JST: Engineering and Technology for Sustainable Development
Volume 33, Issue 4, October 2023, 056-062

Optimizing Dump Diffuser in Gas Turbineby Bleeding Air Method and Vortex Generators

Duc-Huy Ta, Van-Minh Le *
Hanoi University of Science and Technology, Ha Noi, Vietnam
* *Corresponding author email: minh.levan@hust.edu.vn*

Abstract

The diffuser is a component that connects the combustion chamber and the compressor in a gas turbine engine. Its main function is to slow down the airflow supplied by the compressor to enhance efficient combustion and avoid excessive total pressure loss. Therefore, the effectiveness of the diffuser is evaluated based on two values: the total pressure loss coefficient and the static pressure recovery coefficient. The smaller the total pressure loss coefficient, the better the static pressure recovery coefficient. In previous studies researchers have given direct attention to the inlet angle of the diffuser. The results indicated that there is a clear flow separation on the diffuser wall before the angle of the front diffuser wall is extended to a certain degree. Consequently, there is an increase in the region of recirculation process, thereby increasing the total pressure loss of the system and adversely affecting the performance of the diffuser. However, only a small number of studies have focused on addressing this issue. In this research, two methods are introduced to improve the performance of the diffuser: the bleeding air system and vortex generators. In this study, numerical methods are used in simulating the 3d model of the diffuser. The computational results indicate that swirl generators and air injection methods can be used to delay the separation process in the front diffuser and reduce the total pressure loss and improve the performance of dump diffuser by over 20%.

Keywords: Dump diffuser, pressure loss, flame tube, vortex generator, bleeding air.

1. Introduction

In the past ten years, the demand for aviation transport and energy has significantly increased. An essential component of a gas turbine engine is the combustion chamber, which consists of a diffuser, shell ring, and flame tube. The diffuser plays a crucial role in reducing the velocity of gas entering the combustion zone to maintain the flame. However, the efficiency of the diffuser is influenced by the total pressure loss in the diffuser. Among different types of diffusers, the render diffuser is commonly used due to its ability to reduce flow velocity while minimizing pressure loss, attracting the attention of researchers.

Several studies have examined the relationship between combustor walls and total pressure loss. Sanalkumar *et al.* [1] presented an idealized physical model of a dump diffuser and analysed the static pressure distribution on the upper walls of diffusers with different combustor walls. Karki *et al*. [2] employed computational techniques based on the Navier-Stokes equations to study the dump diffuser and obtained outcomes closely matched the experimental data. Honami *et al.* [3] demonstrated the effectiveness of inclined walls in regulating the inlet flow of the diffuser. Carrotte *et al.* [4] conducted a study on various diffuser models and found a correlation between large total pressure loss and the distance between the diffuser outlet and the short dome tip. Ghose *et al.* [5] investigated the influence of pre-diffusion angles on static pressure recovery and observed that the dump gap had a significant impact on static pressure recovery only at small prediffuser angles. Garrotte *et al.* [6] provided test data for the combustion chamber diffusion system, observing flow separation and total pressure loss after passing through the pre-diffuser with a high mean angle. Hestermann *et al*. [7] and Honami *et al.* [8] investigated the influence of dump gas distance and the positive effect of the flame tube head on the upstream diffuser flow. Xu et al. [9] studied the effects of pre-diffuser wall angle and length on dump diffuser performance and identified pressure loss mechanisms. Shen *et al.* [10] conducted numerical simulations of a full-scale single annular combustor diffusion system, focusing on the cowling geometry, pre-diffuser area ratio, and axial length of the dump gap. Wang *et al*. [11] investigated the influence of uniform and non-uniform flow conditions on diffuser characteristics, emphasizing the role of pressure loss caused by circulating eddy currents within the diffuser. However, these studies have not thoroughly investigated the treatment of these unfavourable eddies.

The bleeding air method is a well-established technique used in various fields, including aviation, to reduce blockages in system operations. In gas turbine engine compressors, applying the bleeding air method at a rate of 4.0% has been shown to improve

ISSN 2734-9381
https://doi.org/10.51316/jst.169.etsd.2023.33.4.8
Received: June 6, 2023; accepted: August 9, 2023

compressor efficiency near stall by 1.0% to 2.0% at different rotation speeds. Ponick *et al.* [12] investigated the impact of bleeding air near the stator shroud suction surface on a low-speed compressor, using simulations and experiments to analyze losses and pressure rise. They found that the static pressure ratio improved with increasing bleeding air ratio. Peltier *et al.* [13] conducted experiments to study the effects of a bleeding air system behind the stator on the shroud surface. They found that the static pressure ratio improved with increasing bleeding air ratio, and they also performed simulations with bleeding ratios ranging from 4% to 14% on the total pressure ratio of an axial compressor, showing an increase in the total pressure ratio with higher bleeding ratios. Zhao *et al.* [14] examined the influence of bleeding air on a transonic single-stage compressor and found that increasing the bleeding rate led to improvements in total pressure ratio and efficiency. Grimsha *et al.* [15] studied the impact of bleeding airflow on the stability of a low-speed single-stage axial compressor and observed slight improvements in total pressure and stall margin with increasing bleeding rate. The bleeding air method eliminates turbulent flows that can be detrimental to the system, making it a viable option for application in the diffuser. There has been limited research conducted on the design of diffusers with higher pre-diffuser wall angles and their impact on flow separation. Increasing the pre-diffuser wall angle can effectively reduce the length of the pre-diffuser while minimizing total pressure loss. However, when the pre-diffuser wall angle is increased beyond a certain threshold, flow separation becomes apparent on the pre-diffuser wall, resulting in an increase in total pressure loss. Vortex generators (VGs) are small, inclined vanes that are used to energize "sluggish" boundary layers. This means that they can delay the onset of flow separation and increase the maximum lift coefficient. VGs are typically designed to be about half the height of the local boundary layer thickness [16]. VGs have certain drawbacks, such as the potential to increase minimum drag and reduce the maximum lift-to-drag ratio. However, they typically lead to an increase in the maximum lift coefficient and delay the occurrence of stall at higher angles of attack. In a study by Bragg and Gregorek, VGs were evaluated and found to improve the degraded performance caused by a trip strip back to the "clean section" level [17]. A trip strip is a device that is used to simulate contamination in a flow field. Wheeler-type vortex generators have shown less drag penalty compared to typical vortex generators for similar levels of flow control. This means that they can be more effective at improving diffuser performance without significantly increasing drag. Although Wheeler-type vortex generators have shown less drag penalty compared to typical vortex generators for similar levels of flow control, their design is slightly more complex. However, it is evident that vortex generators (VGs) and bleeding air have positive effects on aerodynamic phenomena similar to the issues faced by the dump diffuser. Therefore, this study will investigate the effects of VGs and bleeding air on the performance of dump diffusers without the need to change its length and overall size. The study will use a 3D model to discuss the aerodynamic efficiency of the diffuser and propose an optimal solution for the diffuser.

2. Geometry Description

A typical simplified shape of the annular rendering diffuser combustion chamber is adopted. Fig. 1 illustrates the geometry under consideration and its geometrical parameters. Table 1 describes the values of geometrical parameters of the basic 3D model. This 3D model was developed based on the 2D model proposed by Wang *et al* [11]. The selection of the computational domain width was determined based on a similar experimental model at "Fluid Mechanics and PIV Laboratory (FMPL)" of Xiamen University [16]. The ratio of gap width D/h_2 in this study is 1. When the phenomenon of vortex circulation occurs at the corners of the inner and outer walls, they are equal. In addition, an expansion angle of 6 degrees is chosen for both the upper and lower walls, where the early separation phenomenon begins to occur.

Table 1. The parameters of the 3D dump diffuser models.

Parameter	Value
Inlet height h_1, m	0.028
Width computation domain, m	0.028
Inlet inner radius $R_{1,i}$, m	0.156
Flame tube depth W	0.096
Inner annulus height $h_{4,i}$, m	0.028
Outer annulus height $h_{4,0}$, m	0.028
Pre-diffuser length	0.173
Angle $\theta_1 = \theta_o$	6^o

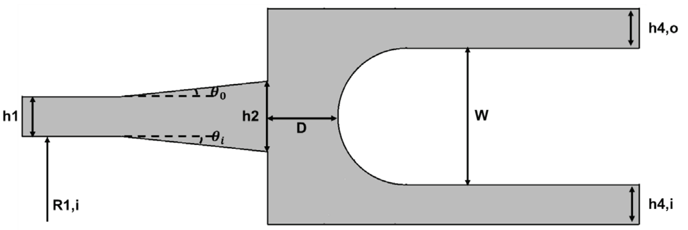

Fig. 1. Geometry of 3D basic dump diffuser

3. Numerical Method

In this paper, the effects of bleeding on the performance of dump diffuser are investigated by three-dimensional steady-state CFD simulations and three-dimensional RANS equations using the $k - \omega$ SST turbulence model, were solved for the aerodynamic analysis using the commercial software ANSYS FLUENT 19.1. The mesh quality is commonly assessed with determinant and angle. Jacobian determinant has been calculated on six sides and then standardized the matrix of determinant to characterize the deformation of the unit. The value 1 indicates an ideal hexahedron cube, while 0 indicates the transcube with negative volume, which is represented on the *x*-axis from 0 to 1. The solution is to use the C-type grid and wall maps. To improve the mesh quality, O-grid is used to build the grid, which effectively improved the mesh quality and controlled the boundary wall of prediffuser. All of three models have the mesh quality indicator ratehigher than 0.9. The growth rate of element size is set to 1.2 to ensure gradual expansion of adjacent cells. The boundary layer near the wall is fixed at 10e-6 m to fit the SST model, where the value of *y*+ on the wall is less than 1 (maximum *y*+ is 0.93). It is demonstrated at Fig. 2.

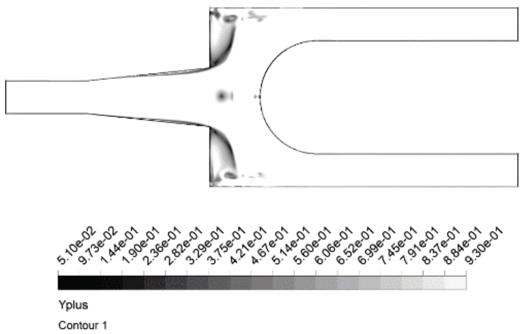

Fig. 2. Contour y plus of numerical model

4. Boundary Condition

The simulations are performed using the total temperature and total pressure input boundary conditions. At the outlet boundary, atmospheric pressure is applied. The inner and outer annulus wall are set with an adiabatic no-slip wall. These conditions satisfy the experimental conditions in a typical low-speed aerodynamic tube. The boundary conditions used in this study are the same as the boundary of 2D model in the previous study. The velocity inlet is 95 m/s. These conditions used in the current study can be easily replicated and validated in a typical laboratory-sized wind tunnel.

For the model using the bleeding air method, the bleedingpipes are integrated to extract gas flow ranging from 1% to 20% in increments of 1% of the inlet diffuser mass flow inlet.

5. Result and Discussion

In the present work, the aerodynamics performance of the annular dump diffuser combustor is of interest. Two parameters are recorded and compared, which is the static pressure recovery coefficient C_p and the total pressure loss coefficient λ. The formula for the calculation of static pressure recovery coefficient Cp is defined as follows:

$$C_p = \frac{m_{outlet} \times \bar{P}_{outlet} - m_{inlet} \times \bar{P}_{inlet}}{\frac{1}{2} \times m_{inlet} \times V_{inlet}^2} \quad (1)$$

The formula for the calculation of total pressure loss coefficient λ is defined as in following:

$$\lambda = \frac{m_{inlet} \times \overline{Po}_{inlet} - m_{outlet} \times \overline{Po}_{outlet}}{\frac{1}{2} \times m_{inlet} \times V_{inlet}^2} \quad (2)$$

The performance of the diffuser is indicated by the value of efficiency coefficient η. The lower the total pressure loss and the higher the static pressure recovery, the better the diffuser performance. Therefore, the goal is to find the optimal design point where the value of the η is maximized:

$$\eta = \frac{C_p}{\lambda} \quad (3)$$

5.1. Validation Results

In this research, four different meshes were investigated to find the optimal mesh. The number of nodes for Mesh 1, Mesh 2, Mesh 3, Mesh 4 are 1M, 2.3M, 4.3M, 5.6M, 6.7M elements, respectively. Fig. 3 illustrates the convergence of the total pressure loss and static pressure recovery for four criteria. Moreover, after Mesh 3 (4.3 million elements), the result of static pressure recovery is approximately the same (almost constant), which demonstrates that Mesh 3 is selected as the optimal mesh for the simulation.

Fig. 4 and Fig. 5 depict the aerodynamic characteristics of the diffuser, and a qualitative assessment reveals that the observed phenomena are entirely reasonable and consistent with reality. As high-speed airflow exits the compressor and enters the diffuser, it undergoes deceleration and pressure preservation to maintain stable combustion in the combustion chamber. Thus, at the diffuser throat (indicated by the black dot in the figures), the velocity is reduced while the pressure is higher compared to the surrounding area. Two large-scale recirculation vortices exist at the corners, which result from the sudden geometric expansion causing the airflow to rotate clockwise and generate local vortices, leading to total pressure losses. Additionally, flow separation is detected at the expansion section of the diffuser when the expansion angle reaches a certain value. Early flow separation also increases the size of the recirculation region and reduces the efficiency of the diffuser. To address these issues, two methods are proposed for investigation to improve the diffuser's performance.

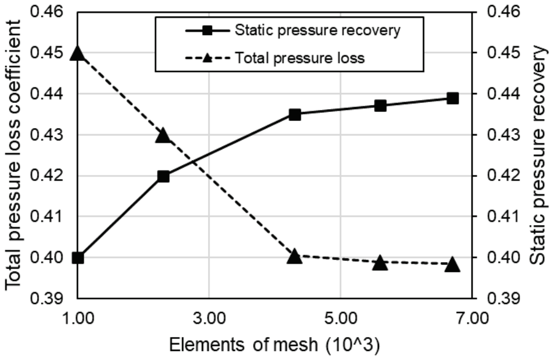

Fig. 3. Mesh sensitivity study results of four mesh levels

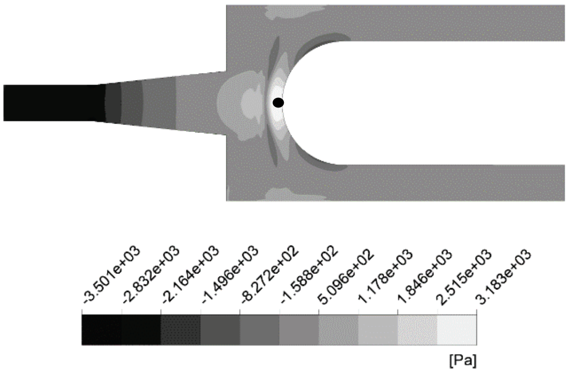

Fig. 4. Pressure contour of dump diffuser

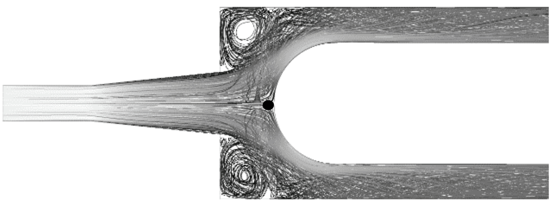

Fig. 5. Streamlines of dump diffuser

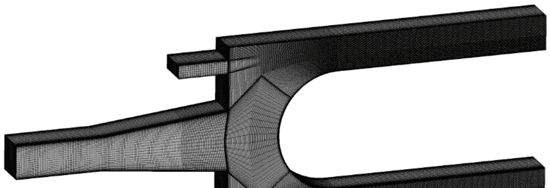

Fig. 6. Mesh of diffuser with bleeding tube

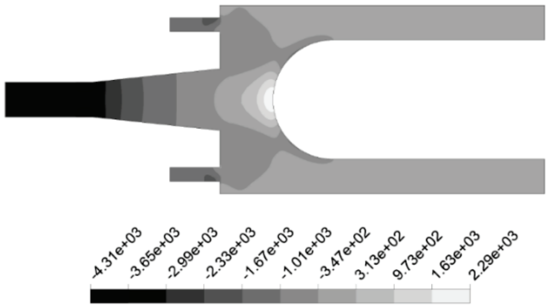

Fig. 7. Pressure contour of model with bleeding air method (10% mass flow rate) at middle plane.

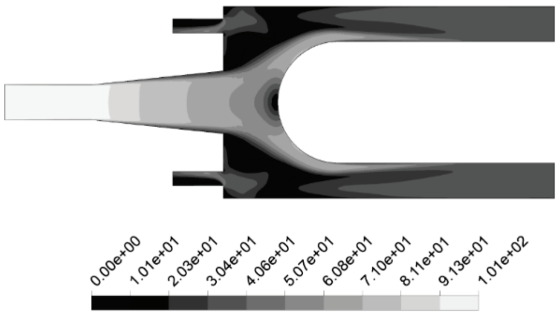

Fig. 8. Velocity contour of the model with bleeding air method (10% mass flow rate) at middel plane

5.2. Bleeding Air Method

Fig. 6 illustrates the grid model of the dump diffuser using the bleeding air method. The grid generation method is the same as the basic model. The bleeding air is set at mass flow rates ranging from 1% to 20% of mass flow inlet. The results in Fig. 7 show that the pressure gradually increases and reaches a maximum at the diffuser point. The area of low-pressure region at the two corners decreases due to the use of bleeding air tubes.

The results indicate that bleeding air from the outer annulus path effectively reduces the velocity in the main area and promotes a more uniform velocity distribution. As the bleeding air increases, the pressure decreases in the outer dump region while the inner dump region moves upward, resulting in a higher proportion of air flowing through the outer annulus path in relation to the total airflow. These results demonstrate that bleeding air from the dump region enhances the flow field distribution, improves velocity uniformity near the flame tube, and reduces total pressure loss. The total pressure loss coefficient tends to decrease and the static pressure recovery coefficient tends to escalate simultaneously with increasing air bleeding. Since the air bleeding tubes are in the circulation area, the flow is partially withdrawn to prevent vortex circulation, reducing the vortex in the circulation area and expanding the mainstream area (Fig. 8 and Fig. 9). When the bleeding percentage is 10%, the diffuser has the best performance: the total pressure recovery coefficient decreases 30.06% and the static pressure recovery coefficient increases 36.32%. (Fig. 10). The performance efficiency of the diffuser is shown in Fig. 11. We can see that efficiency tends to increase at first and then decrease. The maximum efficiency value is at the case of 10% gas extraction with a value of 2.12.

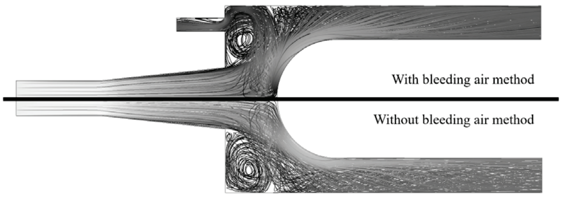

Fig. 9. Comparison of the velocity streamlines between bleeding and non-bleeding models

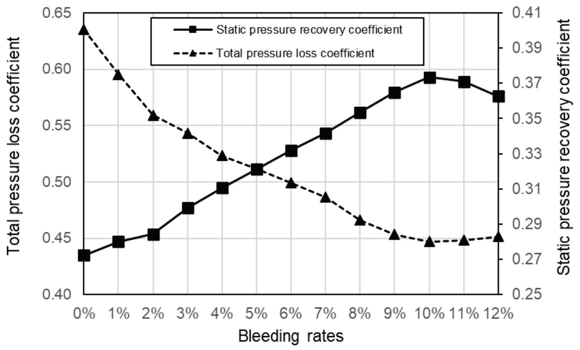

Fig. 10. Effect of bleeding rate to total pressure loss coefficient and static pressure recovery coefficient.

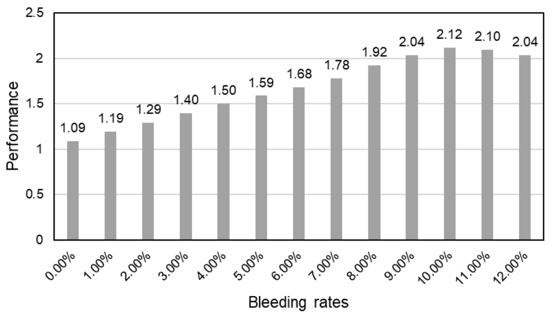

Fig. 11. Performance of dump diffuser with bleeding air method

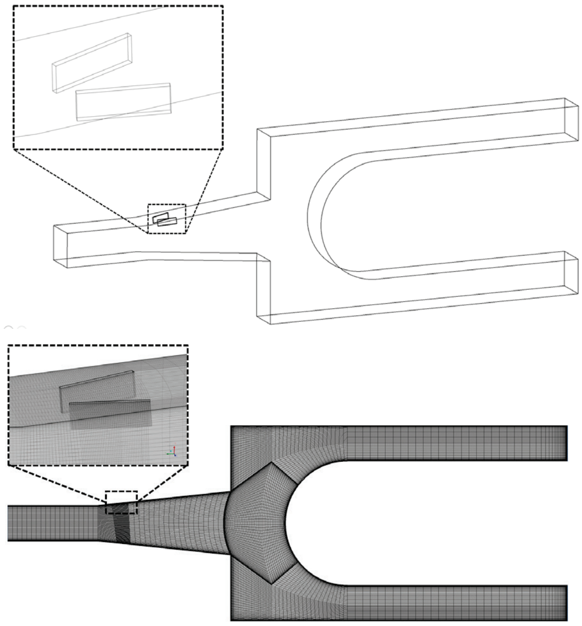

Fig. 12. Geometry and meshes of diffuser with vortex generator.

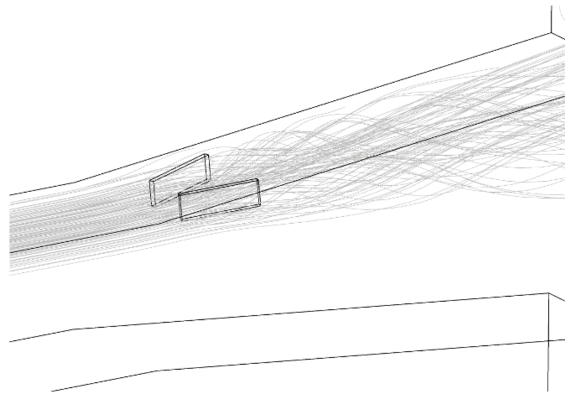

Fig. 13. Streamlines of pre-diffuser through vortex generator

5.3. Dump Diffuser with Vortex Generator

Vortex generators are integrated into the dump diffuser model for the purpose of computation. With a simple rectangular shape measuring 0.015 m x 0.001 m, two rectangular panels are placed diagonally in the expansion region of the dump diffuser inlet to generate vortices. The geometry and mesh of this model is shown in Fig. 12.

Through Fig. 13 and Fig. 14, the vortex generator creates effectively a high-pressure region near the wall, which is generated by the small vortices formed when the airflow passes through the VGs. Thanks to these small vortices, the flow separation phenomenon occurs at a slower rate. This difference can be observed at the lower wall of the pre-diffuser, where there is a thick region of low pressure compared to the thin boundary layer near the upper wall. Precisely by slowing down the flow separation process in the expansion region of the pre-diffuser the efficiency of the diffuser is significantly improved. In the Fig., with VGs the total pressure loss coefficient decreases 25.11% and the static pressure recovery coefficient increases 11.4% (Fig. 15).

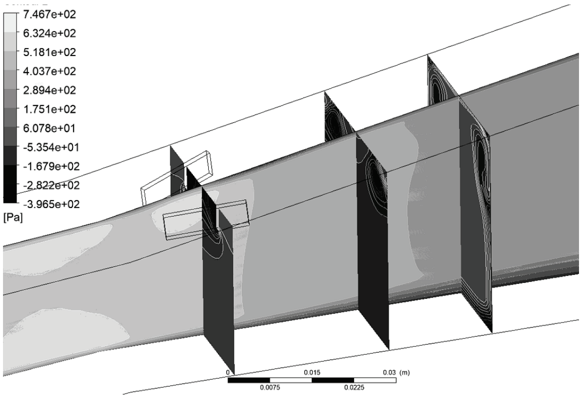

Fig. 14. Contours of pressure at middle plane and section plane of pre-diffuser

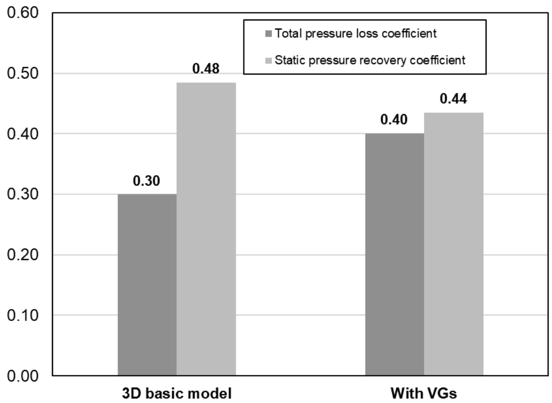

Fig. 15. Total pressure loss and Static pressure recovery of Basic dump diffuser and the diffuser with VGs.

6. Conclusion

Clearly, the diffuser plays a crucial role in the operation of an aircraft engine, consequently improving the diffuser is indispensable. The conical diffuser has two main issues. Firstly, an early separation phenomenon occurs at the diffuser entrance when the opening angle reaches a certain value. The early separation leads the flow to no longer adhering to the geometric shape of the diffuser, resulting in the expansion of the swirling region downstream and pressure losses. Secondly, in the dump diffuser, a large swirling region always appears at the right-angle region, and these swirling regions cause significant local pressure losses. In this article, a numerical study is presented a numerical study on flow in the diffuser and two methods to address these two issues in order to enhance the diffuser performance, namely flow generation and vortex creation. Air escaping from the diffuser region is beneficial to improve the diffuser performance as it helps narrow down the swirling region. With an increase in the bleeding ratio, the total pressure loss coefficient initially decreases and then increases, while the initial static pressure recovery coefficient increases and then decreases. The optimal bleeding ratio is approximately 10% of the inflow velocity, which significantly improves the performance from 1.08 to 2.12. Regarding the use of vortex generators to delay the flow separation process at the diffuser inlet, the results demonstrate that VGs are valuable aerodynamic tools capable of slowing down the flow separation process, improving the diffuser performance. Small vortices are formed and adhere to the diffuser wall surface. By slowing down this flow separation process, the swirling region is also narrowed, resulting in a significant improvement in diffuser performance, specifically a 25.11% reduction in total pressure losses and an 11.4% increase in static pressure recovery coefficient. In the future, we will investigate the geometric shapes of VGs to determine the optimal geometry for diffuser utilization.

Acknowledgments

This project has been carried out at Hanoi University of Science and Technology (HUST), and financially supported under the grant number T2021-TT-009.

Reference

[1]. V. R. S. Kumar, M. Abhijit, Y. Khan, A. Arokkiaswamy, and R. M. O. Gemson, Studies on dump diffusers for modern aircraft engines in Collection of Technical Papers - 43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference, 2007, https://doi.org/10.4028/www.scientific.net/AMM.232.246

[2]. K. C. Karki, V. L. Oechsle, and H. C. Mongia, A computational procedure for diffuser-combustor flow interaction analysis, J Eng Gas Turbine Power, vol. 114, no. 1, 1992, https://doi.org/10.1115/1.2906301

[3]. S. Honami, W. Tsuboi, and T. Shizawa, Effect of the flame dome depth and improvement of the pressure loss in the dump diffuser, in Proceedings of the ASME Turbo Expo, 1998, https://doi.org/10.1115/98-GT-225.

[4]. J. F. Carrotte, D. W. Bailey, and C. W. Frodsham, Detailed measurements on a modern combustor dump diffuser system, J Eng Gas Turbine Power, vol. 117, no. 4, 1995, https://doi.org/10.1115/1.2815453.

[5]. L. L. Xu, C. Ruan, X. Y. Fang, F. Xing, and C. L. Zhao, Effects of pre-diffuser on performance of dump diffuser, Tuijin Jishu/Journal of Propulsion Technology, vol. 36, no. 7, 2015, https://doi.org/10.13675/j.cnki.tjjs.2015.07.013.

[6]. J. F. Garrotte, P. A. Denman, A. P. Wray, and P. Fry, Detailed performance comparison of a dump and short faired combustor diffuser system, J Eng Gas Turbine Power, vol. 116, no. 3, 1994, https://doi.org/10.1115/1.2906850.

[7]. R. Hestermann, S. Kim, A. Ben Khaled, and S. Wittig, Flow field and performance characteristics of combustor diffusers: A basic study, J Eng Gas Turbine Power, vol. 117, no. 4, 1995, https://doi.org/10.1115/1.2815454.

[8]. S. Honami, T. Shizawa, A. Sato, and H. Ogata, Flow behavior with an oscillating motion of the impinging jet in a dump diffuser combustor, J. Eng Gas Turbine Power, vol. 118, no. 1, 1996, https://doi.org/10.1115/1.2816551.

[9]. P. Ghose, A. Datta, and A. Mukhopadhyay, Effect of prediffuser angle on the static pressure recovery in flow through casing-liner annulus of a gas turbine combustor at various swirl levels, J Therm Sci Eng Appl, vol. 8, no. 1, 2016, https://doi.org/10.1115/1.4030734.

[10]. S. Shen, M. Shang, P. He, and M. Ronghai, The effects of cowling geometry, area ratio and dump gap on a combustor diffusion system, in Proceedings of the ASME Turbo Expo, 2014. https://doi.org/10.1115/GT2014-26652.

[11]. H. Wang and K. H. Luo, Numerical investigation of dump diffuser combustor performance at uniform and non-uniform inlet conditions, in Proceedings of the ASME Turbo Expo, 2020, https://doi.org/10.1115/GT2020-15982.

[12]. S. Pönick, D. Kozulović, R. Radespiel, B. Becker, and V. Gümmer, Numerical and experimental investigations of a compressor cascade flow with secondary air removal, J. Turbomach, vol. 135, no. 2, 2012, https://doi.org/ 10.1115/1.4006570.

[13]. V. Peltier, K. Dullenkopf, and H. J. Bauer, Experimental investigation of the performance of different bleed air system designs, in Proceedings of the ASME Turbo Expo, 2012, https://doi.org/10.1115/GT2012-68242.

[14]. B. Zhao, S. Li, Q. Li, and S. Zhou, The impact of bleeding on compressor stator corner separation, in ASME-JSME-KSME 2011 Joint Fluids Engineering Conference, AJK 2011, 2011, https://doi.org/10.1115/AJK2011-22001.

[15]. S. D. Grimshaw, G. Pullan, and T. Walker, Bleed-induced distortion in axial compressors, J. Turbomach, vol. 137, no. 10, 2015, https://doi.org/10.1115/1.4030809.

[16]. F. Xing, H. Su, S. Chan, L. Xu, and X. Yu, Optimization study of the dump diffuser in gas turbine to reduce pressure loss, International Journal of Aerospace Engineering, vol. 2018, 2018, https://doi.org/ 10.1155/2018/6070782.

[17]. M. B. Bragg and G. M. Gregorek, Experimental study of airfoil performance with vortex generators, J Aircr, vol. 24, no. 5, 1987, https://doi.org/10.2514/3.45445.