10th ANKARA INTERNATIONAL AEROSPACE CONFERENCE **AIAC-2019-018**
18-20 September 2019 - METU, Ankara TURKEY

EXPERIMENTAL VALIDATION OF AN AIR INTAKE PERFORMANCE

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

ABSTRACT

An experimental study is presented to validate the performance characteristics of an air intake. The investigation is carried out in the Trisonic Wind Tunnel (TMK) of DLR. The model was an external compression, 2D rectangular air intake. Experiments were carried out in order to obtain the performance and flow characteristics of the test model, which was designed with a back-pressure control system. In CFD simulations back-pressure is simulated with a pseudo-nozzle. Performance map is obtained by parametric variation of pseudo-nozzle throat area and steady flow simulation at each nozzle parameter. The results are compared in order to fine-tune and validate the CFD model.

INTRODUCTION

The required air mass flow, i.e. oxygen for combustion, and pressure recovery for supersonic air-breathing propulsion is maintained by air intakes. Air intake performance important in order to maintain a stable combustion. Therefore, one must gain insight on the evaluation of the intake performance. Experimental investigation is a convenient tool regarding the air intake performance. However, it is expensive and resolving 3D flow details, which is ubiquitous in an air intake flow path is challenging. Therefore, flow simulations via CFD methods should be employed with experimental investigations in unison. In order to do that, it is necessary to use the experimental data as a comparison case to fine-tune and validate the CFD methods.

Evaluation of the air intake performance is done by three parameters, mass flow ratio, pressure recovery, and stability point [Herrmann & Gülhan, 2015]. The mass flow ratio is the ratio of mass flow rate of air passes through the combustion chamber to the mass flow rate of air ingested by the air intake. The experimental evaluation methods for air intake performance are well established in terms of pressure recovery and mass flow ratio by Herrmann et al. [Herrmann, Siebe, & Gülhan, 2013].

In order to improve the inlet performance, one must enhance and increase the energy of the boundary layer flow within the inlet flow path. One of the most common ways to do that, is to

[1] Engineer, Propulsion System Design Department, *corresponding author, email: tugrul.akpolat@roketsan.com.tr

implement boundary layer bleed to avoid high boundary layer thicknesses and/or boundary layer separation which triggers inlet unstart. Hermann et al. [Herrmann, Siebe, & Gülhan, 2013] conducted an experimental study comparing air intake performance with and without bleed at Mach numbers between 2.5 and 3.5. The results show that a stable air intake operation with better pressure recovery and mass flow ratio is achieved by employing bleed. The design of bleed in terms of bleed entrance and exit area ratio [Herrman, Blem, & Gülhan, 2011, Herrmann & Gülhan, 2015], bleed position [Soltani, Daliri, Younsi, & Farahani, 2016], and, bleed slot types [Bauer & Kurth, 2011], influence of bleed entrance perforation [Bauer & Kurth, 2011, Fukuda, Roshotko, & Hingst, 1975] are studied by various authors.

The mass flow ratio is affected by spillage and bleed, which is primarily employed to overcome the adverse effects of shockwave boundary layer interactions, e.g. boundary layer separation. The pressure ratio is affected mostly by the combustion chamber pressure, which is simulated by employing a throttling plug [Herrman, Blem, & Gülhan, 2011] or a valve [Trapier, Duveau, & Deck, 2006] plug in the experimental studies. However, both of the parameters influenced with each other.

CFD is a crucial tool for supersonic inlet design, especially when the cost of supersonic wind tunnel experiments is considered. Moreover, CFD allows designers to resolve the shockwave boundary layer interactions and other 3D flow behaviors, which are by far challenging to resolve with experimental methods [Slater, Davis, Sanders, & Weir, 2005]. Eriksson et al. [Eriksson, Johansson, & Borg, 1993] proposed to use CFD as a design tool for supersonic ramjet inlets, and employed Euler and Navier-Stokes solvers. The results of simulations were then compared with results of water tank, wind tunnel and direct-connect experimental runs. The results of CFD are mostly captured the flow patterns and inlet performance in terms of pressure recovery and pressure distribution with good overall comparison. However, the authors mentioned the need of a 3D solver and computational power to capture a reliable flow pattern for especially inlet flowpath. Srinivasan and Sinhamahapatra [Srinivasan & Sinhamahapatara, 2017], carried out a Large Eddy Simulation study of a 3D supersonic inlet designed for Mach 2.2. They have employed fifth order freestream preserving WENO scheme for the convective terms and forth order central scheme for viscous diffusive terms. The results indicate that the flow features such as shockwave boundary layer interactions and separation bubble are very well captured.

Within the scope of this study a 2D, rectangular external compression air intake model was tested in the Trisonic Wind Tunnel (TMK) of DLR with the aim of assessing the performance of the air intake model. The other purpose of the study is to fine-tune and validate the CFD simulations in order to minimize the experimental work, reducing the air intake design cost. First the experimental setup, the air intake model, and the experimental procedure including the performance assessment method is explained. As numerical method, Reynolds-averaged Navier Stokes approach with standard k-ω turbulence model is used.

EXPERIMENTAL SETUP & PROCEDURE

Experiments were conducted in the Trisonic Wind Tunnel (TMK) of DLR (Figure 1). The TMK wind tunnel has a closed test section of 0.6x0.6 m^2 with a flexible nozzle that can be adjusted during testing. The standard Mach number range of the TMK is from Mach 0.5 to Mach 4.5. The model was tested at a Mach number range of $2 \leq M_0 \leq 4$, at an angle of attack of 0° and an angle of yaw of 0°. For the purpose of this paper, the experimental results at M_0=2.5 are presented.

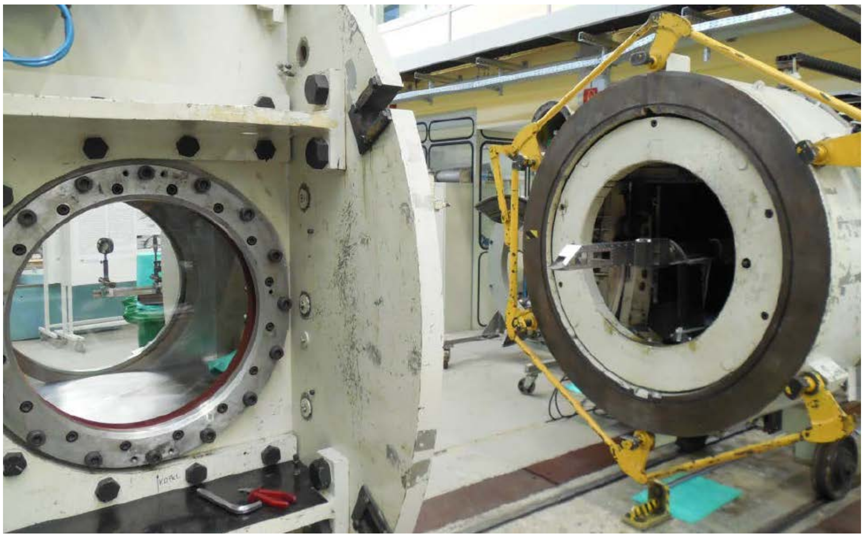

Figure 1: The air inlet model in TMK of DLR.

The simulation of the combustion chamber pressure was conducted by employing a throttle mechanism driven by a hydraulic system. The throttle was moved in order to vary the pressure by changing the critical throttle cross section. The static pressure measured from the upstream of the throttle was used to calculation of the inlet mass flow ratio and the pressure recovery. For the calculation, the total temperature data of the wind tunnel and the critical throttle cross section geometry are also employed. The details of the calculation can be found in [Herrmann, Blem & Gülhan, 2011].

The model is equipped with two Schlieren windows for flow visualization, and with Kulite pressure sensors for unsteady pressure measurements. Four pressure sensors are located at each ramp and 10 pressure sensors are located at the top of the model. The calibration of the sensors was conducted using an automated pressure calibrator. The accuracy of the pressure sensors are $\Delta p_{st}/p_{st}$=0.03%. The details of pressure sensor locations and Schlieren windows are given in Figure 2.

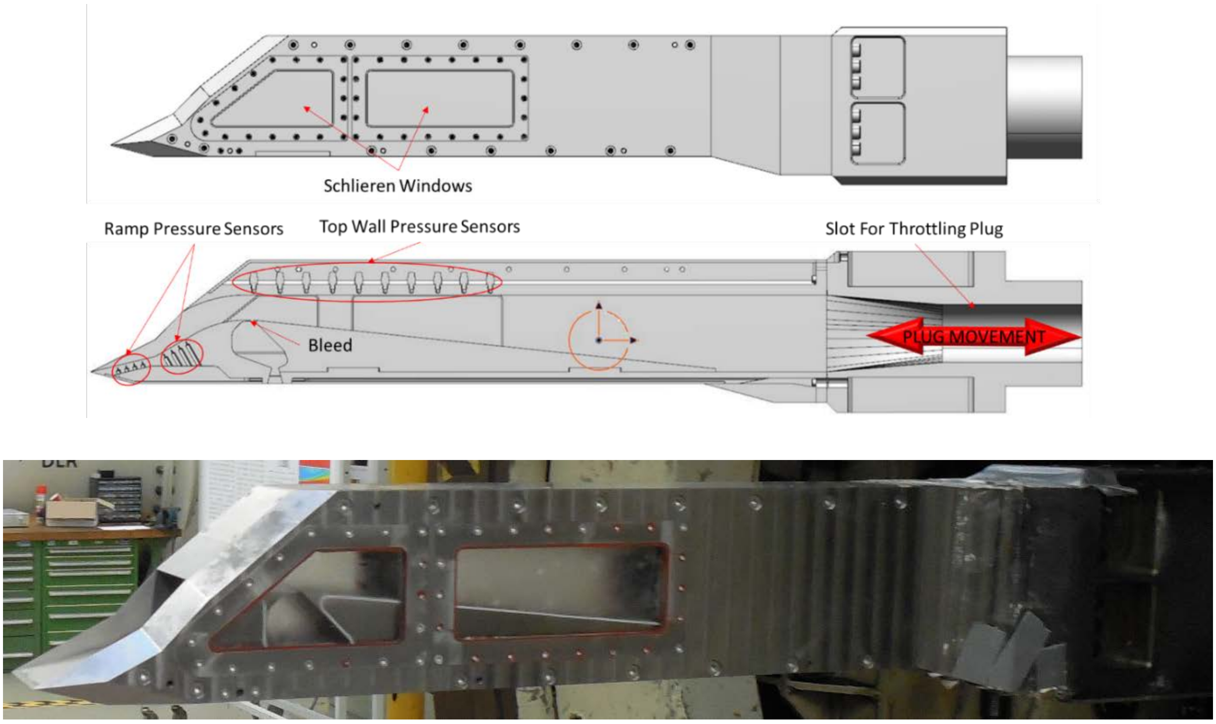

Figure 2: Test Model

<u>Performance parameter calculations:</u> Plug moves back and forth changing the area at the end of the test model as shown in Figure 3.

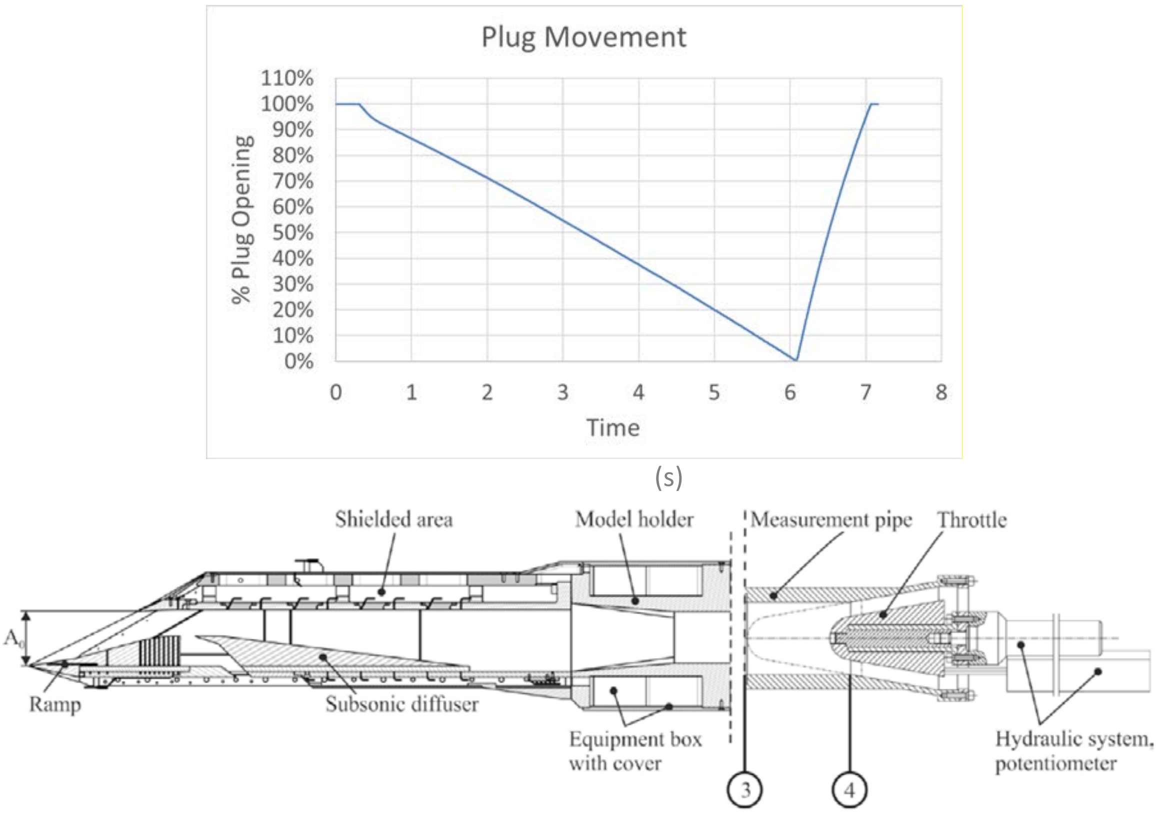

Figure 3: Plug movement (left), back area control scheme (right)

The tests start with plug in the backward position, fully opened, such that the A_{11} has its maximum value. Then plug starts its movement forward, closes the area entirely, stays fully closed for a moment then starts to go backward until it reaches fully opened position. The entire plug movement is symmetric and lasts for 7.5 seconds in total. Meanwhile pressure and Schlieren system start recording data simultaneously with the plug. Performance calculations are done using static pressure readings from the Kulite pressure sensor located at measurement pipe denoted as 3. Total pressure as well as mass flow rate at this location were derived using analogy described in [Herrmann, Blem & Gülhan, 2011].

NUMERICAL METHOD

RANS equations are solved with a commercial software (FLUENT) and k-ω turbulence model is selected to close the Reynolds-averaged conservation equations. Numerical model and equations are described and emphasized in [Can Çıtak, Özgür Harputlu, Ezgi Arısoy, Tekin Aksu and M. Tuğrul Akpolat, 2019]

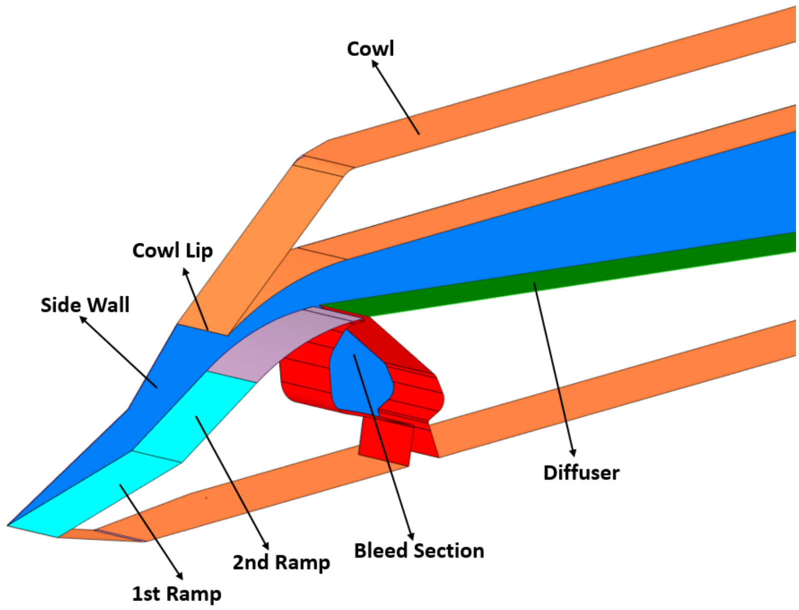

Figure 3: Elements of Air-Intake Model

Figure 3 displays the elements of the air-intake used in numerical calculations. Ramps are external compression surfaces, which decelerates the flow by oblique shock waves. Cowl section reflects oblique shocks coming from ramps. At design point, oblique shocks are focused on the cowl lip. At Mach numbers greater than design value shock waves intersect at downstream of the cowl-lip, whereas at lower Mach number values than design point, shock waves intersect at upstream of the cowl-lip. Pressure sensors at the upper wall and their connection equipment are placed on the top wall, which results in a thicker cowl section than operational air-intake model.

Since the aim of this study is to compare experimental and numerical results, geometrical model with the same cowl thickness employed in experiments is used in numerical simulations Computational domain is discretized with unstructured tetrahedron cells with structured grid at near wall for resolving the boundary layer.

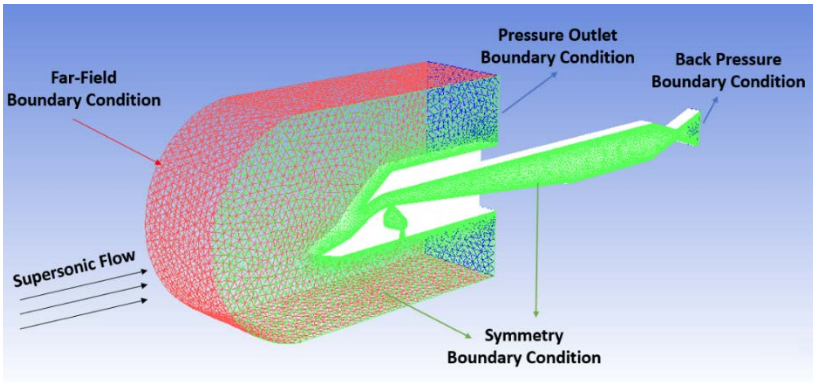

Figure 4: Boundary Conditions of the Simulation Model.

Quality of mesh elements in computational domain is a compromise between element quality and CPU usage. A mesh independency study is conducted on a model case to ensure that simulation successfully treats flow physics and results are calculated in a reasonable CPU time. Average number of mesh elements for each simulation is about 8 million. Boundary conditions of simulation model are presented in Figure 4.

Properties of supersonic freestream flow are defined in far-field boundary condition in terms of Mach number, static temperature and static pressure. Ambient static pressure inside wind tunnel is assigned to pressure outlet boundary conditions. A half-symmetric model is used in simulations in order to reduce computational time. Mid-plane of the air-intake is defined as the symmetry plane. Section view of computational domain is represented in Figure 5.

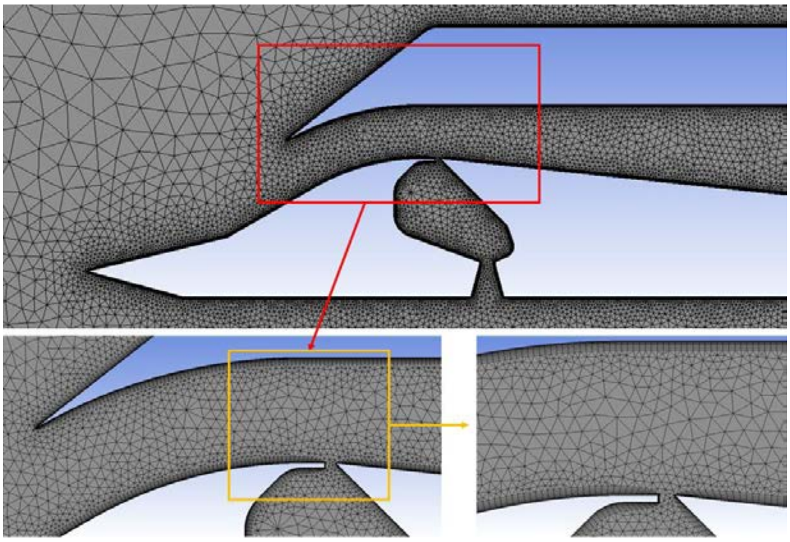

Figure 5. Section view of the computational domain.

Conditions corresponding to different backpressure values must be simulated to obtain performance map of air-intake. For this purpose, a pseudo nozzle is attached to the end of divergent section and throat is changed parametrically. At each nozzle, throat value a different backpressure is simulated. Pseudo nozzle and nozzle parameters is shown in Figure 6.

Figure 6: Pseudo Nozzle for the CFD Simulations and Nozzle Parameter for Back Pressure Adjustment

RESULTS

At wider pseudo nozzle throat values, intake operates at supercritical condition. As the nozzle throat is throttled, position of terminal shock moves upstream. Figure 7 shows variation of terminal shock position with nozzle throat area. The lower number of nozzle parameter (NP) is the wider the throat area is. (e.g NP1>NP2 …>NP5) At supercritical operation normal shock is positioned in the isolator section. Shock-train region may be observed according to the nozzle parameter.

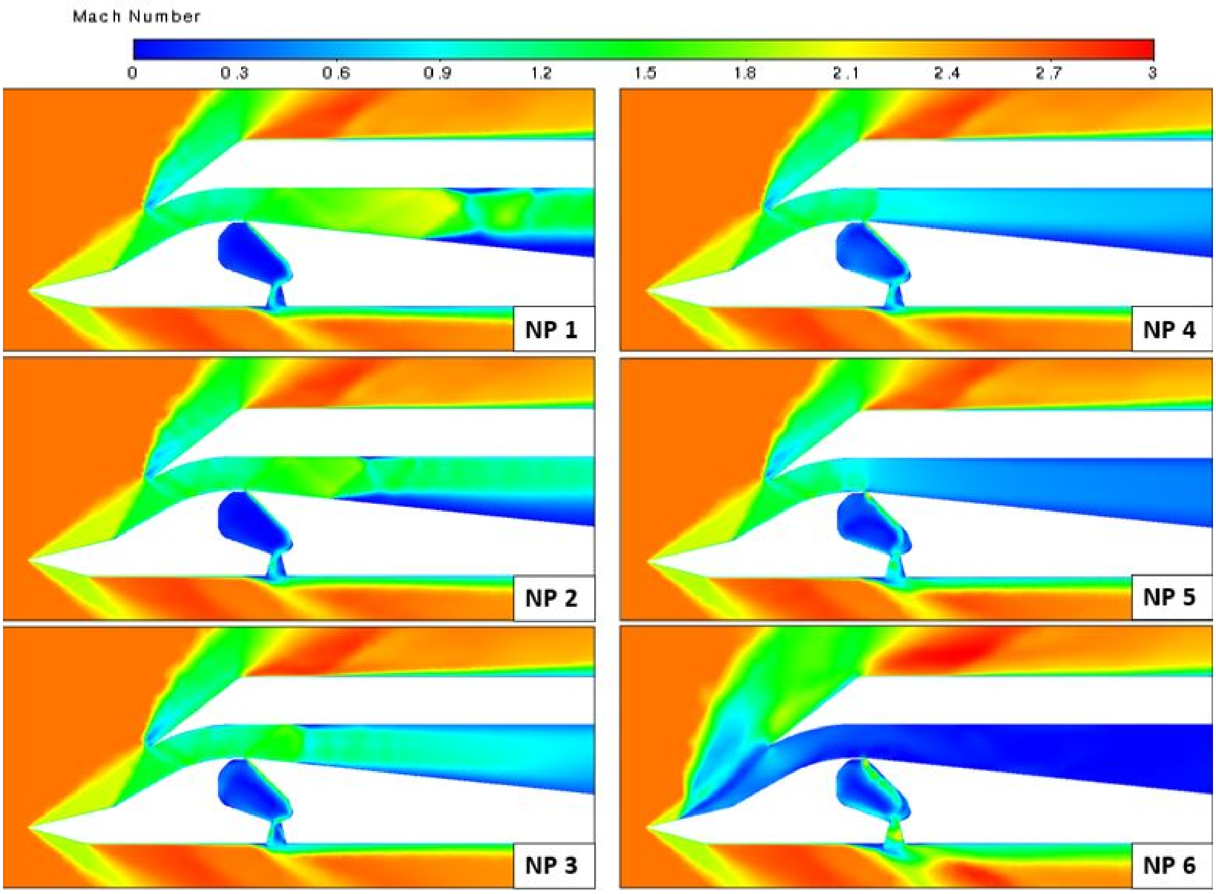

Figure 7: Variation of Terminal Shock Position with Nozzle Parameter

A Mach number contour plot with narrowed range to identify the subsonic and supersonic regions is presented for NP3 in Figure 8. Range is limited between sonic condition and a low supersonic value. Terminal shock location and shock-train structure in isolator can be identified from this figure. Pseudo-nozzle with a large throat operates in supercritical regime. Throttling of nozzle results in simulation of greater combustion chamber pressure and the shock system in the isolator section moves through the upstream. Further throttling eventually results in expulsion of the shock system, which leads to unstart of intake.

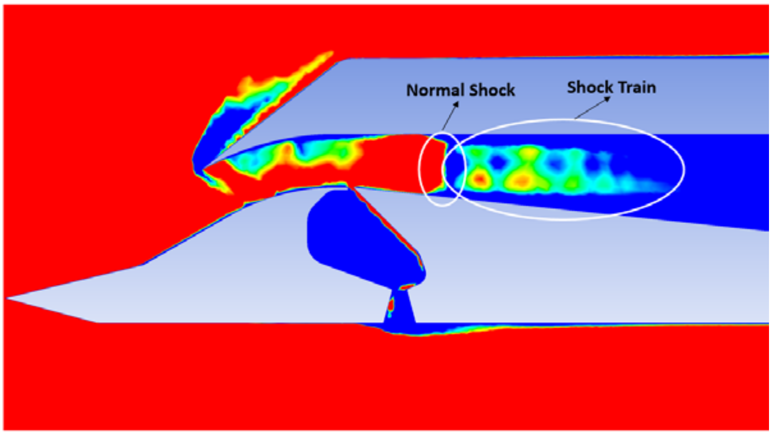

Figure 8: Terminal Normal Shock and Shock Train for NP3

Static pressure contours at different nozzle parameters are presented Figure 9. Shock structure and pressure level on external compression surfaces are same until the critical point. When inlet operates at subcritical condition a strong shock is generated on ramps. Critical point corresponds to maximum pressure recovery, and a further increase of back pressure results in buzzing of the air intake.

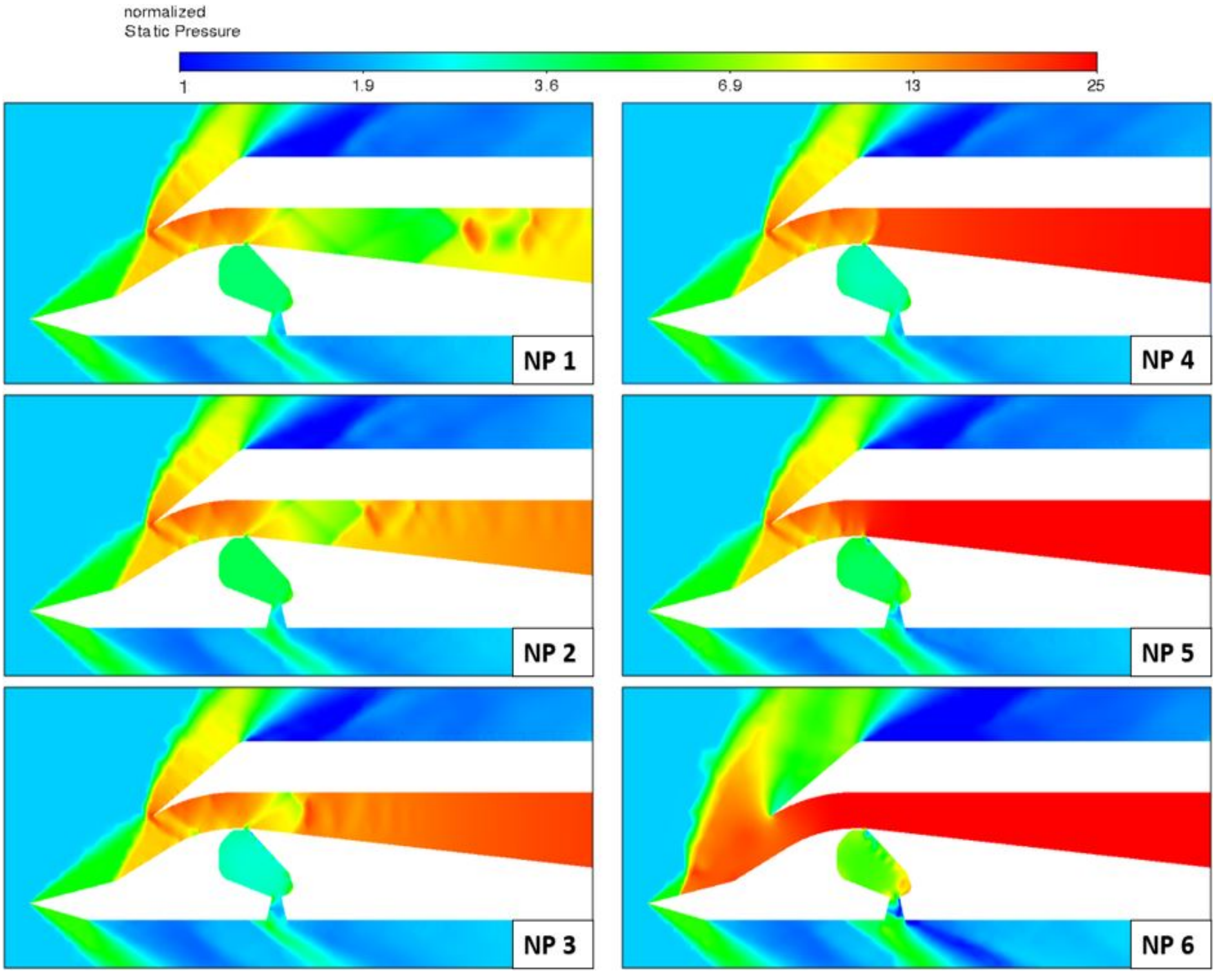

Figure 9: Static Pressure contours at different nozzle parameters (logarithmic scale)

Performance map of air intake obtained with wind tunnel tests and CFD simulations is shown in Figure 10. Pressure recovery and mass flow ratio results of CFD simulations and wind tunnel tests are in good agreement in terms of mass flow ratio and pressure recovery. According to the observations In tests, when the shock system expelled out of intake there exists an unsteady flow regime in which position of shock waves change in time. Steady numerical calculations are not adequate to simulate unsteady characteristics in buzzing mode, whereas time-accurate transient simulations are too expensive for this problem in terms of CPU usage. Since unsteady operation is not desired during the mission of air intake, steady CFD is still a powerful tool to obtain critical point before buzzing region.

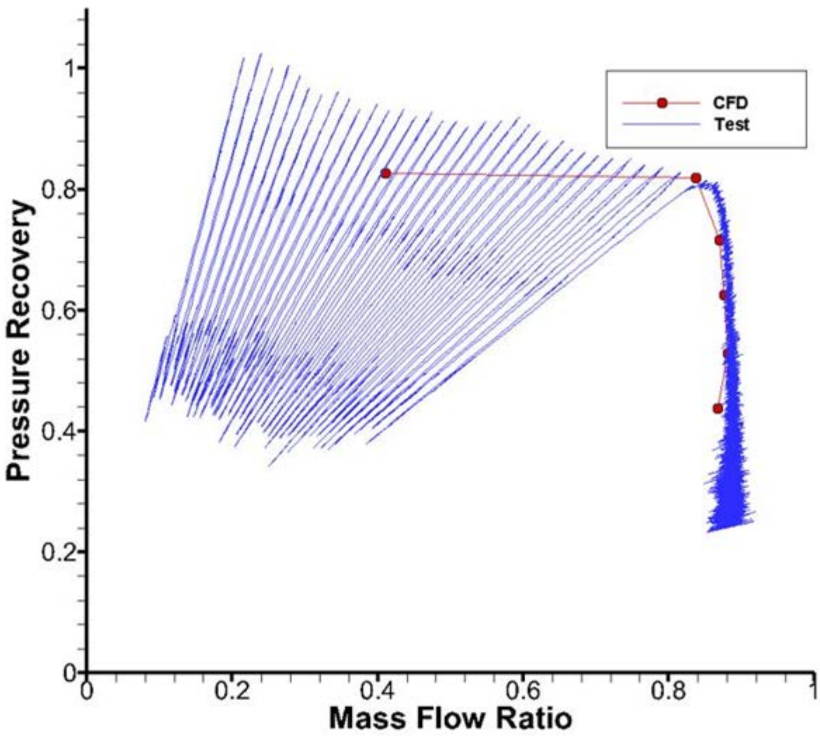

Figure 10: Comparison of Performance Curves

CONCLUSION

In order to design an air intake via wind tunnel tests only is inefficient and expensive. Many conceptual alternatives are needed to observe the performance characteristics to ‘freeze’ the design. Manufacturing wind tunnel test models, adjustment of the experimental equipment and testing require time and effort. Employing validated CFD tools beside the wind tunnel tests could make this process ‘reduced time and make process efficient. For this purpose, experimental results are compared with CFD tools in this study. Comparison is conducted via performance curves of the air intake. It can be seen that performance curves are similar both obtained from experiment and CFD analysis at the stable region. Although behavior of an air intake at the buzzing region could point out some characteristics, performance curves at the stable region is superior. In conclusion, number of wind tunnel tests on the design process could be reduced with CFD tools. It provides more agility to observe more design alternatives in a certain time.

REFERENCES

Bauer, C., & Kurth, G. (2011). Importance of the Bleed System on the Overall Air Intake Performance. *47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit.* San Diego, CA: AIAA 2011-5759. doi:10.2514/6.2011-5759

Can Çıtak, Özgür Harputlu, Ezgi Arısoy, Tekin Aksu and M. Tuğrul Akpolat. (2019). Computation Methodology of Performance Characteristics for a Supersonic Air Intake. *Ankara International Aerospace Conference.* Ankara.

Eriksson, L., Johansson, U., & Borg, R. (1993). CFD Analysis and Testing on a Twin Inlet Ramjet. *AIAA/SAE/ASME/ASEE 29th Joint Propulsion Conference and Exhibit.* Monterey, CA: AIAA 93-1839. doi:10.2514/6.1993-1839

Fukuda, M., Roshotko, E., & Hingst, W. (1975). Control of Shock-Wave Boundary-Layer Interactions by Bleed in Supersonic Mixed Compression Inlets. *AIAA/SAE 11th Propulsion Conference.* Anaheim, CA: AIAA 75-1182. doi:10.2514/6.1975-1182

Herrman, D., Blem, S., & Gülhan, A. (2011). Experimental Study of Boundary-Layer Bleed Impact on Ramjet Inlet Performance. *Journal of Propulsion and Power, 27*(6), 1186-1195. doi:10.2514/1.B34223

Herrmann, D., & Gülhan, A. (2015). Experimental Analyses of Inlet Characteristics of an Airbreathing Missile with Boundary-Layer Bleed. *Journal of Propulsion and Power, 31*(1), 170-189. doi:10.2514/1.B35339

Herrmann, D., Siebe, F., & Gülhan, A. (2013). Pressure Fluctuations (Buzzing) and Inlet Performance of an Airbreathing Missile. *Journal of Propulsion and Power, 29*(4), 839-848. doi:10.2514/1.B34629

Slater, J., Davis, D., Sanders, B., & Weir, L. (2005). Role of CFD in the Aerodynamic Design and Anlaysis of the Parametric Inlet. *ISABE Pağer 1168*.

Soltani, M., Daliri, A., Younsi, J., & Farahani, M. (2016). Effects of Bleed Position on the Stability of a Supersonic Inlet. *Journal of Propulsion and Power, 32*(5), 1153-1166. doi:10.2514/1.B36162

Srinivasan, G., & Sinhamahapatara, K. (2017). Large Eddy Simulation of Supersonic Intakes Using Fifth Order Free-Stream Preserving WENO Scheme. *23rd AIAA Computational Fluid Dynamics Conference.* Denver, CO: AIAA 2017-3614. doi:10.2514/6.2017-3614

Trapier, S., Duveau, P., & Deck, S. (2006). Experimental Study of Supersonic Inlet Buzz. *AIAA Journal, 44*(10), 2354-2365. doi:10.2514/1.20451