International Journal of Natural and Engineering Sciences 6 (3) : 87-94, 2012
ISSN: 1307-1149, E-ISSN: 2146-0086, www.nobel.gen.tr

Investigation of Total Pressure Distribution at Aerodynamic Interface Plane of an "S-shaped" Air Intake at Sideslip Condition

S. M. Nima SHOJAEE* Hooman Amiri HAZAVEH

Department of Mechanical and Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, IRAN

***Corresponding Author**
e-mail: nimashojaee@srbiau.ac.ir

Received : April 29, 2012
Accepted : June 02, 2012

Abstract

An appropriate performance of an air intake is a critical parameter for the best flight of an aircraft using a jet engine. Since then it is expected to ensure of this performance clearly, otherwise, the engine could probably have difficulties in breathing due to lack of proper functioning of the air intake. An Unmanned aerial vehicle - like any other aircraft – takes over a variety of flight regimes, since then the analysis of intake performance should be done in each condition. Due to the necessity of utilizing a jet engine in some of UAVs, it has been set to investigate the performance of an "S-shaped" air intake in the subsonic flight regime using numerical methods. Regarding to the lack of a professional experimental apparatus and the high level of knowledge required in the experiments the results then have to be compared and to be validated with experimental and numerical data obtained in previous similar efforts accomplished by aerodynamic and propulsion scientists all over the world.

Keywords: Intake Aerodynamics, S-shaped air intake, Compressor face pressure distribution, Sideslip condition.

INTRODUCTION

Air intake is one of the most essential parts of a jet propulsion system in an air vehicle. It has the responsibility of guiding the uniform free stream through the ducts and delivering it with the most effective and suitable velocity distribution to the engine face while converting kinetic energy to static pressure. Appropriate and optimized design of an air intake could affect on jet engine performance insofar as an inappropriate design and manufacturing could have possibly led to whirling stall of compressor by the turbulence and fluctuation previously generated. Design affiliations and manufacturers of air vehicles with jet engines, use experts and researchers of design, optimization and manufacturing of air intakes all over the world. Nearly since 1980s remarkable advancements in the design, optimization and production process of a jet engine air intake has been performed, and the internal aerodynamics of the air intake has been developed.

There are two main categories of air intakes: subsonic and supersonic. On both flight conditions, each air intake has particular characteristics that would be altered it to act properly in that flight regime. Geometrically, there will be various shapes of air intakes exist: Straight, "S-shaped", Serpentine and, etc. The geometry and characteristic of air intake depend significantly on the mission, and designer provisions.

The pressure distortion in the compressor face also is needed to reach a uniform condition that is the margin of engine tolerance. These distortion changes in various flight regimes and conditions, hence the most critical conditions should be spotted and applied to the calculations of air intake as a substantial parameter. Therefore, the entire system should be designed by a way that the supplied air enters the compressor with a minimum total pressure loss and maximum flow uniformity. These characteristics quietly depend on the shape and the size of the air inlet, and it s diffuser.

THEORY

In flight, the function of an air inlet is almost like a compressor. In fact, the air inlet bleeds the free stream air and supplies it to the compressor with appropriate properties. Several parameters have been introduced about the air intake. In this study, the most important intake parameter that is the total pressure loss – or total pressure recovery – and total pressure distortion are being discussed.

Aerodynamic Interface Plane

Experimentally, it is almost difficult to measure flow parameters at a compressor face when the engine is running. "The aerodynamic interface plane (AIP) is a plane forward of the compressor face but sufficiently close to the compressor face to have a very similar flow field" [1]. Hence, it is normal to allocate flow properties to this plane instead of compressor face.

Total Pressure Recovery

In general, any duct has its specific friction and loss. The pressure loss is defined as the ratio of pressure difference through the duct to the dynamic pressure [2]:

$$Pressure\ Loss = \frac{\Delta P}{q} = \frac{P_i - P_f}{q} \qquad (1)$$

Where, is the mean total pressure at AIP, is the total pressure at inlet and is the dynamic pressure.

Based on the definition above, total pressure recovery can be defined as below [2]:

$$\eta = 1 - \frac{\Delta P}{q} = 1 - \frac{P_i - P_f}{q} \qquad (2)$$

The maximum allowed pressure loss for an air intake is reported only smaller than 0.02 in many sources. Thus, a well designed air intake should have a total pressure recovery of 0.98 and more [2].

Total Pressure Distortion

One of the main tasks of the air inlet is to deliver air uniformly to the compressor blades. This uniformity consists of total pressure, static pressure, total temperature, or a combination of these parameters. If one of these parameters becomes non-uniform, distortion is said to be occurred. Furthermore, if the flow passing the duct has an angle with the engine longitudinal axis, it is said that another distortion is occurred called swirl [3]. In this study, only total pressure distortion has been investigated at the AIP of an "S-shaped" subsonic air intake.

There are two types of total pressure distortion:

A) Steady distortion. Steady distortion is defined as any non-uniformity of total pressure distribution in any section of the air intake. There is always a radial non-uniformity of pressure in sections of the duct because there is a boundary layer on the walls due to the viscosity of the air even in the absence of flow separation. It is common to neglect radial distortion in computations but to consider circumferential distortion because the aerodynamic loading on the compressor blades is on the circumference direction.

B) Dynamic distortion. If the total pressure distortion at AIP changes with time or if is of a kind of spatially non-uniform, it is called dynamic distortion. This type of distortion was first discovered by European scientists.

Three kinds of distortion have been defined by the jet engine manufacturers yet; DC(θ), KDA, and IDC-IDR [3]. In this study only the first definition is used. DC(θ) has been defined by Rolls-Royce Company in TORNADO (1981) and Euro-Fighter (2000) projects and is defined as below [3]:

$$DC\theta = \frac{\overline{P_{t_{AIP}}} - \overline{P_{t_\theta}}}{q} \qquad (3)$$

Where, is the mean total pressure at AIP, is the minimum total pressure of all sectors of θ degrees extent, and is the mean dynamic pressure at AIP. The value of critical θ that is proportional to total pressure depends on the engine design and is generally 60, 90, or 120 degrees (Figure 1) [3].

METHOD

Air Intake Model Description

It is well seen that this air intake has heterogeneous geometrical sections along its structure (Figure 1). The duct length is about 52 cm and outlet cross section diameter is 12.6 cm. The contraction ratio (the ratio of Inlet and outlet cross sectional areas) are 0.434 to 1. This air intake model has been constructed with composite materials and designed in a way that has a capability of supplying small turbojet engines with a mass flow up to 0.2 kg/sec favorably.

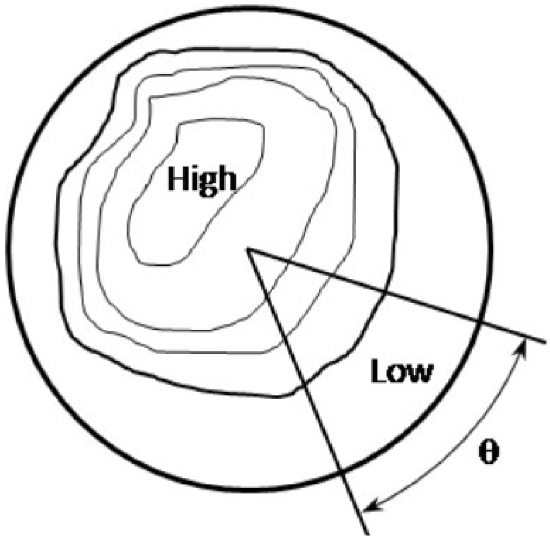


Fig.1. Illustration of total pressure contours to define distortion coefficient [3].

The main model was imported as IGS standard format into GAMBIT. Some preliminary adaptations were made on the model to be suited to the software. Due to some geometric limitations arise from importing all vertexes, faces and volumes are in virtual type. The geometry of inlet and outlet zones was determined from prior 2-D considerations. The main problem of generating a suitable 3-D domain was to get a structured grid in the entire domain. Furthermore it was required to correct some highly skewed cells to avoid wrong calculations. The final computational model had no skewness and was a distinct among all available models.

Physical Domain and Flight Conditions

A typical maneuver envelope for aircraft at subsonic and supersonic speeds is shown below. Based on this fact and the availability of flight condition details of the desired UAV and some cost issues, the following table determines the cases that would be simulated.

Flow Conditions

According to Table-1 it is obvious that the maximum velocity of air supplied to the intake is 40 m/s (Mach 0.12 at sea level). Density variation is about 0.1% in this velocity, thus it could be assumed that all analyses were done in incompressible

Table.1. Predicted Flight Conditions for the Conceptual UAV.

	Flight Regime	Velocity (m/s)	Angle of Attack (degrees)	Angle of Sideslip (degrees)
1	Negative Stall	25	-4	0, 5, 10, 20, 30, 40
2	Cruise	40	0	0, 5, 10, 20, 30, 40
3	Shallow Climb	31.5	+4	0, 5, 10, 20, 30, 40
4	Steep Climb	29.5	+10	0, 5, 10, 20, 30, 40
5	Positive Stall	27	+14	0, 5, 10, 20, 30, 40

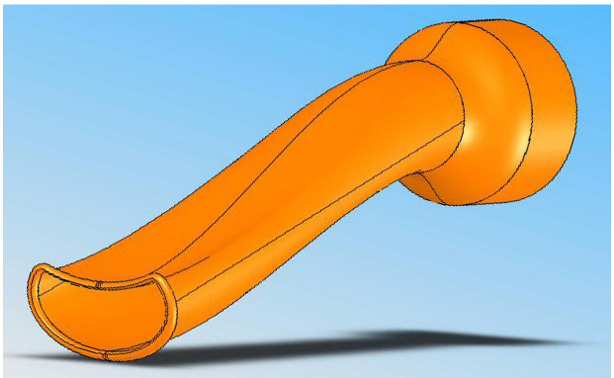


Fig.2. Schematic of the air intake.

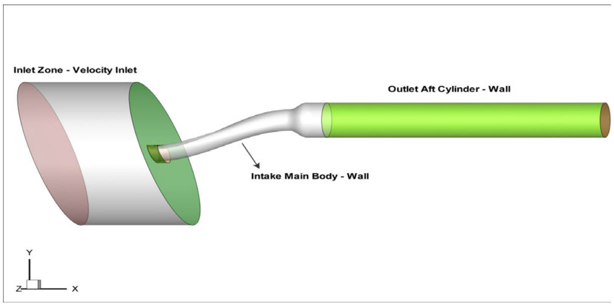


Fig.3. Final 3-D model and boundary definition.

condition. Accordingly, Navier-Stokes equations must be solved in subsonic, steady, incompressible, viscous and turbulent condition whereas the air as an operating fluid

Solver Model

Assuming the minimum circular area of 3 cm diameter, minimum density of 1 kg/m^3, minimum velocity of 25 m/s and average viscosity of air of 1.7×10^{-5} kg/ms^2 the equivalent Reynolds number is 44000. This is the number that ascertains the flow in the duct is fully turbulent.

FLUENT has a lot of predefined turbulence models. Choosing the best turbulent model is very significant in the process of solution since the appropriate model leads to more realistic answers. With some background of prior researches [1], [4], [5], [6], [7], and [8] the appropriate turbulence model was chosen to be k-ω. This model is a good one in FLUENT to predict flow behavior in problems with the fluid passing the rough walls or with high pressure and/or velocity gradients due to geometric variations in model.

Boundary Conditions

The zone forward to the lip is determined as a velocity inlet. Intake body and lip are described as walls and the cylindrical zone which stands after the body has a wall around and pressure outlet at the rear (Figure 3). The inlet geometry was selected a way that either satisfies the incident angles of attack and sideslip or preserves the space of computational domain for the premier zones such as near wall and AIP surfaces. The outlet aft cylinder geometry was chosen through 2-D analysis so that the reverse flow problem would not occur meanwhile the solution process.

Grid Generation

A structured grid was applied to the entire geometry. A number of about 500K cells were obtained after generating the grid. In this model, only two quad meshes were applied so that in the outlet aft cylinder, a circular zone has an inner pave and outer mapped mesh type as sees in Figure 4. This scheme continues as a copper method to the inlet zone as seen in the Figure 5.

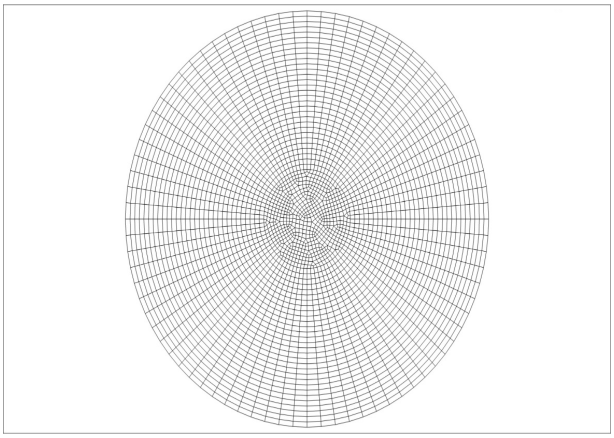


Fig.4. Quad mesh of outlet zone with the inner Pave and outer Mapped grid.

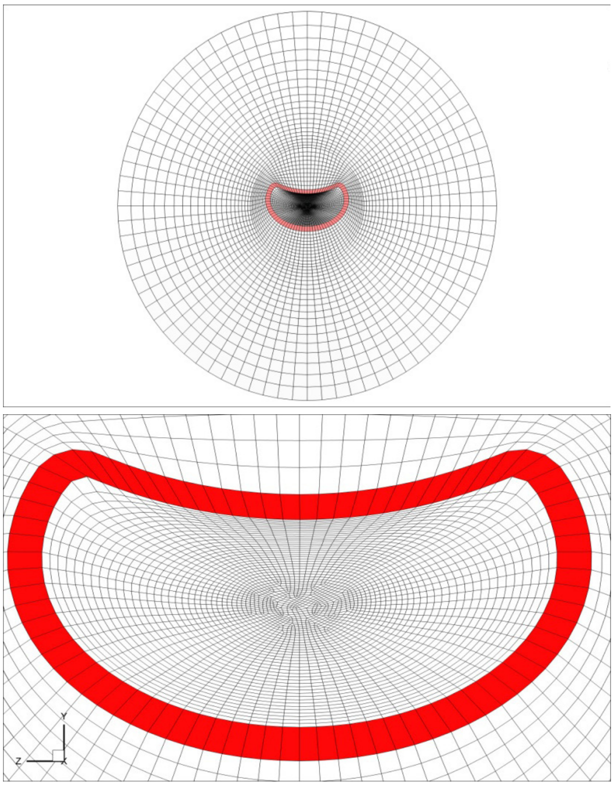


Fig.5. Quad mesh of Pave-Map of inlet zone.

Numerical Parameters Definition

Numerical Parameters defined in this research were operating condition, wall roughness and under relaxation parameters. Operating pressure was selected exactly equal to the pressure of the experimental condition. Composite material used to build ducts, made walls with a roughness of 1×10^{-4}m based on the surveys. This magnitude was applied on the wall roughness setting. For the proper convergence and to minimize convulsions of the residual graph, under relaxation parameter was set to nearly half of their defaults.

Solution

With the assumption of steady and incompressible flow and the fact that the energy equation is no longer used, using a personal computer seems to be sufficient. This PC is supplied with a 2.6 GHz Quad core Intel processor and 3 GB of RAM. All solution procedure consists of overall 30 cases and takes a month to be solved on this machine. Governing equations are solved using FLUENT segregated algorithm. The SIMPLE algorithm is either used to correlate pressure and velocity fields. The upwind scheme is used to relate grid points. This scheme essentially advances the solution to the convergence and it has been approved that this scheme is the best choice on the discretization of convection term in Navier-Stokes equations [9].

It is worth mentioning that in case of having faster and more powerful computer, using second-order schemes such as second-order upstream difference scheme (QUICK) or second-order upwind scheme is recommended to create a better relationship between grid points

Convergence Criteria

In all numerical analysis, a criterion for stopping iterations must be taken. This measure is called the convergence criteria. Since the shrinking value of the difference in mass (residuals in continuity equation) in the governing equations of fluid flow, is not a sufficient condition for convergence, another strategy was devised. That is to consider mass flow rate in the duct at various sections including the inlet edge, the second throat, settling chamber at the entrance to the outlet conduit and the conduit surface, during the iterations. When the mass flow rate of the four major sections remained unchanged and the difference in total mass flow (residual in continuity equation) is less than 1×10^{-5}, the equations are assumed to satisfy the convergence condition and the iteration has been suspended. To achieve this purpose, at least 4000 iterations and a maximum of 11000 iterations for each case -an air inlet on each flight regime (Table 1) - have been conducted.

Apart from the continuity residual values, residuals of velocity in all three directions (Cartesian equations of motion in three axes), "k" (turbulent kinetic energy) and "ω" (specific dissipation rate) are set to be less than 1×10^{-5} to be sure of solution process.

Solution Independency

One of the main arguments in numerical fluid dynamics is the solution independency. Although in the grid generation process, it was determined to get the maximum capabilities of the personal computer, But by reducing the grid down to about five hundred thousand cells, the numerical results did not show remarkable changes in the overall behavior of the flow. It should be recognized that reducing the grid cells, leads to some inconvenience to simulate the boundary layer within the duct walls, and create some problems of capturing swirl within the duct to some extent.

A number of 4 models with different number of grids were used to get the optimum grid that can handle both major flow characteristics and not to create some large round of errors. These models have 3×10^5, 5×10^5, 8×10^5, and 12×10^5 elements respectively. The best result is obtained from the second model. The first model simulates the flow in non-physical form near the walls and the fourth model had high round of error and a large CPU time that could not be performed by a single Personal Computer.

RESULTS

Based on the results obtained from numerical analysis, the following contours and charts are derived.

Total Pressure Loss at AIP

Mean total pressure loss and a comparison between these losses in different flight conditions are as followed (Figures 6 and 7).

Total Pressure Distortion at AIP

Total pressure distortion is demonstrated in Figure 8.

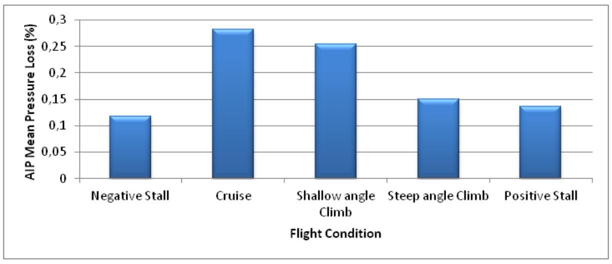


Fig.6. Mean total pressure loss at AIP in different flight conditions.

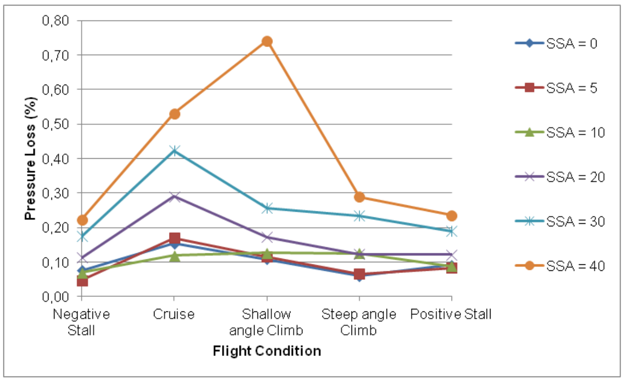


Fig.7. Total pressure loss at AIP in different flight conditions at Sideslip angles.

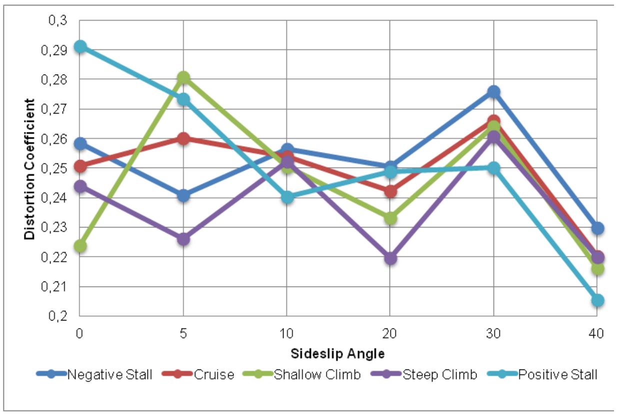


Fig.8. Distortion Coefficient at different flight conditions.

Total Pressure Distribution at AIP

Following tables (Table 2to 6) show the total pressure distribution at the aerodynamic interface plane of the duct at various flight conditions and side slip angles.

VALIDATION

Comparing the results with previous activities, researches and efforts done by other scientist and researchers will increase the validity of the numerical simulations accomplished. It should be noted that aerodynamic behavior of an air intake depends completely on its geometry. Namely, with any alteration in cross sectional area through the duct, internal aerodynamic behavior and performance parameters are changed. Reference materials that have been studied include many different types and shapes of bending ducts and because of that the comparison may be difficult in some aspects.

In reference [10] total pressure loss of royal M-2129 "S-shaped" duct near Mach 0.2 is reported about 0.1%. Total pressure recovery is listed between 0.91 and 0.97 in reference [11]. There is a report in reference [12] with a pressure loss values between 0.0081 and 0.0228 using different numerical and experimental methods. Similarly in reference [13], duct pressure loss in low speeds for a serpentine intake is about 0.1% using computational methods. Results obtained from studies on a single bent duct with a nominal curvature [14] in India, shows a magnitude of 0.3% to 0.4% of pressure loss. In the present study, numerical analysis show that the loss of total pressure is between 0.02% and 0.74%. These numbers are completely valid and acceptable considering what was mentioned so far.

Reviewing the existing literature especially references [10], [15], and [16], it is seen that the maximum non-dimensional total pressure at AIP is always bigger than 0.9 as well as the results obtained in this study.

In case of distortion coefficient, references [10], [11], and [12] report a value of total pressure distortion coefficient of 0.22, 0.1, and between 0.18 and 0.55 respectively. With regards to the values obtained in the present study (Figure 8), it could be declared that the values have a comparative consistency with reported values. In some references, there is a background of design and usage of vortex generator devices to decrease distortion. It is suggested that the effects of using these tools would be considered in the future studies.

Total pressure contours in references [10], [11], [15], [16], and [17] show an acceptable similarity to the contours derived in the present work (Tables 2 to 6).

DISCUSSION

An overview of the results obtained from numerical analysis, gives the following points discussable:

There is a left-right asymmetry in the contours of total pressure, even at zero side slip angle. This asymmetry indicates the presence of swirling flow particularly at the bends and also shows 3-D physical behavior of this phenomenon and the fact that 2-D simulations are not acceptable in order to extend to 3-D case.

Contours of total pressure show that the desired region of pressure (high-pressure area) tends to be at the upper half and is concentrated almost in the center of the AIP in all flight regimes. In the other side, low-pressure region of this section is related to its lower half. Both low and high-pressure zones are asymmetric due to the flow jumps on the first and second bends when passing through the duct.

Table.2. Total pressure distribution at AIP in negative stall condition.

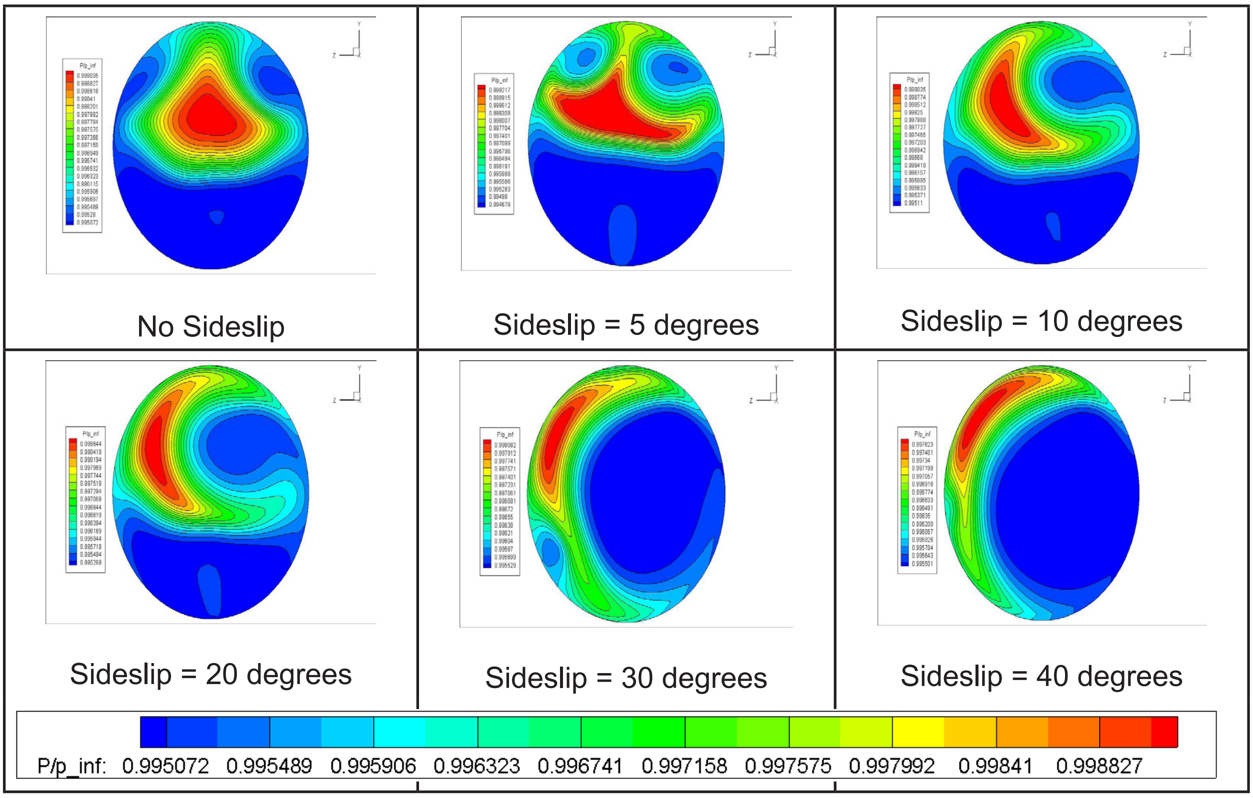


Table.3. Total pressure distribution at AIP in cruise condition.

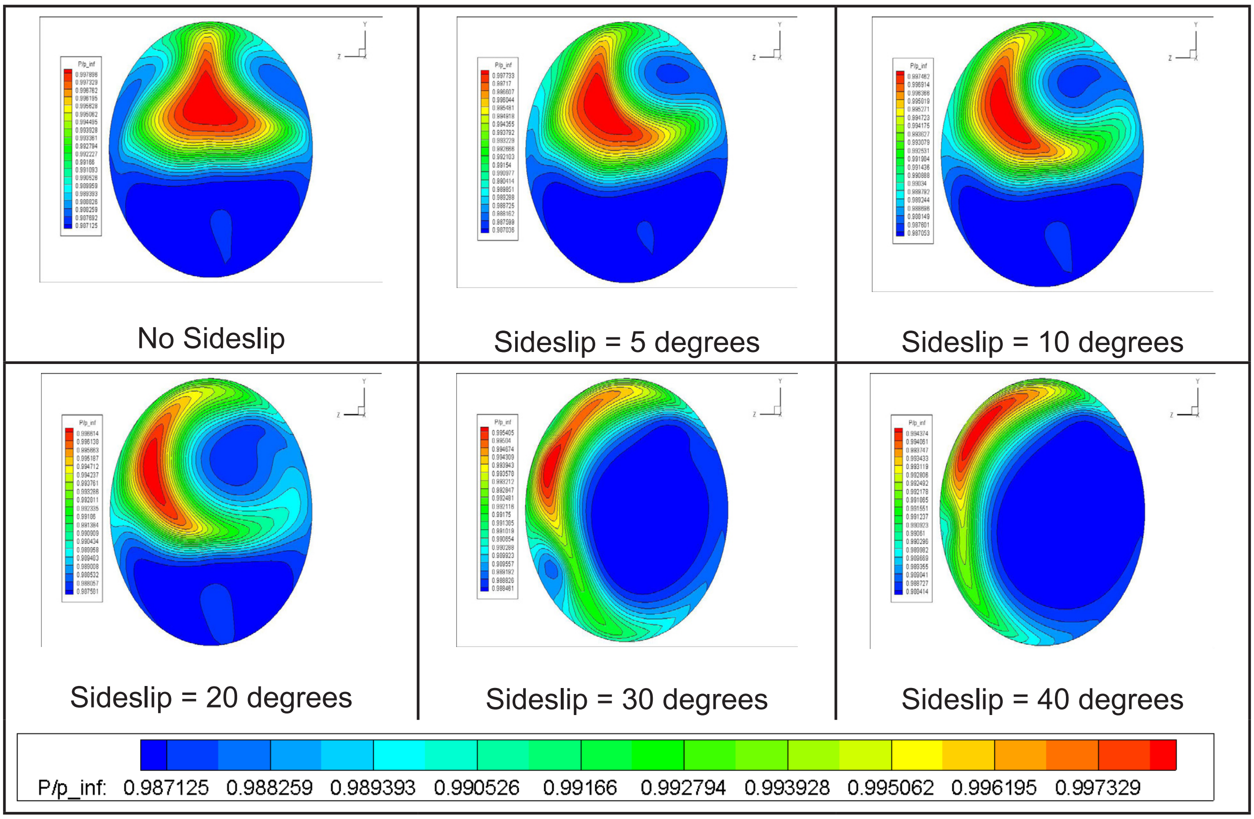


Table.4. Total pressure distribution at AIP in shallow angle climb condition.

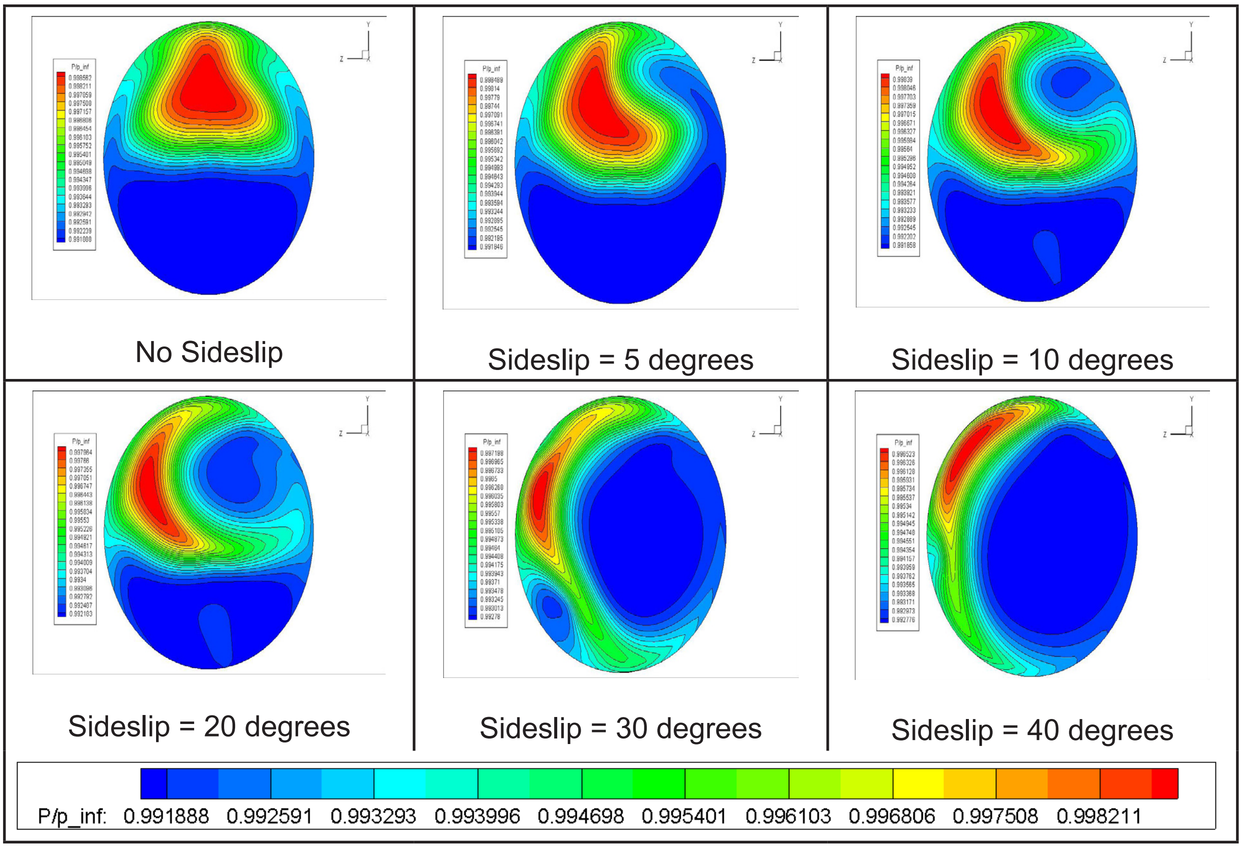


Table.5. Total pressure distribution at AIP in steep angle climb condition.

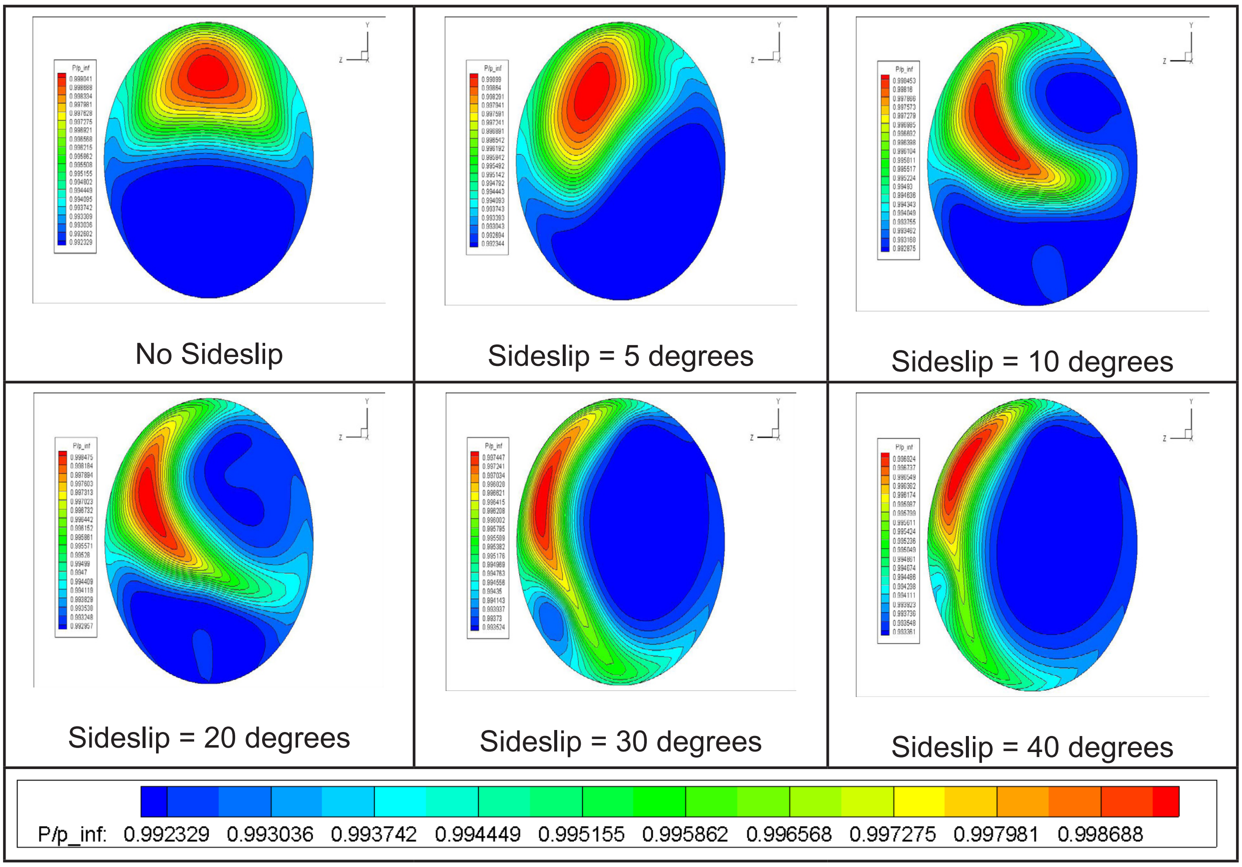
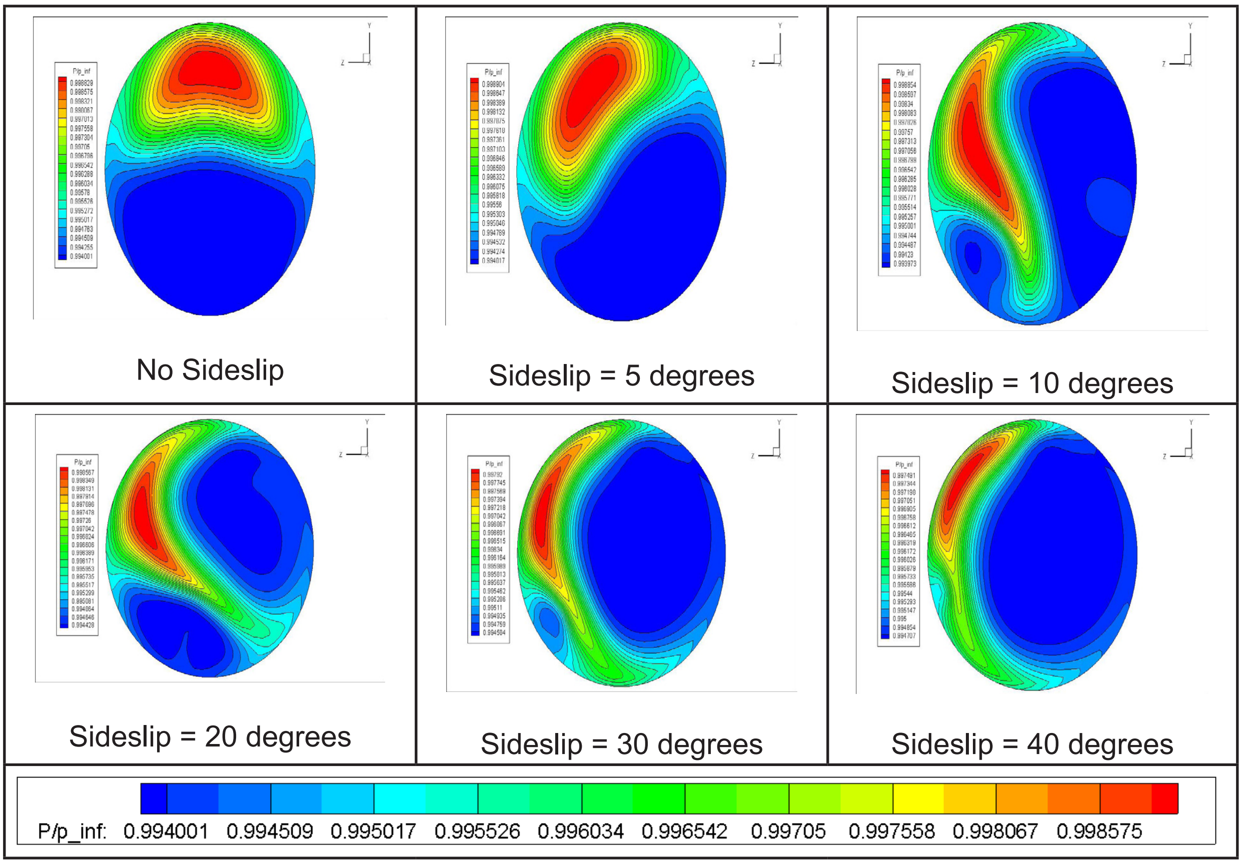


Table.6. Total pressure distribution at AIP in positive stall condition.



The obtained values specified in Figure 7, total pressure recovery is a bit sensitive to side slip angle changes. Therefore it is concluded that this air intake is properly designed in terms of performance at side slip condition.

In the graphs of the pressure distortion, Apart from velocity, angle of attack and side slip angle changes are factors affecting the amount of distortion. Therefore these results could not be considered as absolute results. However, it is clear that the greatest amount of total pressure distortion in AIP is seen at positive stall condition. It can be said in explanation that as the uniformity of flow entering the duct directly influences the total pressure distortion; entering of the turbulent boundary layer to the duct and the effect of angle of attack, eliminates the desired effects of low speed in positive stall condition. Therefore the distortion coefficient of the AIP section is considerably increased in this flight regime. A same condition is seen at cruise flight where high speed state eliminates the desired effect of zero angle of attack and reduces the uniformity of flow entering the duct and totally increases the pressure distortion at AIP again.

The mean distortion coefficient at AIP in all flight regimes is between 0.2 and 0.3 as seen on Figure 8. This magnitude is not desirable anyway. While the average distortion coefficient at the AIP of an air duct, is recommended to be in the range of 0.05 and 0.1 by some jet engine manufacturers and it seems very ideal, but it should be noted that the distortion values obtained in this study are far beyond the acceptable range. Therefore, measures should be devised to reduce this undesirable distortion values.

Some interesting points are obtained considering Figure 7. Total pressure loss is between 0.02% and 0.74% that is very desirable due to the allowed maximum loss referred by [2]. Also, maximum total pressure loss is occurred at cruise flight, and this is true due to the fact of decreasing of pressure recovery ability caused by increasing inlet velocity. Minimum velocity is at the negative stall condition and that is the minimum pressure loss is seen in this regime. The results show that the pressure loss is highly sensitive to the velocity but is not sensitive to the incident angles (angle of attack and sideslip).

Researchers, who have compared the results of numerical analysis of air intake flows with the results of experimental tests, declared that computational fluid dynamics would estimate the flow behavior satisfactorily but, this will be possible only with complete knowledge on the subject and with having an adequate experience in applying CFD correctly to the problems.

REFERENCES

[1] R. D. D. Menzies, Investigation of S-Shaped Intake Aerodynamics Using Computational Fluid Dynamics, Glasgow: University of Glasgow, 2002.

[2] J. Seddon and E. Goldsmith, Intake Aerodynamics Second Edition, UK: Blackwell Science Ltd, 1999.

[3] N. C. Bissinger and T. Breuer, Encyclopedia of Aerospace Engineering - Volume 8 - Chapter EAE487, vol. 8, John Wiley&Sons, Ltd. ISBN: 978-0-470-75440-5, 2010.

[4] D. C. Wilcox, “Turbulence Modelling for CFD,” DCW Industries, Inc., La Canada, California, 1994.

[5] M. J. Brear, Z. Warfield and J. F. Mangus, “Flow Separation within the Engine Inlet of an Uninhabited Combat Air Vehicle (UCAV),” in 4TH ASME_JSME Joint Fluids Engineering Conference, Honolulu, Hawaii, USA,, 2003.

[6] F. R. Menter, “Zonal Two Equation Kappa-Omega Turbulence Model for Aerodynamic Flows,” AIAA, Paper 93-2906, 1993.

[7] B. G. Allan, L. R. Owens and B. L. Berrier, “Numerical Modeling of Active FlownControl in Boundary Layer Ingesting Offset Inlet,” in 2nd AIAA Flow Control Conference, Portland, Oregon, 2004.

[8] Y. Wu, E. Ng and K. Wong, “Numerical study of the swirl flow in F-5E intake with subsonic speeds,” Mathematical and Computer Modelling, vol. 48, pp. 447-467, 2008.

[9] J. L. F. a. M. Peric, “Computational Methods for Fluid Dynamics,” Springer-Verlag, Heidelberg, 1996.

[10] B. H. Anderson, D. R. Reddy and K. Kapoor, “A Comparative Study of Full Navier-Stokes and Reduced Navier-Stokes Analysis for Separating Flows within a Diffusing Inlet S-Duct,” AIAA, Vols. 93-2154, no. Lewis Research Center, 1993.

[11] S.-Y. Cho, Three dimensional compressible turbulent flow computations for a diffusing S-duct with/without vortex generators, Ph.D. Thesis, Case Western Reserve University, 1993.

[12] P. E. H. Abrahamsen, B. A. Pettersson, L. Saetran and J. B. Fossdal, “Air Intake Studies: Experimental Measurements and Computational Modeling,” Kongsberg Deffence and Aerospace AS, Norwegian University of Science and Technology, Trondheim, 1998.

[13] R. Stanley and J. Mohler, “Wind-US Flow Calculations for the M2129 S-Duct Using Structured and Unstructured Grids,” in 42nd AIAA Aerospace Sciences Meeting and Exhibit, Reno, Nevada, USA, 2004.

[14] A. M. Pradeep and R. K. Sullerey, “Secondary Flow Control in Circular S-Duct Diffuser Using Vortex Generator Jets,” in 2nd AIAA Flow Control Conference, Indian Institute of Technology Kanpur, India, 2004.

[15] R. Menzies, “Computational Investigation of Flow in a Diffusing S-Shaped Intake,” Journal of Advanced Engineering Design, Glasgow, vol. 41, June 2001.

[16] B. L. Berrier and B. G. Allan, “Experimental and Computational Evaluation of Flush-Mounted, S-Duct Inlets,” in 42nd AIAA Aerospace Sciences Meeting & Exhibit, Reno, Nevada, 2004.

[17] J. H. Gary, A. R. Bruce and R. W. Steven, “Navier-Stokes analysis and experimental data comparison of compressible flow in a diffusing S-duct,” AIAA, Sverdrup Technology Inc., 1992.