DIFFUSER DESIGN FOR A SUPERSONIC/SUBSONIC MIXING CHAMBER

André Fernando de Castro da Silva, andre.fernando.t10@gmail.com
Departamento de Ciência e Tecnologia Aeroespacial (DCTA) **-** Instituto Tecnológico de Aeronáutica (ITA)
Pça. Marechal Eduardo Gomes, 50 – CEP: 12228-900, São José dos Campos, São Paulo

Marcos Aurélio Ortega, ortega@ita.br
Departamento de Ciência e Tecnologia Aeroespacial (DCTA) **-** Instituto Tecnológico de Aeronáutica (ITA)
Pça. Marechal Eduardo Gomes, 50 – CEP: 12228-900, São José dos Campos, São Paulo

João Batista Pessoa Falcão Filho, jb.falcao@ig.com.br
Departamento de Ciência e Tecnologia Aeroespacial (DCTA) Instituto de Aeronáutica e Espaço (IAE)
Divisão de Aerodinâmica (ALA), Pça. Marechal Eduardo Gomes, 50 – CEP: 12228-904 , São José dos Campos, São Paulo

Abstract. *The Pilot Transonic Wind Tunnel (PTT) is a modern installation, that can operate continuously by means of a main compressor, but also intermittenly with the assistance of an injection system which operates in a combined way with the main compressor. So the tunnel operation is a very complex issue, and many parameters must be set to obtain a perfect match. To get acquaintance with the tunnel operation and propose possible improvement modifications, a supersonic/subsonic mixing chamber was built with very similar design comparing with the one that already exists in the PTT´s aerodynamic circuit. This work presents the main steps of the mixing chamber diffuser´s design. For simplicity, it was chosen a flat walls diffuser, which allows divergence angle variation, although the diffuser in TTP circuit is conical. Reasoning for this choice is presented along with the main flow phenomena that take place during a diffuser operation. The four primary flow regimes that can occur during operation are discussed as they have a direct impact in the performance of stalled and unstalled diffusers. The final design parameters are confronted with other wind tunnel diffuser designs in a special graphic that registers the geometric configuration of well-designed diffusers according to specialized literature. The work also presents a CFD analysis of the design with a typical operational configuration. For which it was utilized a numerical code to solve the Navier-Stokes two-dimensional equations using RANS method and with the turbulent effects being accounted for through the use of the Spalart and Allmaras one-equation turbulence model. The main interesting flow parameters (Mach number and turbulent level) were analyzed for vertical longitudinal plane that passes through the diffuser center line.*

Keywords: *diffuser design, mixing chamber, numerical calculation, wind tunnel*

1. INTRODUCTION

The PTT (Pilot Transonic Tunnel) of IAE (Institute of Aeronautics and Space) is a modern installation, with a conventional closed circuit, continuously driven by a main compressor of 830 kW of power. The tunnel also operates in an intermittent mode by means of an injection system, which works together with the main compressor, for at least 30 seconds expanding the tunnel´s operational envelope with the same installed power. The injection system works by means of ten injectors in a choked operation at design Mach number 1.9. Its test section is 0.30 m wide and 0.25 m height with longitudinal slots. The tunnel has several sub-systems that, through acting in many control valves, automatically control pressure (from 0.5 bar to 1.25 bar), Mach number (from 0.2 to 1.3), temperature, and humidity, related to test section. Figure 1 shows a global view of its aerodynamic circuit with 17 m of extension, and its operational envelope, pointing out the region with the injection system operation. Additional technical details regarding PTT can be found in Falcão Filho and Mello (2002).

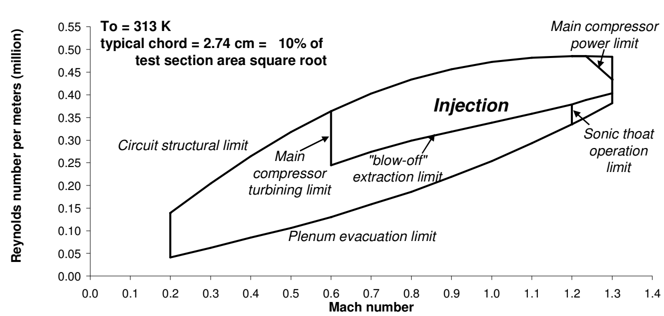


Figure 1. Aerodynamic circuit and the operational envelope of PTT.

The tunnel is an 1/8th scale down from an industrial transonic project, and it was initially designed to study the innovative features of the industrial facility, specifically concerned with the injection system operation in combination with the conventional main compressor operation. It was also designed for training the technical team in high speed tests, to perform basic research and academic research assessment, to produce tests in developing new aerodynamic transonic profiles, tests with simple geometry vehicles, qualitative tests of airplane basic configuration, anemometric tests, and others. With the aim of studying the injection system and proposing possible modifications without penalizing the tunnel´s campaigns schedule, it was developed the project and the construction of a smaller mixing chamber separated from the TTP´s circuit. Figure 2 shows the mixing chamber installation which resembles the TTP´s mixing chamber in some aspects. With one installed supersonic injector beak working at Mach number 1.9, the mixing chamber has the same length and is equipped with a flat diffuser with an equivalent angle of divergence compared to the TTP conical diffuser – herein, it is called a flat diffuser the one with four sharp corners, in a truncated square base pyramid geometry. The present work in a first point describes the main steps during the project of the flat diffuser to be used in the new mixing chamber. More technical details regarding the mixing chamber can be found in Silva *et al.* (2010).

Diffusers are one of the key components of a wind tunnel and other internal flow systems. They often play an important role in the flow performance because the entire flow is affected by their flow pattern and exit flow conditions. Since the pressure gradient opposes the flow direction, the boundary layer developed in diffuser walls decelerates and thickens rapidly. With the increase of the opening angle the flow can separate from the diffuser walls producing large unsteady eddies that characterizes the stalled flow condition. Some very important experimental works like of Kline *et al.* (1959), Reneau *et al.* (1967), Smith and Kline (1974), and Ashjaee and Johnson (1980), have studied the flow regimes, pressure recovery, and optimum geometry of two-dimensional diffusers. These experiments made it clear that the flow patterns are most affected by the opening angle of the walls and the length of the diffuser (Song *et al.*, 1991).

Figure 2. Mixing chamber construction separated from the aerodynamic circuit of TTP.

Since the flow through diffusers has boundary layers three-dimensional interactions with possibility of separation, it is a very complicated issue to be addressed. Also considering supersonic diffusers with shock waves interacting with boundary layers, even after many decades, their flow behavior is not well understood. It is important to note that diffuser is a device which converts kinetic energy into enthalpy regardless its geometric shape (Zucker, 1977). Diffuser efficiency is influenced by so many parameters, such as area ratio, inlet Mach number, inlet angle, length, shape, etc., that its design for a particular application must be based on empirical data and sometimes is more of an art than a science (Anderson, 1990). Rarely the first attempt done of a new diffuser design is ever completely successful. For subsonic applications, for example in wind tunnel, some simple rules applied to the diffuser opening angle are used as a guide to avoid flow separation and even data collected from other wind tunnel designs are plotted to prescribe a safe range (Eckert *et al.*, 1976).

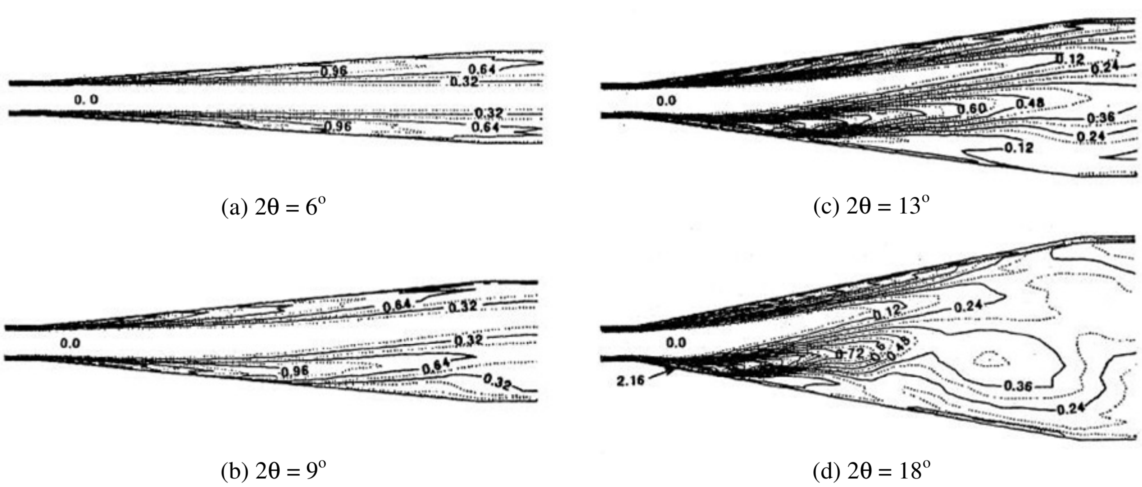
Although very spread in the literature for almost a century, nowadays diffuser design has called the attention of the scientific community because of new CFD tools capable of adequately investigating its performance. So, in addition to the diffuser design development, a numerical code was used to verify the diffuser performance and test it for different design parameters: inlet velocity level, cross section velocity profile (uniform and distorted), opening angle variation, etc. The diffuser performance was investigated with the help of a CFD code based on a finite difference scheme with second order centered spatial discretization to calculate the Navier-Stokes equations. The code make use of the implicit diagonal algorithm proposed by Pulliam and Chaussee (1981), and an artificial dissipation based on the spectral ratio (Pulliam, 1986, Mello, 1994) to better capture the shock passages. The turbulence effects are accounted for through the use of the Spalart and Allmaras (1992) one-equation scheme.

2. DIFFUSER DESIGN

The basic diffuser geometry is very simple and the reason for a long history of investigation by many researchers may be summarized by two-fold problem. The first one is related with the diffusion process itself for which there is a tendency of the boundary layers to separate from the diffuser walls if the rate of diffusion is too high. This results in large losses in stagnation pressure. On the other hand, the second problem appears if the rate of diffusion is too low. In this case the fluid flows through an excessive length of wall and wall fluid friction losses become predominant. So, an optimum rate of diffusion is expected in order to maintain the losses at minimum level. According to the results from many sources, this condition will be reached when the diffuser total opening angle is about seven degrees for both flat or conical wall diffusers. In fact, for flat wall diffuser this angle should be a little bit smaller. Eckert *et al.* (1976) establishes an opening angle for the conical diffuser about 0.5 to 1 degree smaller than that with flat walls.

Four primary flow regimes can occur in diffusers operation as they have been addressed by many researchers. The research interest in diffuser has been sustained for several decades because of its wide applications in industry and complex flow phenomena (He and Song, 1991). The performance of both stalled and unstalled diffusers in many cases are mapped for a wide range of geometries and inlet boundary layer thicknesses. A good understanding of the relationships between flow regime and performance leads to a rational basis for diffuser design (Reneau *et al.*, 1967). Kline *et al.* (1959) used empirically derived transformation of variables between the conical and two-dimensional geometries to correlate all available data for optimum recovery performance.

Based on preceding works, Song *et al.* (1991) classified the flow behavior in a diffuser in four categories as an approximated function of its opening angle: unstalled flow ($2\theta \leq 7^o$), transient stall ($2\theta = 8^o$ and 9^o), small transitory stall ($2\theta = 10^o$ to 14^o) and large transient stall ($2\theta \geq 16^o$). Figure 3 shows the vorticity field for these four classes of performance due to diffuser wall opening. Observe that in the occurrence of detachment, the stream core (denounced by a low vorticity value) bends to follow parallel to one of the side walls.



(a) $2\theta = 6^o$ (c) $2\theta = 13^o$

(b) $2\theta = 9^o$ (d) $2\theta = 18^o$

Figure 3. Vorticity contours field predicted numerically by He and Song (1991), for the four cases of performance due to diffuser wall opening.

In the unstalled flow regime (Fig. 3 (a)), there is no reverse flow throughout the diffuser and the velocity profiles are characterized as potential-like flow plus boundary layer region.

In the transient stall, for $2\theta = 8^o$ symmetric small stall regions start to develop near the exit – flow separation and reattachment occur near the exit on the walls. For $2\theta = 9^o$ the stall regions become asymmetrical – in some parts the stall extents beyond to the exit (Fig. 3 (b)). Within this narrow angle range, the flow changes from steady to unsteady, from symmetrical to asymmetrical, and from unstalled to stalled condition. The transient stall regime is defined as the regime from the first occurrence of symmetric separation bubble to the condition when one of the bubble extends beyond the diffuser exit, and the other separation bubble disappears.

In the transient stall regime, the flow pattern is characterized by the occurrence of flow separation only on one wall and almost periodical vortex shedding generated near the separation point – the single stall region grows larger as the opening angle is increased from 10^o to 14^o (Fig. 3 (c)).

As the opening angle reaches 16°, qualitative change occurs in the flow feature. The vortices shed from near the separation point no longer convect out of the diffuser directly, but they stay in the downstream portion of the diffuser for a few shedding periods. So, the large transitory stall regime is characterized by flow feature containing a short period vortex shedding up front of the stall region and a long period vortex movement in the downstream region. After one large eddy is washed out, a small vortex from the other wall boundary is often induced. As a result, regular vortex movement from both side walls accompanied by corresponding pressure fluctuations takes place (Fig. 3 (d)).

Based on these facts, one may say that diffusers are very sensitive to design errors which may cause flow separation. This separation can cause vibration, upstream oscillations that may increase the velocity disturbances, and higher losses. Ekcert *et al.* (1976) presents some practical results collected from many wind tunnel diffusers designs. These data are presented establishing operational safe limits for flat and for conical diffusers design. However, it is a very important fact that the actual effect of corners is unknown: they may alter the start of separation somewhat and only with very powerful tools (three dimensional CFD approach or precise experimental visualization) they may be addressed.

Figure 4 shows the recommended design region based on many existing wind tunnels designs: the conical stall line was obtained from McDonald and Fox (1964) and the flat stall line was from Reneau *et al.* (1967), as referred by Eckert *et al.* (1976). The upper portion of the design region is the recommended for flat diffusers with rounded corners, and the lower portion for diffusers with sharp corners. In figure, the equivalent radius is defined in function of the inlet cross sectional area of the diffuser (A):

$$R_e = \sqrt{A/\pi}\,. \tag{1}$$

Other wind tunnel diffuser designs are represented in the same figure. Observe that most of these wind tunnels were designed above the flat diffuser stall line but below the conical stall line. The present diffuser design, as shown already installed in Fig. 2, has rectangular inlet cross section 0.173 m wide and 0.228 m height with sharp corners, and with total length of 1.600 m, with outlet cross section of 0.290 m wide and 0.360 m height, resulting in an area ratio of 2.65 and a length to equivalent radius ratio of 14.3. The total opening angles were 6.70° in horizontal plane and 4.72° in vertical plane. As can be seen in figure, the present plotted diffuser design is conservative, below the stall flat diffuser line. At a first look, this fact indicates an expected unstalled flow condition but, on the other hand, with low performance (higher losses). The reason for choosing a conservative design point was that the present diffuser has some particular difficulties. The inlet flow coming from the mixing chamber may be distorted because of the supersonic jet action and this fact will be later experimentally addressed.

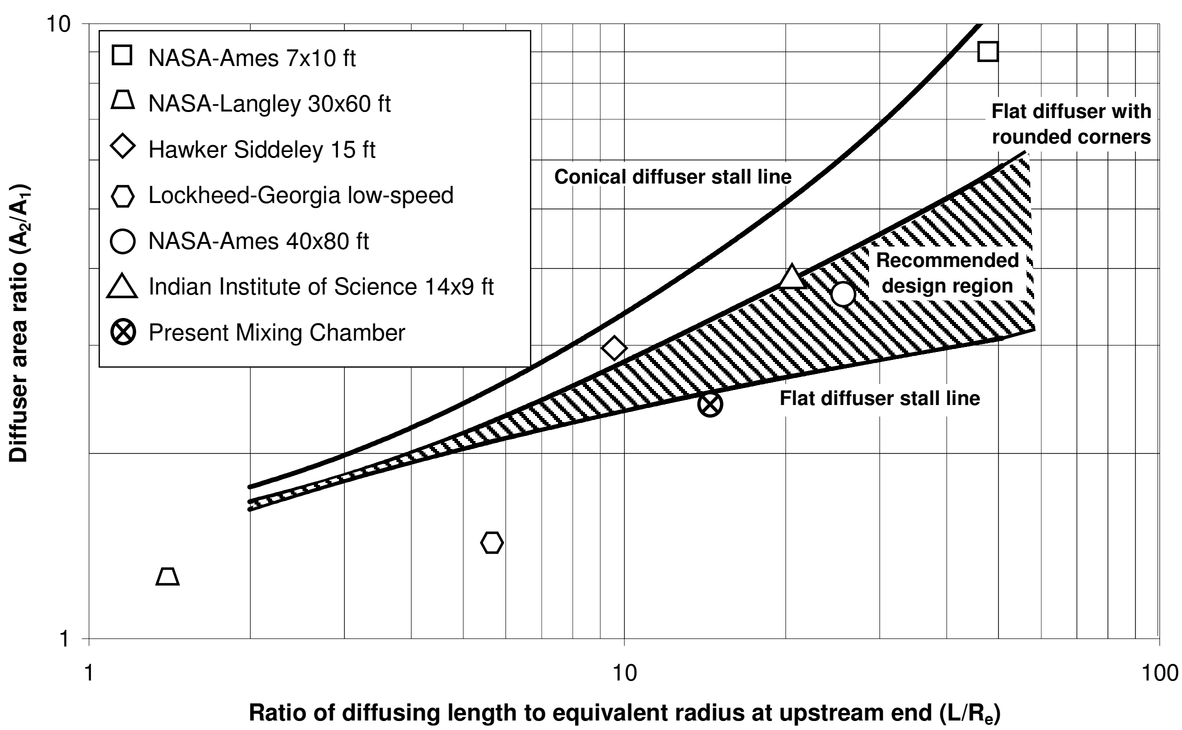


Figure 4. Recommended design region for flat and conical diffusers according to Eckert *et al.* (1976) – logarithmic scales.

3. NUMERICAL ANALYSIS

A numerical code was used to calculate the flow inside the diffuser. The mathematical model is represented by the Reynolds-Averaged Navier-Stokes (RANS) equations, written in two-dimensional generalized coordinates and conservation-law form. By "Navier-Stokes equations" we recognize the collection of the continuity, momentum, energy, and any constitutive equation necessary to represent the averaged flow. The medium is air considered as an isotropic and Newtonian fluid, and as a thermally and calorically perfect gas. The numerical algorithm follows closely the main lines of the finite-difference – second order accurate in space and first order in time –, diagonal scheme due to Pulliam and Chaussee (1981), complemented by a non-linear, spectral-radius-based artificial dissipation strategy due to Pulliam (1986). A one-equation turbulence model as suggested by Spalart and Allmaras (1992) was also aggregated to the system of equations to model the turbulence. For the numerical details concerning to the code utilized, treatment of the original equations, coordinates transformation, numerical implementation, turbulent model parameters etc., as presented by Falcão Filho and Ortega (2008).

The numerical code had passed through some verification and validation procedures to give confidence of its numerical results. For the sake of shortness, the details will be omitted here. It is worth noting that only a three dimensional analysis would be enough to represent the real flow condition in the diffuser, for example, the boundary layers interference in the corners, the complexity of the flow detachment that will occur first in the vertical plane (with bigger opening angle) while still attached in the horizontal plane, etc. However, a two-dimensional represents a first approach for the problem, which can confirm the opening angle choice and the global behavior of the diffuser.

Moreover, as could be seeing in the four cases commented in item 2, to perfectly capture the real physics in high opening angles of diffuser walls it should be necessary a transient simulation. However, a RANS approach will be useful to predict near optimum design situations, like the one in the present work.

3.1. Mesh and Boundary Conditions

The diffuser has a rectangular inlet cross section and the walls have different opening angles. The two-dimensional code was used to simulate the vertical plane, that passes in the central axis of the diffuser, with grid refinement close to the walls to better mimic the viscous effects. Figure 5 shows the scheme of a vertical plane passing at the central axis of the diffuser. The calculation region started 0.036 m before the diffuser inlet in order to allow accommodate the boundary layer imposed at the entrance and the simulation of the expansion corner at the diffuser inlet plane. The vertical plane was chosen to be shown because it represents the worst expected flow condition, with distorted velocity profile, although with smaller opening angle.

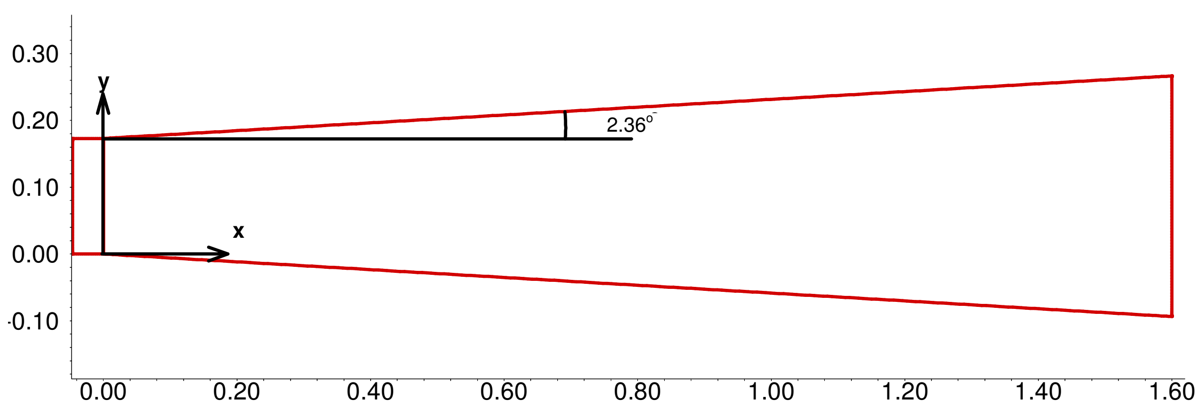
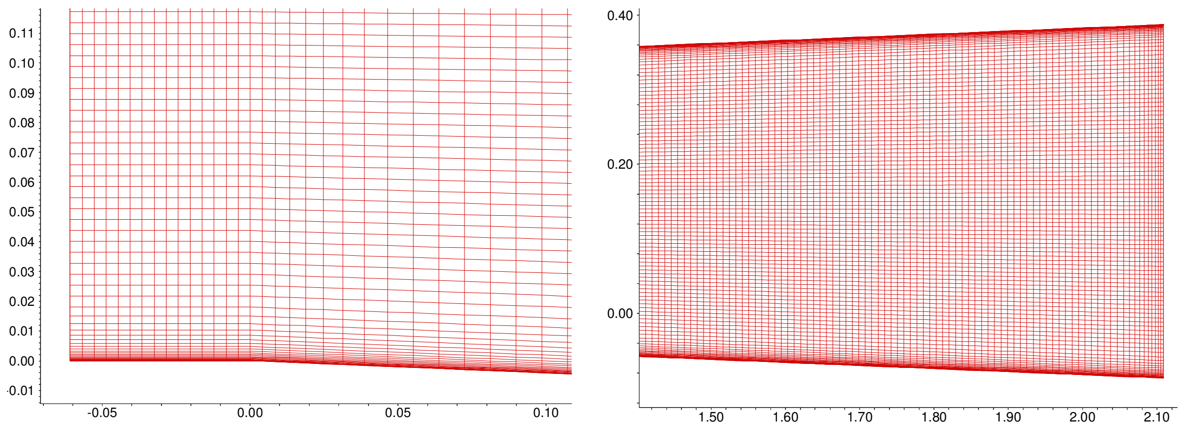


Figure 5. Scheme of vertical plane of the diffuser passing through the center axis, showing the origin of Coordinates. The initial region corresponds to the last 0.036 m of the mixing chamber.

Figure 6 shows some details of the mesh with 265 × 121 points used to simulate the flow in the diffuser. A grid refinement of 20% with a minimum displacement of 5×10^{-5} m was applied to the walls to better account for the turbulent boundary layer region influence (Fig. 6 (a)). Nevertheless, a reasonable grid refinement was applied to the whole mesh to capture regions of flow detachment (Fig. 6 (b)).



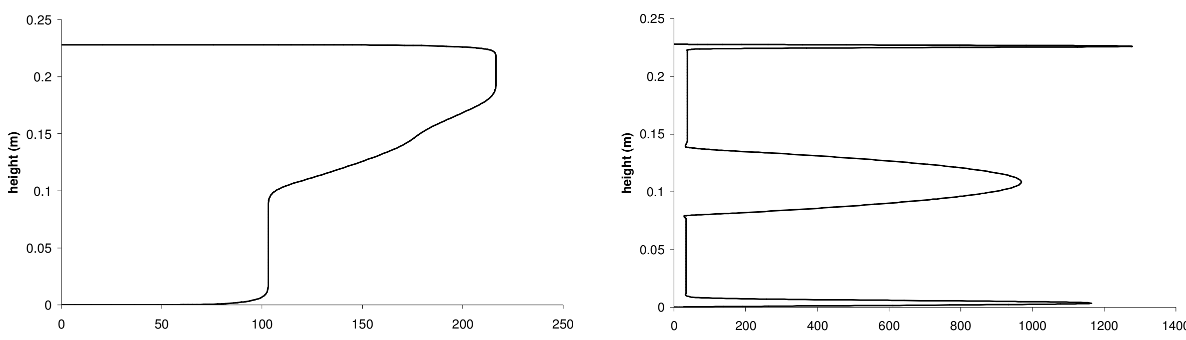
(a) (b)

Figure 6. Details of the diffuser mesh used, with 265 × 121 points: (a) lower part at the entrance, (b) final region.

Values obtained from preliminary experimental results obtained with Pitot probes in the mixing chamber installation were imposed as boundary conditions at the inlet plane. Figure 7 shows the velocity profile at the inlet plane of the diffuser (Fig. 7 (a)) and the turbulent boundary layer (Fig. 7 (b)) that reflects a typical turbulent kinetic energy profile, consistent with a global level for the potential-like region – note that these profiles have already some development from the imposed profiles at 0.036 m upstream.

At this point of the design no turbulent measurements was available, so the same strategy used by Falcão Filho and Ortega (2008) was used to settle the inlet conditions by considering typical situations. There are many details in the implementation, inspired in the law of the wall and in the profile, obtained from collected data by Saffman and Wilcox (1974) in his solution of the interaction of a turbulent boundary layer and an oblique shock wave. It would be too long to describe them all here, and the reader is advised to refer to Falcão Filho and Ortega (2008).

Observe in Fig. 7 (a) the two plateau-like regions determined by the projected flow from the injector that comes from the mixing chamber at the upper region, and the main flow that is being accelerated at the lower region, as obtained from experimental results. At the end of the mixing chamber the flow is still not uniform, and so, it is expected that the diffuser will have a tough task to decelerate the flow in a well behave condition – flow detachment is probable to happen. In Fig. 7 (b) it is clear the regions of high turbulent activity: the boundary layers at the diffuser walls and the mixing layer.



(a) (b)

Figure 7. Inlet profiles of Velocity (a) and turbulence (non-dimensionalized by reference molecular turbulence) (b).

At the outlet plane ambient pressure was imposed as the static pressure value and the other parameters were extrapolated by a zeroth-order scheme.

In the lower and upper frontiers there are the diffuser walls presence, for which non-slip condition were established. Besides that, adiabatic wall condition were established and pressure propagation through the boundary layer allowed to extrapolate pressure from the first calculation point to the wall.

3.2. Results

Figure 8 shows the Mach number field from the numerical simulation. It is interesting to observe that the high velocity flow looses its strength but does not vanish up to the end of the diffuser. The obtained result is very instructive as one can realize the upper boundary layer growth and the flow field.

It is interesting to note that the high velocity flow that comes from the injector was bent following the divergent wall, energizing the boundary layer, avoiding any possibility of detachment in it. So, the present situation is much more complex than that in the TTP´s diffuser, where there are injectors in the ceiling and in the floor of the mixing chamber, allowing the boundary layer on both walls to be energized. Observe also that the flow was not detached anywhere, as can be observed by the streamtrace arrows in the lower boundary layer. However, it is evident that the lower part of the diffuser experienced a high de-acceleration at the lower part (Mach number of 0.02). This fact shows that the diffuser is very close to experience a flow detachment at its lower wall.

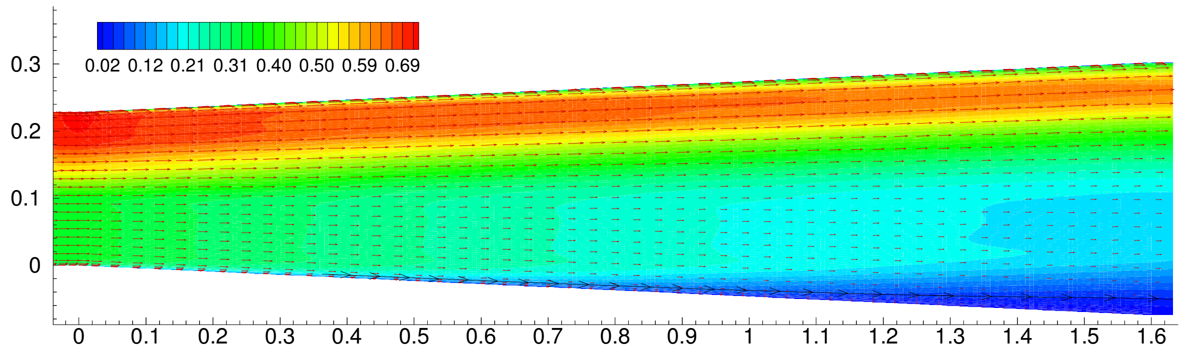


Figure 8. Mach number and velocity field, showing streamtracers in the lower boundary layer.

Figure 9 shows the viscosity field, non-dimensionalized by the reference laminar viscosity. It is impressive the growing activity of the mixing layer, starting with the value of 912 (see Fig. 7 (b)) and reaching 2190 at the diffuser outlet section. The turbulent activity is also notorious in the boundary layers, reaching levels of 810 and 1110 upper and lower, respectively. Observe that the great turbulent activity that showed up in the lower region of the diffuser was caused by diffusion in the extraordinary growth of the boundary layer. Typical non-dimensional turbulent level of 15 was imposed at the entrance in the core regions.

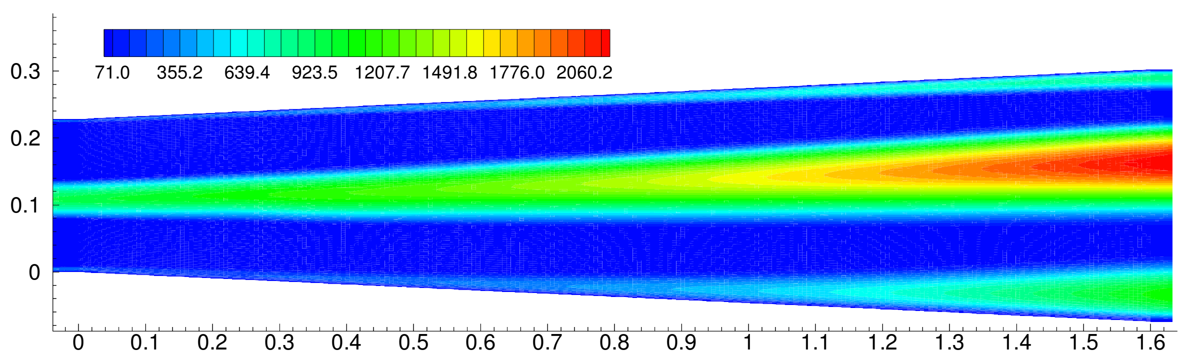


Figure 9. Non-dimensional viscosity field.

One of the most representative parameters in the theory of mixing layers is the growth rate, denoted by db/dx, where b is the mixing-layer thickness. Following Goebel and Dutton (1991), the mixing-layer thickness is defined as the distance between two transversal positions for which the mean streamwise velocity is equal to $(u_1 - 0.1\,\Delta u)$ and $(u_2 + 0.1\,\Delta u)$, being u_1 and u_2 the local mean streamwise velocities ($u_1 > u_2$) at a certain transverse section – as called here potential-like region or core region. Figure 10 shows the mixing layer growth in diffuser longitudinal direction starting at the diffuser inlet cross section. Observe that the mixing layer reaches the diffuser with thickness of 0.066 m and has a growth rate of db/dx = 0.023, which is comparable with the study cases addressed by Goebel and Dutton (1991), whose values varied from 0.020 to 0.059. Nevertheless, it is important to note that the present problem is very complex, in which mixing layer and diffusion phenomena interact with each other.

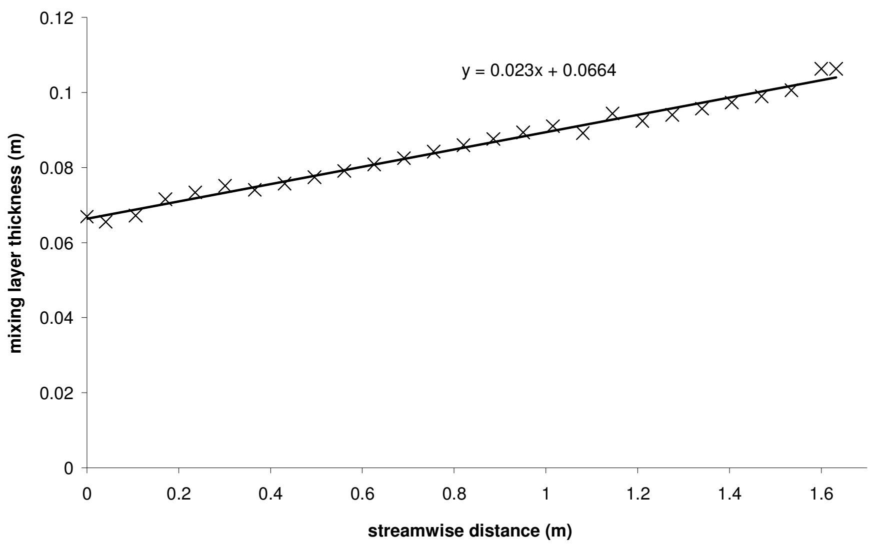


Figure 10. Mixing layer thickness growth along the streamwise diffuser direction.

4. CONCLUSIONS

A flat diffuser design for the supersonic/subsonic mixing chamber was presented. The main issues concerning it were applied to the real installation. Some very noticeable works were also investigated in order to describe the main phenomena in the diffuser operation. The four primary flow regimes that can occur in diffusers operation (unstalled flow, transient stall, small transitory stall and large transient stall) were described as they have direct impact in design.

The present flat diffuser design was plotted in graphic along and it was compared with some other diffuser design application in wind tunnels.

A numerical code based on RANS with Spalart and Allmaras turbulence model was used to model the flow inside the diffuser for a typical operational configuration. The numerical results is not yet concluded, and they already point to a better understanding of the physical phenomena inside the diffuser. An incipient detachment region appeared probably causing a poor diffuser performance. Other numerical cases will be verified in order to clarify the situation.

The work presents a design methodology for a flat diffuser, starting with the establishment of its global geometrical parameters until the final performance verification by using numerical simulation.

5. ACKNOWLEDGEMENTS

The authors would like to express their gratitude to CTA (Aerospace Technology and Science Department), IAE (Institute of Aeronautics and Space) and ITA (Technological Institute of Aeronautics) for the institutional support.

6. REFERENCES

Anderson, J. D. Jr., 1990, “Modern Compressible Flow – with historical perspective,” 2nd Edition, McGraw-Hill Publishing Company, 650 p.

Ashjaee, J., Johnston, J. P., 1980, “Straight-Walled, Two-Dimensional Diffuser – Transitory Stall and Peak Pressure Recovery,” Journal of Fluids Engineering, Trans. ASME, Vol. 102, No. 3, pp. 275-282.

Eckert, W. T., Mort, K. W., Jope, J., 1976, “Aerodynamic Design Guidelines and Computer Program for Estimation of Subsonic Wind Tunnel Performance,” Ames Research Center and U. S. Army Air Mobility R&D Laboratory Moffett Field, California, 94035, NASA TN-D-8243, Washington, D.C.

Falcão Filho, J. B. P., Mello, O. A. F., 2002, “Descrição Técnica do Túnel Transônico Piloto do Centro Técnico Aeroespacial,” Anais ... IX Congresso Brasileiro de Ciências Térmicas e Engenharia, ENCIT-2002, Caxambu-MG, artigo CIT02-0251.

Falcão Filho, J. B. P., Ortega, M. A., 2008, “Numerical Study of the Injection Process in a Transonic Wind Tunnel. The Numerical Details,” Computers and Fluids, Vol. 37, Issue 10, December 2008, pp. 1276-1308, available online in: www.sciencedirect.com. DOI: 10.1016/j.compfluid.2007.10.015.

Goebel, S. G., Dutton, J. C., 1991, “Experimental Study of Compressible Turbulent Mixing Layers,” AIAA Journal, Vol. 29, no. 4, pp. 538-546.

He, J., Song, C. C. S., 1991, "Numerical Simulation and Visualization of Two-Dimensional Diffuser Flows," ASME Technical Paper 330, Series A, pp. 355-361.

Kline, S. J., Abbott, D. E., Fox, R. W., 1959, "Optimum Design of Straight-Walled Diffusers," Journal of Basic Engineering, Trans. ASME, pp. 321-331.

McDonald, A. T., Fox, R. W., 1964, "Incompressible Flow in Conical Diffusers," Tech. Rep. No. 1, Army Research Office (Durham), Project No. 4332 (Available from Armed Services Technical Information Agency, U. S. Department of Defense).

Mello, O. A. F., 1994, "An Improved Hybrid Navier-Stokes: Full-Potential Method for Computation of Unsteady Compressible Viscous Flows", PhD Dissertation, Georgia Institute of Technology, Atlanta.

Pulliam, T. H., 1986, "Artificial Dissipation Models for the Euler Equations", AIAA Journal, Vol. 24, nº12, pp. 1931-1940.

Pulliam, T. H., Chaussee, D. S., 1981, "A Diagonal Form of an Implicit Aproximate-Factorization Algorithm", Journal of Computational Physics, Vol. 39, pp. 347-363.

Reneau, L. R., Johnston, J. P., Kline, S. J., 1967, "Performance and Design of Straight, Two-Dimensional Diffusers," Trans. ASME – Journal of Basic Engineering, Vol. 89, pp. 141-150.

Saffman, P. G., Wilcox, D. C., 1974, "Turbulence-Model Predictions for Turbulent Boundary Layers," AIAA Journal, Vol. 12, No. 4, pp. 541-546.

Silva, A. F. C., Braz, R. O., Rocha, A., Falcão Filho, J. B. P., 2010, "Descrição de Câmara de Mistura Supersônica/Subsônica e Primeiros Resultados," Proceedings of VI National Congress of Mechanical Engineering (accepted for publication).

Smith, C. R., Kline, S. J., 1974, "An experimental Investigation of the Transitory Stall Regime in Two-Dimensional Diffuser Including the Effects of Periodically Disturbed Inlet Conditions," Journal of Fluids Engineering, Trans. ASME, Series 1, Vol. 96, No. 1, pp. 11-15.

Song, C. C. S., He, J., Larson, E., 1991, "Numerical Simulation of Flow Regimes in Two-Dimensional Diffusers," University of Minnesota Supercomputer Institute Research Report UMSI 91/78, Minneapolis, Minnesota.

Spalart P. R., Allmaras, S. R., 1992, "A One-Equation Turbulent Model for Aerodynamic Flows." AIAA Paper 92-0439.

Zucker, R. D., 1977, "Fundamentals of Gas Dynamics," Matrix Publishers Inc., Beaverton, Oregon – USA, 443 p.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.