33RD CONGRESS
OF THE INTERNATIONAL COUNCIL
OF THE AERONAUTICAL SCIENCES
STOCKHOLM, SWEDEN, 4-9 SEPTEMBER, 2022

ICAS 2022 SWEDEN

DEVELOPMENT OF THE LOW-COST AND LIMITED SPACE SEA-LEVEL RAM AIR TEST (RAT) MACH 0.8 SYSTEM

Truong-Giang Nguyen[1*], Quoc-Huy Nghiem[1], Quang-Hai Nguyen[1], Nhu-Van Nguyen[1], Duy-Lanh Chu[1], Phi-Minh Nguyen[1]

[1*]Corresponding author, giangnt84@viettel.com.vn
[1]Jet Propulsion Center, Viettel Aerospace Institute, Viettel Group, Vietnam

Abstract

The low-cost and limited space sea-level RAT Mach 0.8 system design process is proposed and presented for both small turbojet engines and flight vehicle system development. Overall steps from requirements to the final RAT system layouts are addressed and discussed briefly. The main plenum chamber design improvement section is presented with the extensively using CFD as the analysis solver. The special treatments are performed on the guide vane aero-dynamical design and pipe flow straightener for turning corner to minimize the turbulence and loss in order to match for the limited space. The diffuser, honeycomb and screen for plenum chamber are thoroughly considered and design for the customized length of plenum while maintaining the stability, uniform of flow and Mach 0.8 of exit. The proposed methodology shows the feasibility and effectivity of the low-cost and limited space sea-level SAT Mach 0.8 system for small turbojet engines and flight vehicle system development.

Keywords: Ram Air Test (RAT), small turbojet engines, Limited space, Low-cost, CFD, Plenum design.

1. Introduction

Sea level ram air test (RAT) is the subset of engine altitude test facility (ATF). While the ATF system has more capability to test engine at various altitudes to complete engine's flight envelope [1], sea level RAT focuses only on re-creating flight condition at sea level (h < 1000m) with free jet stream. Therefore, sea level RAT's building cost is considerably less expensive than that of complete ATF.

On another hand, sea level RAT is essential, especially for small start-up teams building their very first engines at early phases because sea level RAT offer capabilities to test in-flight start, sea level engine slam acceleration, engine-inlet compatibility and so on.

Being aware of the potential sea level RAT can offer, the sea-level RAT that can offer Mach 0.8 free stream speed system has been researched and developed. The main objectives of this RAT facility are exit flow up to Mach 0.8 at sea-level condition, maximum mass flow rate of 25 kg/s and the duration of minimum 30 seconds for testing time.

Surveying ATF test cells in large organization such as KARI [2], they are often installed in new, dedicated area with built-in purpose. Those systems often feature long, straight and round pipe upstream of plenum chambers. Electric power those facilities consume are in order of several Megawatts.

To further lower cost and to relieve the burden of huge electric power needed, the intermittent blowdown sea level RAT configuration was chosen instead of continuous running configuration. The intermittent configuration basically features pressure vessels for air storage and the throttling valve to control stagnation pressure inside the plenum chamber. Running duration depends essentially on the amount of air stored, thus it can be extended easily.

In contrast to creating new building, the situation in the author's organization is to upgrade the available static sea level in-door engine test bench and there is maximum 5.5 m length in front of the engine inlet. Designing a proper plenum chamber is the key thing to meet this restrictive condition. Because of that, creative solution to design a feasible plenum chamber has been developed. This paper focuses to present in detail the aerodynamic design process of the plenum chamber implementing CFD as analysis tool.

2. The design process of low-cost and limited space sea-level RAT Mach 0.8 system

The overall design process of low-cost and limited space sea-level RAT Mach 0.8 system is shown in the Figure 1 1. The brief descriptions are presented as follows:

Step 1: Requirements

The requirements are composed of laboratory space limits, RAT system requirements, aerodynamic qualities, vibration and noise level by control valve operation, and the regulations for the flow after honeycomb as shown in the Figure 1.

Step 2: Air-tank sizing

The total air volume required is sized to satisfy the requirements of test duration. Then, the numbers of tank are considered to use the existing industrial air-tank while considering the cost factors.

In fact, air pressure vessels are designed to contribute parallel to the main pipe; thus, test duration could be easily extended by increasing the number of air tanks.

Step 3: Valve sizing

Valve sizing is composed of butterfly and ball valves. The valve selection, pipes and hoses are performed at this step.

Step 4: Plenum chamber design

Plenum chamber function is designed to stabilize and supply uniform air flow for engine consumption.

Plenum chamber includes of chamber design and flow straightener as shown in the Figure 1.

The CFD solvers, modification of plenum chambers, and evaluation are considered at this step.

The focus discussion of this article is presented according to the Plenum Chamber Design for the case study.

The optimum plenum chamber configuration is then fixed.

Step 5: Sensor selection and control system

Sensors and control system are selected and performed for control of entire system.

Step 6: Supporting components

The supporting structure, tubes, pipes and hoses are selected for support the entire system.

Step 7: Overall and final RAT system layout

The drawings and COTS components are provided.

The total cost consideration is made to help the decision-maker.

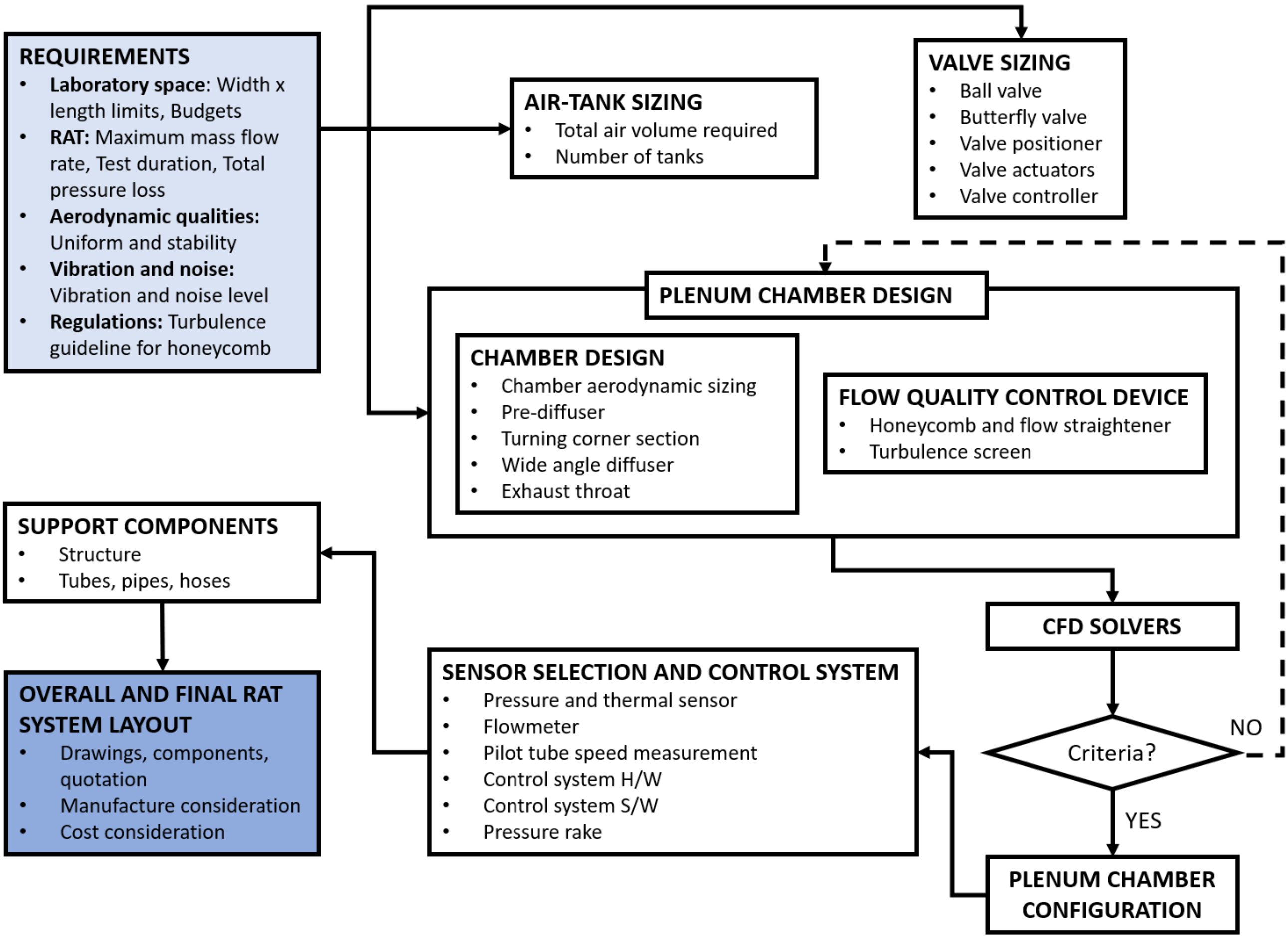


Figure 1 – The low-cost and limited space sea-level RAT Mach 0.8 system design process.

3. System's technical requirement

The requirements for the system are presented as follows:

+ Space requirement: Length of all flow stabilizing and accelerating devices in front of the engine must be shorter than 5.5 m length.

+ Performance requirements:

- Testing duration: At least 30 sec (intermittent test)
- Nominal Mach number at system's output: 0.8M
- Exhaust throat size: 300 mm diameter
- Aerodynamics qualities: Uniform and stability
- Vibration and noise level: Specified by the butterfly valve's manufacturer

The demand of discharging speed and throat size determine the system's discharging mass flow rate of 25 kg/s. This in combination with minimum duration needed determine sizing for pressure vessels of 100 m^3 volume at 12 bar. Air pressure vessels are charged by popular commercial screw compressors having power of 75-200 kW. By adding up more tanks parallel, system's running duration could be extended linearly. It should be noted that for the same level of discharging air mass flow rate, KARI [2] uses approximately 5.4 MW power source for the continuous running system with considerably higher electric distribution complexity.

The detailed steps are presented in the previous section. The main discussion for the plenum chamber design with extensively using CFD methods is addressed as follows.

4. CFD Model

The CFD solution is used to analyze the aerodynamic properties of the air flow and to compute its quality parameters using the ANSYS FLUENT 19.1 software. The simulation model is simplified by subtracting the air tanks, controlling valves, honeycomb grid and turbulence screen. Figure 2 below illustrates the simulation model with uniform inlet assumption during design process development. Details of designing components are presented in the plenum chamber design section.

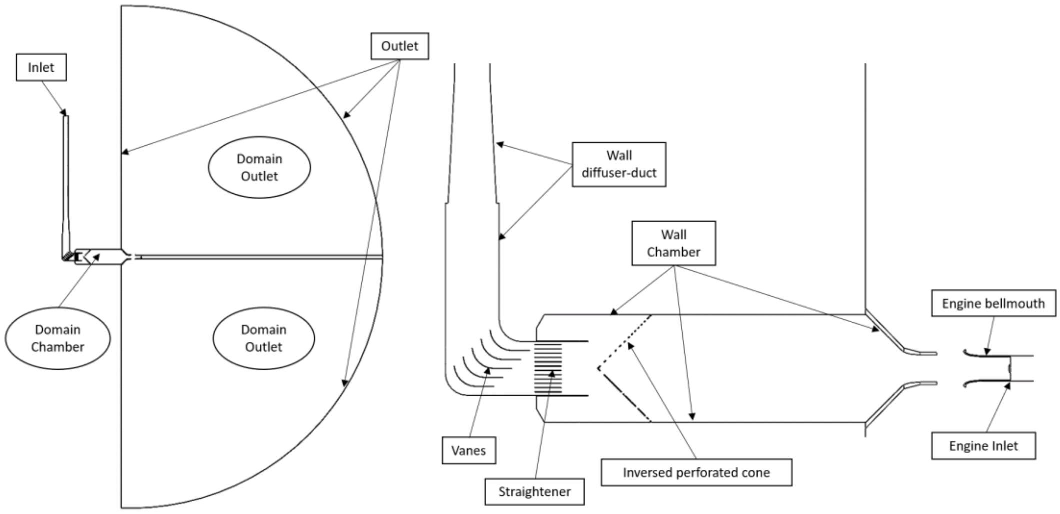


Figure 2 – Simulation model's domains and boundary.

The model is processed and meshed to obtain the hybrid poly-HEXA elements. This approach reduces the number of elements in comparison to the method in which only tetra element is used. Mesh in pre-researching process shown that the results are independent of the mesh if the total elements is above 14 million.

The 3D Navier-Stoke equations is solved by FLUENT's solver, using a k-e turbulence model to enclose the equations. The Enhanced Wall Treatment model is applied to handle the boundary layers. The fluid is considered as ideal gas and compressible. The following table summarizes the boundary conditions:

Table 1 - Boundary Condition (BC) for CFD model

Boundary	Type	Value
Inlet	Mass Flow Rate	25 kg/s (maximum)
Outlet	Ambient Pressure	1 atm
Valve, Wall Pipe-Turning Corner-Chamber, Engine Intake	No slip, adiabatic wall	

The inlet initially was set as DN400 round pipe. Later, since it was learned the two facts that the inlet is partially blocked by the butterfly valve's openings and the valve causes strong disturbance downstream, the inlet was intentionally blocked to simulate one of valve's opening cases and to artificially induce disturbance into valve's exit flow. Also, the job is to design flow control devices along the pipe such that pipe's exit condition is approximately that of uniform inlet round pipe.

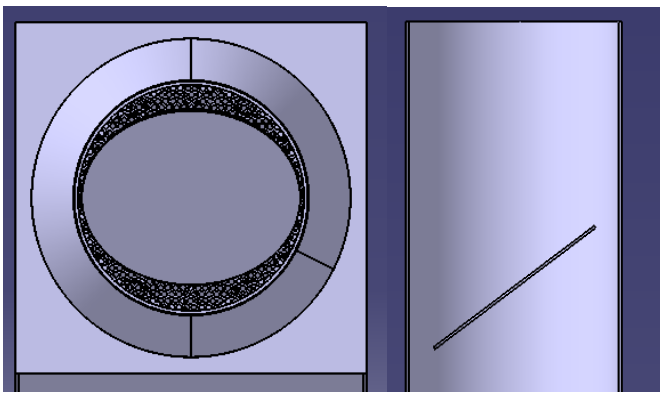


Figure 3 - Partially blocked inlet condition shape (left) projected view and (right) sectional view

This partially blocked inlet BC has contributed to one of major design changes since early wide-angle diffuser designs experienced undesired long lasting flow fluctuation effect inside the plenum chamber when tested with the partially blocked inlet. This may indicate that macroscopic flow-induced vibration may happen in real life, and chamber's structure may be vulnerable to this vibration. After changing of the diffuser design, evolved chamber's designs were able to damp any incoming disturbance quickly, showing phenomenal stabilizing capability.

5. Plenum chamber design

In general, in order to deliver stable, uniform and high-speed flow, the method is to handle it efficiently at low speed and then accelerating it to the desired value. Thus, it mainly comprises 04 phases: damping and straightening highly disturbed flow downstream of the control valve, slowing down high speed, stabilizing-removing disturbance and then accelerating.

These 03 phases are executed via the following sub-components:

- Damping and straightening pipe flow:
  + Baffle
  + Tube bundle flow straightener
- Slowing down:
  + Pre-diffuser
  + Turning vanes
  + AMCA square flow straightener
  + Wide angle diffuser
- Stabilizing and removing disturbance:
  + Chamber
  + Passive flow quality control devices: a honeycomb and a turbulence screen
- Accelerating:
  + Throat

During design process, flow stability is preferred to efficiency, and in total, 25 evolutions of the plenum chamber have been designed and CFD-run before concluding the final configuration.

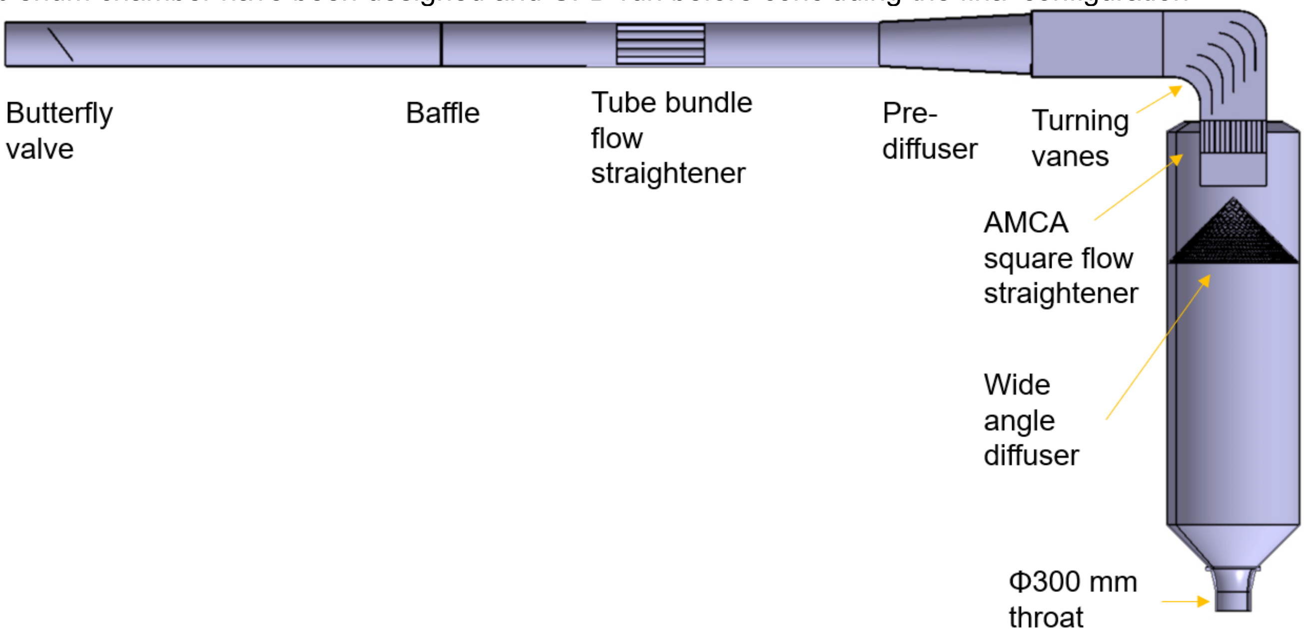


Figure 4 – Overall aerodynamic design, horizontal sectioning (final version)

5.1. Plenum chamber aerodynamic sizing

The plenum chamber is to settle flow to make it uniform before accelerating it to exhaust nozzle. In this region, static pressure dominates, and that total pressure can be considered as equal to static pressure. Pope [4] suggested that flow velocities here should be in range of 10-80 ft/s. Having in mind the Mach 0.8 stream's total pressure of being 1.524 times ambient pressure, the plenum chamber's diameter is chosen to be 1.2 m, which gives average velocity of approximately 40 ft/s (12 m/s) or Mach 0.035.

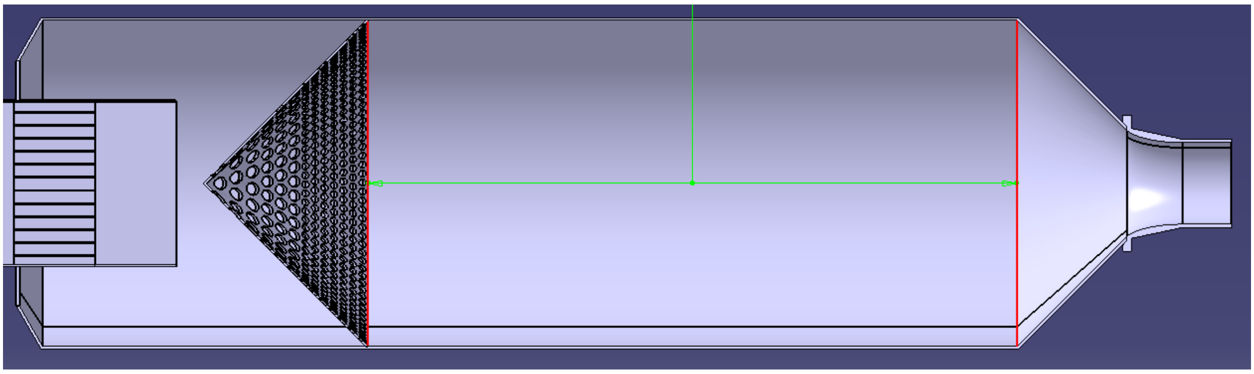


Figure 5 - Plenum chamber's aerodynamic sizing

Given this diameter and restrictive length allowed, the straight portion downstream of the wide angle diffuser is approximately 2 times of plenum chamber diameter, which is sufficient to arrange uniform flow and installing of passive flow correcting devices as recommended by Ferri [9].

Throughout design process and CFD analysis, there is no problem associated with inadequate sizing of the plenum chamber and the sizing choice above is maintained throughout all versions.

5.2. Exhaust throat section

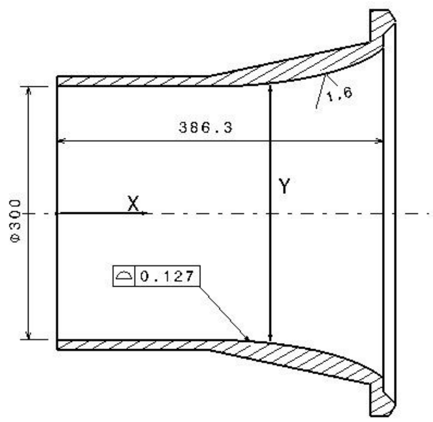
At the end of the plenum chamber, there is convergent section to accelerate flow to desired speed. Since flow acceleration takes place here in preferred pressure gradient region with almost no probability of flow instabilities triggered, the exhaust throat was designed first.

The exhaust section comprises a simple 90° conical shape at the beginning welded to the contoured bell mouth. The contoured bell mouth's function is to minimize loss and it is based on AEDC's model guidelines [12] applied on their altitude test cell with scalability. This bell mouth design showed high working reliability and it allowed ample profile tolerance of 0.127 mm, facilitating its construction process.

The bell mouth contour follows the ellipse shape with standardization radial diameter of 1:

$$\frac{x^2}{0.724^2} + \frac{y^2}{0.448^2} = 1$$

Expanding into 300 mm diameter, the design of the exhaust bell mouth is as below:



Elliptical Countour	
X (mm)	Y (mm) (+0.254 -0.000) ID
0.0	300.0
180.0	300.0
196.1	300.3
212.1	301.4
228.1	303.3
244.2	306.0
260.2	309.4
276.2	313.9
292.3	319.3
308.3	326.0
324.4	334.0
340.4	343.8
356.4	356.0
372.5	372.1
386.3	391.8

Figure 6 - Design of the exhaust bell mouth

This design performed satisfactorily during design process and its flow properties will be presented later.

5.3. Incoming pipe design

The discharged air flow is throttled by the butterfly valve. Throttling process causes pressure differential across the valve, causing considerable acceleration of air velocities. Pope [4] suggested that the Mach number in the pipe downstream of the control valve should be less than 0.4 to avoid undesirable whistling (noise) and high-pressure losses at the maximum designed mass flow rate. Bhatia [5] establishes an upper limit of Mach 0.33 at the valve body's exit due to noise concerns, stating that this is one of the most critical factors to consider when sizing a control valve. The DN400 Sch10 pipe satisfies both pipe’s speed limit as well as valve manufacturer’s noise, vibration, and controllability requirement, so it is chosen as incoming pipe size for the plenum chamber having average speed of Mach 0.34. This speed is calculated from valve calculation program, preliminarily assuming no loss across pipe, i.e., pipe pressure = 1.524 bar. But as shown below, there is ~0.7 bar pressure loss across the pipe, making valve’s downstream speed is lower than Mach 0.34.

As mentioned above about two types of BC, uniform pipe inlet BC and partially blocked inlet BC have been used during development process. The uniform inlet was used first to test other aerodynamic devices in ideal working condition. And then, the nonuniform inlet was used to test system’s stability under extremely challenging working condition.

Finally, to damp disturbances caused by valve’s opening and movements, the pipe is attached the distribution baffle and the tube bundle flow straightener. The 20 mm thickness distribution baffle has hole diameter of 10 mm and opening area ratio of approximately 54%. The tube bundle flow straightener is of 19 tubes type according to ISO 5167 standard, each tube has ID of 75 mm and 2 mm wall thickness. The results gained are promising since the overall system performance with nonuniform inlet BC and pipe flow control devices is nearly equivalent to that with uniform inlet BC.

The cost for this flow correction is approximate 0.5 bar pressure drop across the baffle and 0.7 bar pressure drop in total.

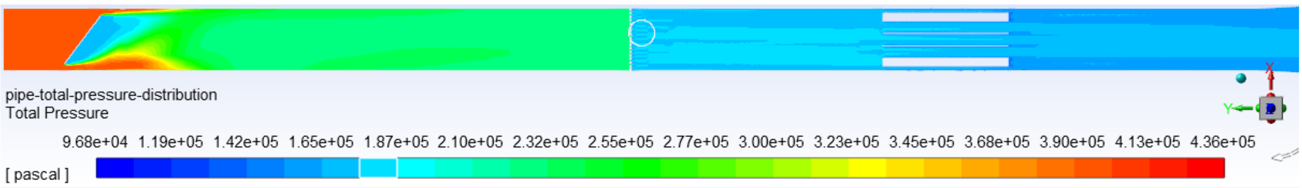


Figure 7 - Total pressure distribution in pipe before pre-diffuser

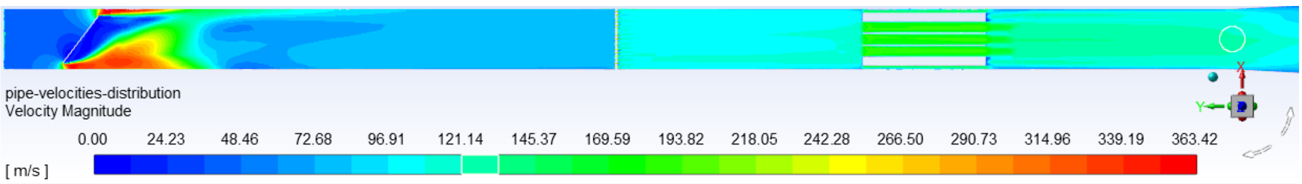


Figure 8 – Velocities distribution in pipe before pre-diffuser

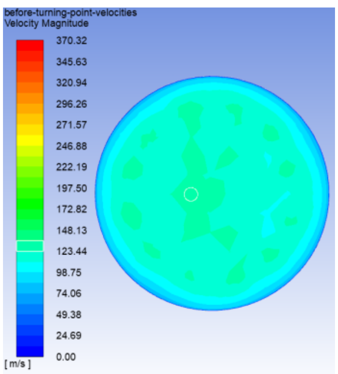


Figure 9 – Velocity contours at the end of straight pipe section

5.4. Pre-diffuser section

In the next step, in order to deal with the limited space of the laboratory, the discharged air must be turned 90° using a turning corner with guide vanes. Since in the turning corner, loss is proportional to velocity squared and structural stress is proportional to velocity cubed, it is crucial to slow down air flow speed prior to the turning corner.

Thus, to turn the air efficiently and safely, it is decided to slow down valve's exit velocity of 0.34 Mach to turning section's incoming velocity of 0.1 Mach order. In addition, the turning section must be in form of square shape for welding of turning vanes into section. Hence, the resulting shape for the turning section is 0.6m x 0.6m square.

A pre-diffuser is a device being responsible for slowing down air velocity from Φ0.4m circular section to 0.6m x 0.6m square section stably and as efficiently as possible. Flow stability is of prime concern since a lot of fluctuation has been observed in various designs' simulation which may harm the structure.

Several pre-diffuser designs were tried with some examples are showing on Figure 6, but all of them except for the configuration formed by conical diffuser followed by dump region (bottom right corner) experience flow instabilities even with the uniform circular inlet BC. In this configuration, the conical diffuser has conical angle of 6^o with area ratio of 1.88 and length per inlet radius ratio of 7.1, lying well inside "no stall zone" as guided by Lefebvre [6] and Osborn [7]. AEDC test cell T-2 [10] used the similar approach to decelerate flow downstream of airflow measuring device.

Figure 11 shows the Mach number distribution in the pre-diffuser, it is easy to observe that flow is stably decelerated before coming to the turning section.

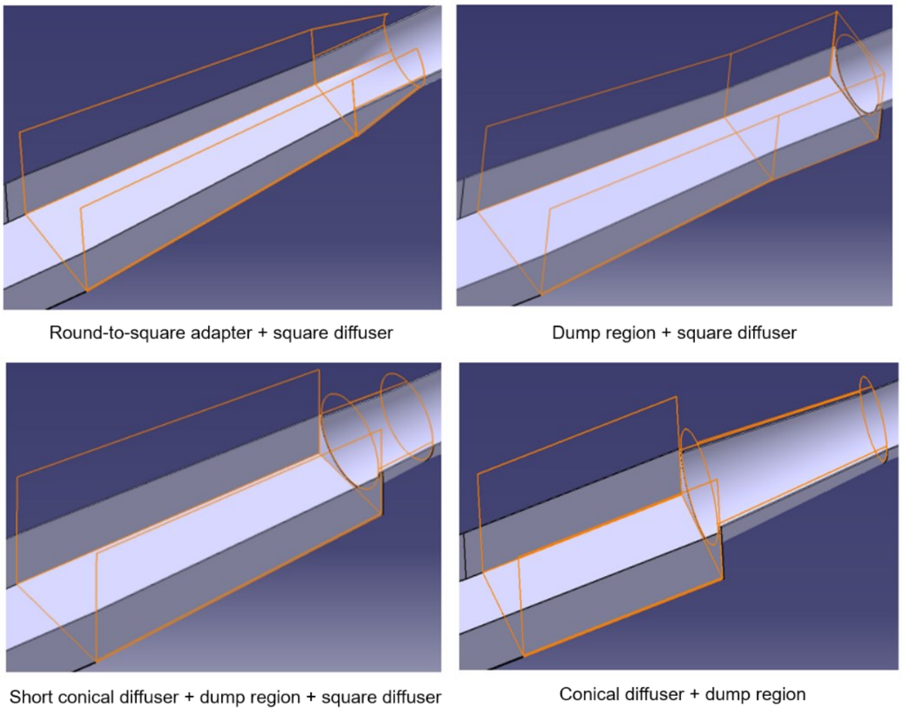


Figure 10 – Some design configurations of a pre-diffuser

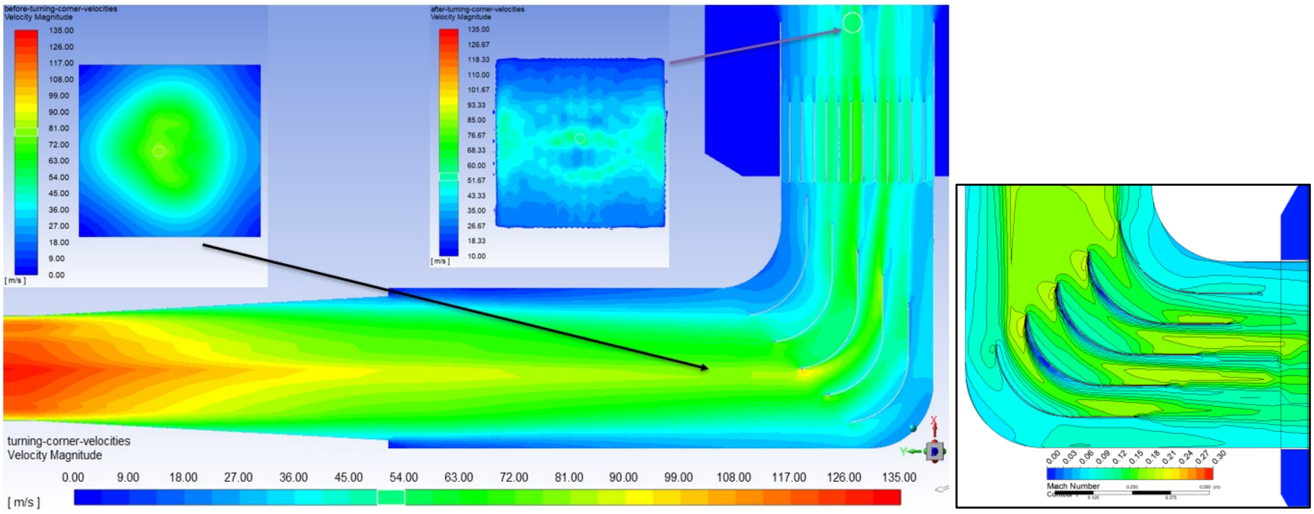


Figure 11 – CFD results show Mach number distribution over the pre-diffuser and the turning corner

5.5. Turning vane design

Turning vane design is expected to distribute flow stably, to have low fabricating cost and to work as efficient as possible. To seek for lowest fabricating cost, instead of expensive airfoil shape, turning vanes are made from 3 mm steel sheet, constructed by bending machine.

Design practice of ¼ circle turning vane is proposed by Johl et. al [11] with detailed description on the figure below.

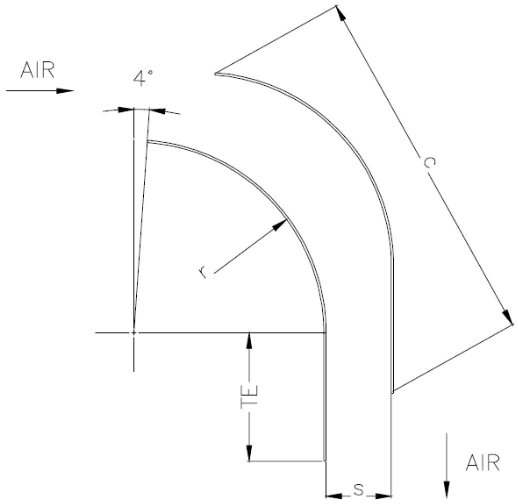


Figure 12 - Plain camber circular arc turning vane definition [11]

The vanes were designed having an angle-of attack of 4° for minimum pressure loss coefficient K_L across vanes. Space to chord ratio (s/c) is targeted to be 0.20 for great flow stability. Large chord vane is desired for the reasons of aerodynamic benefit, rugged structure and ease welding. The aerodynamic aspect is that K_L tends to decrease with increasing Reynolds chord-based number (Re_c) [11]. Given vane height of 600 mm, larger chord allows stronger structure. And finally, given fixed s/c ratio, larger chord c means larger distance s between adjacent vanes, making welding accessible and a smaller number of vanes in the turning corner. Compromising these factors, the resultant vane has radius of 245 mm, training-edge extension TE of 200 mm, height of 600 mm. There are 5 vanes on the turning corner, thus distance s of 100 mm, chord c of 500 mm and Re_c of 1.94×10^6 well inside turbulent region.

Figure 11 shows that there are some flow separation regions on mid-vanes' suction sides but flow exiting tuning vanes is quite stable. No modification is needed on Johl's method.

Also, on Figure 11, probably due to flow's high momentum there appears not straight exit but inclined flow angle downstream of these turning vanes. Therefore, the AMCA square flow straightener (45 mm cell size, 300 mm length) according to ISO 5167 was added downstream of turning vanes to correct flow direction as desired. Thanks to low local velocity of Mach 0.1 order average, pressure loss across this flow straightener is negligible.

5.6. Wide angle diffuser

A wide angle diffuser is needed to further slowdown flow velocity of order Mach 0.1 from the turning corner's exit plane to that of Mach 0.035 (12 m/s) inside the plenum chamber as sizing above. This device focuses on not aerodynamic efficiency but short length instead. In general, Pope [4] and Ferri [9] mentioned 02 feasible methods. The first one utilizes multiple 2D small angle diffuser channels to form a large angle diffuser. The second one consists of an inverse cone having random distribution of perforations to obtain downstream uniform flow/pressure by means of jets coalescence.

5.6.1. Assessment criteria of wide angle diffuser

At certain distance downstream of a wide angle diffuser within the plenum chamber length sized, there are mainly 03 criteria must be met in order to set up a honeycomb flow straightener:

- Turbulence intensity (TI) should be less than 10% at honeycomb installation plane
- Uniformity index (UI) should be > 70%, at honeycomb installation plane
- Plenum chamber's flow stability in case of unsymmetrical flow inlet which simulates valve's leaving flow condition

Both wide angle diffuser configurations were tested by CFD method with initially uniform inlet BC and later asymmetrical inlet BC.

5.5.2. Multi-channels diffuser (MCD)

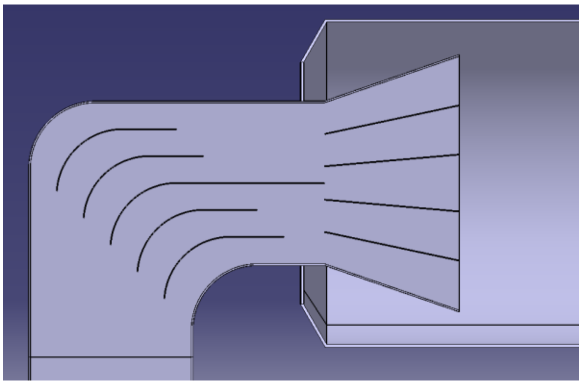


Figure 13 - Multi-channels wide angle diffusers

Each element diffuser channel was designed so that it lies well inside "no stall zone" as recommended by Lefebvre [6] with divergent angle of 7° to 10° in vertical direction only, thus total divergent angle of 38°. This diffuser is expanded until it nearly touches chamber's wall. The reason to choose this design as initial idea is its low cost and simple construction.

This design works well with uniform inlet BC but showing unstable properties with partial block inlet BC. The instability is mainly rooted from dysfunctional capability to slow down flow due to flow separation in diffusing channels with asymmetrical inlet condition, seeing Figure 10.

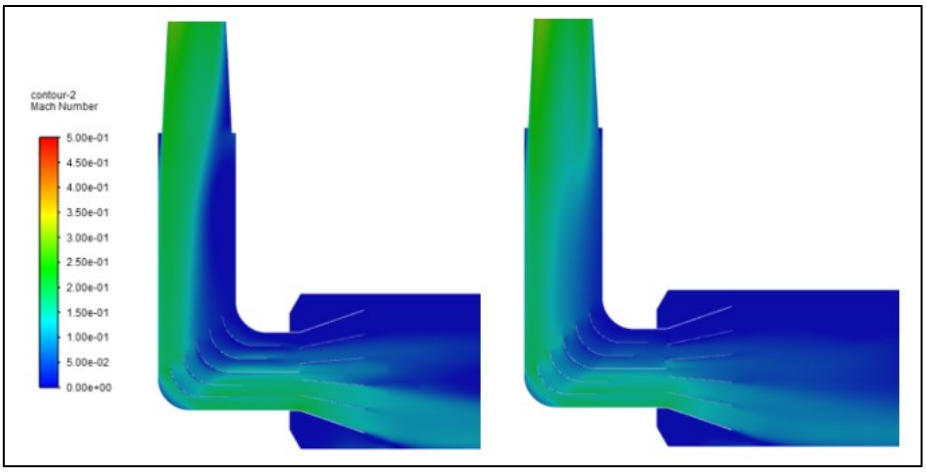


Figure 14 – Multiple channels diffuser's flow instability (fluctuation) with partial block inlet BC.

Therefore, the author had to switch design method to the inverse perforated cone type.

5.5.3. Inverse perforated cone diffuser (IPCD)

Pope [4] and Ferri [9] summarized the inverse perforated cone diffuser's (Figure 14) working features:

- Individual jets of air from individual perforations coalesce to slow down jets and to form a uniform flow
- Random distribution of holes is used in order to obtain uniform pressures in the region downstream of the cone
- The inverse cone allows ample perforations, thus resulting low pressure drop ΔP_t of order 1.0 q_e where q_e is the dynamic pressure upstream of the diffuser.

Because there is no specific guideline existed, trial-and-error method is used with the first design iteration as bellows on Figure 15 (left):

- Cone angle of 90° as [4] and [9] suggested
- Randomly distributed of 935 holes of 25 mm diameter give total area of 0.46 m^2, i.e., 40% opening area, thus average perforation exit velocity = 30 m/s, which is well below Mach 0.5

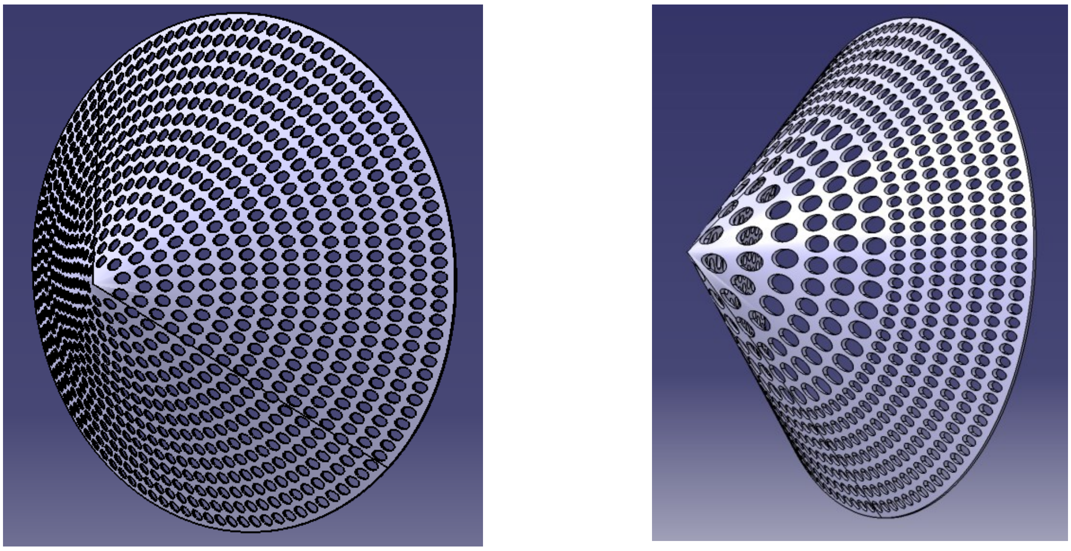


Figure 15 - First design (left) and the final design (right) of IPCD

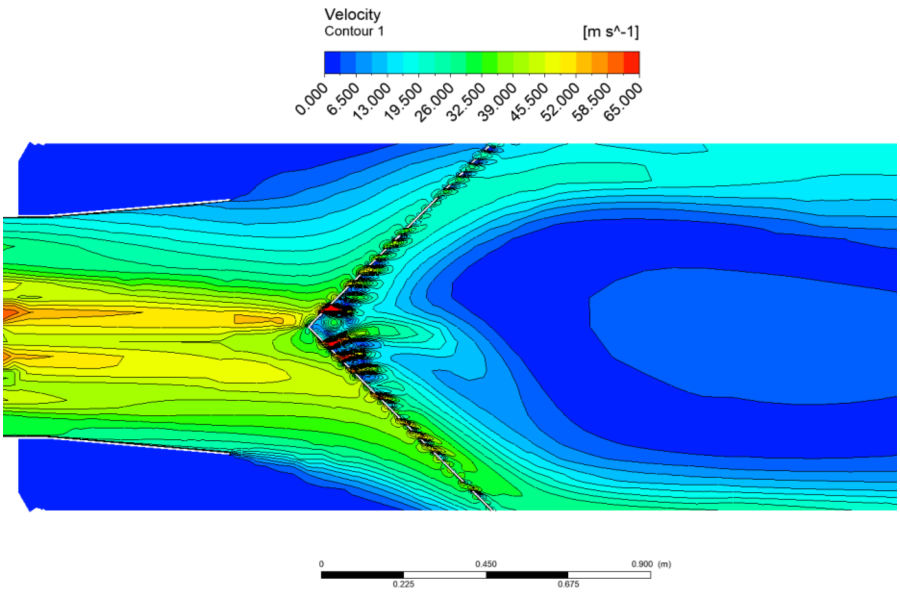


Figure 16 - Velocity contours caused by the first design iteration, uniform inlet BC

Results showed that high velocities emanating from center holes quickly coalesce and lose their kinetic energy, thus creating high static pressure region which pushes other jets away in radial direction. This, in consequence, makes high velocity region off-center. In addition, dynamic pressure q_e just upstream of the cone is 402 Pa, total pressure upstream and downstream of the cone are 157863 Pa and 157199 Pa, respectively. So $q_e/\Delta P_t$ = 1.65, which is higher than the recommended value of 1 in [4].

Therefore, it is decided to further increase size of holes near the center so that flow over these holes possesses higher dynamic pressure, delaying loss of kinetic energy. By trial-and-error method, center holes finally are chosen of mixed of 45 mm and 40 mm diameter and the rest are still 25 mm diameter (Figure 15, right). Testing with uniform inlet BC showed that flow was no longer pushed away to chamber's wall and pressure drop ΔP_t was approximately equal to q_e.

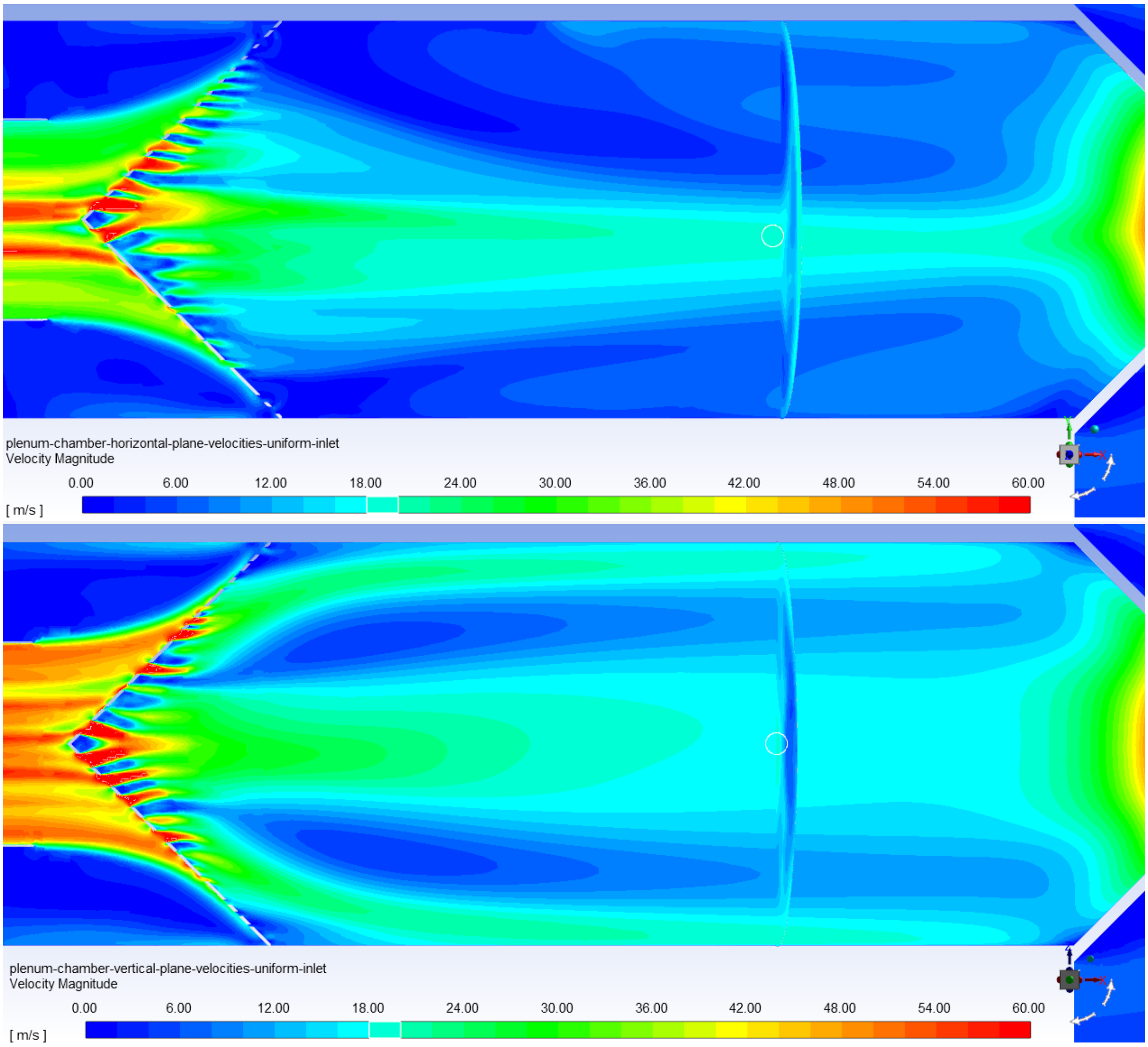


Figure 17 – Velocity distribution inside the plenum chamber under uniform inlet condition, horizontal plane (upper) and vertical plane (lower)

So far, except for honeycomb flow straightener and turbulence screen, all components in the plenum chamber design have been completed, and then the whole system has been undergone CFD simulation with partial block butterfly inlet BC.

By the baffle and tube bundle flow straightener inserted downstream of the butterfly valve, qualitatively, flow distribution inside the plenum chamber is virtually as the same as those with uniform inlet, see figures below. Flow is still highly stable with neither large flow separation region nor large-scale fluctuation observed. Because of that, it could be deduced that flow would not induce dangerous vibration to the plenum chamber's thin-walled structure.

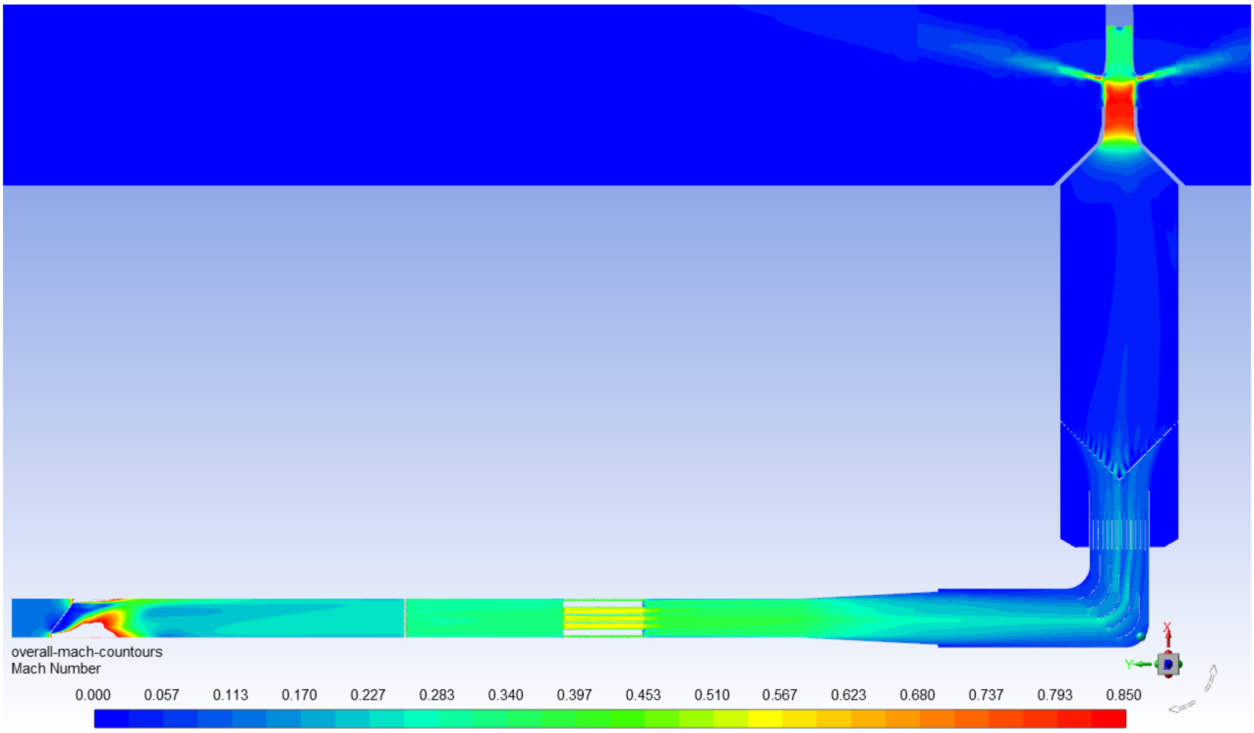


Figure 18 - Mach number contours overview with partial block inlet BC

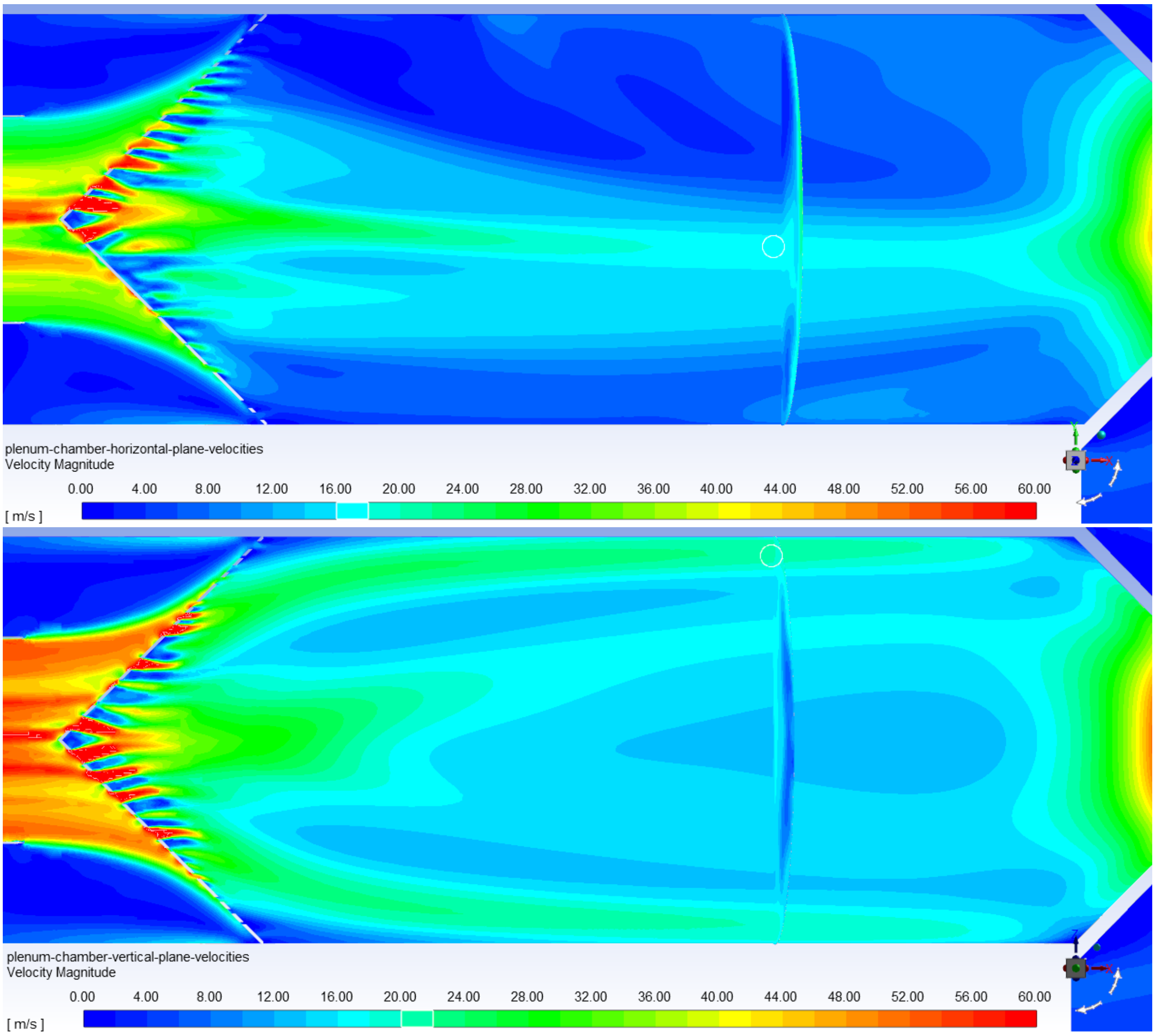


Figure 19 - Velocity distribution inside the plenum chamber under partial block inlet BC, horizontal plane (upper) and vertical plane (lower)

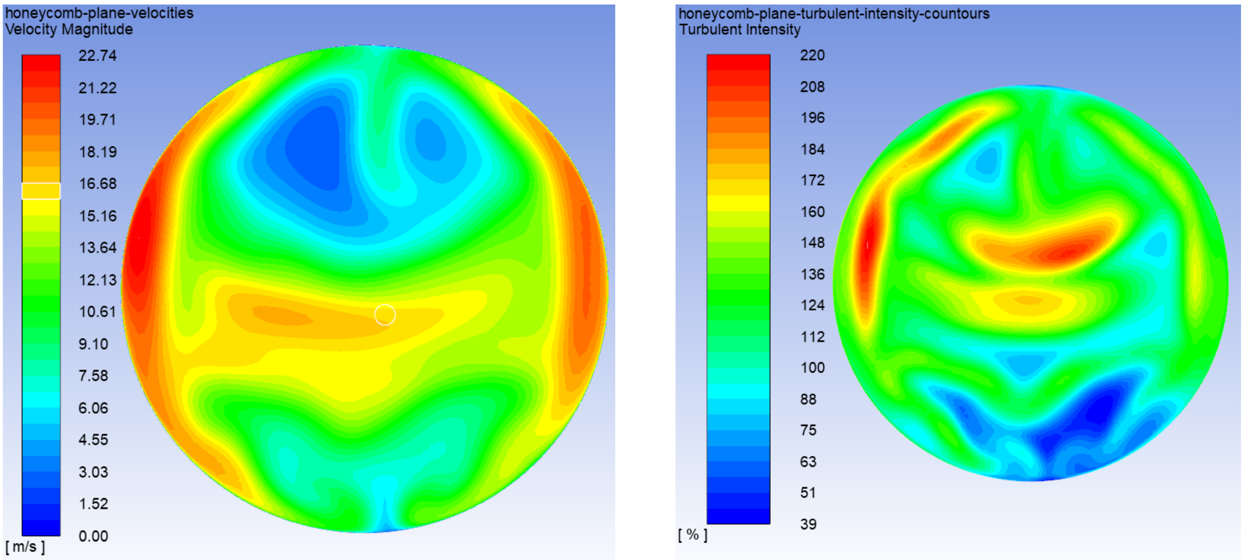


Figure 20 - Velocity and turbulence intensity contours at 1.5 m downstream plane, partial block BC

At the plane to set up honeycomb flow straightener, which is 1.5 m downstream of the IPCD, flow properties are satisfied pre-requisite condition by offering velocity uniformity of 86% and turbulent intensity of 1.3%.

In short, what differentiates the IPCD from the MCD is its aerodynamically stable feature during throttling action of the butterfly valve, damping any fluctuation induced. So, despite of having more complex process and expensive fabricating cost, the IPCD configuration is chosen as wide angle diffuser design.

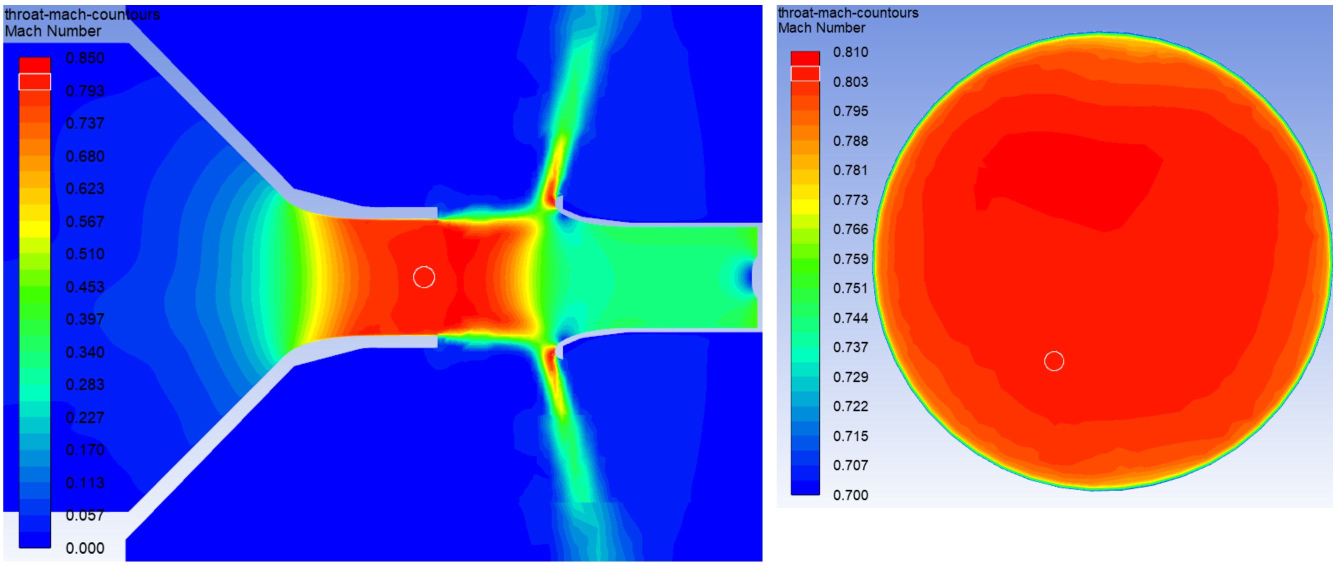


Figure 21 - Mach number contours in the exhaust section and its exit plane (final design)

With the final design configuration is fixed, even with the partial block inlet BC, flow behavior at the throat exit plane is satisfactory with velocity uniformity of order 99% without honeycomb flow straightener and turbulence screen.

6. Flow quality control devices

Simulating full plenum chamber design with honeycomb flow straightener and turbulence screen is out of the author's computing capability. Therefore, those devices were chosen based on practical "design rules" by others and their availability on the market.

6.1. Honeycomb flow straightener

Barlow [13] hinted that about 150 honeycomb cells per diameter are adequate. However, only 6 mm cell size (200 cells per plenum chamber diameter) and 10 mm cell size (120 cells per plenum chamber

diameter) are available. In order to avoid excessive frictional losses over honeycomb's wall on the smaller cell size design, the 10 mm cell size was chosen so 120 cells are abreast chamber diameter. Its length is of 80 mm as Barlow [13] also recommended the length-cell size ratio of 8 being preferred.
This honeycomb is constructed from 0.2 mm wall thickness stainless steel brazed, offering >70% opening area.

6.2. Turbulence screen

Scheiman [14] has conducted experiment to compare level of integral scale of turbulence produced by combination of honeycomb-screen and according to that, the combination of honeycomb and 28M mesh gave quite good performance over wide range of incoming velocity. Thus, the 28M mesh, which is widely popular on the market, is chosen as turbulence screen.
This 28M mesh has 28 mesh per inch, having wire diameter of 0.019 mm and open area of 62%.

7. Conclusions

The aerodynamic design process of the plenum chamber in the low-cost and limited space sea-level RAT Mach 0.8 system has been presented. All these components except for the honeycomb flow straightener and the turbulence screen were designed to be fabricated from steel plate formed with extensive welding applied, thus satisfying low-cost requirement.

Given limiting axial distance of less than 5.5 m in front of an engine for the system installation, special treatments for the pre-diffuser, the turning guide vanes and the inverse perforated cone diffuser were provided. CFD models and the special inlet boundary condition were setup to test system's behaviors in the most extreme working conditions and to estimate flow characteristics of plenum chamber accurately and reliably.

Flow's aerodynamic stability in the region upstream of honeycomb and uniformity at the throat exit plane are of prime consideration. The plenum chamber system has met these two requirements successfully. Mach 0.8 speed discharging into ambient air with uniformity better than 99% at the throat exit plane has been achieved.

References

[1] Grannan, N.D., Moosmann, K.J., Hoke, J.L., McClearn, M.J. and Schauer, F.R., 2018. Small Turbojet Altitude Test Facility with Two Stage Turbocharger Inlet Air Cooling. In *2018 AIAA Aerospace Sciences Meeting* (p. 2119).
[2] Kim, C., Yoon, M., Yang, S. and Lee, D., 2001. An altitude test facility for small jet engines. In *37th Joint Propulsion Conference and Exhibit* (p. 3680).
[3] Toot, L., 1981. *Cruise Missile Engine Program Contract Data Requirements List Sequence Number 95. Qualification Test Report. Volume XXXIX. Vibration and Mission Simulation Testing on Engine 828. XF-107-WR-400 Cruise Missile Engine*. WILLIAMS RESEARCH CORP WALLED LAKE MI.
[4] Pope, A. and Goin, K.L., 1978. High-speed wind tunnel testing.
[5] Bhatia, A., 2014. Control Valve Basics: Sizing and Selection. *Continuing Education and Development, Inc*, *9*.
[6] Lefebvre, A.H. and Ballal, D.R., 2010. *Gas turbine combustion: alternative fuels and emissions*. CRC press.
[7] Mitchell, J.G., Smith Jr, R.E., Osborn, A.R., Rudnitski, D.M. and Vleghert, J.P., 1990. *Comparative Engine Performance Measurements Held in Torino, Italy on 14-15th May 1990, London, United Kingdom, 17-18 May 1990, Montreal, Canada, 4-5th June 1990 and Monterey, California, 7-8th June 1990 (Mesures Comparatives des Performances des Moteurs)*. ADVISORY GROUP FOR AEROSPACE RESEARCH AND DEVELOPMENT NEUILLY-SUR-SEINE (FRANCE).
[8] Reardon, W.H. and Cifone, A.J., 1992. Development and qualification of the US Cruise Missile Propulsion System. *In AGARD*.
[9] Ferri, A. and Bogdonoff, S.M., 1954. *Design and operation of intermittent supersonic wind tunnels* (No. 1-3). North Atlantic Treaty Organization, Advisory Group for Aeronautical Research and Development.
[10] Ashwood, P.F., 1990. *Propulsion and Energetics Panel Working Group 15 on The Uniform Engine Test Programme*. ADVISORY GROUP FOR AEROSPACE RESEARCH AND DEVELOPMENT NEUILLY-SUR-SEINE (FRANCE).
[11] Johl, G., Passmore, M. and Render, P., 2007. Design and performance of thin, circular arc, wind-tunnel turning vanes. *The Aeronautical Journal*, *111*(1116), pp.115-118.
[12] Reed, J.A. and Hiers Jr, R.S., 1995. *Subscale Study of Engine Bellmouth Inlet Vortices in Test Cell R1D*. ARNOLD ENGINEERING DEVELOPMENT CENTER ARNOLD AFB TN.
[13] Barlow, J.B., Rae, W.H. and Pope, A., 1999. *Low-speed wind tunnel testing*. John wiley & sons.
[14] Scheiman, J. and Brooks, J.D., 1981. Comparison of experimental and theoretical turbulence reduction from screens, honeycomb, and honeycomb-screen combinations. *Journal of Aircraft*, *18*(8), pp.638-643.