A CFD Study on Scramjet Intake Using HiFUN

Pravin Bappaji Havale

B.Tech in Aeronautical Engineering, Annasaheb Dange College of Engineering and Technology, Ashta

Abstract—**SCRAM jet engines are external compression engines used for hypersonic flight vehicles. They comprise of an inlet spike over which most of the compression takes place due to the formation of shockwaves and cowl that deflects shocks into the engine. Now that space exploration has matured, there is a need to study and develop faster methods of propulsion. In this study we are going to validate the results from K. Sinha et al. (2016), simulate the case at on-design Mach No. for Different Angles of Attack, and Compare the variation of pressure in the isolator region at different Angles of Attack.**

I. INTRODUCTION

Supersonic combustion ramjets (scramjets) are essential for achieving hypersonic speeds. Unlike conventional engines, scramjets rely on the high-speed compression of incoming air through the intake, without the need for rotating parts. The geometry of the intake plays a significant role in determining the efficiency of the engine by influencing the air compression process.

This study aims to evaluate the performance of a 2D scramjet intake using HiFUN (High Resolution Flow Solver on Unstructured meshes Flow solver). The intake is analyzed at four different Mach numbers (4.5, 5.5, 6.5, and 7.5) to observe how shockwaves interact and influence pressure distribution in the isolator region. The study draws inspiration from research by Krishnendu Sinha et al. (2016) and aims to contribute to the development of efficient hypersonic propulsion systems.
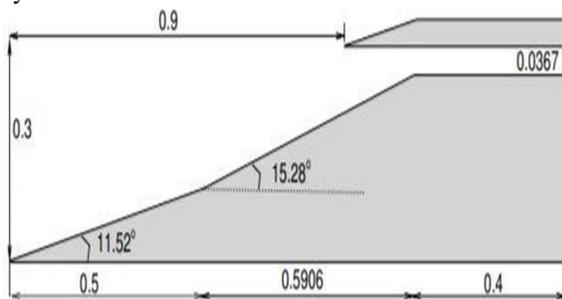

Fig 1. Hypersonic Intake Geometry [1]

II. PROBLEM STATEMENT

The objective of this study is to analyze the airflow through a scramjet intake using HiFUN CFD at Mach numbers 4.5, 5.5, 6.5, and 7.5. The analysis is conducted to observe the pressure distribution and Mach number variations in the isolator region, which plays a critical role in the performance of the engine. This study also aims to validate the results against theoretical data, identifying potential improvements for scramjet intake designs.

III. GOVERNING EQUATIONS

Conservation of Mass equation follows directly from the control volume equation; by applying Gauss Divergence theorem, we can transform the surface integral into a volume integral finally becoming the Equation shown below

3.1 Conservation of Mass

$$\frac{\partial \rho}{\partial t} + DIV(\rho v) = 0 \tag{1}$$

3.2 Momentum Equation (Inviscid Euler Equation)

$$\frac{\partial(\rho u)}{\partial t} + \nabla(\rho u u^t) + \nabla p = F \tag{2}$$

Where ρ is density, p is Pressure
u is velocity
F is the volume Force

3.3 Energy Equation

$$\frac{\partial e}{\partial t} + \nabla.\left((e + p)\,u\right) = Q \tag{3}$$

Where e is the total energy per unit volume
u is velocity
p is the pressure
Q is the heat source

3.4 Ideal Gas Law

Using equation 4 we get pressure as 2130 pa. These parameters will remain the same for all the cases that are going to be run.

$P = \rho \times R \times T$ (4)

Where ρ is the density of air

R is the ideal Gas constant

T is the Temperature

3.5 Geometry and Mesh

The scramjet intake geometry consists of the inlet, outlet, spike, and cowl. The scramjet intake geometry was modeled for 2D simulation with a total length of 1.4906 meters and a breadth of 0.3 meters. The spike and cowl were angled at 11.52° and 15.28°, respectively, to induce oblique shocks for air compression. A minimal thickness of 0.035 meters was assigned to simulate the flow in a quasi-2D environment.

The mesh consisted of 110,000 hexahedral cells, ensuring detailed resolution of shockwave interactions and pressure gradients within the isolator. Mesh refinement was applied to key areas like the shock formation regions and isolator to accurately capture flow dynamics. The mesh was converted into OpenFOAM readble format that is "fluentMeshToFoam" for use in the HiFUN solver.
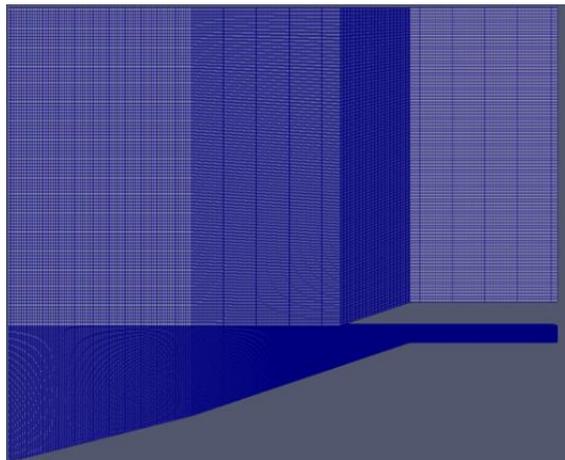

Fig 2. Mesh Region

3.6 Boundary Conditions

The boundary conditions used for the patches are as shown below in Table 1. The Temperature at inlet is 219.3K and the Pressure is 2162pa.

| Boundary Name	U	T	P
inlet	fixedValue	219.3 K	2162 Pa
outlet	supersonicFreeStream	inletOutlet	waveTransmissive
top	supersonicFreeStream	inletOutlet	Zero
outlet_spike	Zero	Zero	Zero
cowl	slip	Zero	Zero
spike	slip	Zero	Zero

Table 1 Boundary Conditions

3.7 HiFun solver and simulation controls

The HiFun solver was employed for simulating inviscid flow through the intake. No turbulence model was applied, and the simulation was kept inviscid. Thermophysical properties were defined using a mixture model, and the solver was set up for steady-state simulations. For each Mach number (4.5, 5.5, 6.5, 7.5), a steady simulation was created, and the solver executed 10,000 iterations to ensure convergence. Parallel processing was enabled to reduce computational time, and the CFL control was automatically adjusted for each case.

IV. RESULT AND ANALYSIS

4.1 Pressure Contours at various Mach No.

In Fig 3 we can see the pressure contour comparing the result with literature. From Fig 3 we can see the two oblique shocks intersect at the tip of the cowl and gets reflected into the isolator region of the engine. This condition is known as the Shock-on-lip condition. This shows the pressure contours at Mach 6.5 which is the On-Design condition.
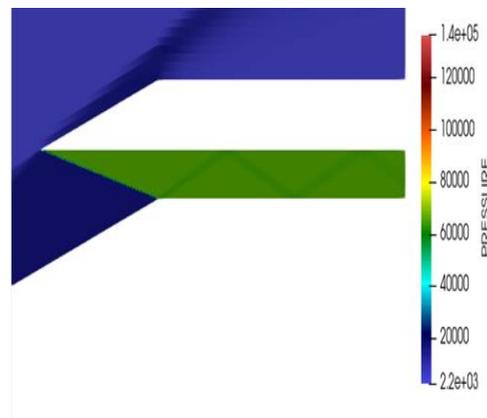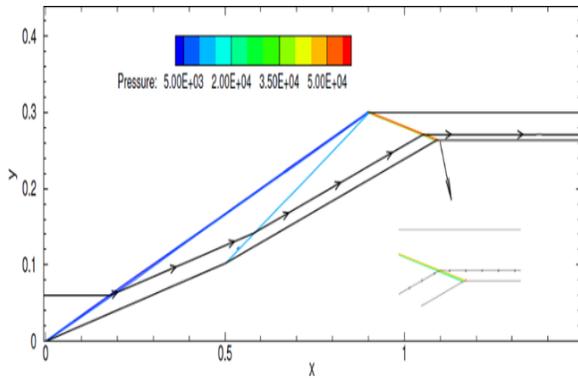
Fig 3. Simulated Pressure contour at Mach 6.5 (top)

Pressure contour at Mach 6.5 from literature [1] (bottom)

The pressure contours for the off-design conditions are given in fig 4. The comparisons will be clearer if we view the results in a table with the values of Mach No. as shown in the table 2.
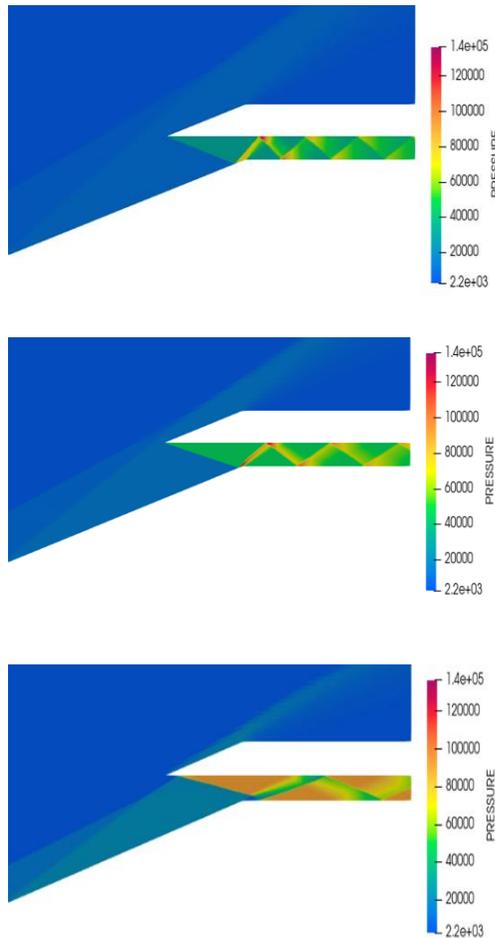

From Fig 4, we can see the variation of the pressure contour inside the isolator region as Mach No. is varied. This leads to uneven distribution inside due to the reflected shock waves. At lower Mach Numbers (Mach 4.5, Mach 5.5), the shocks formed by the two compressions wedges do hit the cowl wall and thus leads to reduction in capture area and at higher Mach number, the shocks intersect and hit the cowl resulting in reflected shock waves continuing throughout the isolator region.

In table 2 below, the Mach No. inside the isolator region is compared with the results obtained in literature [1] and the error percentage between the simulated results and literature is calculated.

Mach No.	Mach No. isolator	(Error %)
4.5	2.10774	13.8
5.5	2.65361	6
6.5	3.12478	0.07
7.5	3.524	4.6

Table 2 Mach No. inside isolator at various Free-stream Mach No.

4.2 Pressure Contours at different Angles of Attack at Mach 5.5

The pressure contours at $-2°$ and $2°$ Angle of Attack are shown below in Fig 5. The pressure at the isolator outlet is going to be viewed in table 3.
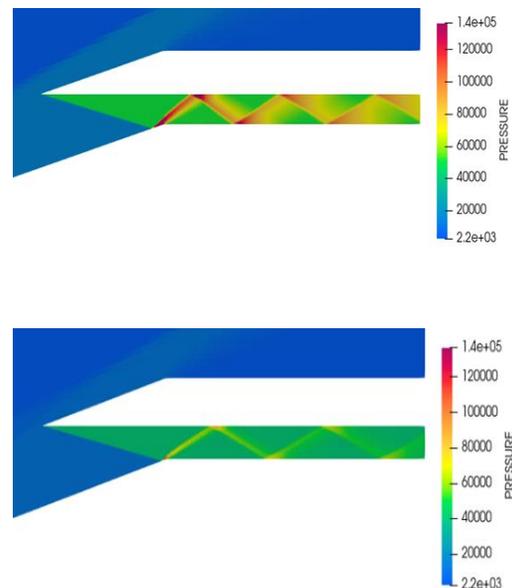

Fig 4. Pressure contour at various off-design Mach No. Mach 4.5 (top), Mach 5.5 (middle), Mach 7.5 (bottom)

Fig 5 Pressure contour at -2 AOA (top), 2 AOA (bottom)

Angles of Attack	-2	0	2
Pressure(pa) isolator (CFD Simulation)	67433.73	56570.59	47101.24
Pressure (Pa) isolator	65314.2	53816	43905.1
Error (%)	3	5	7

Table 3 Pressure variation (Isolator region) at different AOA

As Angle of attack increases the intersection points of the two-shock formed by the wedges moves upstream and away from the cowl leading edge. This causes a reduced capture area resulting in a drop in the pressure in the isolator as shown in Table 3.

V. CONCLUSION

This study presents a detailed CFD analysis of a scramjet intake using HiFUN. The intake performed optimally at Mach 6.5, achieving the shock-on-lip condition necessary for efficient scramjet operation. While the results at lower Mach numbers showed higher errors, the overall accuracy of the HiFUN solver was validated against theoretical models. Future work could focus on including viscous effects and turbulence models to improve the accuracy of the simulations, particularly for off-design conditions.

REFERENCES

[1] Krishnendu Sinha, V. Jagadish Babu, Rachit Singh, Subhajit Roy, Pratikkumar Raje, "Parametric Study of the Performance of Two-Dimensional Scramjet Intake," 18th Annual CFD Symposium, Bangalore, 2016.

[2] Sahil Deepak Kukian, "A CFD Study on 2D SCRAM Jet Intake Using OpenFOAM," FOSSEE Fellowship 2020, IIT Bombay.

[3] Anderson, J.D., *Modern Compressible Flow*, McGraw Hill, New York, 1984.