AERODYNAMIC DESIGN METHODOLOGY OF A NON-UNIFORM PERFORATED INVERSE CONICAL FLOW SPREADER IN INTERMITTENT BLOW DOWN WIND TUNNEL

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

[1]Viettel Aerospace Institute, Viettel Group, Hanoi, Vietnam
[2]Correspoding author, huynq80@viettel.com.vn

Abstract

This study presents the aerodynamic design method of the non-uniform perforated inverse conical flow spreader (non-uniform PICFS) used in intermittent blowdown wind tunnel. This device, which is installed inside the wind tunnel's plenum chamber, is to slow down and to spread uniformly flow coming from the inlet pipe by tearing apart it into many small jets and letting those jets coalesce. However, despite widely being described in various documents, there has been no overall design guideline so far mentioning PICFS design variables, for example, hole sizes, number of holes and perforation uniformity. This study discovers that at least for square inlet pipe shape, a non-uniformly perforated device offers superior downstream flow uniformity to that of a uniformly perforated one. Therefore, the design process has been proposed to obtain improved freestream velocity distortion at the plane finding at one plenum chamber's diameter downstream of the flow spreader.

1. Introduction

Wind tunnels are an important part of any testing facility. It is used to reduce fluctuating flow from incoming source, which is not always ideal to gain more uniform flow. One of the specific parts used in wind tunnel is the perforated inverse conical flow spreader or PICFS.

The function of (PICFS), as its name suggests, is to expand the incoming flow into a wide angle and short distance path. The flow is now forced downstream through several small holes perforated in the PICFS's wall. By going through these holes, the flow is essentially torn into several corresponding small jets. These jets coalesce to form a relatively stable and slow speed flow, i.e., without large scale flow fluctuation, which may harm a wind tunnel structure. A demonstration of a PICFS in an intermittent blowdown wind tunnel is shown in Figure 1. Detail of the function and how the system work is presented in the CFD method section below.

Without a PICFS, a slowing flow must undergo inverse pressure gradient process which have long narrow angle diffuser channel(s) yet still susceptible to upstream flow's fluctuation source such as flow control valve's movement. The biggest advantage of PICFS over other flow decelerating devices is its stability over upstream flow's fluctuation by self-damping characteristics of these jet's coalescence.

PICFS has been widely incorporated successfully in many ground-based testing facilities such as Air Force Aero Propulsion Laboratory [1], Aerospace Research Laboratory [2], Florida Center for Advanced Aero-Propulsion [3]. However, the details shown are insufficient to re-create similar function design.

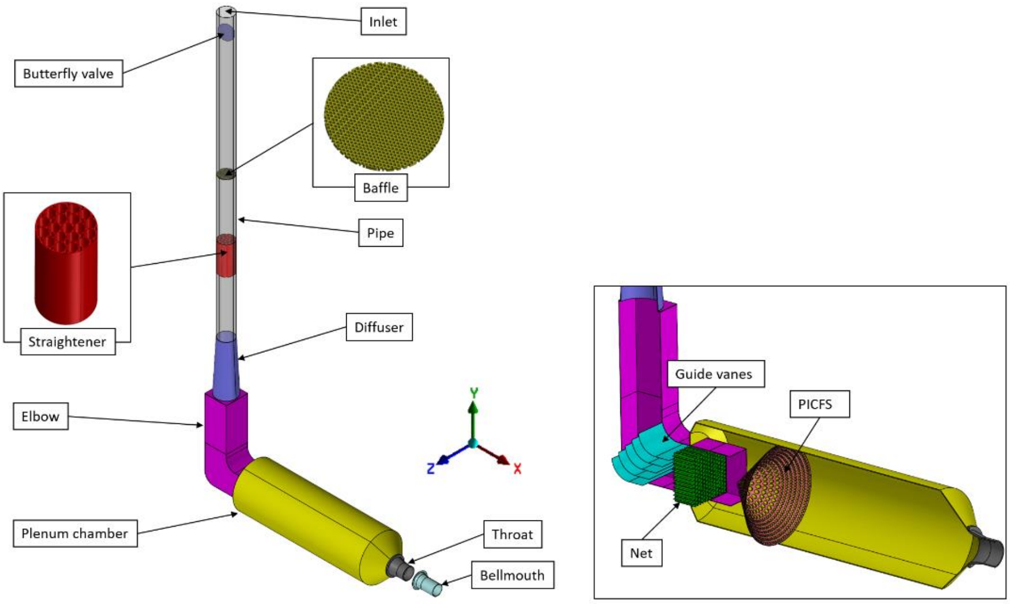

Figure 1. Plenum chamber in blowdown wind tunnel and PICFS position.

Two important academic books firstly proposing PICFS are the one by Gollin et al. [4] and the one by Ferri et al. [5]. As pointed out firstly in the books of Gollin et al. [4] and Ferri et al. [5], the PICFS mechanism is used for the following three purposes:

Firstly, it stably decelerates a high-speed incoming flow to low speed in a plenum chamber, i.e., no macro flow fluctuation in both upstream and downstream of a PICFS regardless of incoming condition such as shock pressure due to upstream control valve's movement.

Secondly, it deceleraté uniformly that flow to certain level that enables installation of aerodynamic devices such as honeycomb and turbulence screen inside a plenum chamber, downstream of a PICFS. The experience-based criteria are such that freestream velocity distortion should be and could be adjusted to be less than, for example, 150 % where freestream velocity distortion is defined as follows [6].

$$V_{distorsion} = \frac{v_{max} - v_{min}}{v_{avg}} \qquad [1]$$

While v_{max}, v_{min} and v_{avg} are the maximum, minimum and average velocity at the calculation surface.

Finally, total pressure drop across a PICFS should be 1.0 qe where qe is dynamic pressure before entering the PICFS. A too high-pressure drop would cause a decrease in wind tunnel's running duration.

Despite the general suggestion, only a brief introduction of general working principles of this device exists. The recommendation that jets velocity through perforation should be less than Mach 0.5 seems automatically satisfied at room temperature. There is no overall design guideline mentioning all the following parameters that designers must consider during the design process.

Dump gap ratio d/D, which is defined as the distance from an incoming pipe's exit plane to a PICFS's apex. Dump gap is chosen to be the shortest length possible to distribute flow to PICFS's conical surface.

Porosity (openness ratio), which decides the average jet velocities across perforated holes on a PICFS. Porosity is chosen to offer sufficient low jet velocities to be well below Mach 0.5 while balancing PICFS's mechanical strength.

The number of holes perforated affects the level of pressure drop as well as level of flow stability across a PICFS. A higher number of perforations produces a higher level of aerodynamic stability but also costs higher loss. Mechanical fabrication tooling size is also considered for convenience.

Perforation uniformity in radial direction determines the level of flow's uniform distribution downstream of a PICFS. Unfortunately, uniform perforation on PICFS does not always result in an elevated level of flow uniformity. Due to the inverse shape of a PICFS, those holes near its center often ask for ampler size to compensate quick loss of dynamic pressure. Non-uniform PICFS performance has been published in [7]. In addition, it has not been recorded other published papers discussing design process that calls the four variables above at the same time.

For practical design and fabrication purposes, in this study, a proposed design process for the non-uniform perforated inverse conical flow spreader is used in wind tunnel's plenum chamber for decelerating flow. This design features reliable aerodynamic quality as well as fabrication convenience. This design method focuses on the installation of the PICFS downstream of a square cross-sectional inlet as it demands unconventional perforation techniques. Figure 1 illustrates this design concept.

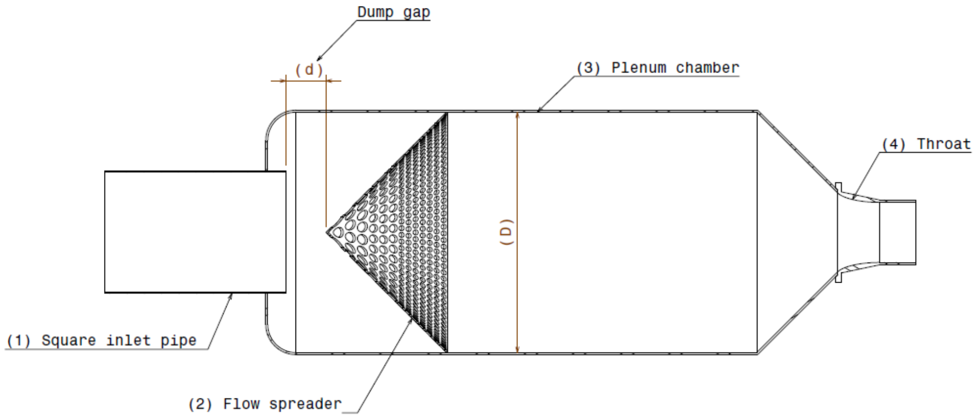

Figure 2. A non-uniform PICFS inside a plenum chamber with square inlet pipe.

2. Methodology

The proposed design process for a non-uniform PICFS is shown in the following Figure 3.

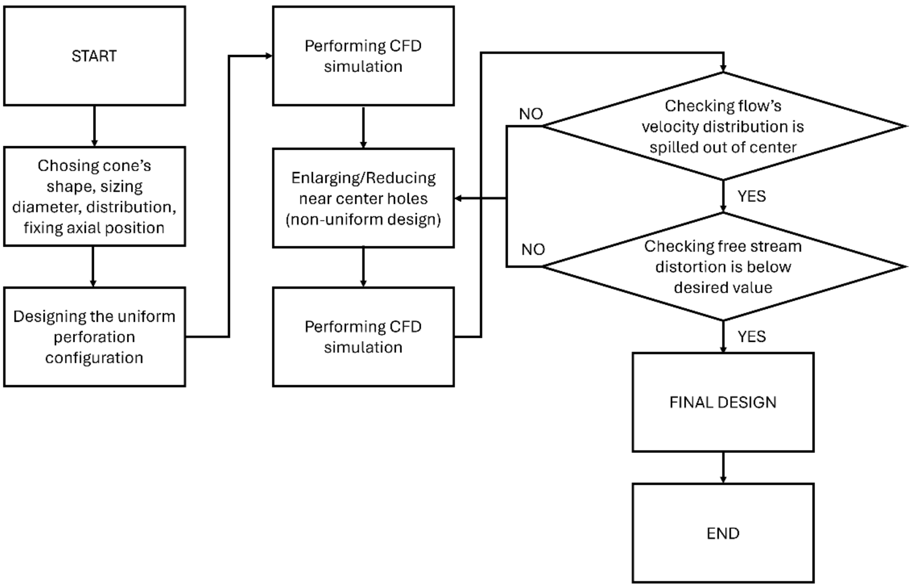

Figure 3 – Flowchart of the design process.

The entire process can be divided into 6 steps:

(1) The first step is to design a fixed PICFS in axial position. The shape and diameter of the PICFS is also chosen.

(2) The second step is to build a uniform configuration, which is the base for further modification. The design parameters in this step include porosity and number of perforations. A dump-gap ratio is fixed for conveniently analyzing.

(3) An interim uniform PICFS is undergone CFD analysis for performance feedback. Based on the aerodynamics performances and behavior, an innovative design of perforation with different diameter is proposed.

(4) The CFD results in the step (3) suggest that the perforations in the central region should be larger than original uniform hole size. A new configuration is built without changing the first porosity value of the uniform PICFS.

(5) The non-uniform PICFS undergoes CFD simulation to analyze its properties.

(6) In terms of aerodynamics performance, if high velocity flow is no longer pushed away radially, then freestream velocity distortion at a target plane is checked to be within desired criterion or not. Other properties like Turbulence Intensity and Uniformity are also calculated.

(6) The concluded non-uniform PICFS design.

3. CFD method

The CFD is the main tool for investigating the aerodynamics performance of the PICFS. Due to typically ample size of perforations and typically non-compressible flow characteristics inside a plenum chamber, it is decided that commercial Computational Fluid Dynamics (CFD) software tools such as ANSYS Fluent offers high fidelity result in PICFS performance analysis. Thus, ANSYS Fluent is used as a simulation and assessment tool to compare various results.

In this method, the PIFCS is not studied alone, but is coupled with overall system as presented in Figure 1 to investigate the interaction between components. This increases the reliability of the PIFCS and has a practical meaning in the system. The following figure describes the CFD method used in this paper.

In this study, various components of the simulation model were eliminated or simplified to reduce the overall amount of computation work. These components comprise air pressure tanks, ball valves, in-between connecting pipes, a butterfly valve system (with only a simple flat disc representing the opening valve) and honeycomb mesh.

For the CFD model, the butterfly valve at the intake is meant to open 45°. This valve generates turbulence mostly in the upstream direction. A baffle and a series of straightener pipes significantly slow the flow before it reaches the square 90-degree elbow. The elbow including guides vanes that turn the flow and directs it to the plenum chamber. Before entering the PICFS in the plenum chamber, the flow passes via a square straightener. At this moment, inertia causes the flow to move downhill. The PICFS uniformly redistributes the flow throughout the chamber, reducing turbulence and velocity before it reaches the honeycomb. Because of the criterion for applying honeycomb, the quality of the air must be verified prior to its placement, and this is the government parameters when designing the PICFS. The air is then discharged into the surrounding environment via the exhaust throat. A bellmouth is located 1D from the exhaust throat to draw discharged air into the engine compressor.

The main zone is built based on the above-mentioned model and takes up the whole space within the plenum chamber and bellmouth. The area around the system or zone is represented as a hemisphere, enabling air to exit in a radial direction. The outermost face of the hemisphere depicts the area where static pressure equals ambient pressure. An annulus plane is placed at the end of the bellmouth to serve as a compressor intake. Figure 4 depicts the whole fluid zone for CFD simulation as well as its boundaries.

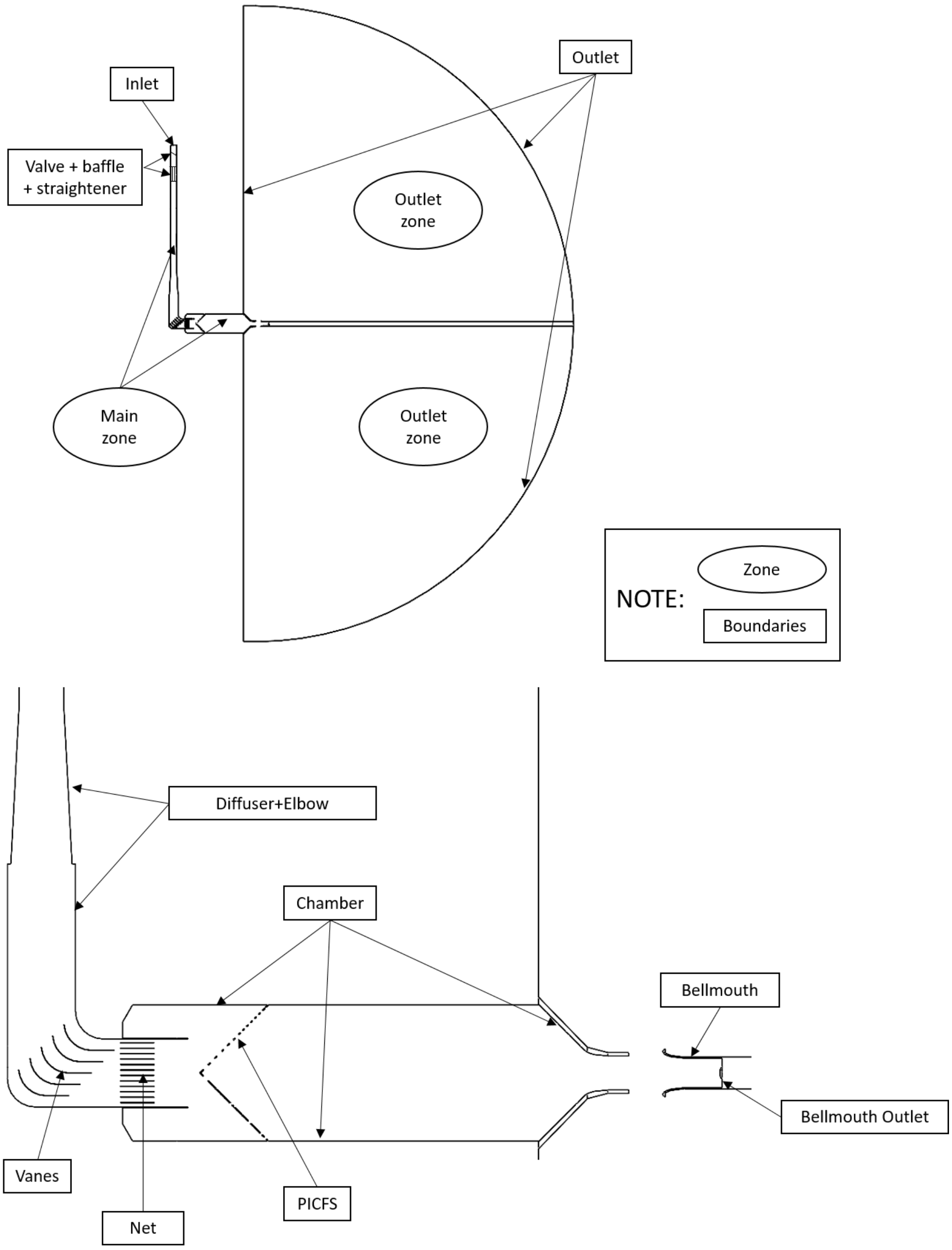

Figure 4. Fluid domain and boundaries for CFD simulation.

The model is meshed to archive the poly-Hexa core type, with a skewness quality of 0.95 and an orthogonal quality of 0.05. Figure 4 shows the full mesh. The air inside the plenum chamber is where the phenomenon of pressure balance and velocity suppression in the radial and axial directions occurs, therefore the region directly behind the PICFS will be meshed evenly using the Body of Influence (BOI) method, as shown in Figure 5, which describes the BOI placement, and the picture below is the meshing result. The size of the elements of the BOI is set to 20 mm. For the PICFS, the surface sizing is 2 mm for the perforation and 5 mm for the conical surface. The growth rate of the mesh is 1.1 for all boundaries. The final mesh is also presented in Figure 5.

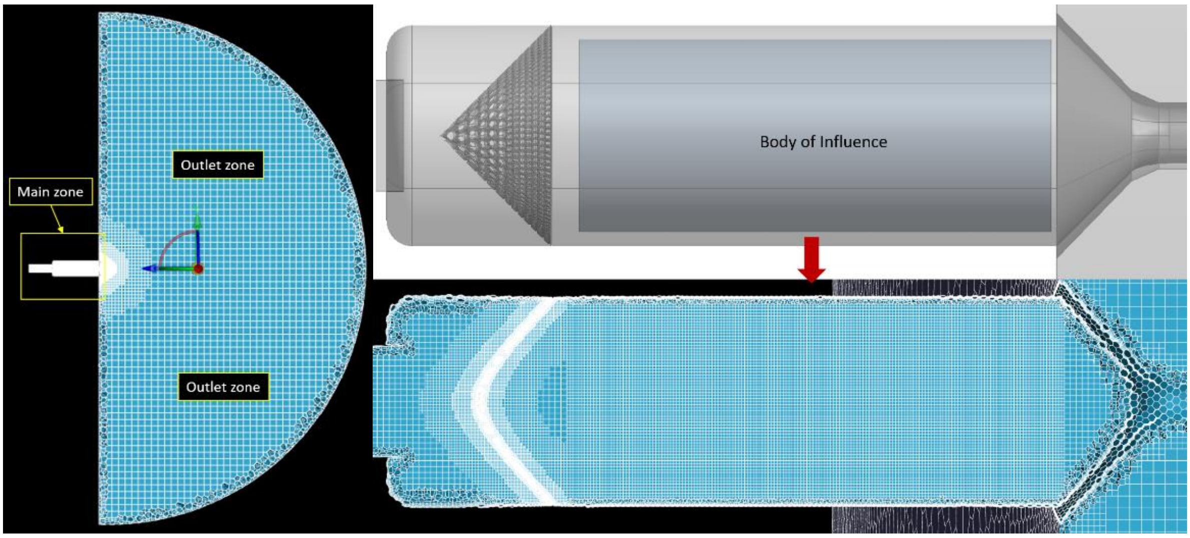

Figure 5. Mesh of the fluid domain. The left one is the main and outlet domain. The right one is the mesh inside the plenum chamber.

The 3D Navier-Stoke equations are solved using commercial FLUENT software. The k-ε scalable wall function is employed to fully close the equations and to handle the boundary layers. Fluid is the ideal gas since it is compressed with high pressure before being released to the ambient environment and reaching transonic. All walls are set as non-slip and adiabatic. The boundary condition is the mass flow inlet – pressure outlet type and based on the working conditions. The flow at the inlet is assumed to be continuous and unlimited. The pressure at the outlet is at ambient air at reference temperature of 15°C. And finally, the pressure outlet is set at bellmouth exit with a target mass flow rate which obtained from requirement of the compressor. For convergence criteria, the residuals are set to below 10^{-4} for continuity and 10^{-6} for other variables. Mach number at the bellmouth exit and mass flow at the exhaust throat are plotted for further convergence judgement.

The outcomes of the CFD simulation are the quality of the air at the position of the honeycomb and the velocity distribution inside the plenum chamber.

4. Design a uniform PICFS

The first thing is to fix a PICFS in axial position. There is a term called "dump gap" d being the distance from an incoming pipe's exit plane to a PICFS's apex. The higher dump gap value, the more likely that incoming flow has chance to spread over a PICFS's conical surface. But this will cost of higher system's overall length. Based on practical experience, the satisfied dump gap d is chosen between one-third to one-half of an incoming pipe's hydraulic diameter.

The next step is to build a uniform configuration, which is the base for further modification. Since the overall shape of a PICFS is fixed by its overall diameter being equal to plenum chamber's diameter and the conical angle being equal to 90°, there are 02 parameters left are porosity and number of perforations.

Porosity (openness ratio) is chosen by balancing jet velocities through perforation's requirement and PICFS's mechanical strength. Porosity here is defined based on projection of perforations axially on cross sectional area of a plenum chamber. High porosity reduces jet velocities, which is asked to be well below Mach 0.5. Also, high porosity depletes PICFS's mechanical strength. Based on experience, porosity value around 40% ± 2% is chosen because plenum chamber's velocity is often less than 80 ft/s.

Number of perforations, on the other hand, contributes largely on flow stability and level of pressure drop across a PICFS. The higher number of perforations, the higher level of flow stability but the higher cost of pressure loss. Bearing in mind that the total pressure drop across a PICFS should be in order of 1.0 qe where qe is the dynamic pressure upstream of a PIFCS, the number of perforations is recommended between 900-1000 ones. Level of 40% ± 2% porosity and number of perforations determine the size of average perforation. Since perforation is often fabricated by drilling method, the number of perforations is chosen for convenient tool size. Note that these holes are randomly distributed to obtain stabilized flow, i.e., to prevent harmonized symptoms. An example of a uniform PICFS is demonstrated in Figure 6.

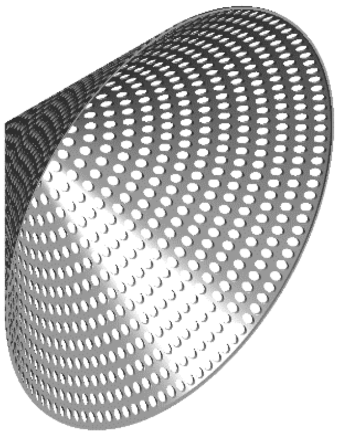

Figure 6. A uniform PICFS.

An interim uniform PICFS is undergoing CFD analysis for performance feedback. The CFD results detect that high velocity stream is pushed off-center. Flow is very stable and decelerates quickly, however, the center space is occupied by low-velocity stream. An example case is presented in Figure 7. This is less desirable because it causes high freestream velocity distortion. High velocity distortion again limits working efficiency of aerodynamic devices such as honeycomb flow straightener and turbulence screen. This flow behavior could be explained as jets emanating from near center holes undergo longer distances than that of jet in rear area. They quickly coalesced and get lost their dynamic pressure. Loss of dynamic pressure raises static pressure. And high static pressure, in turn, pushes other jets away in radial direction. Thus, high velocity regions are off-center as shown in Figure 8. Then the distortion is determined in target plane by equation [1] and give the value of 142%, which is near limited value which is 150%. The pressure loss is $3q_e$ after the PICFS which is three time of limited value.

The reason above suggests that center perforations should be enlarged to give their corresponding jets more energy for deeper travelling distance. Those center perforations should be within one-half of plenum chamber diameter D and based on experience, they should be enlarged up to near 1.5 times of the original uniform hole size, tooling size convenience should be paid attention also. Noted that the overall porosity of 40% should be kept, so that the number of perforations decrease. By doing so, total pressure loss across a PICFS is also reduced.

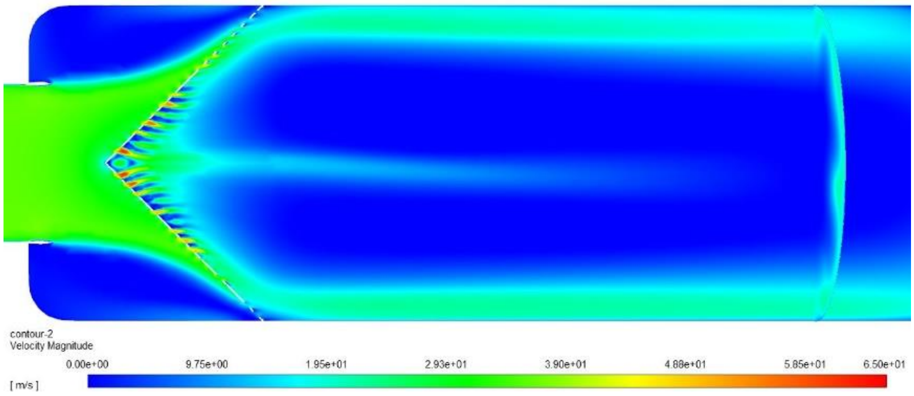

Figure 7 – Velocity distribution at cross section of the plenum chamber when using non-uniform PICFS.

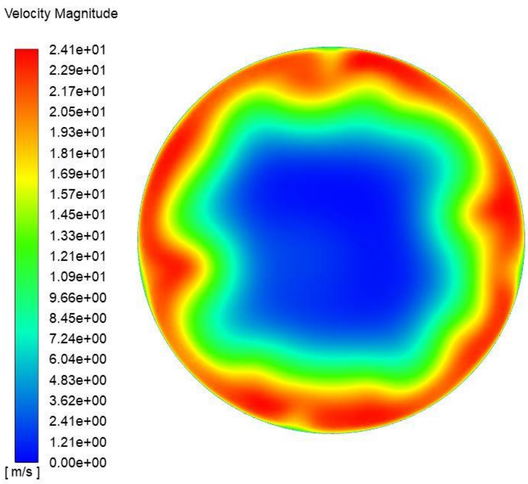

Figure 8 – Velocity distribution at target plane where the honeycomb is installed using uniform PICFS.

5. Design a non-uniform PICFS

Based on flow behavior and characteristics, the diameter of the perforated hole is then enlarged in one-half of the PICFS. It should be noted that the porosity is kept the same as the uniform PICFS, which mean the diameter of the hole is enlarged but the number of hole decreases. There is

Based on the CFD results, if high velocity flow is no longer pushed away radially, then freestream velocity distortion at a target plane is checked to be within desired criterion or not. Typically, this target plane is from 1D to 2D downstream of a PICFS, depending on the specific application and aerodynamic devices installed. If freestream velocity distortion is higher than desired value, center perforations are larger than necessary size. So, their size should be decreased. This starts a new design fine-tuning loop until satisfied result appears which based on quick CFD calculation. Figure 9 demonstrates a enlarged perforation hole in haft-center of a PICFS with the same porosity as previous uniform one.

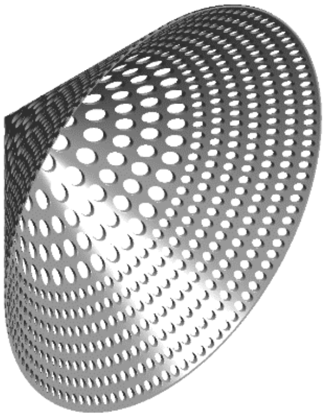

Figure 9. A non-uniform PICFS

On the Figure 10, freestream velocity distortion is 130% at 2D downstream of a desire PICFS for installation of honeycomb flow straightener. This value is calculated on the same target cross sectional plane of previous section 4. The non-uniform PICFS offers superior result than that of uniform one. In other hand, the flow moves straight and not being bended downward in the chamber. On Figure 11, velocity distributions are seen more uniform. Peak velocity in Figure 10 is lower than that on Figure 8 while bottom velocity is higher; thus, significantly improving freestream velocity distortion. And the pressure loss is 0.67 q_e which is less than the limited value.

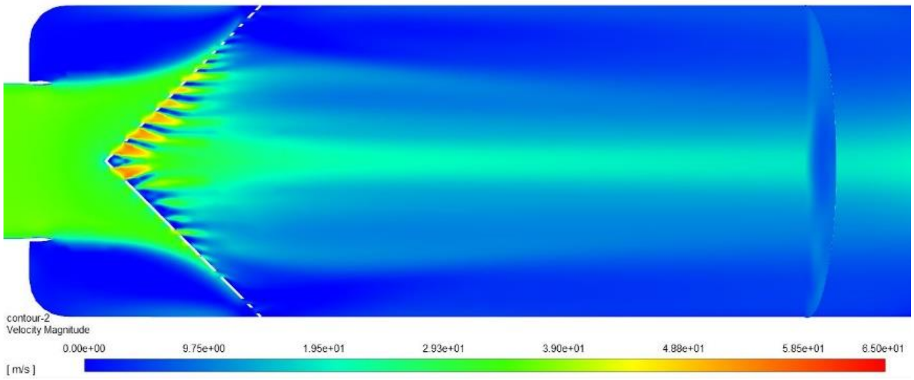

Figure 10 – Velocity distribution at cross section of the plenum chamber when using non-uniform PICFS.

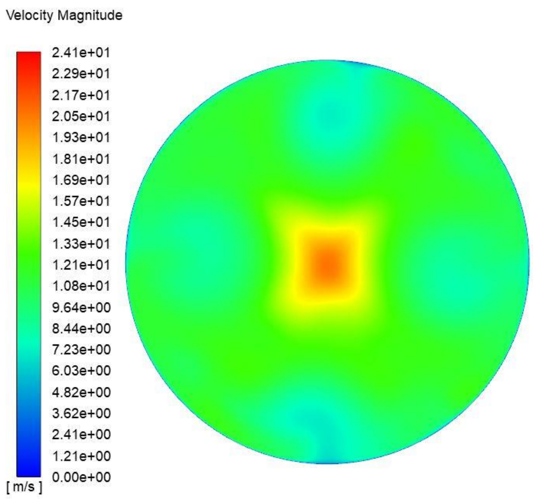

Figure 11 – Velocity distribution at target plane where the honeycomb is installed using non-uniform PICFS.

At this stage, a suitable non-uniform PIFCS has been archived and may proceed to the next manufacturing phase to create a prototype for testing.

6. Conclusion

This study proposes and presents the PICFS design methodology in terms of aerodynamic performance by applying numerical method. This part is useful for small test facilities where installation space is restricted, and the flow highly turbulence or suddenly changes direction.

For the presented systems, the uniform PICFS is first investigated and developed. The CFD results demonstrate that the flow is not evenly distributed, the distortion is close to the limitation value, and the loss pressure is three times the allowable value. This leads to change the perforated hole diameter in the PICFS. The resulting design is a non-uniform PICFS capable of handling unstable flow, spreading it uniformly, low pressure loss and providing adequate flow at the plenum chamber's outlet throttle.

7. Copyright statement

References

[1] Wennerstrom, A.J., Derose, R.D., Law, C.W. and Buzzell, W.A., 1976. Investigation of a 1500 ft./sec. Transonic, High-Through-Flow, Single-Stage Axial-Flow Compressor with Low Hub/Tip Ratio. *Aerospace Research Laboratories, Wright-Patterson Air Force Base, AD B*, *16506*.

[2] Fiore, A.W., Moore, D.G., Murray, D.H. and West, J.E., 1975. *Design and calibration of the ARL Mach 3 high Reynolds number facility* (p. 0184). Aerospace Research Laboratories.

[3] Arora, N., Ali, M.Y. and Alvi, F.S., 2016. Flowfield of a 3-D swept shock boundary layer interaction in a Mach 2 flow. In *46th AIAA fluid dynamics conference* (p. 3649).

[4] Pope, A. and Goin, K., High-speed wind tunnel testing, 1965.

[5] Ferri, A. and Bogdonoff, S.M., 1954. Design and operation of intermittent supersonic wind tunnels: AGAR Dograph AG-1. *NATO Advisory Group for Aeronautic Research and Development*.

[6] Lominac, J.K. and Boytos, J.F., 1998, June. Aeropropulsion environmental test facility. In *Turbo Expo: Power for Land, Sea, and Air* (Vol. 78637, p. V002T02A015). American Society of Mechanical Engineers.

[7] Nguyen, T.G., Nghiem, Q.H., Nguyen, Q.H., Nguyen, N.V., Chu, D.L. and Nguyen, P.M., DEVELOPMENT OF THE LOW-COST AND LIMITED SPACE SEA-LEVEL RAM AIR TEST (RAT) MACH 0.8 SYSTEM.