NUMERICAL METHOD FOR DESIGNING AND MODELING AN EXHAUST CLUSTER FOR A SMALL TURBOJET ENGINE TEST CELL

Truong-Giang Nguyen[1, 2], Quoc-Huy Nghiem[1] and Phi-Minh Nguyen[1]

[1]Viettel Aerospace Institute, Viettel Group, Hanoi, Vietnam
[2]Correspoding author, Giangnt84@viettel.com.vn

Abstract

This paper describes the process of designing, modeling, and simulating a jet engine test cell's exhaust cluster including an augmentor tube, a blast basket, a deflector cone, and a square-cross-section exhaust stack. This system is intended for testing a small and uncooled turbojet engine at sea level condition with exhaust gas temperature (EGT) of around 700°C. The studying method used in this research is Computational Fluid Dynamics (CFD). The design target is to dilute engine's hot exhaust gas by induced air via an augmentor tube to the extent that allows installation of supervising camera at the nose of a deflector cone i.e., 150 ºC. After that, a deflector cone and a blast basket are used to divert and decelerate direction of now mixed and cooled exhaust gas into an exhaust stack. Blast baskets have been widely used in engine test cell; however, there has been limited information on flowfield emanated from this device. Based on CFD analysis, it has been detected that in the traditional case where perforations meet a deflector's surface, high velocity and high temperature regions leaned unsymmetrically against one end wall. By sliding a basket axially inside an exhaust stack while keeping a deflector cone on a rearward wall, thus creating dump region between a deflector cone and a blast basket, it is attempted to promotes different mixing schemes between high and low velocity jets inside an exhaust stack.

Keywords: jet engine test cell, augmentor tube, blast basket, deflector cone, exhaust stack

1. Introduction

Augmentor tubes are used in gas turbine engine test cell to mix hot exhaust jet flow from an engine with induced ambient air to decrease temperature. A blast basket is then inserted at the end of the augmentor tube and connected to an exhaust stack to spread cooled flow out by aid of a deflector cone. Flow is quickly slowed down by escaping through vast porous surface area of a blast basket. This type of augmentor system was originally used in high bypass-ratio turbofan testing and proved successful [1].

This study explores the implementation of the exhaust system with integrated blast bucket for testing small turbojet engines, taking advantage of its good mixing and flow redirecting capabilities. The exhaut gas undergoes the process of cooling jet stream's high EGT, decelerating it, and redirecting it out into an exhaust stack. The primary goal is to lower the exhaust gas's temperature sufficiently to avoid overheating an augmentor tube surface and a deflector cone surface, where a camera will be installed.

The concept of using a blast basket it due to its simple construction, low cost fabrication and easy installation as well as maintenance.

In addition, the air flow should escape an exhaust stack with as uniform as possible velocity and temperature profile on an exit plane. The reason is that for the same overall mass flow rate, nonuniform flow with higher peak velocity and higher peak temperature will produce higher noise level and possible damage to brick wall's structure [6].

The use of blast basket has been applied widely with satisfaction in case of high bypass turbofan engine where low speed - cold flow dominates in articles [1], [2] and [9]. However, in case of small jet engine where high speed – hot flow dominates [6], the blast basket still works but with less success and aerodynamic modification has been made. Analyzing [6], the core flow was not well mixed with secondary flow to fill cylindrically most area of the augmentor tube. In addition, in [6], the blast basket's length-to-diameter ratio is around 1, which is quite short compared to designs mentioned above where that ratio of the blast basket is around 2 [12].

Furthermore, in [11] and [8], downstream of a blast basket, a deflector having conical shape and specifically having 25° half-conical angle was pointed out to offer less noise and less velocity distortion in an exhaust stack's exit plane. Velocity distortion is defined in [10] as follows:

$$FC_{dist} = \frac{V_{max} - V_{min}}{V_{avg}}$$

Considering all experience collected abouve, this led to design concept for an exhaust cluster of a small turbojet engine test bench, illustrated in Fig. 1. the augmentor tube is defined by its diameter and length. These parameters mainly influence mixing and cooling capacity. The blast bucket has the same diameter as the augmentor tube and has length of two times of its diameter. The deflector cone turns flow 90° to radial direction.

Due to abrupt turn, flow leaves the blast basket in a highly distorted manner [3], [8], [6]. This makes outflow be blowing unsymmetrically against 4 walls of the exhaust stack. And then, it was found that by moving the blast basket relative to the deflector cone, mixing between high and low velocity jets inside an exhaust stack could be changed to make the outflow less distorted and less noisy. This method was chosen because of economic reason where it does not change overall design sizing.

Traditionally, the blast basket's perforation ends when it meets the deflector cone, so there is no empty gap between these two devices. In this study, the empty gap between the blast basket and the cone after axial movement is called adump region and it is shown in Fig. 2 below. This design creates a recirculation zone at the end of the deflector cone, which keeps the gas stream away from the wall. The effectiveness of this design has been demonstrated by CFD calculation results. Parametric study was conducted to find out the appropirate dump size in this specific exhaust cluster geometry.

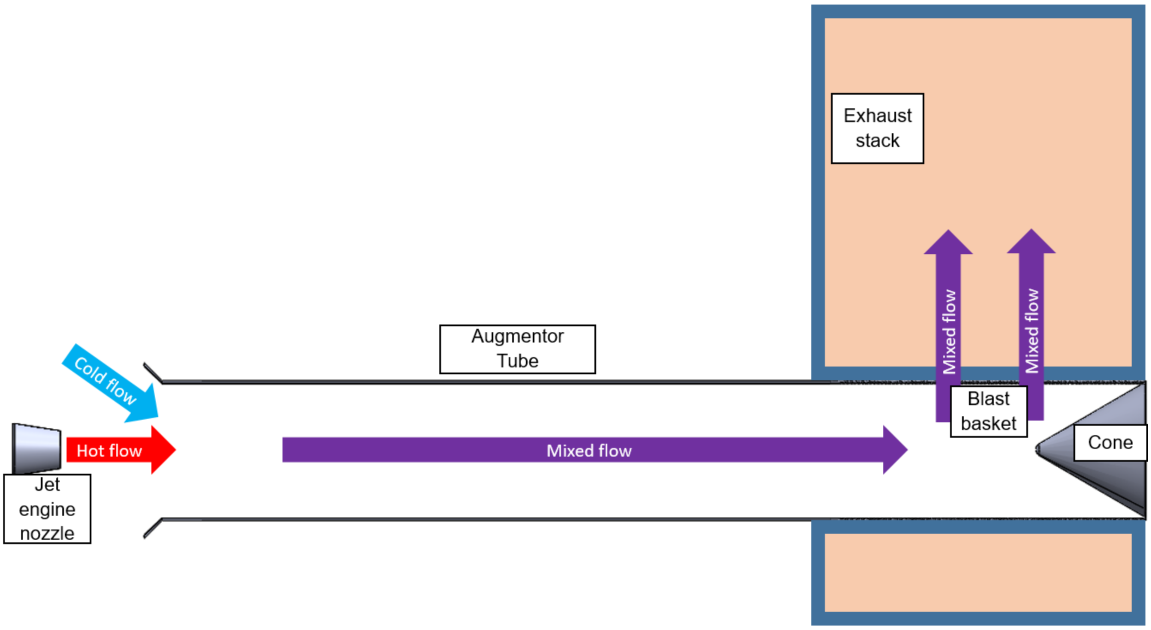

Figure 1: An exhaust cluster with a blast bucket

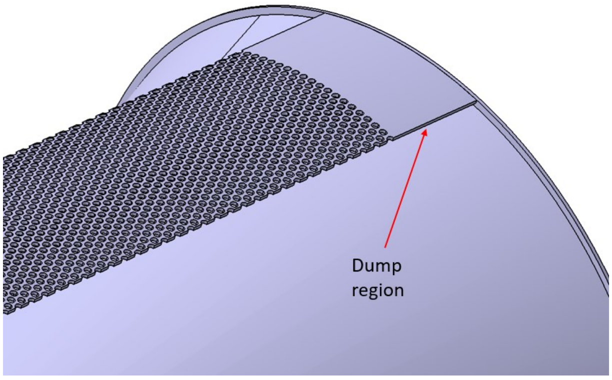

Figure 2: Dump region

2. Design methodology

The exhaust cluster was design for the following 03 purposes: pumping out hot exhaust gas to the outside environment without reverse flow or hot gas re-entrainment, structures are durable for long terms use and allowing installation of supervising camera from backward view being concentric to engine nozzle.

The criteria above all lead to demand of slowing down a high speed exhaust gas and diluting a hot gas with cold flow. Cold flow is essentially environmental air, induced by high speed exhaust gas to an augmentor tube.

Tube diameter determines the amount of secondary flow and tube length determines level of mixing. A starting point was based on guidelines in [7] for expendable engine's test cell where:

+ Ratio of augmentor tube diameter to nozzle diameter at least 4 times
+ Ratio of augmentor tube's length-to-diameter at least 4 times

The turbojet nozzle geometry is a simple convergent type having diameter 220 mm, flow is choked with pressure inlet of 2.35 bar, inlet temperature is approximately 720 ^{o}C.

The assessment criterion is that temperature just in front of a deflector cone must be below 150 ^{o}C. After several iterations, the augmentor tube was chosen to be of 1.0 m diameter and 5.0 m length. So, the blast basket diameter consequently is of 1.0 m diameter is 2.0 m diameter length.

The blast basket is low cost made from standard punched steel 5mm thickness plate having perforations of 10 mm diameter. Openess ratio is of about 54%. Total perforation open area is approximately 4.5 times of tube's cylindrical area. That helps to explain flow's quick deceleration across a basket surface.

Due to construction reason, the internal dimension of the exhaust stack is slightly longer than a blast basket's length. Therefore, there appears the idea of sliding the blast basket relative to the deflector cone to see how different mixing scheme inside the exhaust stack could be to choose the most effective case. The 03 cases are no dump length, dump length being 15% basket diameter and dump length being 30% basket diameter.

3. Numerical method

The system's performance is estimated using the CFD technique. Many studies have been undertaken and reported in [4], [5], [6e], and [7]. It is worth noting that the computational model comprises multiple tiny hole surfaces, creates large numbers of mesh elements, and increases the workload for the calculation operation. Thus, a one-half section 3D model is created, with the phenomena assumed to be symmetric. Symmetry assumption has been confirmed in [8].

The model features a square-section exhaust stack with a total height of 7 m and a depth of 2.5 m measured from the center line of the augmentor tube. It generates the ground effect, which has a substantial impact on the uniform distribution of air flow upwards and cannot be ignored in practice.

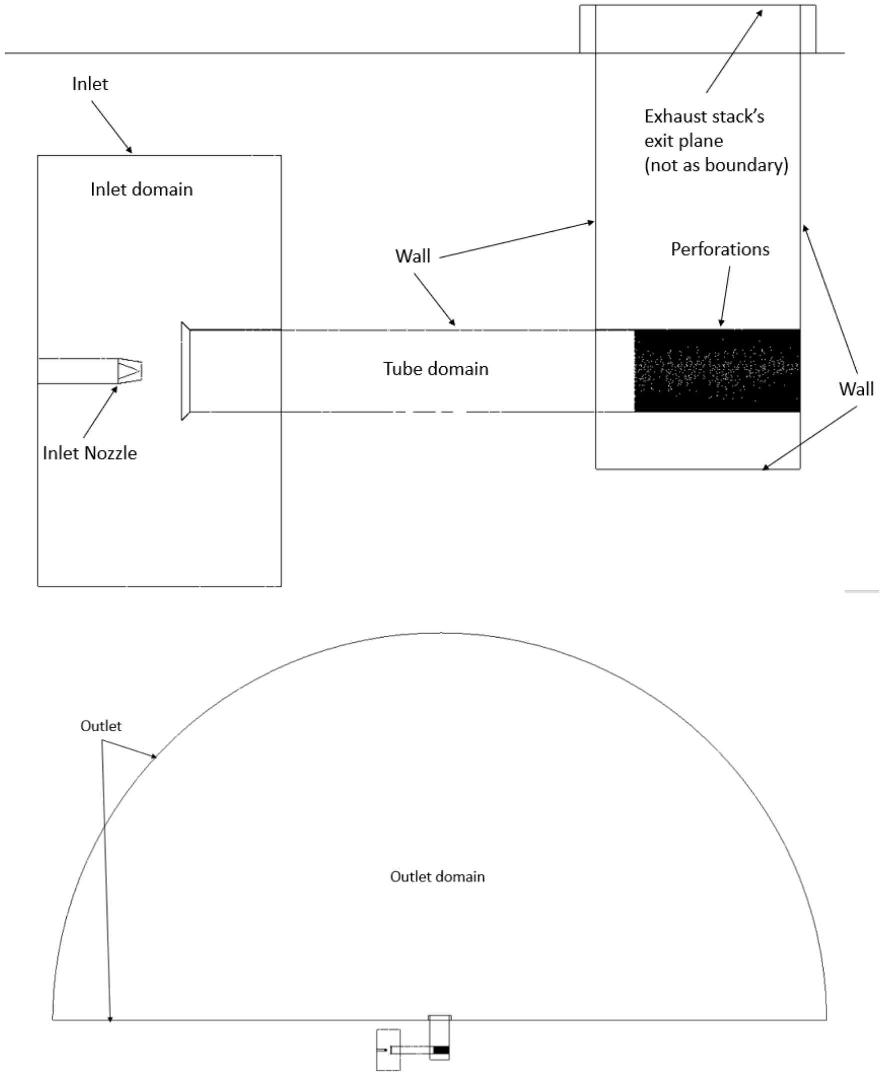

Figure 3: CFD domains

Fig. 3 shows all the surfaces as boundary condition (BC). The exhaust gas enters the nozzle through the nozzle's inlet which is set as pressure inlet BC. The intake domain represents test cell's indoor environment, and a large half-hemisphere represents the output domain. The inlet and outlet's outermost surfaces are considered to be at ambient pressure. Because of the high velocity exit from the exhaust stack, the outlet domain has a radius equal to ten times the stack's dimensions (50 m radius). All walls are assumed as non-slip and adiabatic. The values of BCs are shown in the table below.

Table 1: Boundary condition

Total pressure inlet at engine nozzle's inlet (bar)	2.35
Total temperature inlet at engine nozzle's inlet (°C)	723
Ambient pressure (atm)	1
Ambient temperature (°C)	27

Surveying published papers, there are 02 methods of setting pressure outlet BC. Authors in [3] and [6] set the exhaust stack's exit plane as outlet pressure boundary while the author in [8] set the external space as outlet pressure BC. It has been tried these two methods above during design process. Interestingly, results agree with the authors above but contracdicting each other. For those setting exhaust stack's exit plane as outlet pressure BC, it was seen high speed region lean against rearward walls. For those setting external space as outlet pressure BC, it was seen high speed region lean against forward walls. Since numerical results in [8] were backed by experimental validation, the setting external space as outlet pressure BC was chosen.

The domain was meshed for achieving the poly-hexa core type. This form of mesh can lower the number of elements while still managing a small or complicated surface. The jet stream expands and slows down in the domains between the nozzle and the deflector cone, as well as abruptly changes direction after the bucket. Thus, two bodies of influence (BOI) are placed here to mesh these domains equally in size, as illustrated in Fig. 4. The element size of the BOI within the augmentor tube is 20 mm, whereas the BOI inside the exhaust stack is 40 mm. The blast basket has a 1 mm hole and a 5 mm cylinder surface. The nozzle element ranges from 1 to 5 mm. The other surface is meshed with elements ranging in size from 1 to 100 mm. The growth rate is 1.1 over all surfaces. With this meshing process, the total number of elements is around 30 million. The quality is maintained by keeping the degree of skew below 0.95 and the orthogonal value over 0.05.

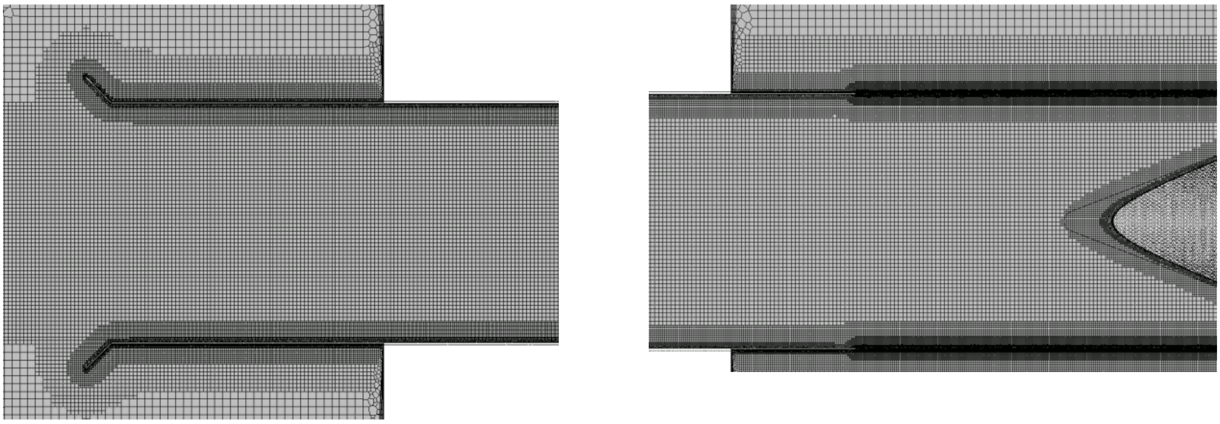

Figure 4: Meshing on internal tube space

Due to the large number of cells, HPC was employed for simulation. The 3D Navier-Stoke problem is addressed using the commercial ANSYS FLUENT program. The k-epsilon, realizable, and scalable wall function is used to close the equations and manage the near wall boundaries. This method has been shown to be quick and feasible during simulation in different research cases. The fluid is chosen as an ideal gas.

4. Results and discussion

4.1 General design result

Analyzing CFD results, it is pointed out that moving the blast basket relative to the deflector cone alters flowfield downstream of the blast basket only. Augmentor tube design controls the flowfield upstream of the blast basket. Therefore, in all 03 cases, flowfields here are nearly the same.

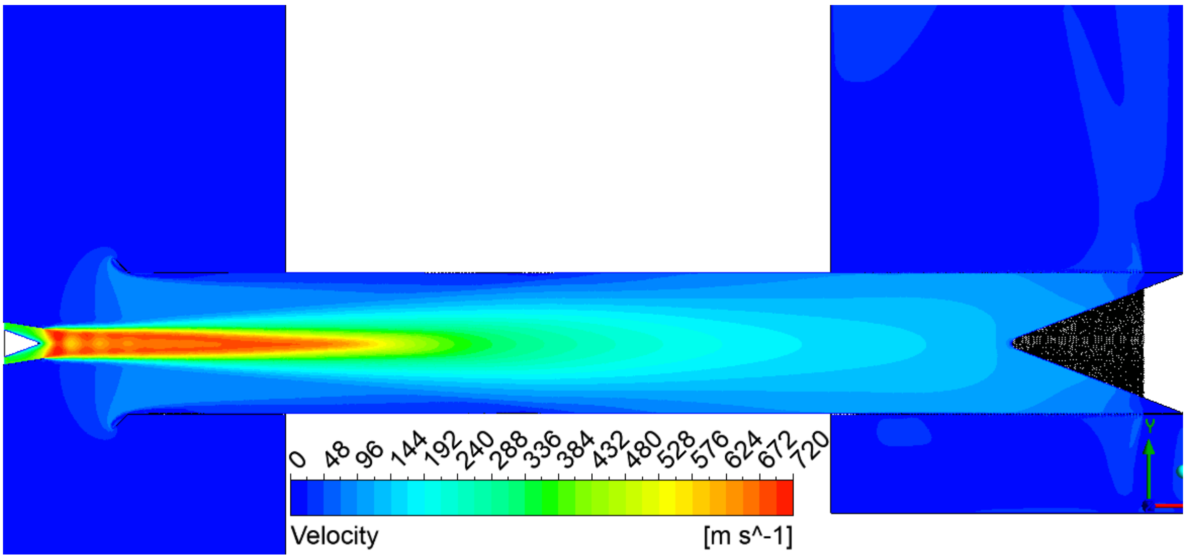

Figure 5: Velocity distribution inside the sugmentor tube and the blast basket

By comparing mass flow rate in 02 inlet BCs, the amount of induced cold flow is about 10 times that of hot flow. The augmentor length is proved to be sufficiently long by observing that the mixed flow occupies the whole cylindrical space almost entirely.
No reverse or rebound flow is occurred; this is significantly better than the original flow field in [6].

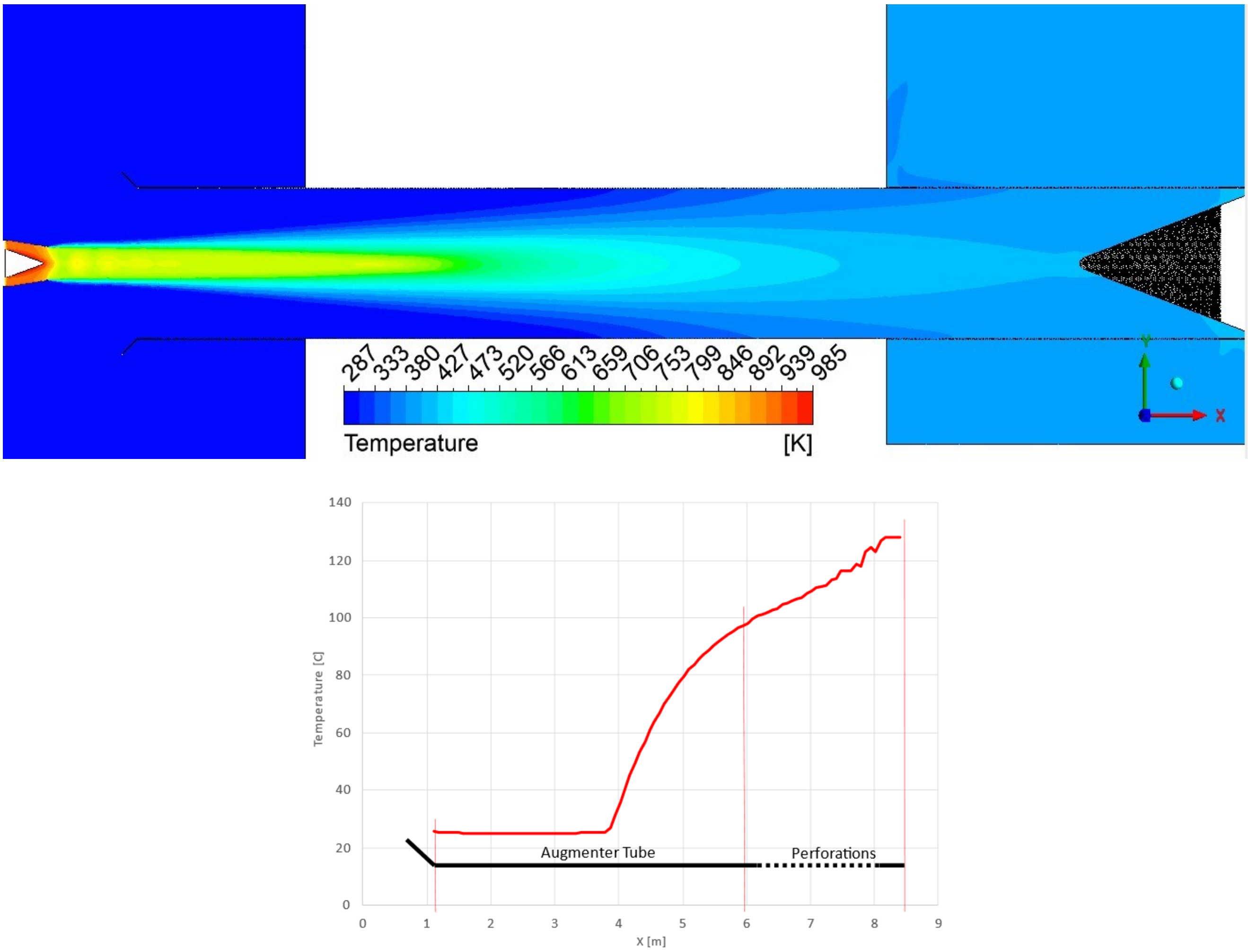

Figure 6: Temperature distribution inside the augmentor tube and the blast basket

Temperature shows that all walls are protected from hot gas. Flow field's temperature is cooled to below 150°C entirely inside the augmentor tube.
Jet escaping from the blast basket is chaotic, it also quickly looses kinetic energy thanks to small size, as illustrated in the following figures.

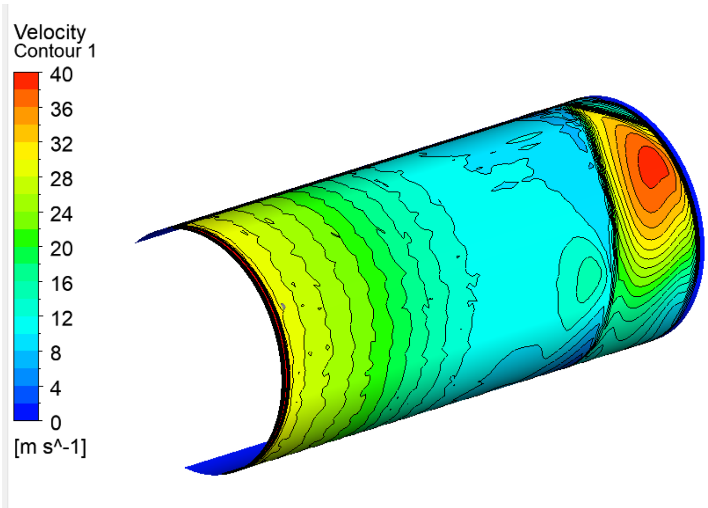

Figure 7: Flow velocity distribution outside of the blast basket

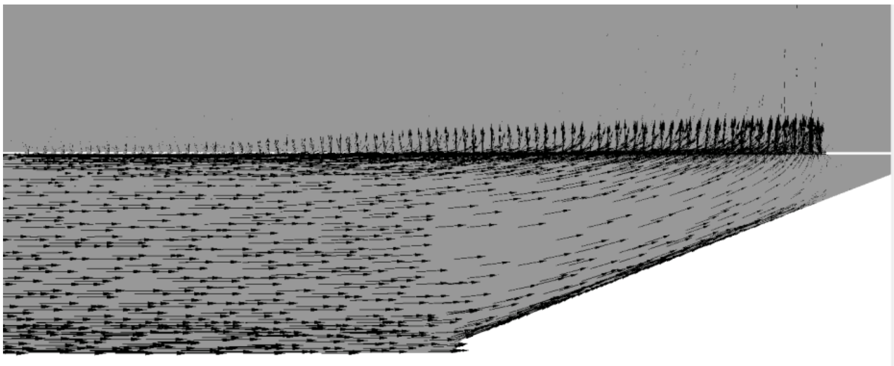

Figure 8: Flow vectors emanating from perforations

4.2 Effect of blast basket's axial position to flow uniformity on exit plane

By sliding the blast basket relatively to the deflector cone, from 15% diameter to 30% diameter, i.e., 150 mm to 300 mm, the flowfield both inside and outside the exhaust stack changes significantly.

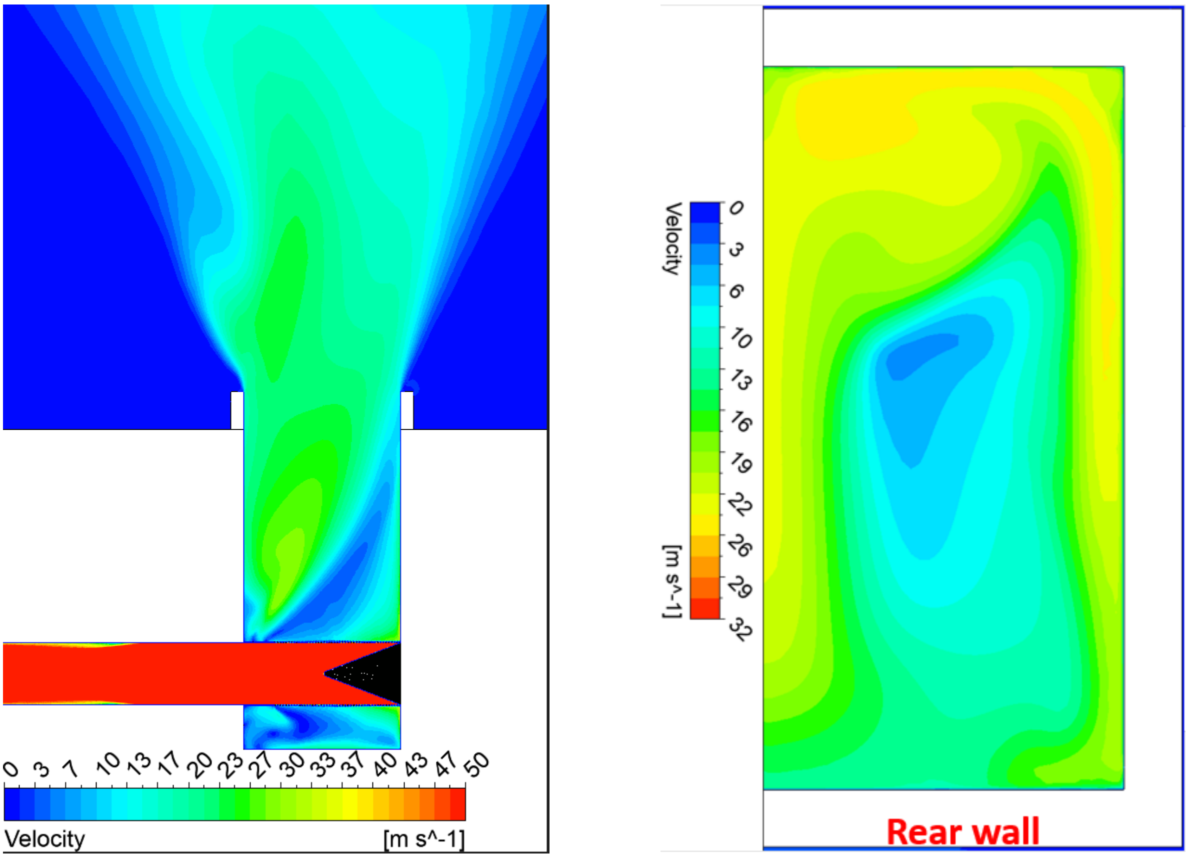

Figure 9: Flowfield with no dump gap

It could be seen that in the exhaust stack's exit plane, the no dump gap case produces lowest peak velocity, thus offers the best velocity distortion of 116%., following by 134% in case of 150 mm dump gap and 137% in case of 300 mm dump gap.
In case of no dump gap, flow tends to move forward. The dump gap creates strong oblique jet to prevent this movement. However, high velocity region drifts into side walls. For the vast majority of area, the 300 mm dump gap case offers the higher local uniformity.
Outside of the exhaust stack, flow in case of no dump gap tends to dissolve quicker due to larger expanded area.

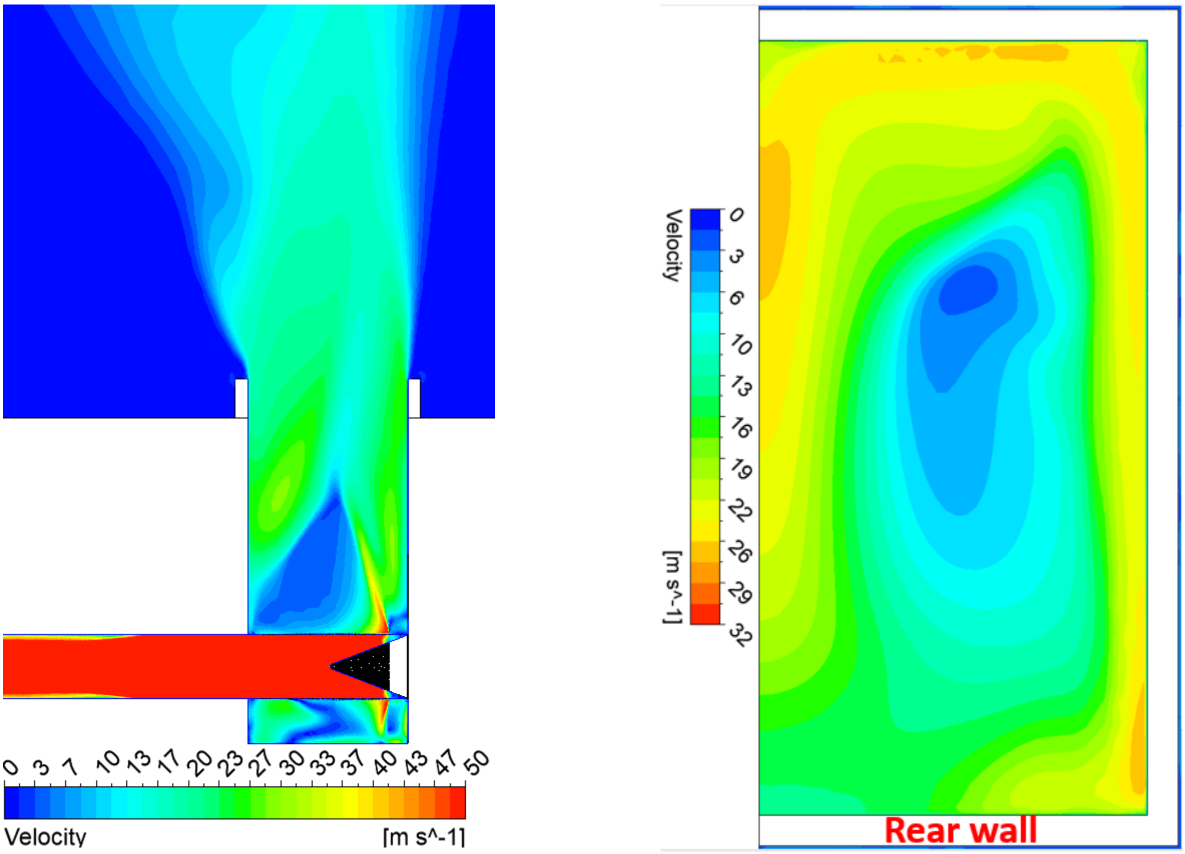

Figure 10: Flowfield with 150 mm dump gap

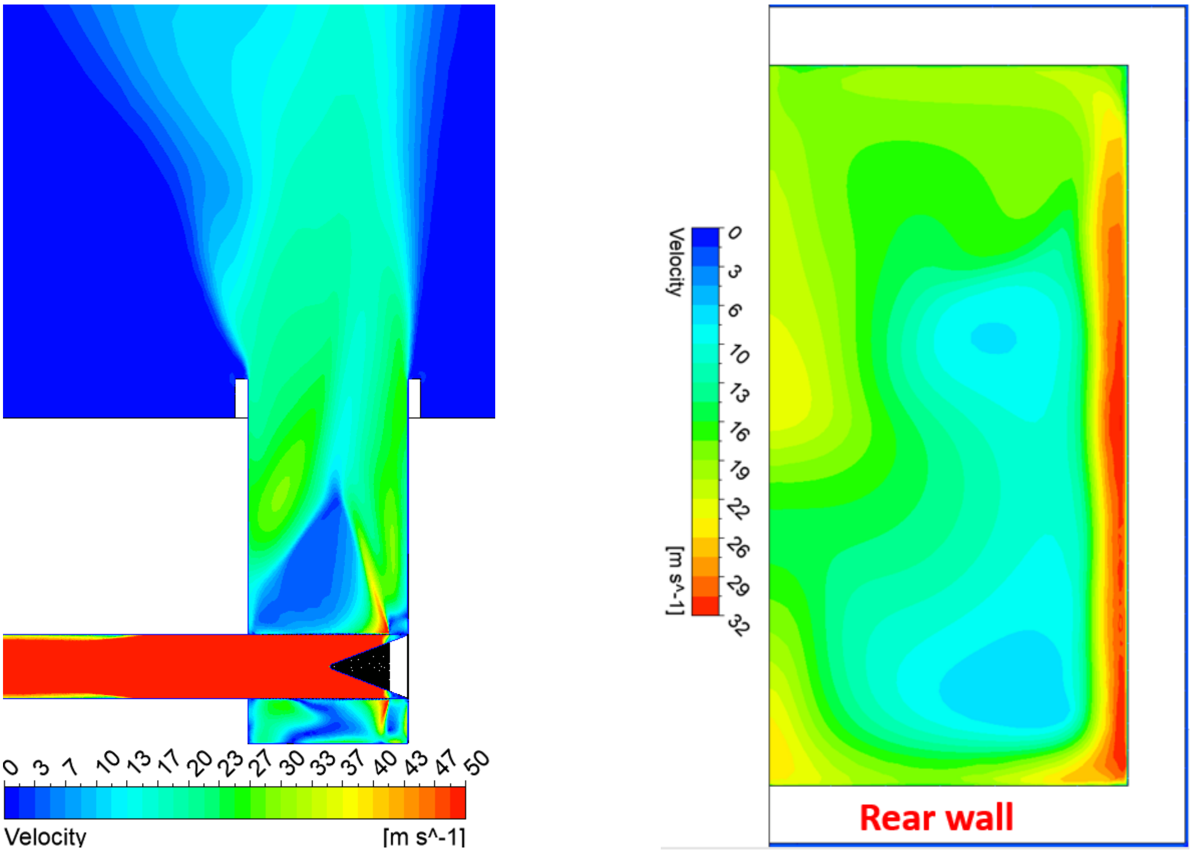

Figure 11: Flowfield with 300 mm dump gap

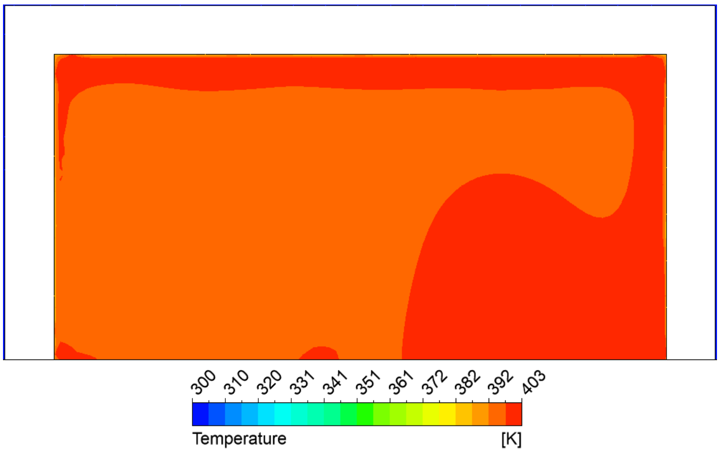

Figure 12: Temperature distribution in the exhaust stack's exit plane

Since temperature gradient was well mixed form the augmentor tube already, there is no temperature difference across exhaust stack's wall. Therefore, temperature distribution is of no concern here.

5. Conclusions

The fundamental principles and guidelines for designing an exhast cluster for small size turbojet engine test cell utilizing a blast basket concept have been discussed in this article. The augmentor tube is responsible for inducing secondary airflow to cool down and slow down an exhaust hot flow. It is found that a blast basket can work well for hot exhaust flow as long as there is sufficient amount of cooling flow via a proper designed augmentor tube. This allows installation of supervising device having working temperature below 150^{o}C.

The CFD method have been proposed facilitating analysis of flow inside an exhaust stack. Via CFD analysis, the best position for a blast basket in terms of exit flow's profile is next to a deflector cone leaning against a rearward wall.

6. Contact Author Email Address

Mailto: giangnt84@viettel.com.vn or giangkstnk55@gmail.com.

7. Copyright Statement

References

[1] Adkins, R.C. and Anas, A., 1996. *The Blast Basket: A New Component Used in Testing Large, High Bypass Ratio, Fan Engines* (Vol. 78736, p. V002T02A002). American Society of Mechanical Engineers.

[2] FREULER, R., 1993. Recent successes in modifying several existing jet engine test cellsto accommodate large, high-bypass turbofan engines. In 29th Joint Propulsion Conference and Exhibit (p. 2542).

[3] Gilmore, J.D., 2012. Computational fluid dynamics analysis of jet engine test facilities.

[4] Spuhler, T., Friedrichs, J., Städing, J., Reitz, G., AG, M.A.E. and Marx, M., 2022. CFD Modelling of a Complete Enclosed Jet Engine Test Bed and Validation with Pass-Off Data.

[5] Horoufi, A. and Boroomand, M., 2007. The effect of augmenter tube configuration on jet engine test cell performance. In *43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit* (p. 5336).

[6] Pearson, J.T., Sheoran, Y. and Schuster, B., 2015, November. CFD Analysis of a Gas Turbine Engine Test Cell to Understand and Alleviate Infrasound. In ASME International Mechanical Engineering Congress and Exposition (Vol. 57564, p. V013T16A001). American Society of Mechanical Engineers.

[7] Milatovic, B., 1994. Portable Static Test Facility for Small, Expendable, Turbojet Engines. Phase 1. *Performed by: Test Devices, Inc. Hudson, MA*.

[8] Allenstein, J.T., 2020. *An investigation of jet engine test cell exhaust stack aerodynamics and performance through scale model test studies and computational fluid dynamics results*. The Ohio State University.

[9] Desmarais, G. and Rocha, J., 2018. Discrete tones in subsonic jet engine test cells. *Progress in Aerospace Sciences*, *101*, pp.49-60.

[10] Gas Turbine Test Facilities and Equipment. “Modeling Techniques for Jet Engine Test Cell Aerodynamics." *SAE Aerospace Information Report (AIR) 4827.*

[11] Gullia, A., 2006. Thrust and flow prediction in gas turbine engine indoor sea-level test cell facilities.

[12] Desmarais, G. and Rocha, J., 2018. Discrete tones in subsonic jet engine test cells. Progress in Aerospace Sciences, 101, pp.49-60.