & Mucci, A. (2018, June). Numerical analysis of industrial gas turbine secondary air systems employing vortex reducers. *ASME Turbo Expo: Turbomachinery Technical Conference and Exposition*, Oslo, Norway. GT2018-76313.

Sousek, J., Riedmuller, D., & Pfützner, M. (2014). Experimental and numerical investigation of the flow field at radial holes in high-speed rotating shafts. *Journal of Engineering for Gas Turbines and Power, 137*(3), 1–13.

Sun, Z., Amirante, D., Chew, J. W., & Hills, N. J. (2015). Coupled aerothermal modeling of a rotating cavity with radial inflow. *Journal of Engineering for Gas Turbines and Power, 138*(3), 032505.

Wittig, S., Kim, S., Jakoby, R., & Weibert, I. (1996). Experimental and numerical study of orifice discharge coefficients in high-speed rotating disks. *Journal of Turbomachinery, 118*(2), 400–407.