FOSSEE, IIT Bombay
FOSSEE Fellowship 2020
June 20, 2020

A CFD study on 2D SCRAM jet intake using OpenFOAM

Sahil Deepak Kukian

B.Tech in Aeronautical Engineering

Manipal Institute of Technology, Manipal

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.

1. Introduction

Supersonic flow is characterized as flow that is above 1.2 Mach. For this project we are going to study the shock interaction at High supersonic flows with Hypersonic Intake that is designed for optimum Operation at Mach 6.5. For this we use the rhoCentralFoam Solver and ParaView for the visualization. One of the major issues with solving solutions at such high velocities is that we then need to consider the viscous interaction effects and high Temperature effects which add another level of complexity to the solution. Although we will not be considering those effects in this study, it does play a major role in real world hypersonic aerodynamics. The Intake design we are going to be considering is similar to the Design from K.Sinha et al (2016)[1]. This design is known as mixed compression intake which is also the most widely used design due to the shorter length and lower Drag. The other intake types are External Compression intake and Internal Compression Intake.

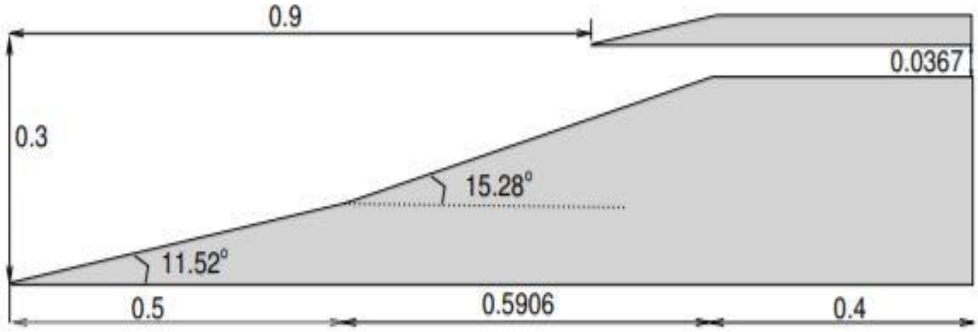


Fig 1. Hypersonic Intake Geometry [1]

2. Problem Statement

To Study a 2D SCRAMJET intake design at various Mach Numbers and Angles of Attack (AOA) using the compressible OpenFOAM solver “rhoCentralFoam”. The case is simulated at 26km altitude with temperature 219.3k and air density of 0.03436 kg/m^3.

3. 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

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

The Inviscid Euler equation is given below

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

Where ρ is density, p is Pressure

u is velocity

F is the volume Force

The energy equation is given below

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

Where e is the total energy per unit volume

u is velocity

p is the pressure

Q is the heat source

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.

$$P = \rho \times R \times T \tag{4}$$

Where ρ is the density of air

R is the ideal Gas constant

T is the temperature

We are going to be evaluating the flow at various different Mach No. and comparing the performance parameters with the on-design parameter (i.e. Mach No 6.5).

Also to calculate the Velocity values at various Mach numbers we use the equations shown below. From equation 5, we can calculate the speed of sound.

$$a = \sqrt{\gamma \times R \times T} \tag{5}$$

Where R is gas constant (287 J/kgK)

T is temperature (K)

γ is Specific Heat ratio (assumed 1.3)

Once the Speed of sound is calculated, we use Equation 6, shown below to calculate the velocities at their respective Mach numbers.

$$V = M \times a \tag{6}$$

Where M is Mach number

a is speed of sound (m/s)

4. Case Setup

4.1 Geometry and Mesh

The SCRAM jet intake geometry consists of inlet, outlet, top, spike, cowl and oulet_spike. The total length of the model is 1.4906 m and 0.3 m in breadth. The angle of the first wedge is 11.52^0 and second wedge is 15.28^0 as shown in Fig 1. Since we want to simulate only the 2D simulation for this case but OpenFOAM operates only in 3D, so we assign a thickness of 0.035 m. The mesh can be seen in Fig 2. It consists of 110000 hexahedral Mesh elements made using Ansys meshing tool and the mesh has been designed in such a way to capture the oblique shocks and also the region inside the cowl. The mesh was exported into .msh format and then converted into OpenFOAM readable mesh by using the built-in function “fluentMeshToFoam “.

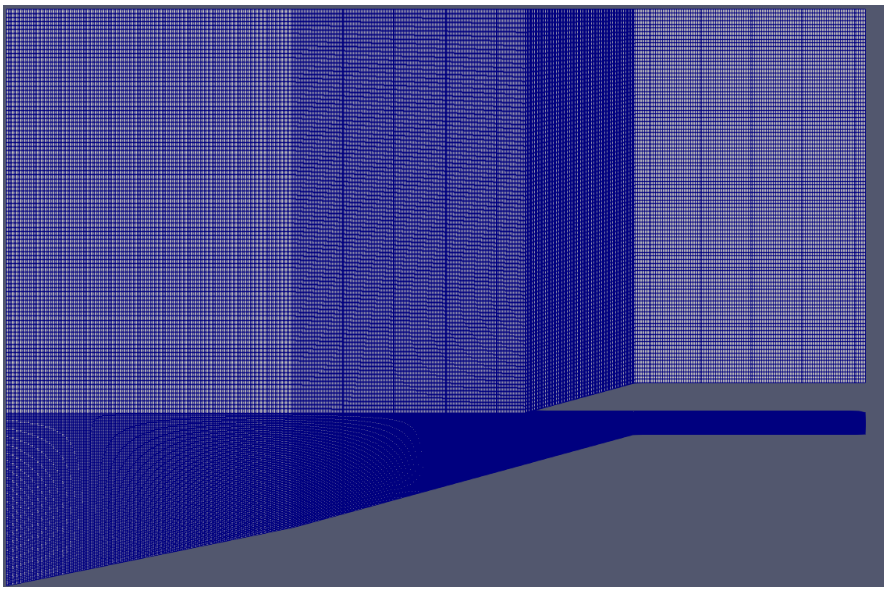


Fig 2. Mesh Region

4.2 Boundary Conditions

The boundary conditions used for the patches are as shown below in Table 1.

Selecting boundary conditions was one of the most difficult part of the simulation as incorrect selection will lead to the model diverging and not giving a good result.

Before being able to make the changes in the boundary conditions the necessary changes to the top, spike, outlet, spike_outlet and cowl must be done in the polyMesh folder after importing the mesh into OpenFOAM format. The frontAndBackPlanes must be changed into empty. All the others should be changed to patch. Inlet velocities are changed according to the Mach No to be simulated.

The Temperature at inlet is 219.3K and the Pressure is 2162pa.

Boundary Name	U	T	P
inlet	fixedValue	fixedValue	fixedValue
outlet	supersonicFreeStream	inletOutlet	waveTransmissive
top	supersonicFreeStream	inletOutlet	zeroGradient
outlet_spike	zeroGradient	zeroGradient	zeroGradient
cowl	slip	zeroGradient	zeroGradient
spike	slip	zeroGradient	zeroGradient
frontAndBackPlanes	empty	empty	empty

Table 1 Boundary Conditions

4.3 Solver and Simulation Controls

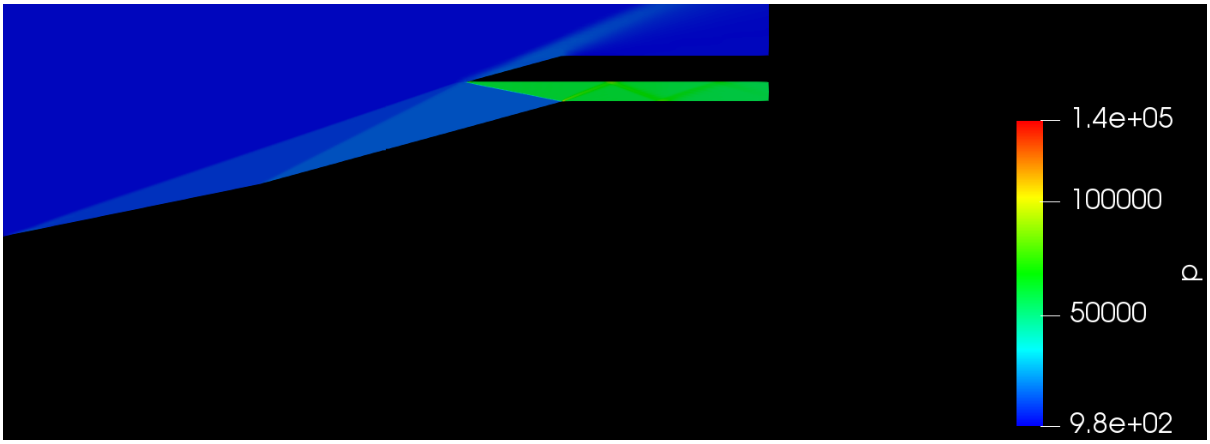
There is no special Turbulence model applied to this simulation. So in the turbulence type dictionary it is set to laminar.

As for the thermophysical properties, we are going to be using a mixture model with the properties as set in the dict file.

5. Result and Analysis

5.1 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.



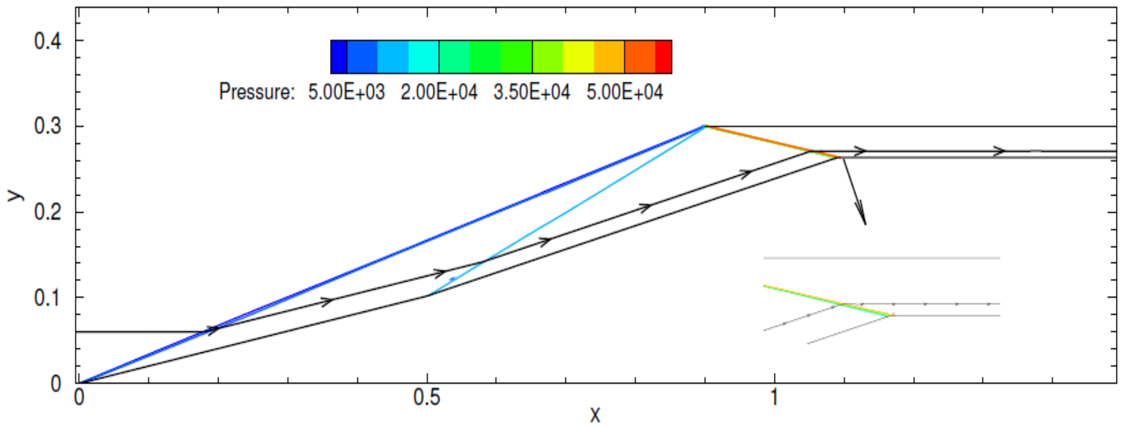


Fig 3. Simulated Pressure contour at Mach 6.5 (top)

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.

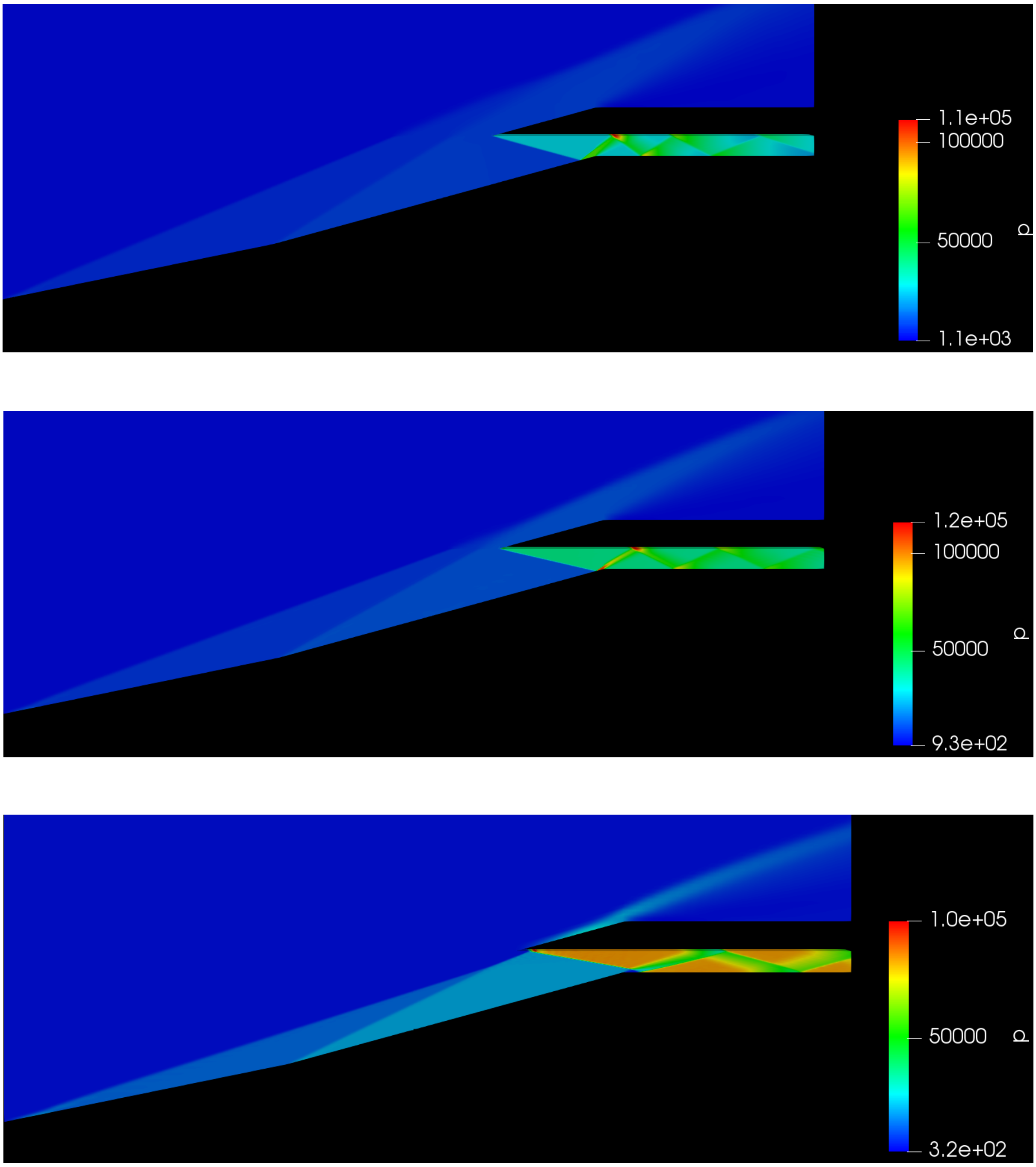


Fig 4. Pressure contour at various off-design Mach No.

Mach 4.5 (top), Mach 5.5 (middle), Mach 7.5 (bottom)

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 compression wedges does 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 (given in brackets).

Mach No.	Mach No. $_{isolator}$ (Error %)
4.5	2.5650 (4.69 %)
5.5	2.8376 (0.267 %)
6.5	3.1613 (1.324 %)
7.5	3.3521 (0.531 %)

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

5.2 Pressure Contours at different Angles Of Attack

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.

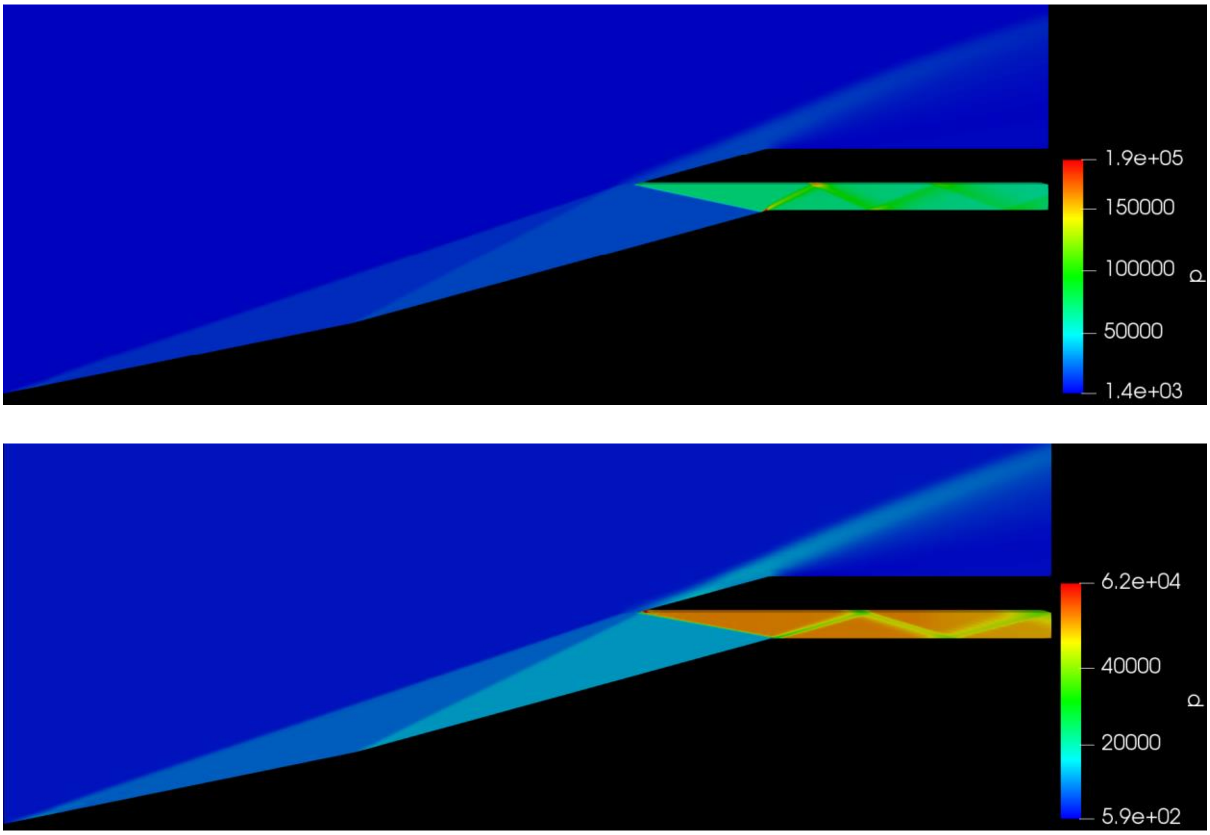


Fig 5 Pressure contour at -2 AOA (top), 2 AOA (bottom)

Angles of Attack	-2	0	2
Pressure(pa) isolator	67433.73	56570.59	47101.24

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.

6. Conclusion

This case study has explored the SCRAM jet intake design that has been validated from K.Sinha et al (2016) at the Different Mach No. At Mach No. 6.5 and at an Angle of attack 0^0, the shock-on-lip condition is achieved that results in optimal Air capture Area. When the results obtained by ‘rhoCentralFoam’ was compared with the results from literature, we found that errors percentages were below 5%. This means that rhoCentralFoam was able to accurately simulate complex flows at high velocities. This error percentage can be further reduced by using a refined mesh, turbulence models etc. Also we have simulated the case at Different Angles of Attack (AOA) and from the result we can infer that increasing the AOA results in a reduction in the pressure inside the isolator region(as shown in table 3) thus reducing the efficiency of the intake.

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, August 10-11, 2016, Bangalore

2. Anderson, J.D., Modern Compressible Flow, McGraw Hill Inc., New York, 1984.

3. J. H Perziger , M .Peric , Computational Methods Of Fluid Dynamics ,Springer, ISBN 3-540-42074-6 Springer-Verlag Berlin Heidelberg NewYork