

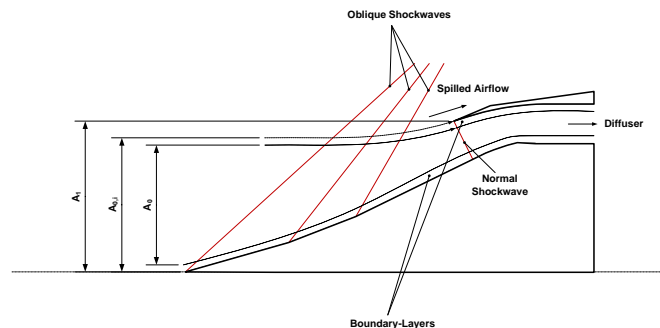
## CHARACTERIZATION OF SUPERSONIC INTAKE FOR SPIN-STABILIZED FLIGHT VEHICLES ALONG A BALLISTIC TRAJECTORY USING CFD

ADRIAAN JT STEENKAMP<sup>1</sup>, MADELEINE L COMBRINCK<sup>1</sup>  
<sup>1</sup>*Flamengro, a division of Armscor SOC Ltd., [adriaans@armscor.co.za](mailto:adriaans@armscor.co.za)*

**Keywords:** *External Aerodynamics, Compressible Flow, Supersonic Flow*

The supersonic intake for ramjet applications is characterized by complexities of high speed compressible flow and its transition from supersonic to subsonic flow. CFD is used to capture the flow phenomena such as shock reflections, flow separation, and shock-boundary layer interaction. Operating parameters like pressure recovery, mass flow rate, and the terminal shock location are estimated through this analysis.

Compression is achieved by decelerating the air stream through multiple shocks. The first step is an external compression which is achieved by a conical shock generated by the fore-body of the intake. The internal compression is achieved by one or more reflected oblique shocks followed by a normal shock which decelerates the supersonic flow to subsonic. A typical shock structure inside a supersonic intake is illustrated below in Figure 1.



**Figure 1: Typical shock structure inside a supersonic intake**

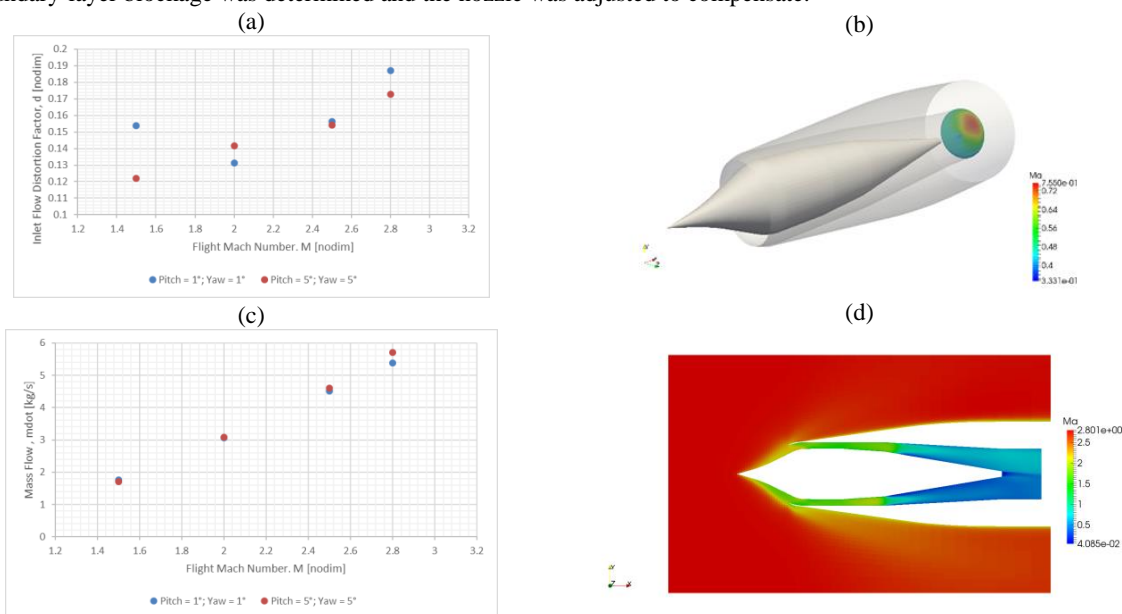
The situation is complicated significantly by the fact that the application for which the intake is envisaged requires adequate performance for a spin-stabilized flight vehicle traveling along a ballistic trajectory where due to the varying interactions between the aerodynamic and dynamic forces a significant departure from the design conditions for the intake. General, one-dimensional flow techniques are inadequate to describe the intake function and therefore three-dimensional CFD must be utilized for the analysis of the ERM intake.

Numerical computations on supersonic intakes have been carried out by several researchers and the complexities involved in applying CFD-based methods for a supersonic intake analysis is summarised in [1]. Emphasis is laid on the overall solution approach, application of outlet boundary conditions and appropriate turbulence modelling. The validation of the NASA WIND CFD code with emphasis on supersonic intakes are presented in [2]. One-equation Spalart-Allmaras [3] and two-equation Chien  $k-\epsilon$  [4] turbulence models are recommended for capturing the boundary layer. Finally [5] carried out numerical computations on an axisymmetric supersonic intake. The modelling approach specified in [5] was adopted for this work. The primary difficulty for the CFD simulation was found to be the establishment of the terminal shock. We present a novel approach. In CFD methods, the boundary condition at supersonic intake are fixed and outflow conditions are adjusted to get the applied exit static pressure at every iteration. The simulation involves time marching of the initial guess flow field. If the flow field is initialized with supersonic flow, then the backpressure is not sensed by the solution domain and a fully supersonic exit would be obtained, which is undesirable. However, if subsonic conditions are initialized, then a high static pressure at exit generates a transient flow into the duct from the exit. This highly reflective nature of pressure boundary condition produces non-physical transients with abnormally high total pressure in the upstream. These transients ultimately lead to divergence of simulation. Several code specific strategies exist for applying the outflow pressure boundary condition [1]. These include moving the outflow boundary downstream of the flow domain so that the boundary condition has minimal impact on the local flow field near the intake entry. This increases the number of iterations for converged solution and corresponding computational time; the mass flow rate at the exit may be specified and the exit static pressure is continuously adjusted. Though [1], [6] reported this to be efficient for finding performance plots, this method does not provide the solutions with various terminal shock locations within the duct.

Finally, a converging-diverging nozzle may be attached at the end of the intake section to obtain supersonic exit out from the modified domain. The procedure requires changing the area of nozzle to obtain terminal shock and specified exit pressure. The flow-field was initialized by intelligently assuming a terminal shock position and setting the flow upstream of the terminal shock to the freestream condition. The flow downstream of the terminal shock was determined by applying the Rankine-Hugenoit equations across the normal shockwave.

A final complication relates the effect of the rotating walls on the internal flow through the intake. For the present applications spin-rates of 1900 rad/s are nominal. The effect of the spin-stabilization on the intake internal flow was assessed using two methods which were subsequently compared – it was found that the internal flow is largely irrotational and the mass flow reduction due to spin is therefore approximately negligible.

The 3D domain was discretized using approximately 700 000 volumes. The flow-field was analysed using a derivative of the compressible flow solver *sonicFoam* which is a part of OpenFOAM® [7]. The governing equations describing unsteady, turbulent fluid systems comprise balance equations for mass continuity, momentum and energy. The simulations were executed on the Flamengro IBM BladeCenter system. Each case was deployed across 2 cores and 10 cases were executed in parallel. A total of 150 cases were deployed and we present the use python scripts to control case deployment, management and post-processing. Satisfactory temporal convergence was reached in 5 hours CPU time as measured on the convergence of the pressure forces and the viscous forces indicating that both the inviscid and viscous portions of the solutions had reached convergence. The grid resolution on the intake surfaces was not sufficiently fine and relatively high  $y^+$  values were observed. This however is considered a fair trade-off against the computational cost related to a finer resolution in the flow-field near the walls. Through the use of OpenFOAM® it was possible to provide rapid and accurate estimates of the intake performance across its anticipated operational envelope. The flow distortion at the intake exit was found to be minimum at the design flight Mach number for all pitch and yaw angles considered (see Figure 2(a) and (b)). The reduction in the design mass-flow (see Figure 2 (c)) due to boundary-layer blockage was determined and the nozzle was adjusted to compensate.



**Figure 2: Results for the intake performance at sea-level conditions (a) distortion; (b) location of the intake exit plane; (c) mass-flow; (d) Mach number at Mach 2.8**

**References**

- [1] J. W. Slater, “CFD methods for computing the performance of supersonic inlets,” *ratio*, vol. 9, no. 2, p. 5, 2004.
- [2] J. W. Slater, J. M. Abbott, and R. H. Cavicchi, *Validation of WIND for a Series of Inlet Flows*. National Aeronautics and Space Administration, Glenn Research Center, 2002.
- [3] P. SPALART and S. ALLMARAS, “A one-equation turbulence model for aerodynamic flows.”
- [4] K.-Y. Chien, “Predictions of channel and boundary-layer flows with a low-Reynolds-number turbulence model,” *AIAA J.*, vol. 20, no. 1, pp. 33–38, 1982.
- [5] S. S. Gokhale and V. R. Kumar, “Numerical computations of supersonic inlet flow,” *Int. J. Numer. Methods Fluids*, vol. 36, no. 5, pp. 597–617, 2001.
- [6] J. W. Slater, *Verification assessment of flow boundary conditions for CFD analysis of supersonic inlet flows*. National Aeronautics and Space Administration, Glenn Research Center, 2002.
- [7] O. OpenCFD, “The Open Source CFD Toolbox,” *User Guide OpenCFD Ltd*, 2009.