ISSN No:-2456-2165

Boundary Layer Interaction in Hypersonic Air Intake

Dr. Manoj Veetil
Ph D (Cranfield University), Head of the Department
IIAEM – Jain University

Giri Pallavi
Assistant Professor
IIAEM – Jain University

Abstract:- The future development in aerospace field will be in hypersonic velocities and consequently hypersonic engines. Since the hypersonic engine is very complicated, design and simulation studies play a vital role in the analysis of flow properties. The main aim of this project was to understand the flow fields for a hypersonic intake. Model analysis using spatial grids were used to understand the effects of parameters which affect the operation of a hypersonic intake, these intakes were strongly influenced by the local grid resolution, and fine grids were generated at the intake section to check different parameters for small region (e.g. Pressure, Mach number, and Temperature). This also helped to reduce spillage and bow shock formation at the tip of cowl intake. Initially an optimum 2-Dimensional (2-D) design was selected, using a trial and error method. 3-Dimensional (3-D) hypersonic intakes were then designed using the final 2-D design dimensions. With the help of gas dynamics, isentropic and geometrical relations, different parameters were calculated. The calculated parameters were compared with the results obtained from the analysis. For designing and meshing, ICEM-CFD was used and numerical simulations were carried out in ANSYS-FLUENT.

Keywords:- *Hypersonic flow, Grid generation, Cowl intake, spillage, shock formation, Boundary Layer*.

I. INTRODUCTION

The hypersonic air breathing propulsion system consists of Scramjet (supersonic combustion ramjet) and ramjet plays a major role in high-speed engines. In the scramjet, the shock wave plays a vital role in the compression process instead of the compressor in a gas turbine engine. The large range of Mach numbers can be operated by employing multiple integrated propulsion systems. Hydrocarbon-fuel scramjet systems use two approaches. One dual mode scramjet system and another is the dual combustion ramjet system. The dual mode scramjet system consists of a combined combustor of ramjet and scramjet combustion whereas the dual –combustion ramjet (DCR) system is a combination of fuel rich gas generator and supersonic combustion. Advantages of both ramjet and scramjet combustion system in DCR helps to achieve higher combustion efficiency and performance.

II. DUAL COMBUSTION RAMJET

Billig from John Hopkins University first invented Dual combustion ramjet (DCR). DCR is the combination of both Scramjet and Ramjet combustion flow patterns into an integrated engine for achieving huge speed range and greater performance. Many experiments of direct-connected and free jet test, the Boeing Company conducted flight tests of a hypersonic flight (HyFly) missile. The DCR consists of two major components one is subsonic pre-heater and another is supersonic combustor. Without these two components, Combustion of liquid kerosene is difficult for supersonic flows. For pre heating liquid kerosene spray is used and divide it to molecular species subsonic pre-burner is used. The peripheral air of supersonic intake is mixed with the high-temperature fuel-rich gas for better performance. The DCR includes the best quality of ramjet and scramjet engine to attain high combustion efficiency and performance.

The Dual combustion ramjet (DCR) consists of many components like intake, pre-burner, combustor, and nozzle. The intake composes of four supersonic intakes and two subsonic intakes. Liquid Kerosene has three positions incorporated for the injector’s .The kerosene is infused into two subsonic intakes to achieve stoichiometric ratio for reliable ignition. The circumference of the pre-burner is used to cool the wall of the pre-burner before infusing kerosene into the centre section to crack into micro molecular holes. Lastly rich-fuel gas is used in the supersonic combustor. The air in the supersonic intake combusts and blends the fuel-rich gas with the blending layer, burning and high compressibility has a tendency to diminish the expanding of the blending layer. Ignition has a tendency to discharge the encompassing air through a most extreme thrust nozzle.

The four supersonic intakes are combined into a singular annular passage before entering in the supersonic combustor and the two subsonic intakes are connected with the head of the pre-burner. The figure of DCR shows the flow pattern and key sections. The DCR is split into 5 sections as the flow field of incoming air, different sections of intakes, the inlet of the pre-burner, intake of supersonic combustion and finally the exit of the combustor and nozzle. The pre-burner holds the same geometry as of supersonic intake with a convergent pipe. The supersonic combustor allows the thickness of the boundary layer due to the slight divergent pipe. An annular cavity is introduced in hypersonic intake. The effects of the cavity include.

- A subsonic zone in the cavity is helpful to stabilize flames
- An oblique shock wave induced by the cavity enhances the mixing of air and fuel.

DCR intake compresses the flow as efficiently as possible, minimizing the vicious and shock losses. The criteria for a hypersonic intake valve, in terms of design aspect, have been well documented in the literature.

- The compression of the flow at the intake should be done effectively in order to reduce the losses of both viscous and shock wave.
- The contribution of intake to vehicle drag has to be reduced.

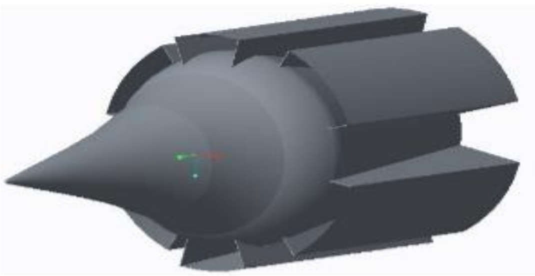

Fig 1:- 3D Hypersonic Air Intake

- The intake ought to act naturally beginning at the scramjet assume control Mach numbers (M1=4) and have the capacity to work on the required scope of Mach numbers, with no critical decay in execution.
- The performance of intake must not be lowered by operation.
- The intake should with stand the back pressures caused by the heat addition.
- The intake should have a higher capacity in order to tolerate the internal pressures and heat loads.
- Uniform speed profiles are by and large attractive at the admission exit.

Based on relations between the parameters namely pressure, velocity, temperature, Mach number, diameter and mass flow rate, theoretical calculations are done to determine the required dimensions of the hypersonic intake. A preliminary design was made using these theoretical calculations.

A structured Mesh was generated for two dimensional model of the hypersonic intake by using ICEM CFD software. The Mesh generated is used for flow simulation using Ansys Fluent. A steady state, density based solver was considered. The types of viscous model used were Spalart-Allamars and Transition kkl -omega model. The dimensions of the intake were modified based on the boundary layer thickness, depending on the required Mach number and pressure at the hypersonic intake from the simulations.

III. MODELLING

The curvature of the conical wall (critical operation) plays an important role in compressing air in the intake section. Since there are no parts to compress the air, formation of shock is essential for compression.

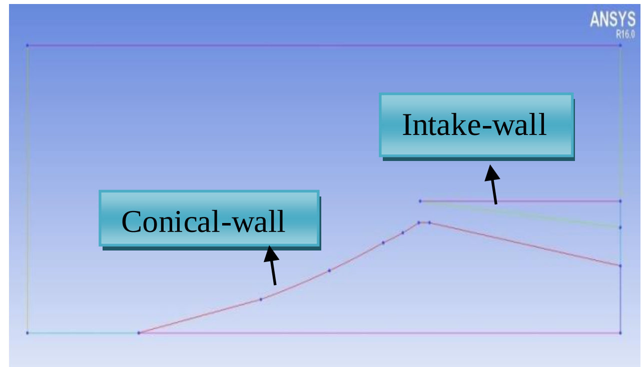

Fig 2:- Design of Hypersonic Intake

Items	VALUES
Total length of the DCR model	1800
Height of the DCR model	600
Length of the conical wall	1300
Height of the conical wall	140
Length of the intake wall	540

Table-1: Dimensions of Hypersonic Intake

Different mesh size generation has been studied for choosing the optimum mesh. As the mesh size goes up, time to compute increases as the complexity increases. The larger the density of meshing the greater is the accuracy of geometry and greater is the difficulty in solving the problem.

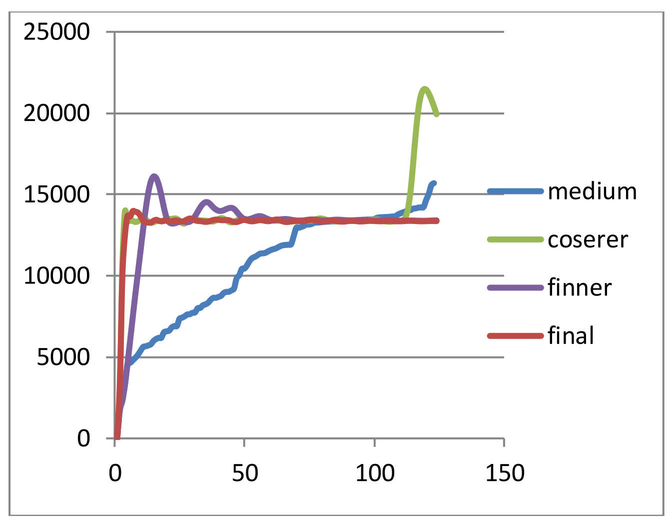

Fig 3:- Mesh study

TYPE OF MESH SIZE	NUMBER OF NODES
Medium	6000
Corserer	26000
Finer	120000
Final	66000

Table 2: Mesh study with Number of nodes

From the graph, it can be seen that the lines of finer and final are same with a different number of nodes. In the final number of nodes are lesser in number than finer so will consider the final mesh to be appropriate. The figure is the final mesh after the study of a different mesh generation with a different number of nodes.

- *Equations*

Mach-6
Stagnation temperature T_0 = 227.75 K
Stagnation pressure P_O = 1500Pascal

$$p_0 = p\left(1+\frac{\gamma-1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}}$$

$$p_0 = 1500\,(1+0.1(6)^2)^{\frac{1.4}{0.4}}$$

p_0 = 2368316.51 Pascal
p_0 = 23.68 Bar

$$\frac{T_0}{T} = 1+\frac{\gamma-1}{2}M^2$$

$$T_0 = 227.75\left(1+0.1(6^2)\right)$$

T_0=1867.55k

By using isentropic relations we can find the total pressure and the total temperature for Mach 6 condition.

IV. BOUNDARY CONDITIONS

- *Mach 6 Condition*

The boundary conditions are as follows:
For Inlet:
Total Pressure (Pascal) = 2368316
Supersonic/Gauge Initial pressure (Pascal) = 1500
Temperature (K) = 1867.
For Outlet
Gauge pressure (Pascal) = 1500
Back flow Total Temperature (K) = 227
Bottom edge is taken as the axis.
Upper edge is taken as far1
Supersonic/Gauge Initial pressure (Pascal) = 1500
Temperature (K) = 227.
For wall:
Operating condition=0.

V. GEOMETRY

This geometry was finalized after researching much geometry with different intake walls and the curved section. Different intake walls were changed to reduce the bow shock and spillage near the intake wall and the conical curve was modified to ensure the flow is uniform and to generate shocks inside the intake section to compress the incoming air.

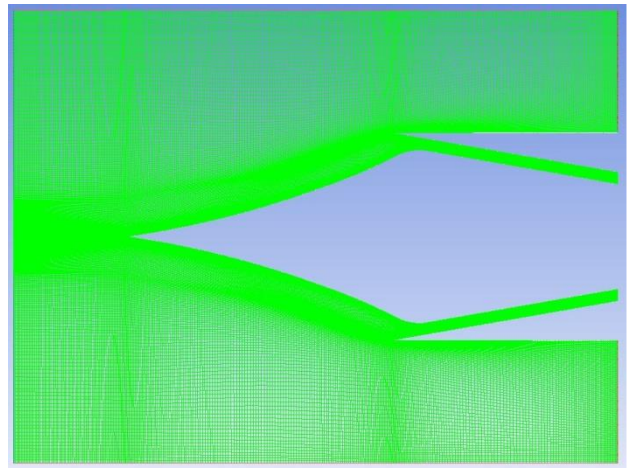

Fig 4:- Complete Mesh Generation

VI. RESULTS

A mesh file of the axi-symmetric model is imported in Ansys Fluent for further flow analysis. The model used in case-1 is Transition kkl-omega.

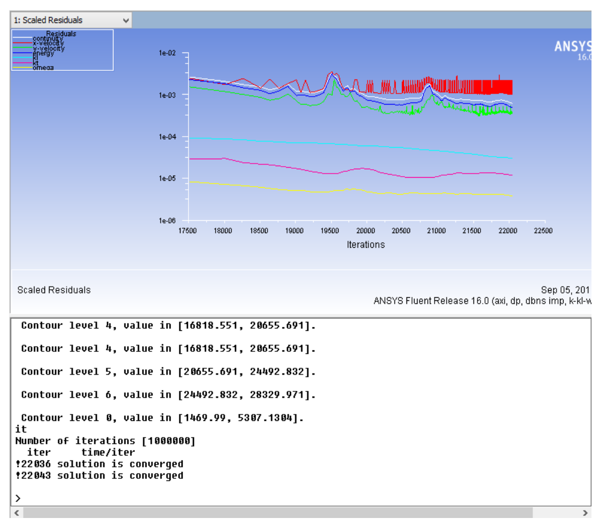

Fig 5:- Scaled Residuals of Mach-6

This picture shows the scaled Residuals of the simulations. Scaled residuals are the correct indicators for showing the convergence of many problems. If there are any disturbances in the flow simulations the graph in scaled residuals depicts it.

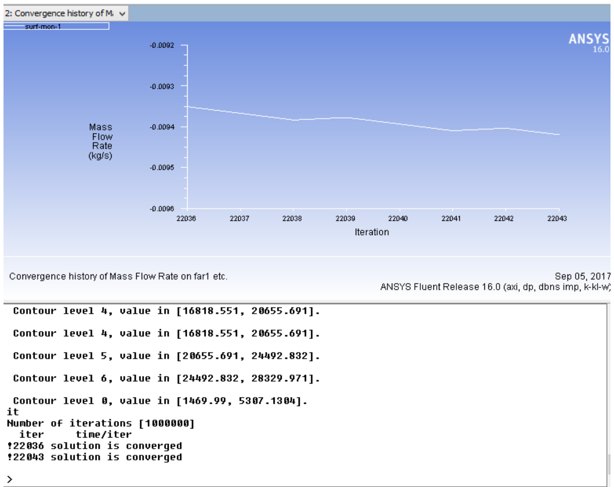

Fig 6:- Convergent history of the mass flow rate for Mach-6.

The above picture shows the convergent history of the mass flow rate. This graph shows whether the simulations have converged or diverged. After the solution has converged, check the difference between the mass flow rate at inlet and outlet as shown in above figure. The difference is very low then we can check various properties like Mach number, pressure, temperature, turbulence.

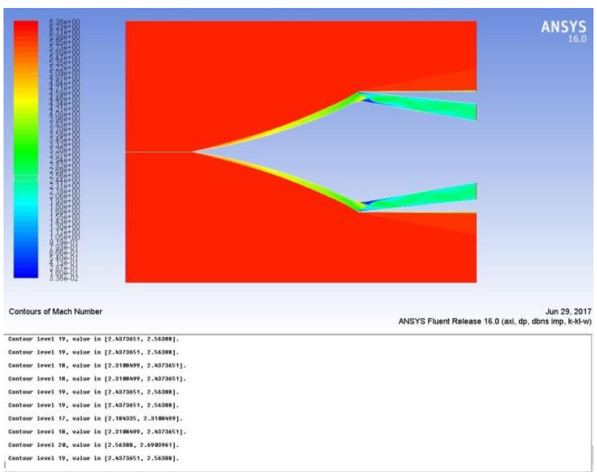

Fig 7:- Mach number Contours

The above picture is a contour of Mach number. In this figure, we see that there is no bow shock formation near the intake section in geometry. The flow is smooth on the conical curve without any spillage. The Mach number is varying in the intake section as we can see in the picture shown. The Mach number at the inlet is Mach is 6 [the red color in the picture] whereas the Mach number at the outlet is 2.4 [the green color in the picture]. The Mach number is varying from [6, 2.437] along the whole geometry as shown in the above figure.

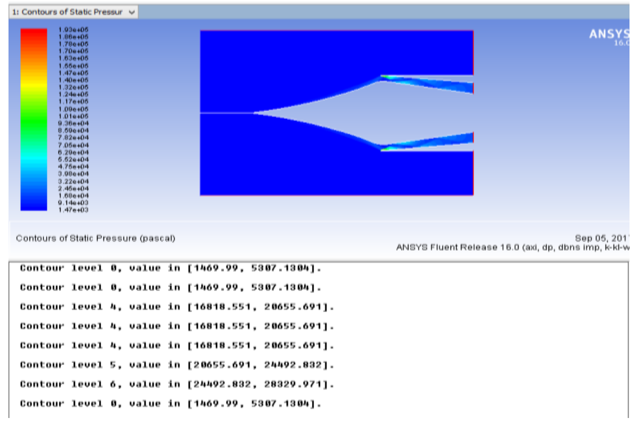

Fig 8:- Contour of Static pressure

The above picture is a contour of Static Pressure. The Static pressure is varying in the intake section as we can see in the picture shown. Static pressure at the inlet is 1500 [Blue color in the picture] whereas Static pressure at the outlet is 28329.971 [light blue color in the picture]. There are shocks generated inside the intake section this helps to compress the incoming air and send the air inside the combustion for further processing. The Static pressure varies from [1400, 24492.832] along the whole geometry as shown in the figure.

VII. CONCLUSION

A Brief study of the hypersonic engine is performed. To test the hypersonic engine in the ground huge high-speed wind tunnels are required, instead of that simulations are done to check the flow of hypersonic engines. ICEM CFD is used for creating the required geometry and mesh have to be generated for the designed geometry for better analysis of flow.

A preliminary design of the hypersonic intake is made for flow analysis and boundary conditions for both inlet and outlet are provided. The inlet and output conditions are calculated based on isentropic flow relations of the parameters such as pressure, Mach number, velocity, the mass flow rate and temperature. Hand calculations are done for both input and output conditions. After designing 2-Dimensional geometry, by using Computation Fluid Dynamics (CFD) flow analysis of Hypersonic intake has to be performed to check the flow in the intake. The calculated input Boundary conditions are used in flow analysis of the hypersonic intake. Simulation is performed for 2D models since both 2D and 3D geometries show the same results.

VIII. LITERATURE REVIEW

Jianguo Tan et al investigated the flow properties and performance of full DCR in Ma4/17Km and Ma6/25Km flight conditions are investigated through different experiments and numerical simulations. In different experiments the parameters of the minimum section of the intakes are simulated, the pressure distribution and the thrust are measured to evaluate the performance. In the numerical simulations, couple implicit

RANS equation, RNG k model,-ε turbulence are adopted .Both the numerical and experimental values are compared and they are appropriate.

Azam che idris and Mohd Rashdan saad et al In this paper,2-Dimensional inlet geometry was chosen to specify the design process. Designing a inlet section for achieving a shock on a lip condition for Mach 6 flow. The inlet was designed to have almost equal compression shock strength in order to maximize the total pressure recovery. The combined methods of experimental and numerical investigation for hypersonic inlet performance are measured. For this research Navier stroke equations are used to model the inlet in the 2-Dimensional environment and are then solved using commercial available software FLUENT.

Sarah Frauholz, Birgit U. Reinartz, Siegfried Muller and Marek Behr et al A brief understanding is given on how the transition modeling for hypersonic intake is analyzed using three-dimensional high-resolution simulations. A mesh-adaptive Reynolds-averaged Navier-Stokes solver is used with the Shear Stress Transport (SST) turbulence model. Computations with the SST transition model are compared to computations with the SST model (no transition model). A comparison to experimental results at M = 7 test condition shows the advantages of using the proposed transition model.

R. Sivakumar and V. Babu et al performance an experiment for the numerical simulations of the compressible, 3-D non reacting flow in the engine inlet section of a hypersonic air-breathing vehicle are presented. These simulations have been carried out using FLUENT. For all the results, the mesh has been refined to achieve area averaged wall. Mass flow rate through the intake and stagnation pressure recovery are used to compare the performance at various angles of attack. The calculations are able to predict the mode of air-intake operation (critical and subcritical) for different angles of attack. The numerical results are validated by simulating the flow through a 2-D mixed compression hypersonic intake model and comparing with the experimental data.

REFERENCES

[1]. R. Sivakumar and V. Babu.”Numerical Simulations of Flow in 3-D Supersonic Intake at High Mach Numbers”. Defence Science Journal, Vol. 56, No. 4, October 2006.

[2]. Azam che idris, Mohd Rashdan saad “Luminescent measurement system for the investigation of a scramjet inlet isolator”. AIAA, DOI: 10.3390/S140406606.

[3]. Jianguo Tan and Yi Wang “Free jet Experimental Investigation on performance of Dual-Combustion Ramjet” Journal of Propulsion and Power. DOI: 10.2514/1. B35263.

[4]. Joongki Ahn and Ryan P. Starkey.”Application of Evidence Theory on Dual combustion ramjet Model”, 49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference July 14 - 17, 2013, San Jose, CA AIAA 2013-3752.

[5]. A. J. Mathews and T. V. Jones. ”Design and Test of a Modular Wave Rider Hypersonic intake” AIAA 2005-3379, 13th International Space Planes and Hypersonics system.

[6]. A. J. Mathews and T. V. Jones. ”Design and Test of a Low Drag Hypersonic intake” AIAA 2003-7045, 12th International Space Planes and Hypersonics system and Technologies.

[7]. J. Fincham and N. V. Taylor “Free stream Disturbance Effects On An Axisymmetric Hypersonic Intake”, AIAA 2011-5760, 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, 2011, San Diego, California.

[8]. Robert D. Zucker & Oscar Biblarz “Fundamentals of Gas Dynamics”, 2nd edition.