Proceedings of ASME Turbo Expo 2011
GT2011
June 6-10, 2011, Vancouver, British Columbia, Canada

GT2011-45975

NUMERICAL STUDY ON VARIED VORTEX REDUCER CONFIGURATIONS FOR THE FLOW PATH OPTIMIZATION IN COMPRESSOR CAVITIES

Xiao-qin Du, Hui-ren Zhu, Zong-wei Zhang
School of Engine and Energy
Northwestern Polytechnical University, Xi'an 710072, China
E-mail: dxq0921@163.com, zhuhr@nwpu.edu.cn

ABSTRACT

In the internal air system of modern aero-engine, the cooling air is generally taken from the compressor platform, transported radially inwards towards the shaft and further transported to the hot parts. Since strongly swirling air is bled in the compressor rotor to a much lower radius, a means of vortices will be created, which lead to very high pressure losses and limit the cooling flow rate. The present work investigates the fluid flow for twelve kinds of vortex reducer geometries which utilize columnar baffles to divide the cavity into several chambers and avoid extensive vortex formation, therefore the pressure loss decreases and the flow mass increases observably. Based on the numerical study, the pressure loss is closely related to the curvature of the curve and the intersection angle (β) at the entry.

Geometries of columnar baffles installed in the cavity include the straight angled degrees (0^0,45^0,60^0) towards the radial and the curved such as Archimedean spirals ($\rho = a\theta$, a=12,20,30), hyperbolic spirals ($\rho\theta = a$, a=50,80,100) and logarithmic spirals ($\rho = e^{\alpha\theta}$, α =0.3,0.4,0.5). For the straight, the attention is focused on six and twelve baffles which are evenly arranged along the circumferential, but six ones for the curved. Otherwise, a key point is that the total volume of baffles is equal for different vortex reducer geometries.

The results show that vortex reducer configurations with twelve baffles perform better than those with six baffles for the straight with the same angle. Compared with the straight radial baffle, the other kinds of baffles play better roles with respect to reducing pressure losses and gaining large flow mass. As a rule, the large pressure loss occurs when the curvature for the curved is large, because the flow direction changes greatly. A dedicated investigation about the influence of varied vortex reducer configurations is presented in this paper, particular discussions and conclusions are shown.

INTRODUCTION

The internal air system of modern aero-engine is defined as those airflows which do not directly contribute to the engine thrust generation. It usually consists of no discrete components, but in place of being composed of passages, drillings, rotating ducts, component interface etc. namely a complex network with a collection of hardware features. Kutz and Speer [1] carried out the simulation and optimization of such a system which requires correlations of pressure losses and heat transfer directly from experiments or CFD analysis, validated by experiments.

Differing from the other major components such as the compressor, turbine and combustor, which generally have only one main function, the air system has several important functions to perform for the safe and efficient operation of the engine. These functions include internal engine and accessory unit cooling, such as the turbine blades and the nozzle guide vanes etc. bearing chamber sealing prevention of hot gas ingestion into the turbine disc cavities, control of bearing axial loads, control of turbine blade tip clearances and engine anti-icing. All of which must to be ensured during engine operation and affect the life cycle of the heat loaded components, or else the performance of engine will be affected seriously.

Associated with the whole engine performance model, the air system performance was analyzed by D.A. Foley [2].

Typically, 10% to 30% of the main core flow may be used for above-mentioned various functions. In many cases, the bleed air from appropriate stages of the compressor in the air system is routed though the inside of the rotor shaft and flows through different components to the turbine. Here air is taken from the narrow slot between the compressor rotors and led downstream to the space between the compressor disc hubs and shaft.

The maximum allowable temperature of the cooling air in the internal air system of most engine applications is determined by the life targets of the rotating components which are cooled or sealed by this air. On the other hand, to reduce the cooling air required, air probably can be taken from compressor stages further upstream, which can make the cooling air temperature drop observably. Whereas the difficulty with any inward flow is that conservation of angular momentum causes any initial swirl to increase as the air flows inwards, with a resulting high pressure loss. Consequently, an aerodynamic optimization of the airflow path is required to maximize the engine power and efficiency.

The primary aim of previous investigations into this area of gas turbine technology has focused on evaluating the flow losses that occur in rotating passages and cavities, for example, the research by Brillert, Reichert and Simon [3]. Young and Snowsill[4] assessed the relative performance of a series of non-circular offtake passages using CFD techniques. Investigations of the complex cavity were also worked on by Owen et al [5]-[8].

In modern jet engine gas turbines two different kinds of vortex reducer, which increase the total weight but make the performance improved, are used: tubeless and tubed vortex reducer systems, shown in figure 1.

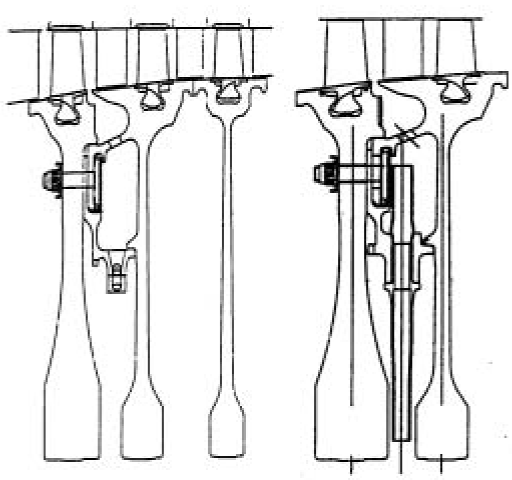


Figure 1: Tubeless and Tubed Vortex Reducer

The tubeless system features a set of nozzles fixed on the compressor discs about halfway down the radial passage. These nozzles which have the advantage of light weight and easy attachment are directed opposite to the direction of disc rotation, thus actively deswirling the air. But the disadvantage is featuring a non monotonous flow characteristic, causing a transient hysteresis effect in extreme operating condition which is concluded by Negulescu and Pfitzner[9]. The basic physics of the tubeless vortex reducer system is analyzed theoretically and experimentally in [10]-[13].

The tubed system consists of an array of radial tubes guiding the air inward from the compressor blade platforms to the disc bores, and reducing the pressure loss. Nevertheless, the tubes increase the additional weight, but also are difficult to attach and prone to vibration. Peitsch et al.[14] numerically simulated this kind of configurations with CFD and a linear flow calculation program. The compares of the fluid flow at different rotational speeds for different vortex reducer configurations with the tubeless or tubed are described by Günther et al. [15].

In the prevent paper the vortex reducer geometries which utilize different kinds of columnar baffles to divide the cavity into several chambers and avoid extensive vortex formation are introduced and simulated. Based on the vortex reducer with the straight radial baffles, the relative mass flow is defined. The assessment and comparison of varied vortex reducer configurations are presented and discussed.

NOMENCLATURE

s	Width of inlet
e	Axial gap between rotating discs
r	Radius
Ω	Rotational speed
γ	Parameter of the straight baffles
a	Parameter of Archimedean and hyperbolic spirals
α	Parameter of logarithmic spirals
ρ	Polar coordinate
θ	Polar coordinate
m	Flow mass
m%	Relative flow mass
β	Intersection angle between tangent lines of baffle and chamber at the point of baffles' entrance

Subscripts

s	the vortex reducer with the straight radial baffles

i Any other cases except the vortex reducer with the straight radial baffles

MODEL GEOMETRIES

The axial view of the compressor disk cavity investigated is shown in figure 2, the air is bled from the narrow slot between the compressor inner blade platforms, afterwards enters into the cavity which is situated between the corotating discs, and then led down to the space between the disc hubs and the shaft, finally axially flows out along the two sides. The distance 'e' between the corotating discs is 4.2mm, and the relations among the geometry parameters are s/e=0.24， r_3/e=13.21， r_2/e=11.67， r_1/e=6.05， r_0/e=4.8.

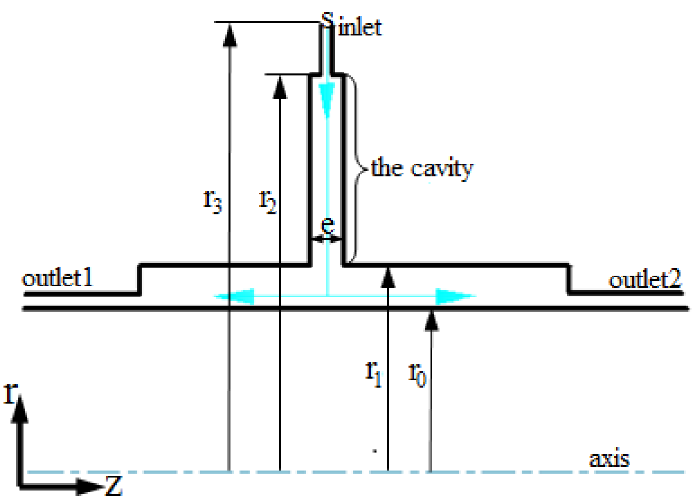
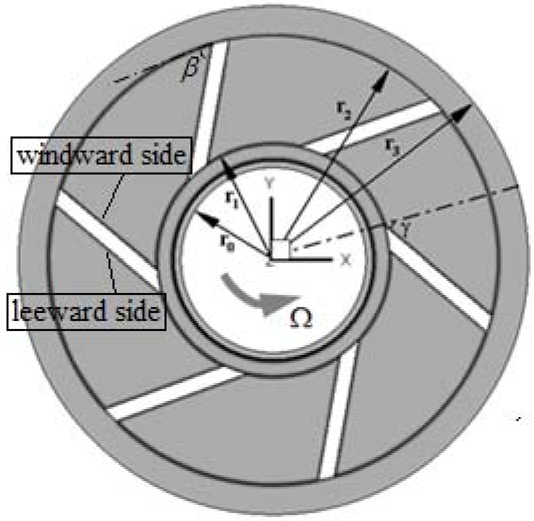


Figure 2: The simplified compressor cavity model

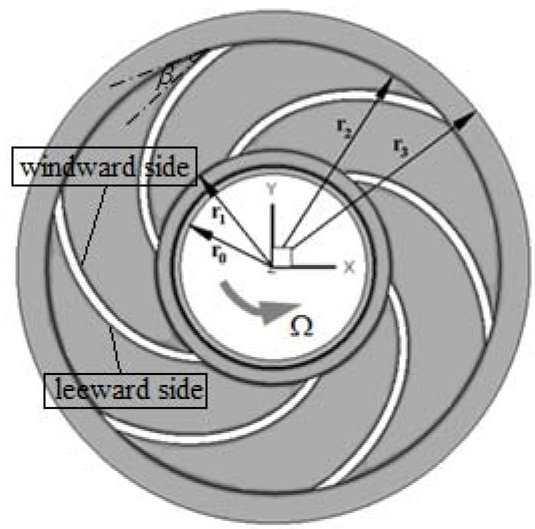
The investigated vortex reducer geometries utilize six columnar baffles which extend axially over the full height of the cavity and divide the cavity into six isometric chambers. Besides the straight radial baffle ($\gamma=0^0$), the baffles include the straight angled degrees ($\gamma=45^0,60^0$) towards the radial and the curved such as Archimedean spirals ($\rho=a\theta$, *a*=12,20,30), hyperbolic spirals ($\rho\ \theta=a$, *a*=50,80,100) and logarithmic spirals ($\rho=e^{\alpha\theta}$， α=0.3,0.4,0.5). Besides, in order to analyze the influence of baffles' number on the fluid flow, the straight of twelve baffles for three different angles are also studied. The circumferential view of the models with the straight ($\gamma=45^0$) and the Archimedean spiral (*a*=20) baffles are visualized in figure 3.

Otherwise, for any aero-engine, the structures of the both shaft ends are complex, that must have difficulty in investigating on fluid flow. The particular attention of this paper is the fluid flow in the cavity with different configurations. To simple the analysis, the annular outflow areas of both exits are controlled to avoid the influence on the flow characteristics in the cavity.

Certainly, columnar baffles in the cavity bring the additional weight, which may affect the performance. To avoid this, the noteworthy is that the total volume of baffles for all the investigated cases is uniform.



(a) the straight of $\gamma=45^0$



(b) the curved of Archimedean spiral(*a*=20)

Figure 3: The examples of models

NUMERICAL METHODS

The present investigations on varied vortex reducers are studied numerically. The software package Fluent in combination with the grid generator Gambit has been used throughout this study for the calculation. The computational domain for all the cases is a circumferential sector which is determined by the geometrical size of the columnar baffle.

Three-dimensional steady state Reynolds-averaged Navier-Stokes equations are solved employing the Fluent 6.3 software with the SIMPLEC algorithm. According to the conclusion drawn by Dieter Peitsch et al [14] who have researched several different kinds of turbulence models and compared the results, realizable *k*-ε turbulence model with standard surface function is adopted in the present study.

The principle of grid generation is the mesh density near the baffles is larger than other regions, because the flow configuration changes possibly greater. A multi-block grid is used in all cases to ensure the highest quality in all regions. In this way, the computational domain for all the cases can be partitioned into several subsections: the narrow slot, the cavity and the axial passage. An appropriate mesh topology is applied for each section. For instance, for the cavity, the quadrilateral mesh on the disc surface is created firstly, then stretching it along axial direction in manner of *cooper*. On the compressor disc surfaces and the baffle surfaces, standard surface function requests that the y^+ value at the wall-adjacent cell should not be between 11 and 60-300, so the density of cells is densified to satisfy this requirement. Among all the cases, the minimum y^+ on the axial passage is 18.45, and the maximum y^+ on the baffle surfaces is 282.62, which is desirable. The computational grid near the baffle surfaces for the computational model of the straight radial baffle is shown in Figure 4. Different meshes have been used to successfully verify mesh independency.

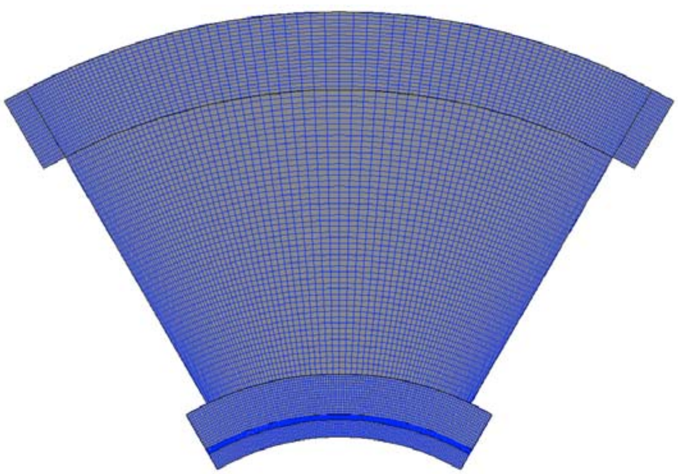


Figure 4: Computational grid near the baffle surfaces for the computational model of the straight radial baffle

All the models are calculated under the same condition, the periodic boundary is employed, and all walls rotate on axis Z with the angular speed of 45000rpm. Pressure boundary conditions are used for both inlet and outlets. At the entry, the air from the compressor must rotate with the compressor platform, therefore the direction cosines of the flow (r, ϕ ,z)=(-0.204,0.979,0) is given in absolute circular cylindrical coordinate, and the positive tangential direction is equal to rotating direction. Such flow style at the inlet, that the tangential velocity is an order of magnitude higher than the axial or radial components, also can make the relative tangential component to the disc reduced, which is good for reducing entry losses. The inlet total pressure is 415896pa and the total temperature is 507K, The static pressures of outlet1 and outlet2 are 267470pa and 317230pa respectively.

The material density is calculated by ideal gas. Separation implicit procedure algorithm is employed to solve all the difference equations, the discretization scheme for all the transport equation's convection terms is the format of Second Oder Upwind, the well-known SIMPLEC method is introduced for pressure-velocity coupling, and pressure interpolation method is standard way. Convergence criteria in all cases are that the residuals of parameters are found to be in the order of 10^{-5} at the end of the calculation.

RESULTS

Flow mass

As we all know that the magnitude of flow mass directly reflects the degree of the pressure loss under the condition of special pressure difference between inlet and outlet. For all the cases numerically investigated in this paper, the only difference is the geometric distinctions in the cavity, which surely make the inward flow path differ and lead to varied flow masses in different cases. Consequently, the particular interest is fixed on the flow characteristics in the cavity and the flow mass into the cavity.

For the assessment and comparison of the flow losses of all the vortex reducer configurations, the vortex reducer with six straight radial columnar baffles in the cavity is used as a basis to define relative flow mass as follows:

$$m\% = \frac{m_i - m_s}{m_s} \times 100\% \quad (1)$$

Where m_s expresses the flow mass into the cavity with six straight columnar baffles and m_i represents the flow mass of any other case. The value of reference mass flow m_s is 23.562g/s. Table 1 shows the computational results.

From table 1, it can be seen that the relative flow mass in any other case is positive. That is to say any other kind of baffles play better roles with respect to reducing pressure losses and gaining large flow mass than the six straight radial baffles. In details, for the straight, the larger the angle the baffle towards the radial, the greater the relative flow mass with the same number of baffles in the cavity; the vortex reducer configurations with twelve baffles in the cavity have better performances of gaining large flow mass than those with six baffles when the angle the baffle towards the radial is the same. For the curved of the same baffle type, there are great differences along with the variation of parameter values. For example, the relative flow mass of the vortex reducer with Archimedean spiral baffles becomes larger as the parameter value *a* increases. The notable differences are also happened among varied baffle types.

Table 1 the results of relative flow mass for all the cases

Baffle type	Parameter value	β	Baffle curvature	Baffle NO.	m%
The straight	$\gamma=0^0$	90^0	1	6	0
			1	12	7.528%
	$\gamma=45^0$	68.497^0	1	6	2.869%
			1	12	9.241%
	$\gamma=60^0$	63.325^0	1	6	12.735%
			1	12	27.261%
Archimedean spiral $\rho=a\theta$	a=12	13.761^0	[0.00056,0.00656]	6	35%
	a=20	22.203^0	[0.00233,0.02025]	6	18.708%
	a=30	31.477^0	[0.00631,0.03579]	6	10.073%
Hyperbolic spiral $\rho\theta=a$	a=50	44.421^0	[0.00510,0.00518]	6	5.298%
	a=80	31.487^0	[0.00189,0.00361]	6	32.177%
	a=100	26.105^0	[0.00107,0.00252]	6	35.507%
Logarithmic spiral $\rho=e^{\alpha\theta}$	$\alpha=0.3$	16.699^0	[0.01955,0.03771]	6	30.334%
	$\alpha=0.4$	21.801^0	[0.01895,0.03655]	6	20.156%
	$\alpha=0.5$	26.565^0	[0.01825,0.03521]	6	12.999%

Analysis

The size of isometric chambers divided by varied columnar baffles is necessarily different, which mainly affects the flow characteristics not only in the cavity but also at the entry. The bleeding air from the narrow slot between the compressor rotors swirls strongly, and its flow direction changes to a certainty due to the action of baffles at the entrance of chambers, which must result in pressure losses, moreover the greater the direction changes, the larger the pressure losses. Flow patterns for all cases in the domain are also used to see how pressure losses occur in the cavity, the following analysis will be carried out according to the flow field in a cut perpendicular to axis of the domain in the middle of the columnar baffles.

It has to be remembered that due to the tangential component of the inlet and the high rotational speed of the compressor, the tangential velocity is an order of magnitude higher than the axial or even radial one before the air enters into the chambers. This high tangential component can be seen in following graphs (note: the frame of reference is static, not corotating with the discs), and it affects the inflow into the chambers significantly. These graphs also show that the air flow inwards along the windward side of baffles, and there are counter clockwise (namely cyclonic) vortices rotating in the same direction with rotational direction of the compressor in the chambers for all the cases.

For the straight, the vortex reducer with bigger intersection angle (β) can make the direction of the flow with high tangential velocity change greater at the entrance of chambers. Therefore, it is easy to understand why the vortex reducer configuration with the straight columnar baffles oblique to the radial obtains larger flow mass than that with the straight radial baffles, and that the relative flow mass becomes larger with the oblique angle increasing, while the number of baffles in the cavity is equal. It can be seen from figure 6(a) and figure 7(a) that there are not only cyclonic vortices close to the windward side of baffles but also anti-cyclonic vortices close to the leeward side of baffles in each chamber when the baffles is oblique to the radial, that means to cause additional pressure losses. However, compared to the cyclonic vortices (shown in figure 5(a)) in the cavity of the vortex reducer with straight radial baffles, the vortices in the chambers clearly weaken when the baffles are oblique to the radial. That is to say, pressure losses introduced by vortices are properly reduced when the baffles oblique to the radial. According to the comparison of the relative flow mass shown in table 1 on the condition of the same number of baffles, it is also illuminated that larger pressure losses happen to the vortex reducers with straight baffles oblique to the radial than that with straight radial baffles even though there are the anti-cyclonic vortices existing by the leeward side of the baffles oblique to the radial.

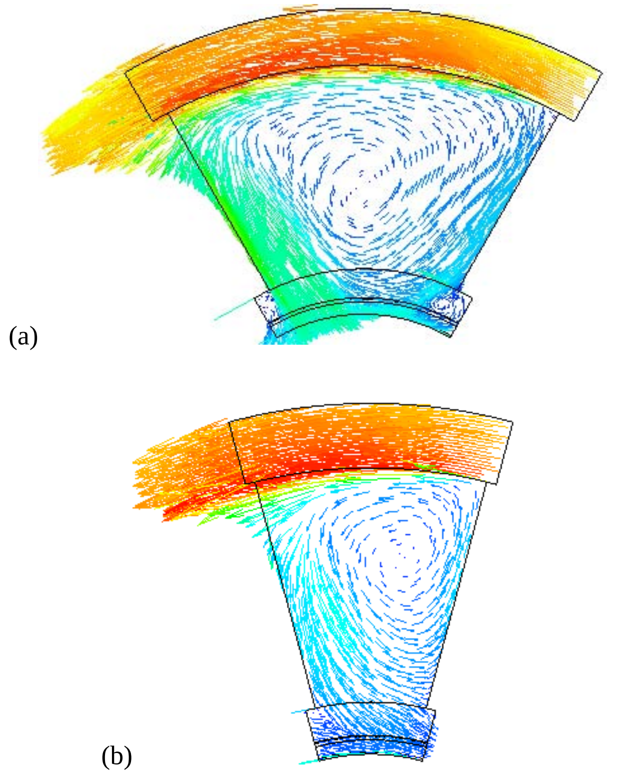


Figure 5: Flow pattern in the cavity with straight radial baffles
(a) Baffle NO: 6 (b) Baffle NO: 12

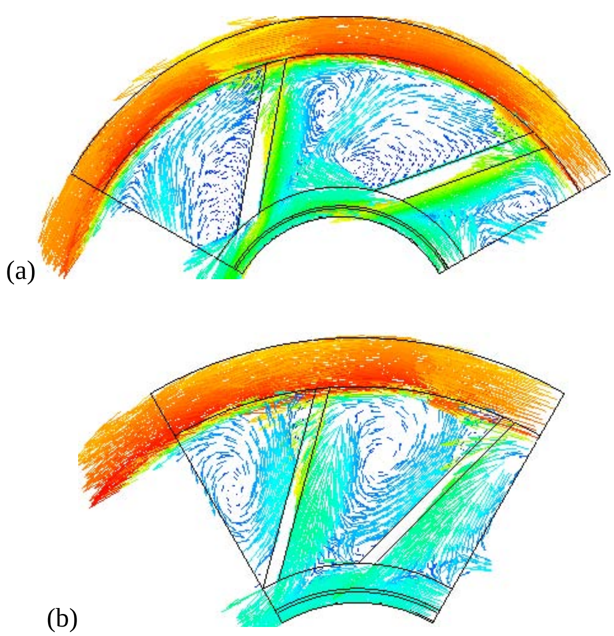


Figure 6: Flow pattern in the cavity with straight baffles ($\gamma = 45^0$)
(a) Baffle NO: 6 (b) Baffle NO: 12

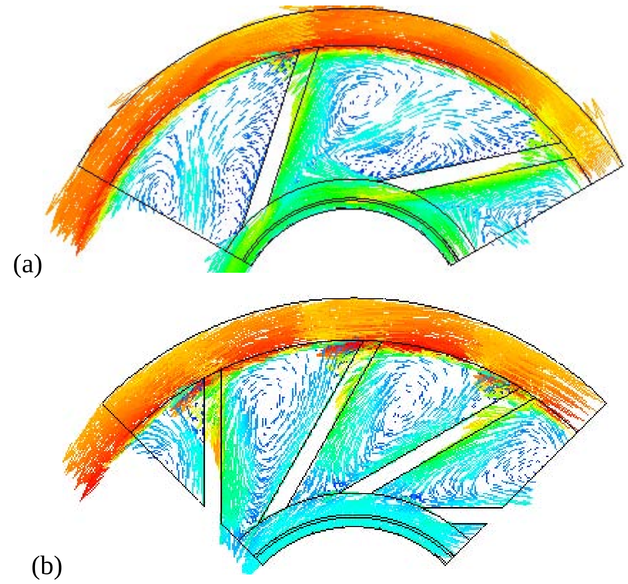
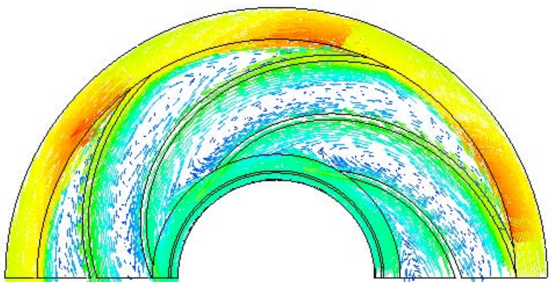


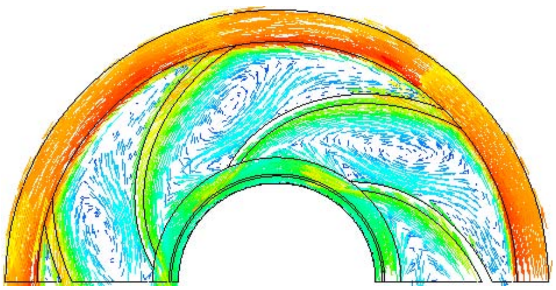
Figure 7: Flow pattern in the cavity with straight baffles ($\gamma = 60^0$)
(a) Baffle NO: 6 (b) Baffle NO: 12

The same as the vortex reducer configurations with the straight baffles in the cavity, the intersection angle (β) at the entrance of the chambers plays the same important role on the flow field in the cavity of the vortex reducer configurations with the curved baffles. The different from the straight is that the flow direction changes continuously when the air flow radially inwards along the windward side of the curved baffles, which may causes another pressure loss. To research the impress of the curved baffles on flow characteristics in the cavity, it is essential to know the ranges of their curvature for all the cases investigated.

Table 1 obviously indicates that the smaller the curvature of the baffles, the larger the relative flow mass into the cavity with the same type of baffles except logarithmic spiral. Compared the vortex reducer configurations with different types of baffles, it can be summarized as follows: for the type of Archimedean spirals, both the intersection angle (β) and the lower limit of the curvature are smaller, but all for that the upper limit of curvature is larger, and the flow direction deviates largely from its foregoing orientation; for the type of hyperbolic spirals, in spite of the curvature is smaller, the intersection angle (β) is larger, the pressure may loss largely at the entry of the chambers; on the contrary to hyperbolic spirals, the curvature is larger and the intersection angle (β) is smaller for the type of logarithmic spiral. Those mean the intersection angle (β) and the curvature of baffles play important and collective roles in reducing pressure losses and gaining large flow mass. It also can be summed up that both the smaller curvature and the smaller intersection angle (β) have better performances of reducing pressure losses and gaining large flow mass.

Figure 8-10 respectively show how the air inwards attaches to the windward side of baffles and outwards flows by the leeward side of baffles within the chambers shaped by the various curved columnar baffles. This creates an anti-clockwise rotating vortex creating the losses. Besides, seen from figure 8(c), another clockwise vortex is established on the leeward inside of the baffles, which affects the flow mass in a detrimental way.


(a)


(b)

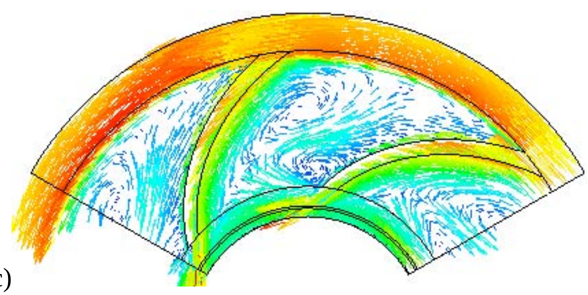
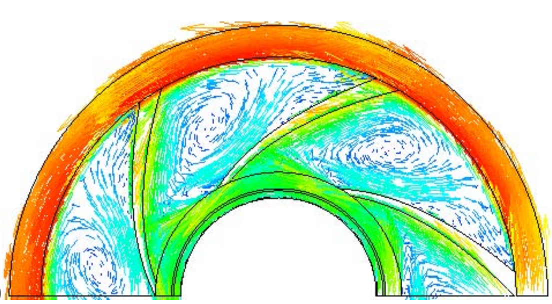
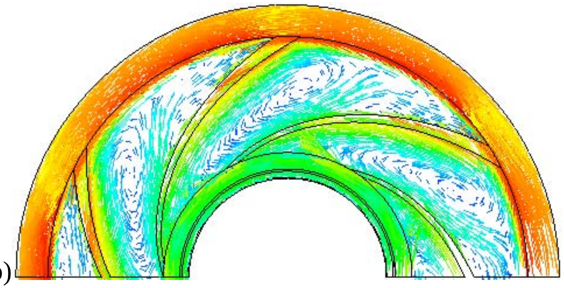

(c)

Figure 8: Flow pattern in the cavity with baffles of Archimedean spiral
(a) $\rho = 12\,\theta$ (b) $\rho = 20\,\theta$ (c) $\rho = 30\,\theta$


(a)


(b)

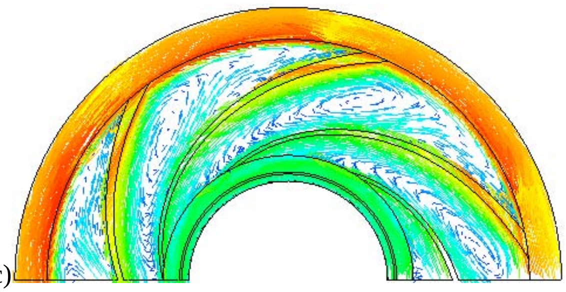
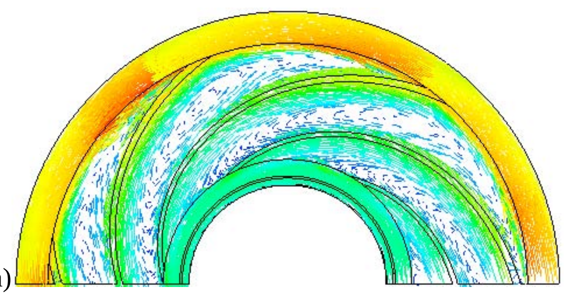
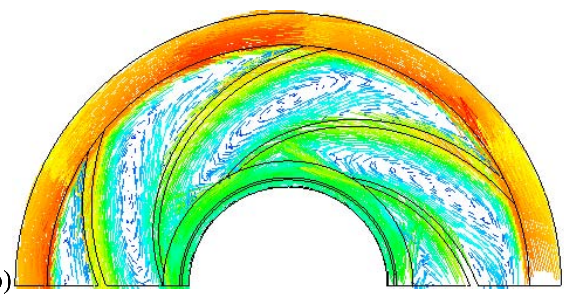

(c)

Figure 9: Flow pattern in the cavity with baffles of hyperbolic spiral
(a) $\rho\ \theta = 50$ (b) $\rho\ \theta = 80$ (c) $\rho\ \theta = 100$


(a)


(b)

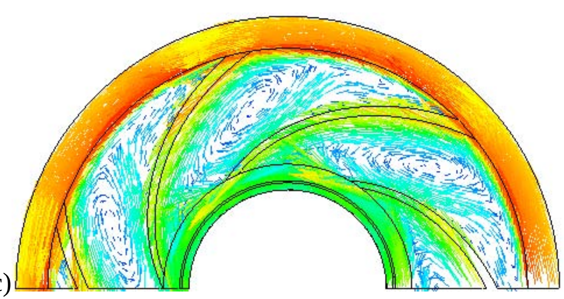

(c)

Figure 10: Flow pattern in the cavity with baffles of logarithmic spiral
(a) $\rho = e^{0.3\,\theta}$ (b) $\rho = e^{0.4\,\theta}$ (c) $\rho = e^{0.5\,\theta}$

Contrastively, the straight baffles have larger intersection angle (β) and constant curvatures, the curved baffles have smaller intersection angle (β) but continuous changing curvatures. However, both the intersection angle (β) and the curvature make the flow direction change just in different locations, and they play important and collective roles in reducing pressure losses and gaining large flow mass. From table 1, on the condition of the same number of baffles in the cavity, there are only two cases of the curved (the Archimedean spiral ($\rho = 30\theta$) and the hyperbolic spiral ($\rho\ \theta = 50$)) with lower flow mass than the case of the straight ($\gamma = 60^0$) whose flow mass is highest among the three straight cases. It seems that the vortex reducer configurations with the curved baffles have better performances with regard to reducing pressure losses and gaining large flow mass.

To investigate the impact of the dimension of chambers shaped by the varied baffles on the flow mass, an additional study was performed. In this study, the number of baffles was increased from six to twelve, but only for the straight baffles. Figure 5-7 respectively give the contrast of flow patterns on the condition of different numbers of baffles in the cavity. It is obvious that the vortices weakens, and there are only cyclonic vortices in the cavity with the baffles oblique to the radial when adding more baffles in the cavity. Which make the pressure losses reduce, and that's why the flow mass increases.

CONCLUSIONS

Both the twelve vortex reducer geometries with different baffles dividing the cavity between the compressor discs into six isometric chambers and the vortex reducer geometries with twelve straight baffles in the cavity were introduced. The computational results of flow mass were presented, and the fluid flow in the cavity was analyzed.

According to the numerical research carried on the vortex reducer geometries, both the intersection angle (β) and the curvature are important to reduce the pressure losses and gain larger flow mass. Although the straight baffles have constant curvature, the intersection angle (β) is larger, so the flow mass is still not very high. The intersection angle (β) of the curved baffles is smaller, but the continuous curvature changing makes the flow direction change continuously. However, the number of baffles in the cavity increases once, the vortices in the chambers weaken, and the flow mass is enhanced observably.

The presented study illuminates that both minimizing the intersection angle (β) at the entrance of the chambers and the curvature changing in the minimal range are the best way to optimization the flow path in the compressor cavity. Meanwhile, the baffles in the cavity growing in number also can improve the flow mass.

This paper is an academic research by numerical method, based on the investigation, the further experimental testing are also planned.

ACKNOWLEDGMENTS

Thanks to the National Basic Research Program of China who sponsored this work.

REFERENCES

[1] Kutz, K.J., Speer, T.M.: Simulation of the Secondary Air System of Aero Engines. J. of Turbomachinery, Vol. 116,1994.
[2] D. A. Foley: On the Performance of Gas Turbine Secondary Air Systems. ASME 2001- GT-0199, 2001.
[3] Brillert, Reichert and Simon: Calculation of Flow Losses in Rotating Passages of Gas Turbine Cooling Systems, ASME 99-GT-251, 1999.
[4] Young, C., Snowsill, G.D.: CFD Optimization of Cooling Air Offtake Passages within Rotor Cavities, ASME GT-2002-30480, 2002.
[5] Owen, J.M.: Thermodynamic Analysis of Buoyancy-induced Flow in Rotating Cavities. ASME GT2007-27387, 2007.
[6] Owen, J.M., Rogers, R.H.: Flow and Heat Transfer in Rotating-Disc Systems, Volume 2: Rotating Cavities. Wiley 1995.
[7] D. Brillert, H.J. Dohmen et al: Application of Conjugate CFD to the Internal Cooling Air Flow System of Gas Turbines. ASME GT2003-38471, 2003.
[8] Cloud, D., Strearns, E.: Probabilistic Analysis of A Turbofan Secondary Flow System. ASME GT2004-53197, 2004.
[9] Negulescu, D., Pfitzner M.: Secondary Air Systems in Aeroengines Employing Vortex Reducers. ASME 2001-GT-0198, 2001.
[10] J. W. Chew, R. J. Snell: Prediction of the Pressure Distribution for Radial Inflow between Co-rotating Discs. ASME 88-GT-61, 1988.
[11] J. W. Chew, P. R. Farthing, J. M. Owen and B. Stratford: The Use of Fins to Reduce the Pressure Drop in a Rotating Cavity with Radial Inflow. ASME 88-GT-58, 1988.
[12] P. R. Farthing and J. M. Owen: De-swirled Radial Inflow in a Rotating Cavity, Int. Journal of Heat and Fluid Flow, Vol. 12, No. 1, March 1991.
[13] M. Pfitzner, W. Waschka: Development of an Aero Engine Secondary Air System Employing Vortex Reducers, ICAS congress August 2000, Harrogate, UK, Paper No. 551.
[14] Peitsch, D. Stein, M., Hein, St. Niehuis, R., Reinmöller, U.: Numerical Investigation of Vortex Reducer Flows in the High Pressure Compressor of Modern Aeroengines. ASME 2002-GT-30674, 2002.
[15] André Günther, Wieland Uffrecht, Erwin Kaiser, Stefan Odenbach: Experimental Analysis of Varied Vortex Reducer Configurations for the Internal Air System of Jet Engine Gas Turbines. ASME GT2008-50738, 2008.

