10th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2019-017**
18-20 September 2019 - METU, Ankara TURKEY

COMPUTATION METHODOLOGY OF PERFORMANCE CHARACTERISTICS FOR A SUPERSONIC AIR INTAKE

Can Çıtak*[1], Özgür Harputlu[1], Ezgi Arısoy[1], Tekin Aksu[1] and M. Tuğrul Akpolat[1]
ROKETSAN Missiles Inc.
Ankara, Turkey

ABSTRACT

A computation method for performance characteristics of a supersonic air inlet of a ramjet engine is presented. In a supersonic air breathing propulsion system, the air inlet is a crucial part in terms of its performance of carrying out its purpose, i.e. compressing the incoming air. Inlet's performance affects all aspects of the propulsion system and the vehicle, such as combustion, range, maneuverability etc. Therefore, it is important to assess the performance characteristics of the air inlets. In order to investigate the air intake performance and computation methodology, Computational Fluid Dynamics (CFD) simulations of supersonic air intake are carried out. Reynolds-averaged Navier Stokes equations are used with standard k-ω *turbulence model. Detailed computation methodology of performance characteristics are examined.*

INTRODUCTION

Ramjet engines employ ram-effect to intake and compress the air via supersonic inlets. Supersonic inlets have the same purpose as a compressor, which is to compress the air and deliver it to the combustion chamber, where the air is mixed with fuel and burned. The high energy gas generated due to the combustion process is then speeded up to generate the desired thrust. For the ramjet propulsion cycle, the air inlets are of crucial importance since they have to provide the needed compressed air for the combustion for a range of Mach numbers and altitudes. The types of supersonic inlets used are internal, external and mixed compression. For relatively lower flight Mach numbers (M=1.4 to 1.6), internal type compression, i.e. pitot type inlet, which generates a single normal shock to compress the air, is considered to be the best choice. At higher Mach numbers up to Mach 2, external compression inlets with multiple compression ramps are usually the best choice. However, for flight Mach numbers higher than Mach 2 the best choice is the mixed type compression inlet, which is a combination of internal and external compression inlets. For a mixed compression inlet, there exists external and internal oblique shocks followed by a normal shock at the inlet throat. Mixed compression inlets are used in order to avoid the high external drag of external compression inlets and excessive boundary layer growth of the internal compression inlets (Seddon & Goldsmith, 1999).

[1] Propulsion System Design Department, *corresponding author, email: can.citak@roketsan.com.tr

Independent from the compression type and flight Mach number, the supersonic nature of the flow inside an inlet imposes some complex flow phenomena such as adverse pressure gradients and shockwave-boundary layer interactions. The complexity of such phenomena makes it difficult and expensive to conduct comprehensive supersonic wind tunnel tests, which take all of the phenomena into account. Therefore, a validated means of CFD methods should be employed in the design process. Some of the most common CFD methods are RANS, LES, DNS etc. Closure models are not needed for DNS, since the Navier-Stokes equations are numerically solved at all scales. LES method models the small eddies, and large eddies are resolved with subgrid scale models, such as Smogorinsky, Wale, Vreman etc. Even though the accuracy of DNS and LES models is pronounced, they may require extremely high CPU power. Therefore, DNS and LES are not suitable for industrial applications. Where the CPU power is of question, a solver for RANS equations may be employed. However, a turbulence model (such as k-ω, k-ε etc.) is needed for a RANS solver in order to compensate the closure problem. k-ω turbulence model is generally recommended and validated for complex flows which involve separation and wall bounded high speed flow (Das & Prasad, 2010). The standard k-ω model in FLUENT, which is employed for the simulations presented in this paper, is based on the Wilcox k- ω model. The Wilcox k- ω model is designed for the application to the boundary layer and is also applicable to wall-bounded flows and free-shear flows (Das & Prasad, 2009). For a supersonic inlet (Reinartz, et al., 2003) and (Coratekin, et al., 1999) compared the k-ω model with experimental results, and reported a good comparison. Within the concept of this study a method for evaluation of supersonic air inlet performance is presented. The results of the CFD method suggested is compared with experimental results.

METHODOLOGY

Air intake decelerates and compresses supersonic flow to a “desired” subsonic flow. As guessed; main objectives of the air intake, which will be mentioned in detail, are defined by combustion chamber. These objectives must be satisfied at the flight envelope. Flight envelope consists Mach number and altitude interval. During design period, checking all the design alternatives whether the air intake design satisfies the requirements by wind tunnel tests makes the design schedule long and expensive. As an objective, performance characteristics of the air intake are numerically computed by using computational fluid dynamics solver. By using this method; fast and accurate simulations could shed a light for the optimization of the air intake design.

Performance characteristics mentioned above can be defined as variation of total pressure recovery with respect to mass capture ratio. Total pressure recovery is defined as the ratio of the total pressure at combustion chamber to freestream total pressure. Mass capture area is the ratio of mass delivered to the combustion chamber to maximum theoretical mass flow rate can enter to air intake’s cross section with freestream velocity.

$$Total\ Pressure\ Recovery\ (TPR) = \frac{P_{t\,cc}}{P_{t0}}$$

$$Mass\ Ratio\ (MR) = \frac{\dot{m}_{cc}}{\dot{m}_0}$$

$$\dot{m}_0 = \dot{m}_{cc} + \dot{m}_{spillage} + \dot{m}_{bleed}$$

Performance curves can be separated into three regimes. Namely; super-critical, critical and sub-critical. The air intake operates critically when the combustion chamber pressure magnitude is such that the normal shock is positioned at cowl lip. If the combustion chamber pressure is less than the critical value, the normal shock is positioned downstream of the cowl lip and such operation regime is called supercritical. When the combustion chamber pressure exceeds the critical value, the terminal normal shock moves forward of the cowl lip and merge with external oblique shocks leading to detached bow shock. This operation regime is called subcritical. Typical air intake system performance curve is shown as Figure 1.

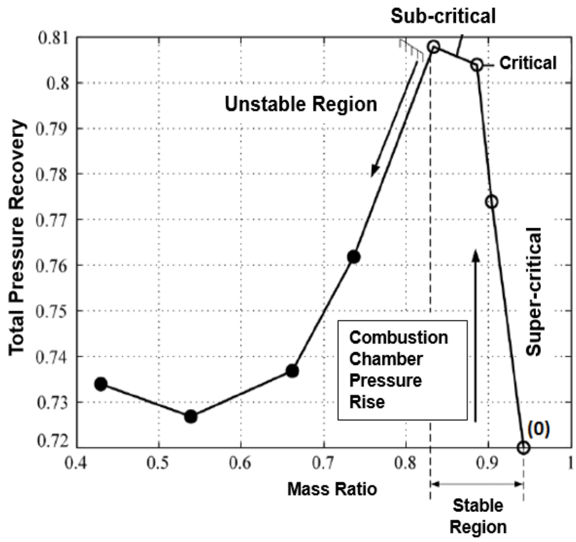

Figure 1: Regions of Performance Curve for Air-Intake System.

Air intake system stations is shown in Figure 2.

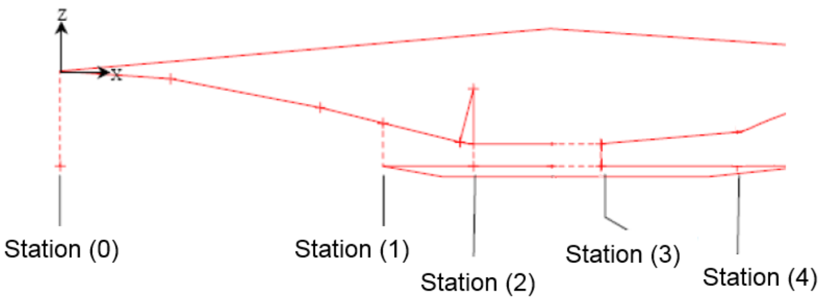

Figure 2: Air-Intake System Stations.

Air intake meets the supersonic flow at Station 0, starts to decelerate and compress until Station 2. Station 2, which is also characterized as “aerodynamic throat”, defines the sub-sections of the performance curves geometrically. Normal shock stays at Station 2 on the “critical point”. Compressed and decelerated flow to subsonic speed passes through Station 4 before entering the combustion chamber.

- Station 0: Free flow region,
- Station 1: Start of internal compression region,
- Station 2: Aerodynamic throat,
- Station 3: End of aerodynamic throat,
- Station 4: Combustion Chamber Entrance

NUMERICAL MODEL

RANS equations are solved with a commercial software (FLUENT) and k-ω turbulence model is selected to close the Reynolds-averaged conservation equations. Standard k-ω turbulence model is found by Wilcox (D.C. Wilcox, 1998). Flow equations are shown below.

$$\frac{\partial \bar{\rho}}{\partial t}+\frac{\partial}{\partial x_i}\left(\bar{\rho}u_j\right)=0 \quad (1)$$

$$\frac{\partial\left(\bar{\rho}u_i\right)}{\partial t}+\frac{\partial}{\partial x_i}\left(\bar{\rho}u_j u_i\right)=-\frac{\partial \bar{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\overline{\tau_{ij}}+\bar{\rho}\overline{u_j^{"}u_i^{"}}\right) \quad (2)$$

$$\frac{\partial\left(\bar{\rho}H\right)}{\partial t}+\frac{\partial}{\partial x_j}\left(\bar{\rho}u_j H\right)=-\frac{\partial}{\partial x_j}\left(\bar{\rho}\alpha\frac{\partial H}{\partial x_j}+\bar{\rho}\overline{u_j^{"}H^{"}}\right)+\overline{S_H} \quad (3)$$

In order to resolve turbulence Favre averaged flow equations are solved and turbulence related $\bar{\rho}\overline{u_j^{"}u_i^{"}}$, $\bar{\rho}\overline{u_j^{"}H^{"}}$ ve $\bar{\rho}\overline{u_j^{"}Y_M^{"}}$ terms need to be modeled. Reynolds stress tensor is defined with Boussinesq approach. According to this approach Reynolds stress tensor in implied as equation 4.

$$\bar{\rho}\overline{u_j^{"}u_i^{"}}=\mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\mu_t\frac{\partial u_k}{\partial x_k}\delta_{ij}-\frac{2}{3}\bar{\rho}k\delta_{ij} \quad (4)$$

Other two term are defined as in equation 5 and 6.

$$\bar{\rho}\overline{u_j^{"}H^{"}}=\frac{\mu_t}{\mathrm{Pr}_t}\frac{\partial H}{\partial x_j} \quad (5)$$

$$\bar{\rho}\overline{u_j^{"}Y_m^{"}}=\frac{\mu_t}{Sc_t}\frac{\partial Y_m}{\partial x_j} \quad (6)$$

Standard *k-ω* turbulence model is used for turbulence modeling. Model first introduced by Kolmogorov and simplified Wilcox (Wilcox, 1998). Turbulent viscosity is shown in equation 7.

$$\mu_t=\frac{\bar{\rho}k}{\omega} \quad (7)$$

$$\omega=\max\left[\omega, 0.875\left(\frac{2\overline{S_{ij}}\,\overline{S_{ij}}}{\beta^*}\right)^{0.5}\right] \quad (8)$$

$$\overline{S_{ij}}=S_{ij}-\frac{1}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij} \quad (9)$$

Specific turbulence dissipation rate is shown with ω in equation 8. At this equation set $\overline{S_{ij}}$ implies averaged strain rate and, β^* denotes turbulence model constant. Turbulence kinetic energy, k, and specific turbulence dissipation rate, ω, is shown in equation 10 and 11 respectively.

$$\frac{\partial\left(\bar{\rho}k\right)}{\partial t}+\frac{\partial\left(\bar{\rho}u_j k\right)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\left(\mu+\sigma^*\mu_t\right)\frac{\partial k}{\partial x_j}\right)+\bar{\rho}\tau_{ij}\frac{\partial u_i}{\partial x_j}-\bar{\rho}\beta^*\omega k \tag{10}$$

$$\frac{\partial\left(\bar{\rho}\omega\right)}{\partial t}+\frac{\partial\left(\bar{\rho}u_j\omega\right)}{\partial x_j}=\alpha\frac{\omega}{k}\bar{\rho}\tau_{ij}\frac{\partial u_i}{\partial x_j}+\frac{\partial}{\partial x_j}\left(\left(\mu+\sigma\mu_t\right)\frac{\partial\omega}{\partial x_j}\right)+\sigma_d\frac{\bar{\rho}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial\omega}{\partial x_j}-\bar{\rho}\beta\omega^2 \tag{11}$$

Model Setup

A mixed-compression air intake with 3 compression ramps and a bleed system is used in simulations. Figure3 shows the geometrical configuration of CFD model.

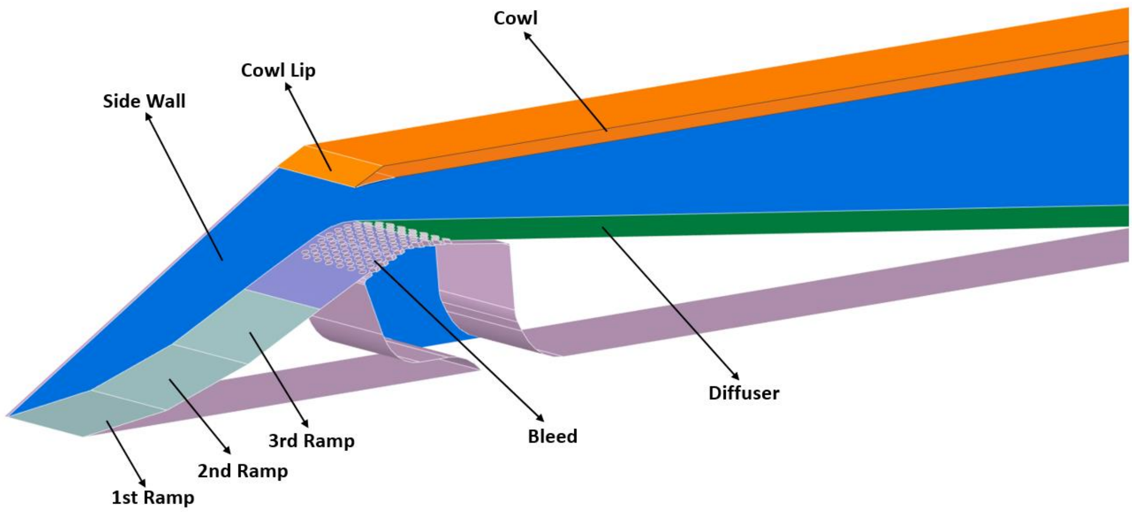

Figure 3: Mixed Compression Air Intake used in CFD Simulations

The aim of numerical simulations is to obtain performance curve of the air intake. For this purpose, characteristics of intake corresponding to different combustion chamber pressure values must be simulated. A pseudo nozzle is attached to end of divergent section and throat area is changed parametrically to create different combustion chamber pressures and, therefore, to obtain performance map. By this method, the pressure at the end section of intake is modified without creating additional disturbance on the boundary layer. Figure 4 shows pseudo nozzle and the nozzle parameter.

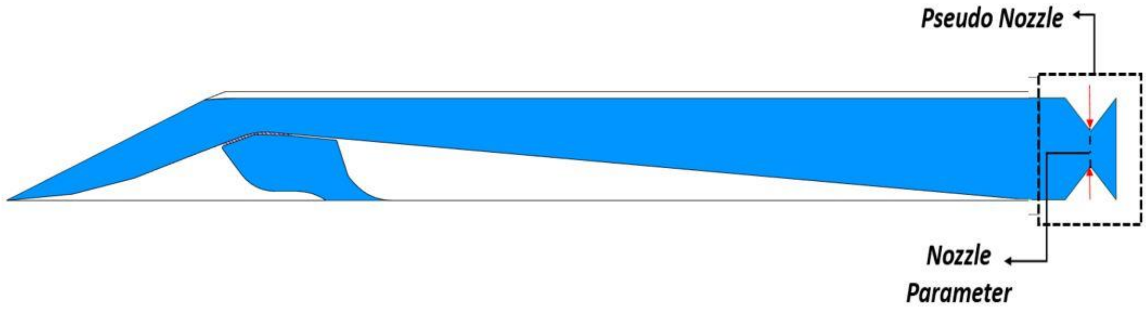

Figure 4: Pseudo Nozzle for the CFD Simulations and Nozzle Parameter for Back Pressure Adjustment

Simulations start with supercritical operation which corresponds to a large nozzle throat opening. As the nozzle is throttled, position of terminal shock moves upstream. After the normal shock passes over bleed section unsteady flow characteristics results in buzzing flow. Buzzing phenomena is not desired during operation of intake, because it may result in insufficient mass flow to the combustion chamber and affect the overall performance of propulsion system. CFD simulations are limited to steady operability region and for the mixed compression intake in this study, critical point is defined as the condition in which normal shock is positioned over the bleed section. A further increase in backpressure will result in buzzing of intake

Near wall cells are structured grid elements to resolve the boundary layer, while remaining elements are unstructrured tetrahedrons. The average cell number for simulations is 7.9 million and a sectional view of mesh at mid-plane of air intake is shown in Figure5. Boundary conditions applied to the solver are shown in Figure 6. Properties of incoming supersonic flow is defined as far-field boundary condition. A half-symmetric model of air-intake is used to reduce the computational cost.

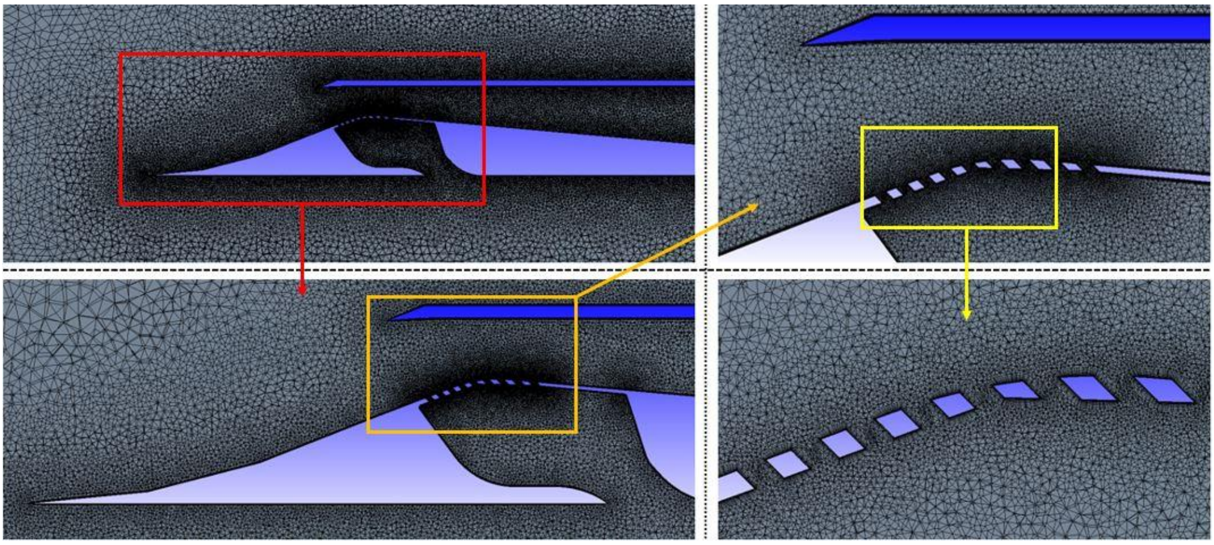

Figure 5: Computational Grid for CFD Simulations

Tetrahedron cells are used for the meshing. It is observed that the boundary layer thickness affects performance characteristics of the air intake. Successful treatment to boundary layer region of the supersonic air intake surfaces are needed. So that, the mesh elements employed near walls are must be high quality. On the other hand, there is a trade-off between meshing quality and CPU usage for the CFD analysis. Since analysis run for high number for configuration for different scenarios, “the baseline” is defined for the number of meshing elements after completing mesh independency study, which is not presented in this paper since mesh independence study was conducted for quite a few studies before.

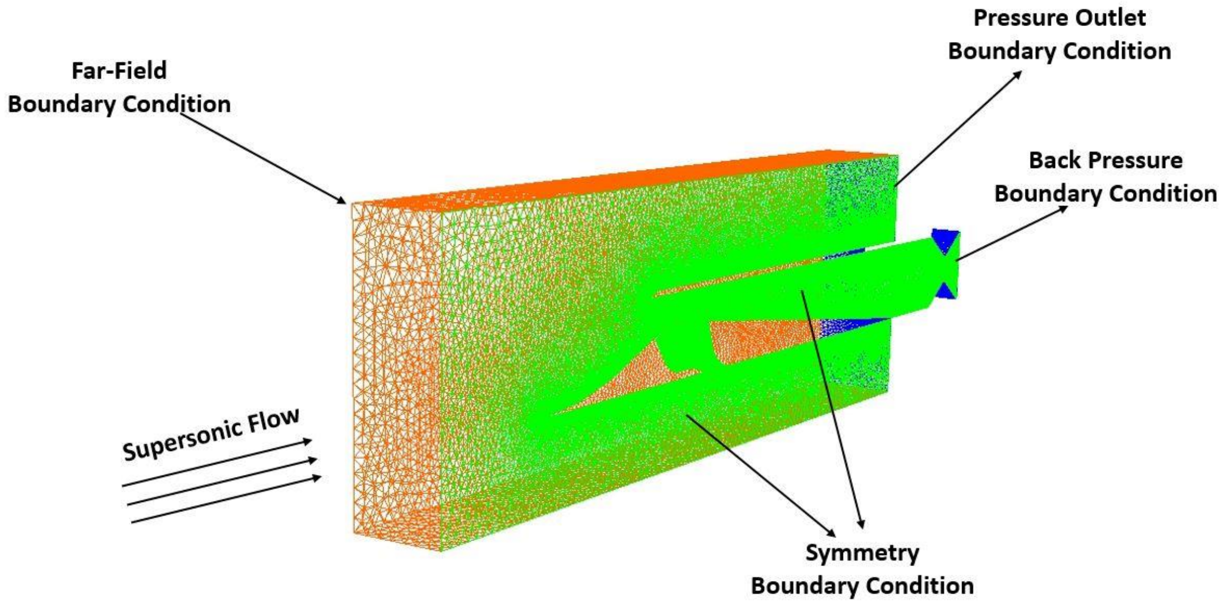

Figure 6: Computational Domain Representation

RESULT

Performance map calculations are conducted for two different flight condition, which are listed in Table 1. In Figure 7, variation of terminal shock position with nozzle throat parameter is represented for a flight condition “1”.

Table 1. Flow Parameters for CFD Simulations

Condition	Mach Number
1	2.9
2	3.2

The lower number of nozzle parameter (NP) is the wider the throat area is. (e.g $NP_1 > NP_2 \ldots > NP_{nmax}$). At large nozzle throat area air intake operates at supercritical condition. As the pseudo nozzle is throttled, a greater value of combustion chamber back pressure is simulated and the shock system in the isolator section moves from the combustion chamber entrance to the cowl lip. Throttling of nozzle eventually results in expulsion of the shock system is outside of the intake and unstart process is observed. In Figure 8, static pressure contours corresponding to nozzle parameters are presented with a logarithmic scale.

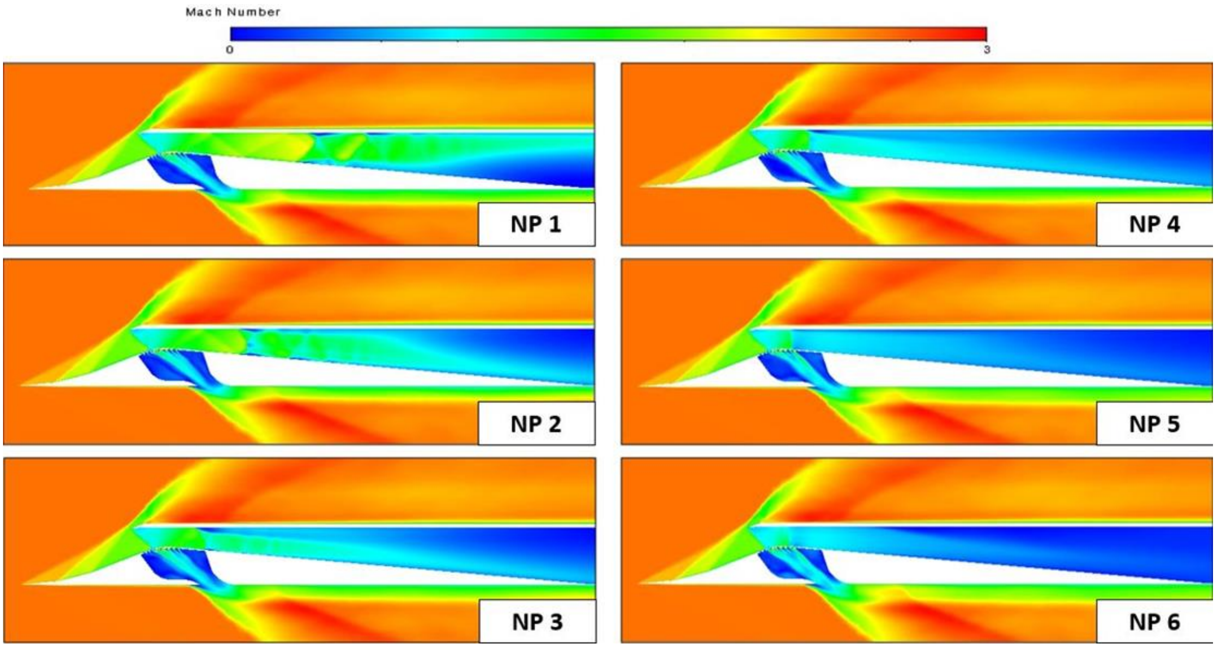

Figure 7: Mach Number Contours with Different Nozzle Parameters for Condition “1”.

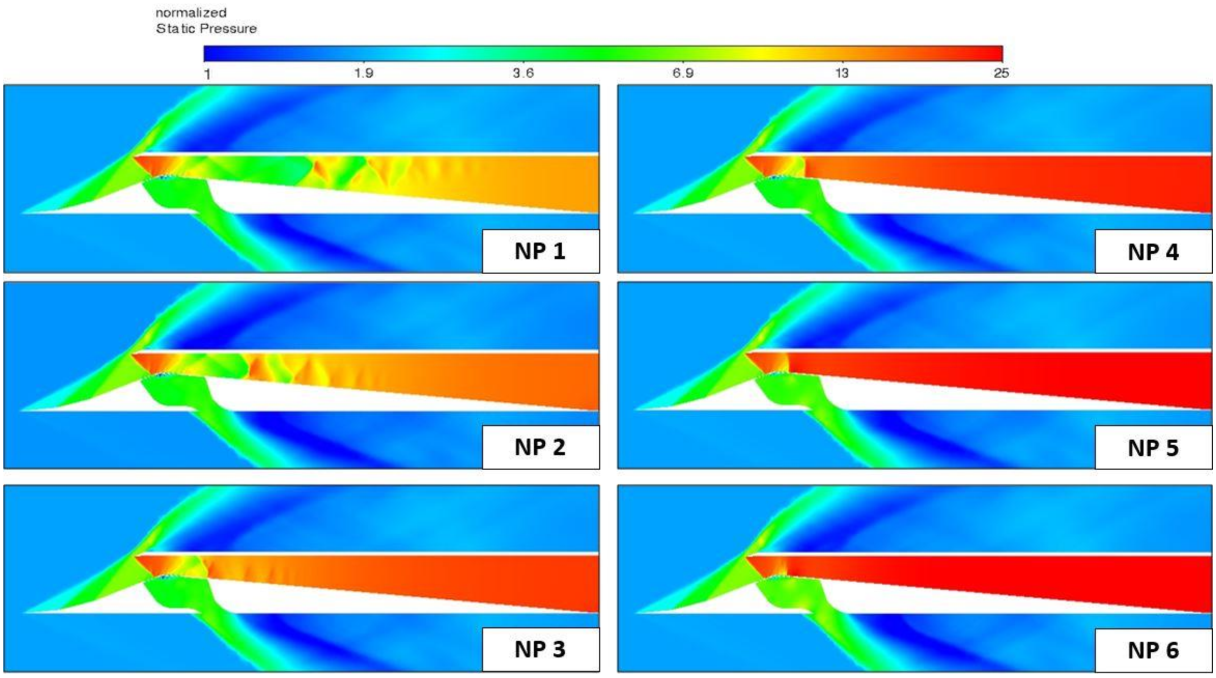

Figure 8: Static Pressure Contours with Different Nozzle Parameters for Condition “1”.

Performance curves obtained for flight conditions are presented in Figure 9. The last stable point of a mixed compression is the condition at which the normal shock positioned on the bleed section. After this point shock system can’t be held inside the intake in a stable manner. This kind of operation results in buzzing and finally unstart of the intake which is not desired during operation. Although a supersonic air-intake with fixed compression surfaces is designed according to on-design condition, it must also provide minimum performance limitations in off-design conditions in order to guarantee operability of the propulsion system.

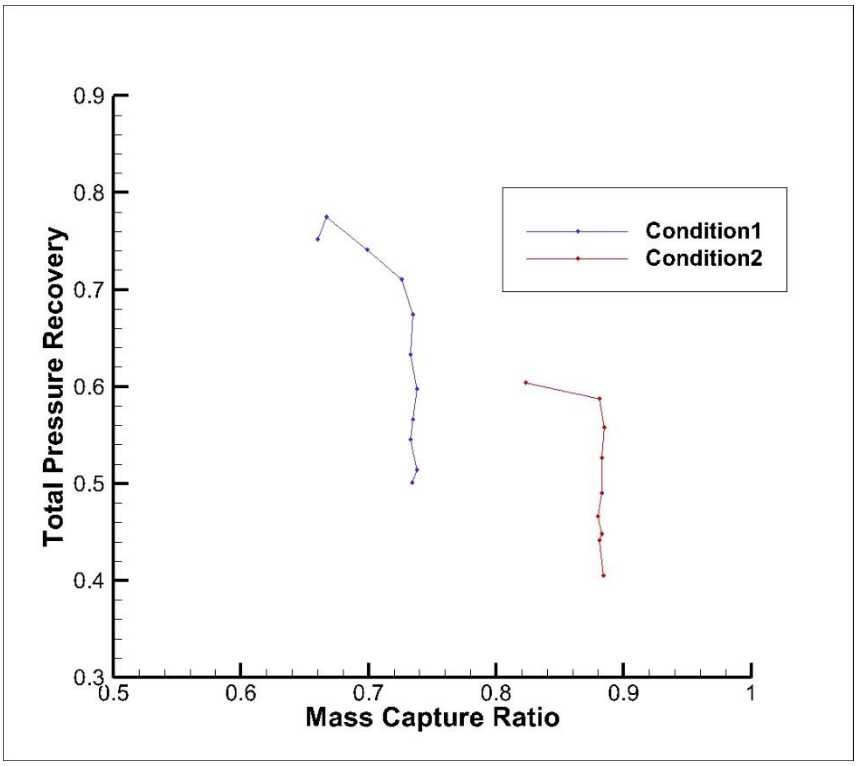

Figure 9: Performance Curves for Flight Conditions.

CONCLUSION

CFD is a viable tool for design process of an air intake system when compared with wind tunnel tests. Despite the fact that wind tunnel tests are compulsory in order to assess performance characteristics of the air intake. In this study, CFD method is examined, in order to compute the performance characteristics of the air intake numerically. This method can adapt back pressure (combustion chamber pressure) by nozzle independently. Meshing and analysis setup are generated by considering similar flow regimes and geometries. Minimization CPU usage for the CFD analysis without losing accuracy and / or stay above acceptable base is one of the major purposes of the analysis process.

FUTURE WORK

Computational Fluid Dynamics (CFD) simulations of supersonic air intake with perforated bleed systems is performed. Simulations are going to conducted with Large-eddy simulation and compared shock structures in depth.

REFERENCES

Coratekin, T., VanKeuk, J. & Ballmann, J., 1999. *Preliminary investigations in 2-D and 3-D ramjet inlet design.* Los Angleles, CA, USA, AIAA 99-2667.

D.C. Wilcox, 1998. *Turbulence Modelling for CFD.* 2nd ed. DCW Industries Inc.: California.
Das, S. & Prasad, J., 2010. Starting characteristics of a rectangular supersonic air-intake with cowl deflection. *The Aeronautical Journal (1968),* pp. 177-189.

Das, S. & Prasad, J. K., 2009. Cowl Deflection Angle in a Supersonic Air Intake. *Defence Science Journal,* 59(2), pp. 99-105.

Reinartz, B. U., Herrmann, C. D., Ballmann, J. & Koschel, W. W., 2003. Aerodynamic Performance Analysis of a Hypersonic Inlet Isolator Using Computation and Experiment. *Journal of Propulsion and Power,* 19(5), pp. 868-875.

Seddon, J. & Goldsmith, E. L., 1999. *Intake Aerodynamics.* 2nd ed. Oxford: AIAA.
Wilcox, D., 1998. *Turbulence Modelling for CFD.* California: DCW Industries Inc..