3rd National Conference on Enhancement in Biomedical Engineering and Healthcare
Organised by Department of Biomedical Engineering,
Adhiyamaan College of Engineering, Hosur, Tamil Nadu, India

Flow Prediction in Scramjet Engine Inlet

T Anand[1], R Selva Kumaran[2], R Vijayan[3], M shalini[4]
[1]Assistant Professor, Department of Aeronautical Engineering, Adhiyamaan College of Engineering, Hosur, Tamil Nadu, India
[2,3,4]Student, Department of Aeronautical Engineering, Adhiyamaan College of Engineering, Hosur, Tamil Nadu, India

ABSTRACT

The primary purpose of an inlet for any air-breathing propulsive system is to capture and compress air for processing by the remaining portions of the engine. In a conventional jet engine, the inlet works in combination with a mechanical compressor to provide the proper compression for the entire driving force. For vehicles flying at supersonic ($1.5 < M < 5$) or hypersonic ($M > 5$) speeds, suitable compression can be attained by the inlet without a mechanical compressor. Because the inlet channel provides the airflow and compression ratios of the scramjet engine, an efficiently designed inlet is critical for the successful operation of the engine. Scramjet inlets are a key component in its function, and their design has stantial in several aspects. A computational study for scramjet inlet with different ramp angles and ramp lengths are analyzed to compress the air by sharp leading edge, fixing the whole cowl up or down. However, the performance of the inlet tends to degrade as Mach number range increases an air intake consisting of different ramps producing oblique shocks followed by a cowl shock. An imposing shock may force the boundary layer to separate from the wall,resulting in total pressure regaining losses and a reduction of the inlet efiiciency. Design an inlety to meet the necessities such as Low stagnation pressure loss, High static pressure and temperature increases and deceleration of flow to the desired value of mach number. A two-dimensional analysis was carried out in this project. ANSYS is used to create the Geometry. ICEM is used to create the mesh & FLUENT is used for analyzing the flow.

Keywords : Scramjet Inlet, Ramp angle, Ramp length, Cowl, Oblique shock, ANSYS

I. INTRODUCTION

Scramjet Flight demands sustained combustion for producing necessary thrust to counter the enormous drag that prevails in hypersonic flight. The design of hypersonic inlet for scramjet engine is pivotal to ensure stable combustion. As the hypersonic flight exceeds Mach 5, the residence time of the air inside the combustion chamber drops to a very low value which engenders the difficulty of burning. Also, very low static pressure prevails at cruise altitude >20km which compounds to this difficulty. The inlet serves to counter this problem by slowing down the head stream and increasing the pressure to provide favorable flow conditions for combustion. The current project involves comparison of performance.

Parameters for scramjet inlet which estimated as a result of FEM Computation of 2-D turbulent flow field for three ramp scramjet inlet geometry with no cowl lip deflection angle.

The boundary and initial conditions are select to the free stream conditions that pertain to a cruise altitude of 30km. The simulations achieved for four free stream Mach number 5, 6, 7 & 8. Thus from the obtained result, comparative studies of performance parameters are carried out by parameterizing geometrical variables and free stream Mach number. It is necessary to simulate the inlet design to obtain the appropriate inlet channel performance. Computational Fluid Dynamics (CFD) is used to analyze flight simulations in both steady and unsteady flow. A time-averaged, viscous, 2 Dimensional, CFD scheme used to compute aero-thermodynamic quantities including boundary layer effects. A variety of turbulent models available ranging from one to four equations transport models. Oblique shock waves, expansion waves and shock wave interactions measured. Accuracy of the solution is dependent on many parameters like size of the control volume, orientation of boundaries, discretization and its order of accuracy.

Scramjet inlet

Hypersonic air-breathing vehicles, the inlet channel is responsible for delivering much of the compression essential for combustion, and, as a result, vehicle function can be very complex to the inlet channel geometry. Variable geometry inlets have often been proposed or tested as a way to improve the performance of a vehicle that must achieve over a wide range of Mach numbers. For if the inlet designed for a single condition, the performance may be poor at all other requirements. At the same time, creating the inlet channel for a wider range of Mach numbers tends to decrease the performance at each condition. With a variable-geometry inlet, the configuration changes with Mach number so that the performance does not fall off as considerably. Constructing a variable-geometry hypersonic inlet presents numerous technical challenges. This paper does not address these physical aspects of the problem, but it aims to measure the potential benefit of allowing simple changes to the inlet geometry (changes ramp length, ramp (slope) angle).

The design of the critical inlet channel component alters the overall performance of the engine the principal purpose of the air inlet is to compress the supersonic flow into the subsonic flow and too rambling the condition such that proper combustion takes place. Also to provide the required amount of air to the engine to make sure a steady flow and to retain the total pressure loss minimum. In hypersonic case, inlets often called as Inlet diffusers. Here the compression is achieved by shocks both external and internal to the engine, and the angle of the outer cowl about the free stream can be made to minimize external drag. These inlets are classically longer than external compression alignments, but also spill flow when operated below the design Mach number. Dependent on the amount of internal compression, however, mixed compression inlets may need variable geometry to start.

Shock Wave

A shock is a discontinuity in a supersonic flow fluid. Fluid crossing a stationery shock front rises suddenly and irreversibly in pressure and decreases in velocity. It also deviates its direction and except when passing through a shock that is perpendicular to approaching flow direction.

Normal Shock

The shock wave is normal to the flow direction. If the shock wave is perpendicular to the flow direction called normal shock wave. After the normal shock wave, the flow will be subsonic whether the upstream of the flow is supersonic.

Oblique Shock

An oblique shock wave is inclined to the incident upstream flow direction. It will occur when a supersonic flow encounters a corner that effectively turns the flow into itself and compresses. The upstream streamlines uniformly deflected after the shock wave. The most common way to produce oblique an oblique shock wave is to place a wedge into the supersonic compressible flow. Similar to the normal shock wave, the oblique shock wave contains a slight region across which nearly discontinuous changes in the flow properties of a gas occur. While the upstream and downstream flow.

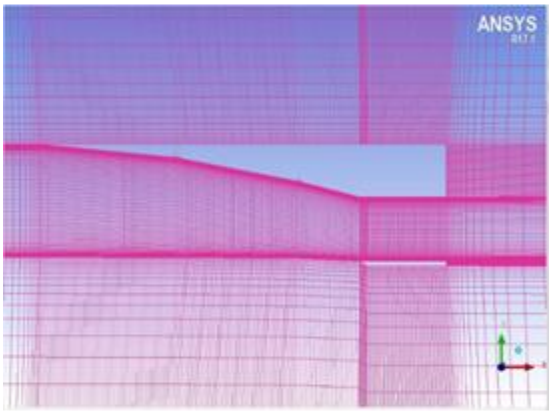
Direction is unchanged across normal shock, they are difficult for flow across an oblique shock wave. The given Mach number M1 and corner angle Θ, oblique shock angle β, downstream Mach number M2 can be calculated. M2 is always less than M1. Unlike after normal shock M2 can still be supersonic or subsonic. Weak solutions often observed in flow geometric open to atmosphere. The Strong solution may found in the confined geometric. A Strong solution is required when the flow essential to match the downstream high-pressure condition. Discontinuous changes also occur in pressure, density, and temperature which all rise download of the oblique shock waves.

Modelling of scramjet inlet in ansys

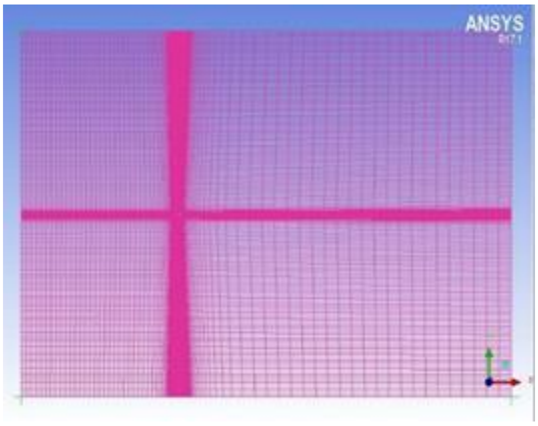
Geometry creation made by ANSYS with necessary commands from the geometry creation tool pad. The geometry creation tool pad contains a specification of scramjet inlet with leading edge, ramps, ramp angle, and length, without cowl deflection and Throat area (CR) to design a three ramp model of scramjet inlet with different Mach numbers.

Create of inlet geometry

Geometry application



Grid generation

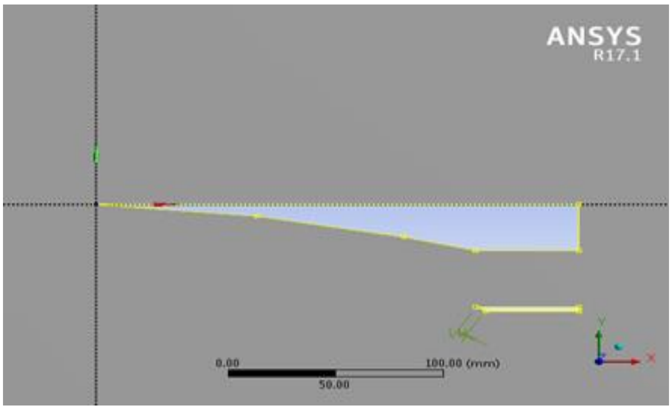


Meshing creation in ICEM done with the help of necessary commands from the meshing creation tool. The meshing creation tool contains control buttons that allow carrying out operations includes creating edge & face meshing.

Meshing creation in ICEM done with the help of necessary commands from the meshing creation tool. The meshing creation tool contains control buttons that allow carrying out operations includes creating edge & face meshing with necessary boundary conditions. For the numerical analyzing, inlet channel geometry parameters such as inlet ramps angles, length, the number of ramps, and contraction ratio are varied.

ICEM Meshing

3 Ramp meshing Geometry model



Computational Domain

The 2D modeling scheme adopted in ICEM. The structured grids were generated using ANSYS Meshing tool.

- Meshing can be done in forms namely edge meshing, face meshing.
- Meshed edge & faces can be copied, moved, linked or disconnected from one another.
- Structured grid cells used for the entire domain. Cells are gathered in the region.
- Grading schemes include successive ratio. The interval count identified for the starting mesh based on the model. In face or 2D meshing, the resulting parameters were stated. Meshing patterns mesh node spacing and face meshing options.

Leading edge	Sharp
No.of ramps	Three
Ramp angles	5.5˚,10.8˚,14.1˚
Ramps length (mm)	75,69,35
Cowl angle	0˚
Throat area (mm)	35

The grid independence test was done which involves transforming the generated physical model into mesh with Number of node points depending on the refinement of the mesh. The various flow properties were assessed at these node points. The extent of accuracy of result depended on largely on the fact that how fine the real domain meshed. After a particular sanitizing limit, the results changes no more. At this point, it is said that grid independence was achieved. The result attained for this mesh is considered to be the best. This mesh formation is made with ICEM.

Boundary conditions

For two-dimensional computations over the model, structured grid consists of quadrilateral cells is made. The overall quadrangular domain formed of numerous iterations is selected for all models. Inlet departure was the part of the outlet boundary face whereas the design base located on the boundary which is reassigned as wall edge. The grid generation scheme is quad/tri type cells of volume meshing. Grid with approximately 3500 to 5000 cells generated for inlet model. The initialize boundary condition for three ramp scramjet inlet model after the meshing can be done.

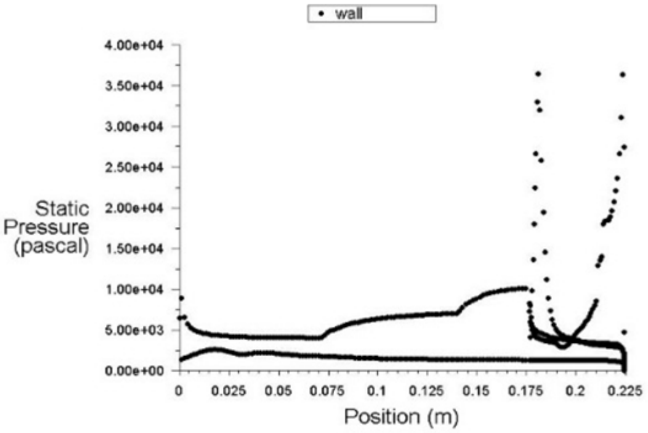
Inlet	Velocity inlet
Outlet	Pressure outlet
Upper boundary	Wall
Lower boundary	Wall
Fore body	Wall
cowl	Wall
Fluid	Air
Surface	Interior

II. RESULTS AND ANALYSIS

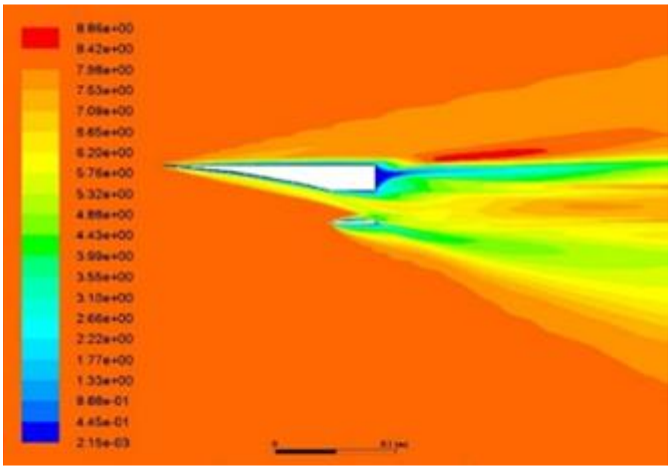
Two-dimensional simulations of the flow field using FLUENT to perform. Computations validated through a Simulation of the hypersonic inlet at desired Mach number. Boundary conditions and properties of the model defined as a reference to the literature

Analysis of scramjet inlet in fluent

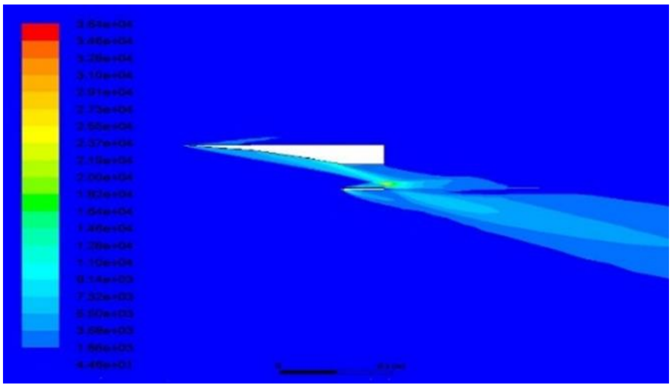
For Mach 8



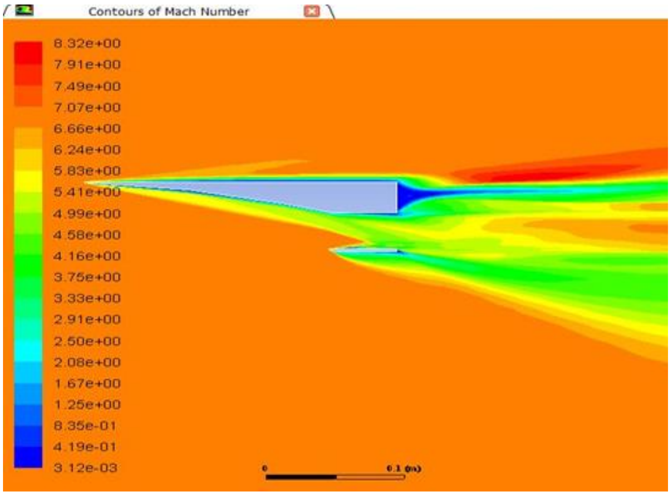
Mach contour

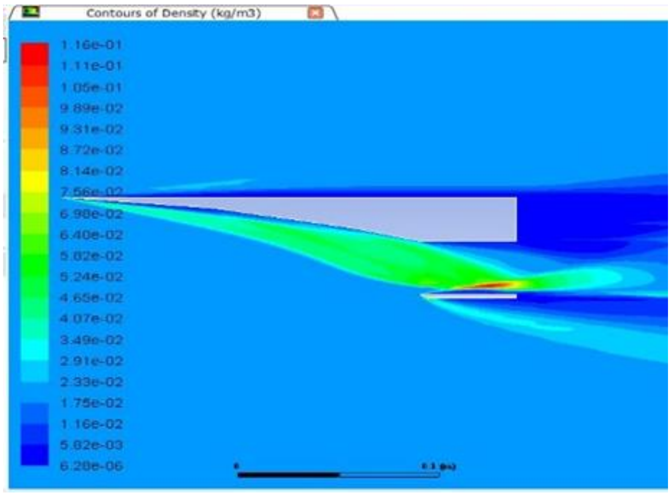


Pressure contour

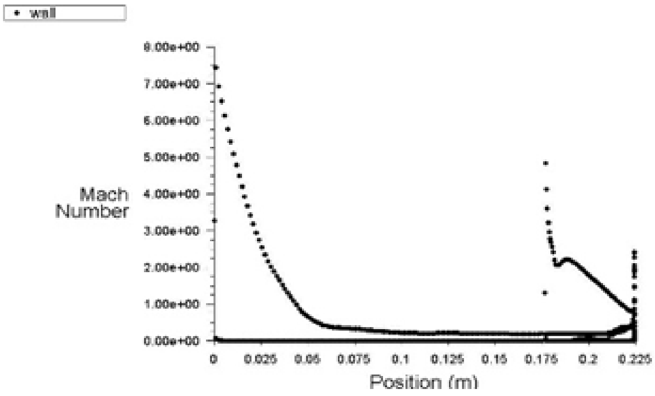


Density contour

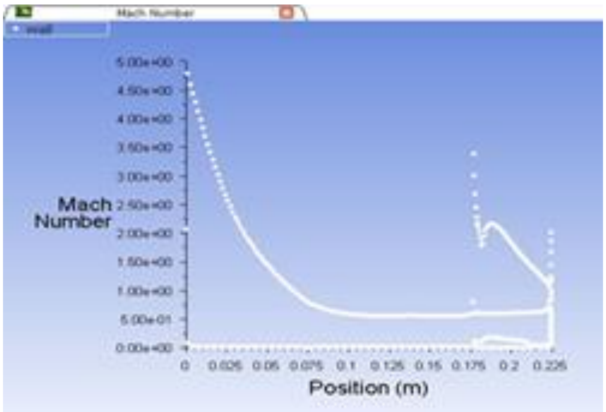




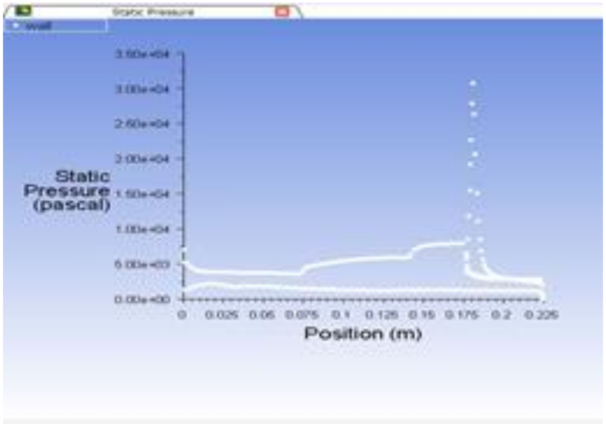
For Mach 7



Mach contour

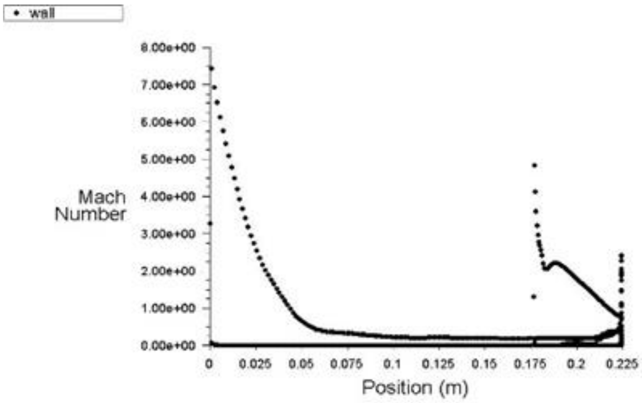


Density contour

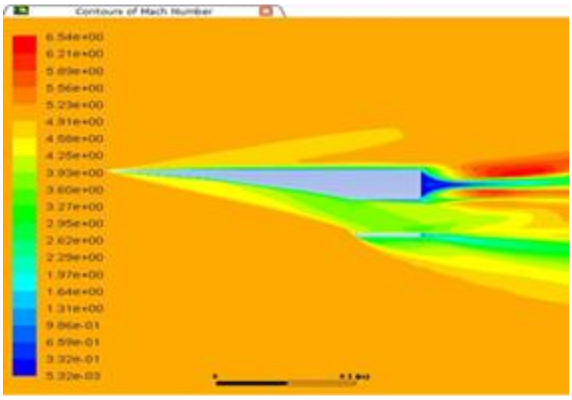


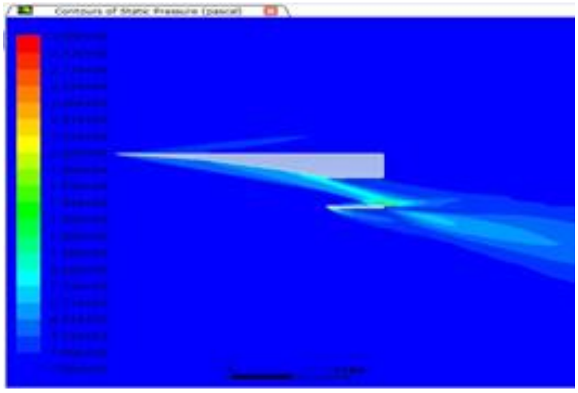
Gauge Pressure	1185 pa
Mach number	7
Reference temperature	226.5 k
Turbulent Viscosity	0.01
Turbulent Ratio	10
Altitude	30 km

For Mach 6

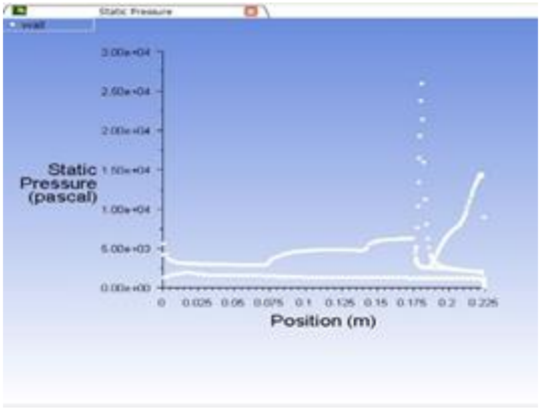


Density contour

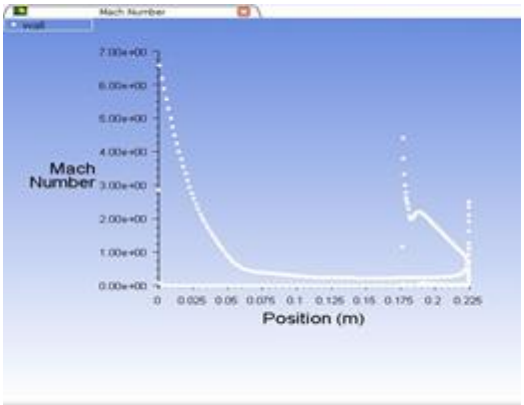




Mach contour

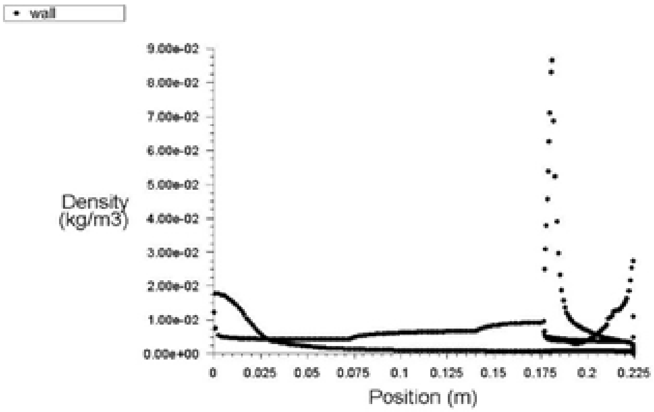


Pressure contour

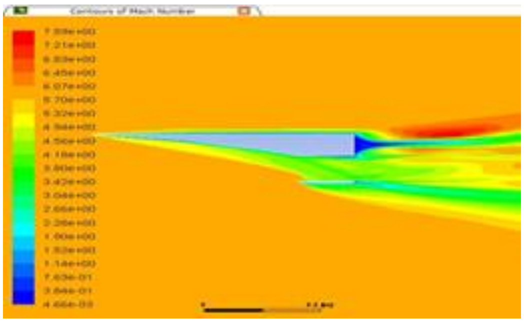


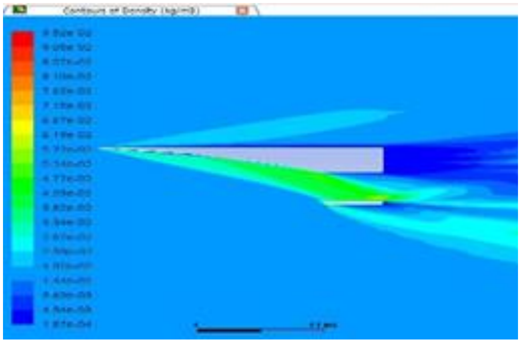
Gauge Pressure	1185 pa
Mach number	7
Reference temperature	226.5 k
Turbulent Viscosity	0.01
Turbulent Ratio	10
Altitude	30 km

For mach5

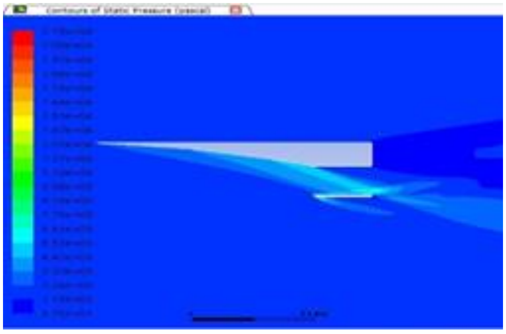


Density counter





Pressure counter



III. RESULTS AND DISCUSSION

The simulation contours obey the flow pattern which analyzed here as plots to compare the performance of the model with respect Mach numbers. Here, is to compare the standard parameters such as Pressure, density, and Mach numbers. The primary objective of this research work is to predict the different Mach flow pattern in 3 ramp inlet model with ramp length and ramp angle.

The comparison of different Mach number in 3 ramp inlet model is given below

Initial Mach no	Wedge angle	Shock angle	Mach no
5	5.5	15.47	4.443
	10.8	21.557	3.543
	14.1	28.08	2.68

Initial Mach no	Wedge angle	Shock angle	Mach no
6	5.5	13.3566	5.24
	10.8	19.6333	4.07
	14.1	25.44	3.03

Initial Mach no	Wedge angle	Shock angle	Mach no
7	5.5	12.24	6.0188
	10.8	18.333	4.55
	14.1	24.44	3.33

IV. CONCLUSION

The evaluation of performance parameters from the numerical simulation predicts that inlet geometry designed with isentropic compression ramp in lower hypersonic limits by providing favorable conditions for combustion as indicated by standard performance parameters. The specific goal is that the efficiency of the scramjet engine can be improved by decreasing the starting Mach number lowered to 3.50. Here the simulation Contours obeys the flow patterns analyzed contours are used to compare the performance of the model with respect to Mach numbers (M=8, 7, 6, 5) and also the standard parameters pressure, density and Mach number for 3 ramp model is also obtained. The purpose ofthis project was to predict the flow pattern for given free stream Mach numbers (M=8, 7, 6, 5) and its give better result for given condition like gauge Pressure, Temperature, Turbulent viscosity and Altitude. Thus the vital performance parameters obtained from FEM numerical simulation are compared and analyzed by parameterizing inlet ramp, Mach number and cowl deflection at low hypersonic limits.

V. REFERENCES

[1]. D. Nirmalkumar, S.Murugesan, Design and Computational Analysis of Scramjet Inlet, International Journal of Engineering Research and General Science, Vol 3, Issue 2, March-April, 2015

[2]. Dilip A Shah and nirmalkumar D, Research Article Computational Analysis of Scramjet Inlet, International Journal of Recent Scientific Research Vol. 6, Issue, 4, Pp.3391-3403, April, 2015

[3]. Derek J. Dalle, Sean M. Torrez, and James F. Driscoll, Performance Analysis of Variable-Geometry Scramjet Inlets Using a Low-Order Model,47th Aiaa/Asme/Sae/Asee Joint Propulsion Conference & Exhibit 31 July - 03 August 2011, San Diego, California

[4]. Reinartz Aachen Institute for Advanced Study in Computational Engineering Science, Performance Analysis Of A 3d Scramjet Intake, Preprint: Aices-2008-13/14/September/2008

[5]. Flock, Johannes C. Riehmer, Ali G¨Ulhan, Axisymmetric Scramjet Engine Design and Performance Analysis International Space Planes and Hypersonic Systems and Technologies Conferences 6-9 July 2015, Glasgow, Scotland 20th Aiaa International

Space Planes and Hypersonic Systems and Technologies Conference

[6]. Sadatake Tomioka, Tetsuo Hiraiwa, Tomoyuki Khisdha, Hiroki Yamasaki Evolution of Vitiation Effects in Scramjet Engines,Trans Isass Tech Japan Vol 7, No Ists Pp. Pa 47 Pa 52 2009

[7]. Jang, R. Peˇcnik and P. Moin, A Numerical Study Of the Unstart Event in an Inlet/Isolator Model,Center For Turbulence Research Annual Research Briefs 2010

[8]. Sarah Frauholz1, Birgit U. Reinartz2, Siegfried Müller3, and Marek Behr Rwth Transition Prediction for Scramjet Intakes Using the Γ-Reθt Model Coupled To Two Turbulence Models, Aachen University, Germany

[9]. Ramesh Kolluru and Vijay Gopal, Comparative Numerical Studies of Scramjet Inlet Performance Using K-E Turbulence Model with Adaptive Grids

[10]. Jeevan Raoȧ, Bhargavȧ, Krupakar Pasalaȧ and Srinivasa Ra, Computational Study of Viscous Flow in a Scramjet, Accepted 10 January 2014, Available Online 01 February 2014, Special Issue-2, (February 2014)

[11]. Yufeng Yao and Daniel Rincon, School of Aerospace and Aircraft Engineering, Shock Induced Separating Flows In Scramjet Intakes. Kingston University Roehampton Vale, Friars Avenue London, UK

[12]. Matthew Flynn Rohweder, A Numerical Investigation of Flow field Modification in High-Speed Air breathing Inlets Using Energy Deposition, Missouri University of science and technology

[13]. M. Krause, B. Reinartz, J. Ballmann,Numerical Computations for Designing a Scramjet Intake, Department of Mechanics, RWTH, Aachen University

[14]. Smart, M. K, Optimization of Two-Dimensional Scramjet Inlets, Journal of aircraft, Vol.36, NO. 2,1999, pp 430-433

[15]. Kristen Nicole Roberts, Analysis and Design of a Hypersonic Scramjet Engine with a Starting Mach number Of 4.00, The University of Texas at Arlington August 2008

Cite this article as :

T Anand, R Selva Kumaran, R Vijayan, M shalini, "Flow Prediction in Scramjet Engine Inlet", International Journal of Scientific Research in Science and Technology (IJSRST), Online ISSN : 2395-602X, Print ISSN : 2395-6011, Volume 5 Issue 5, pp. 64-72, March-April 2020.
Journal URL : http://ijsrst.com/EBHAE020