Available Online at http://www.recentscientific.com

ISSN: 0976-3031

International Journal of Recent Scientific Research
Vol. 6, Issue, 4, pp.3391-3403, April, 2015

International Journal of Recent Scientific Research

RESEARCH ARTICLE

COMPUTATIONAL ANALYSIS OF SCRAMJET INLET

Murugesan S[1], Dilip A Shah[2] and Nirmalkumar D[3]

[123]Department of Aeronautical Engineering, Hindustan Institute of Technology and Science, Chennai, India

ARTICLE INFO

Article History:

Received 2nd, March, 2015
Received in revised form 10th, March, 2015
Accepted 4th, April, 2015
Published online 28th, April, 2015

Key words:

Ramps, cowl, oblique shock and axisymmetric inlet

ABSTRACT

Scramjet inlets are the most vital component of the engine and their design having more effective on the overall performance of the engine. Thus, the forward capture shape of the engine inlet should conform to the vehicle body shape. A 2-D computational study for scramjet inlet with different ramp length and angles are studied to compress the air by blunted and sharp leading edge, moving the whole cowl up and down, deflecting the cowl lip and axisymmetric inlet with sharp and blunted leading edge. These geometric changes have produced a numerous shocks in inlet and remarkable influence on the flow in several aspects. However, the performance of these inlets tends to degrade as higher Mach number to lower Mach number. These inlets consisting of various ramps producing oblique shocks followed by a cowl shock is chosen in order to increase air mass capture and reduce spillage in scramjet inlets at Mach numbers below the design value. An impinging shock may force the boundary layer to separate from the wall, resulting in total pressure recovery losses and a reduction of the inlet efficiency. Design an inlet to meet the requirements such as Low stagnation pressure loss, High static pressure and temperature gain and deceleration of flow to a desired value of Mach number. Fixed geometry inlets can be used only over a relatively narrow range of Mach number while one method to improve this performance is to use variable-geometry inlets which can be used over a wide range of Mach number with reasonably good pressure recovery. A two dimensional analysis is carried out in this project. CATIA is used to create the model. GAMBIT is used to create the mesh. FLUENT is used to cover the flow analysis.

INTRODUCTION

A supersonic combustion ramjet (scramjet) is a variant of a ramjet air-breathingcombustion jet engine. (Ref 1) The definition of a ramjet engine is first necessary, as a scramjet engine is a direct descendant of a ramjet engine. Ramjet engines have no moving parts, instead operating on compression to slow freest ream supersonic air to subsonic speeds, thereby increasing temperature and pressure, and then combusting the compressed air with fuel. Lastly, a nozzle accelerates the exhaust to supersonic speeds, resulting in thrust. Due to the deceleration of the freest ream air, the pressure, temperature and density of the flow entering the burner are "considerably higher than in the freest ream". At flight Mach numbers of around Mach 6, these increases make it inefficient to continue to slow the flow to subsonic speeds. Thus, if the flow is no longer slowed to subsonic speeds, but rather only slowed to acceptable supersonic speeds, the ramjet is then termed a 'supersonic combustion ramjet,' resulting in the acronym scramjet.

To study the inlet performance by evaluating multiple standard parameters. This study involves comparison of performance parameters for scramjet inlet which are evaluated as a result of FEM computation of 2-D turbulent flow field around six different scramjet inlet geometries. The salient geometrical parameters which are varied are; inlet ramp angle and length, cowl lip angle, leading edge and axisymmetric inlet.

The main Objective of the project is to study the shock and shock interaction on the scramjet inlet. To study the inlets performance accordingly with respect to pressure and velocity contours. To study the pressure disturbance of the inlet models with and without the cowl lip deflection. Optimizing the performance of the inlet is to operate over a range of Mach numbers.

The 2-D computation of turbulent flow is obtained by implementing high Reynolds number k-omega compressible turbulent formulation. The boundary and initial conditions are carefully selected to the free stream conditions that pertain to a cruise altitude of 25km. the simulations were performed for three free stream Mach number 5 and 8. Thus from the obtained result, comparative studies of performance parameters are carried out by parameterising geometrical variables and free stream Mach number.

It is necessary to simulate the inlet design to obtain the appropriate inlet performance. Computational Fluid Dynamics (CFD) is used to study flight simulations in both steady and un-

Corresponding author:* **Murugesan S
Department of Aeronautical Engineering, Hindustan Institute of Technology and Science, Chennai, India

steady flow. A time-averaged, viscous, 2 Dimensional, CFD scheme used to computeaero-thermo dynamic quantities including boundary layereffects. A variety of turbulent models available ranging from one to three equations transport models. Obliqueshock waves, expansion waves and shock wave interactions are mainly considered. Accuracy of the solution is dependent on many parameters like size of thecontrol volume, orientation of boundaries, discretizationand its order of accuracy.

Scramjet Inlet

Scramjet inlet is to converts the K.E of the air flow into a static pressure rise that helps in deceleration of flow at lower speeds. This deceleration takes place as the flow passes through a series of oblique shocks that are formed due to the presence of ramps in the inlet, also called as staged compression. (Ref 3) Hence the design of an inlet must be done carefully so as to meet the requirements given below.

- Low stagnation pressure loss
- High static pressure gain
- Deceleration of flow to a desired value of Mach number.
- Achievement of these requirements becomes essential so as to make this concept a reality. These requirements can be achieved by understanding the following concepts of inlet design.

The internal inlet compression provides the final compression of the propulsion cycle. The fore body along with the internal inlet is designed to provide the required masscapture and aerodynamic contraction ratio at maximum inlet efficiency. The air in the captured stream tube undergoes a reduction in Mach number with an attendant increase in pressure and temperature as it passes through the system of shock waves in the fore body and internal inlet. It typically contains non-uniformities, due to oblique reflecting shockwaves, which can influence the combustion process. A scramjet air induction phenomenon includes vehicle bow shock and isentropic turning Mach waves, shock boundary layer interaction, non-uniform flow conditions, and three-dimensional effects.

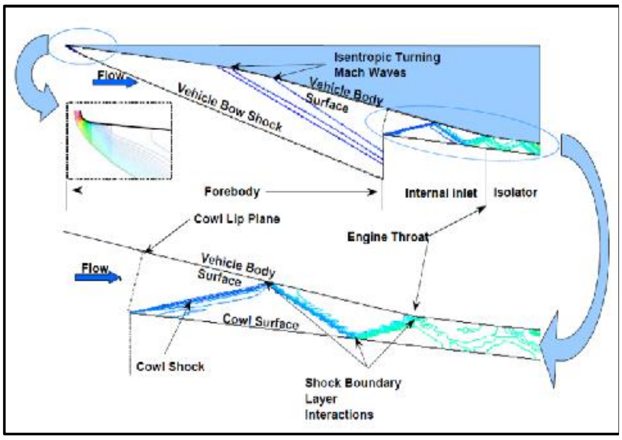


Figure1 Summary of Important Forebody and Internal Inlet Physics

The design of this type of critical inlet component alters the overall performance of the engine. The major purpose of the air inlet is to compress the supersonic flow into subsonic flow and to diffuse the condition such that proper combustion takes place. Also to provide required amount of air to engine ensuring a stable flow and to keep the total pressure loss minimum. In hypersonic case inlets are often called as Inlet diffusers. Here the compression is performed by shocks both external and internal to the engine, and the angle of the external cowl relative to the freestream can be made very small to minimize external drag. These inlets are typically longer than external compression configurations, but also spill flow when operated below the design Mach number. Depending on the amount of internal compression, however, mixed compression inlets may need variable geometry in order 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 changes its direction. Except when passing through a shock that is perpendicular to the approaching flow direction. Such plane normal shocks are easiest to analyze. We are not going to go in detail about the normal shocks as the presence of oblique shocks is applicable for our project. (Ref 5)

Normal Shock

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

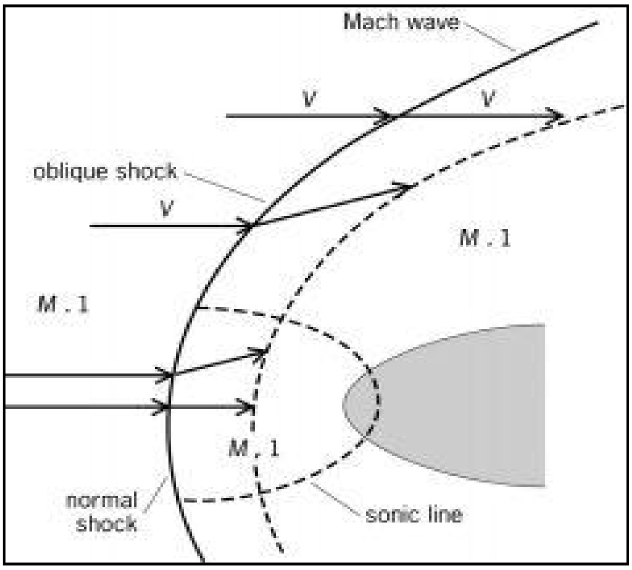


Figure 2 Normal shock formations

Oblique Shock

An oblique shock wave, unlike a normal shock is inclined with respect to the incident upstream flow direction .It will occur when a supersonic flow encounter a corner that effectively turns the flow into itself and compress. The upstream streamlines are uniformly deflected after the shock wave. The most common way to produce oblique an oblique shock wave is to place a wedge into supersonic compressible flow. Similar to normal shock wave the oblique shock wave consists of a very thin region across which nearly discontinuous changes in the thermodynamic properties of a gas occur. While the upstream and downstream flow direction is unchanged across a normal shock, they are difficult for flow across an oblique shock wave. For a 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 a normal shock M2 can be still be supersonic or subsonic. Weak solutions are often observed in flow geometric open to atmosphere. Strong solution may be observed in confined geometric. Strong solution is required when the flow need 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.

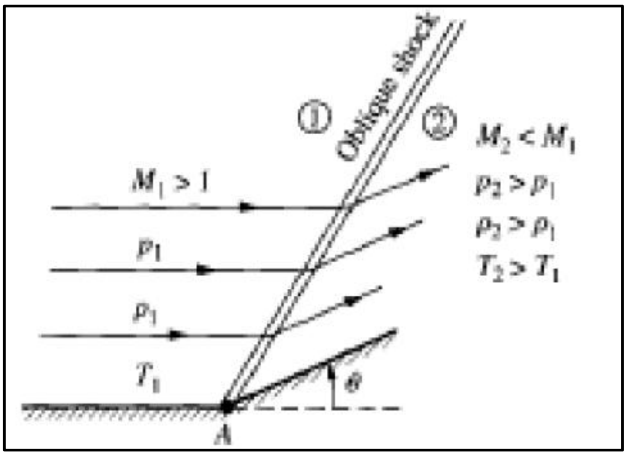


Figure 3 Oblique shock formations

If a plane shock is inclined at an angle to the flow, the fluid passing through suffers not only a sudden rise in pressure and decrease in speed but also a sudden Change of direction.

Inlet Operating Condition

Usually scramjet diffusers are unregulated and designing for a certain Mach number called design Mach number. (Ref 6) Diffuser should prove required compression and mass flow satisfying the conditions about minimum of total pressure losses, Safety and stability of operation.

In the traditional scramjet diffuser system of a number of oblique shocks is realizing. It operates in design mode when oblique shocks hit the engine cowl (Fig 4). Altering the flight Mach number oblique shocks deviate from the engine cowl and two different situations can be observed (Fig 5,6).

In the first case when the flight Mach number higher than the design one oblique shocks deviates inside diffuser (Fig 5) forming intensive reflected shock. In the second case shocks deviates outside the diffuser (Fig 6). In the first case reflected shock/boundary layer interaction causes flow detach, stagnation zone and high heat loads of engine. Flow with higher temperature but without such detaches/reattachment zones are more preferable. High heat loads always appears near the front engine cowl edge but usually it cooling. Appearance of unaccounted "hot spot" on the engine cowl may be catastrophic. Cooling tasks will not be touched upon but one way of heat loads decrease suggested in air inlet.

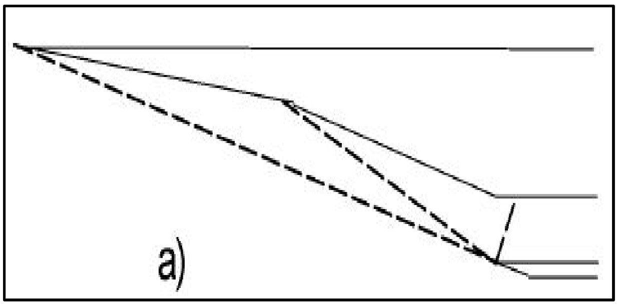


Figure 4 Design mode

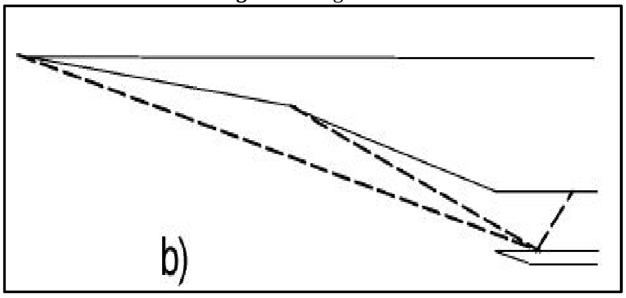


Figure 5 Flight Mach number higher than design one

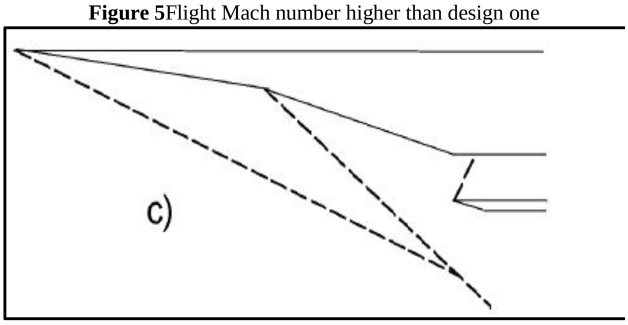


Figure 6 Flight Mach number lower than design one

Modelling Of Scramjet Inlet In Catia

Geometry creation in CATIA is done with the required commands from the geometry creation tool pad. The geometry creation tool pad contains specification of scramjet inlet with leading edge, ramps, ramp angle and length, cowl deflection and contraction ratio (CR). To design a six models of scramjet inlet with different specifications.

A. Create Of Inlet Geometry

The inlet to be optimized in this paper comprises six models,

- Rounded and sharp leading edge with three ramps and without deflection.
- Four Ramped Inlet model with deflection.
- Two Ramped Inlet model with deflection.
- Axisymmetric Inlet model with rounded and sharp leading edge.

The internal geometry is represented by five parameters: the leading-edge, ramp lengths, ramp angle, ramp angle increments, and exit radius. For rounded leading edge the inlet radius is fixed at 0.6mm to ensure constant mass flow entry, which effectively makes one of the ramp parameters dependent on the others for a given value of the combustor radius. Also fixed is the leading edge nose-tip radius 0.6mm in order to focus on the influence of ramp geometries by freezing the entropy layer effect originating from the leading edge. For axisymmetric inlets are two models are sharp and rounded leading edge with three ramps different angles. These assumptions, in effect, leave these parameters as design variables, or decision variables for optimisation.

Scramjet inlet first design model (fig 7) with blunted leading edge with the radius of 0.6 mm, three ramps and without cowl lip deflection this value are clearly explain in the table 1.

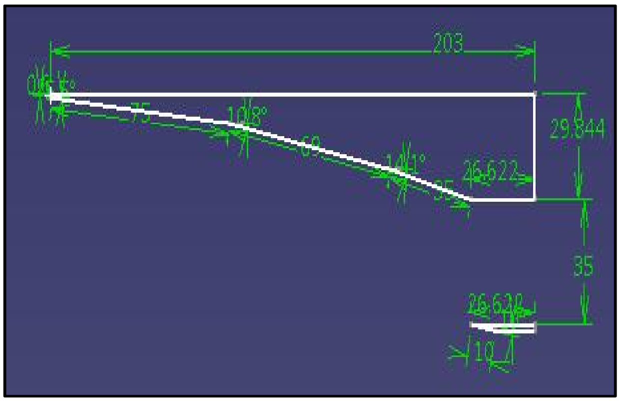


Figure 7 Rounded leading edge with three ramps and without deflection

Table 1 Scramjet inlet 1 Specification

Leading edge	Rounded
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

Scramjet inlet second design model (fig 8) with sharp leading edge, three ramps and without cowl lip deflection this value is clearly explained in the table 2.

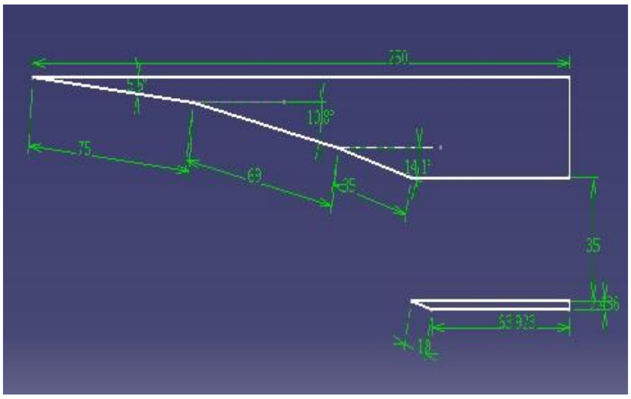


Figure 8 Four Ramped Inlet model with deflection

Table2 Scramjet inlet 2 Specification

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

Third inlet design model (fig 9) with sharp leading edge, four ramps and with cowl lip deflection of 10 degree and this value is clearly explained in the table 3

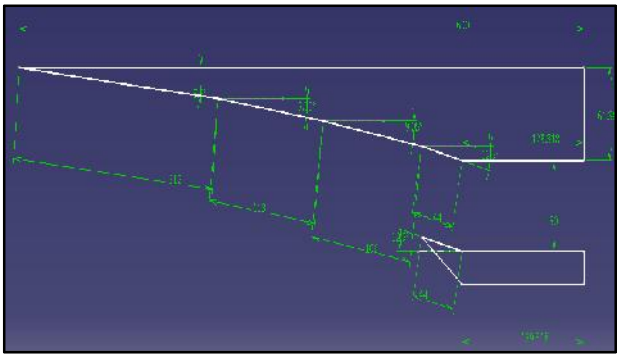


Figure 9 Four Ramped Inlet model with deflection

Table 3 Scramjet inlet 2 Specification

Leading edge	Sharp
No.of ramps	four
Ramp angles (degree)	5.5,7.55,9.05,12.5
Ramps length (mm)	212,113,106,44
Cowl angle (degree)	12.5
Cowl lip length (mm)	44
Throat area (mm)	60

Fourth inlet design model (fig 10) with sharp leading edge, two ramps and with cowl lip deflection this value is clearly explained in the table 4

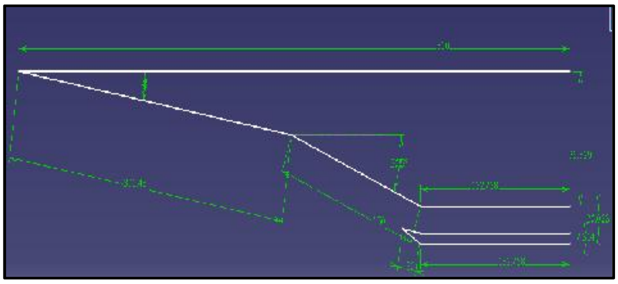


Figure 10 Two Ramped Inlet model with deflection

Table 4 Scramjet inlet 4 Specification

Leading edge	Sharp
No.of ramps	two
Ramp angles (degree)	9,20.5
Ramps length (mm)	300,150
Cowl angle (degree)	10
Cowl lip length(mm)	20
Throat area (mm)	20.066

Fifth model of axisymmetric inlet (fig 11) with blunted leading edge, three ramps and this value is clearly explained in the table 5

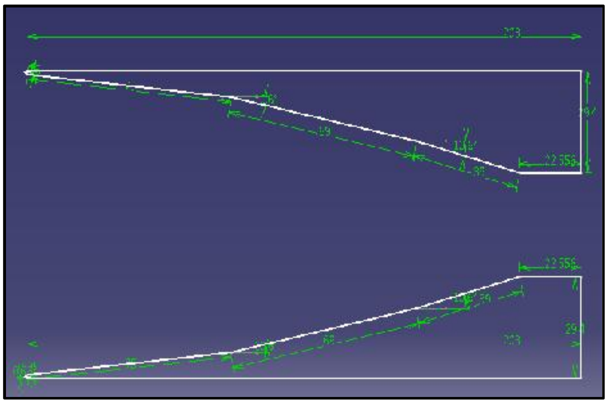


Figure 11 Axisymmetric Inlet model with rounded leading edge

Table 5 Scramjet inlet 5 Specification

Leading edge	Rounded
Inlet type	Axisymmetric
No.of ramps	three
Ramp angles (deg)	5,10.6,13.6
Ramps length (mm)	75,69,39
Throat area (mm)	30

Sixth model of axisymmetric inlet (fig 12) with sharp leading edge, three ramps and this value is clearly explained in the table 6

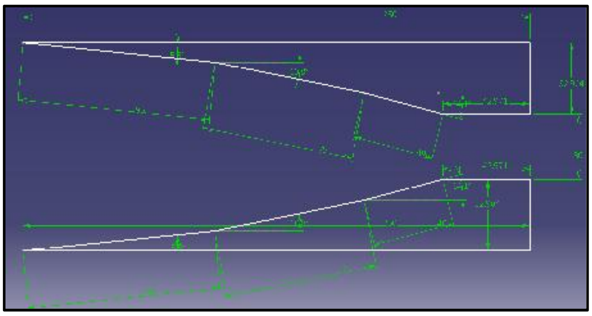


Figure 12 Axisymmetric Inlet model with sharp leading edge

Table 6 Scramjet inlet 6 Specification

Leading edge	sharp
Inlet type	Axisymmetric
No.of ramps	three
Ramp angles (deg)	5.5,10.8,14.1
Ramps length (mm)	95,75,40
Throat area (mm)	30

Grid Generation In Gambit

Meshing creation in gambit is done with the help of required commands from the meshing creation tool pad. The meshing creation tool pad contains command buttons that allows performing operations which include creating edge meshing, face meshing and boundary conditions. For the numerical study, inlet geometry parameters such as inlet ramps angles, length, number of ramps, cowl deflection and contraction ratio are varied. Axisymmetric inlets with sharp and rounded leading edge also meshing with rectangle domain can be create in this Chapter

A.Computational Domain

The 2D modeling scheme was adopted in GAMBIT. The structured grids were generated using ANSYS Gambit 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 are used for entire domain. Cells are clustered at the region.
- Grading schemes includes successive ratio. Double sided grading also can be performed. The interval count can be specified for the starting mesh based on the model. In face or 2D meshing the following parameters can be specified. Meshing schemes mesh node spacing and face meshing options.
- The meshing schemes include the elements and the types. Quadrilateral can be used as the elements. The meshing type pave are used.
- Similar to the edge meshing the grading schemes, mesh node spacing can also be specified for face meshing.

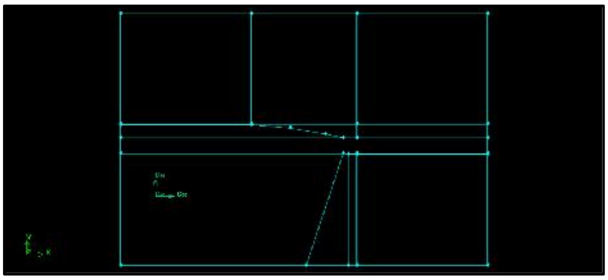


Figure 13 Rectangle domain created around model

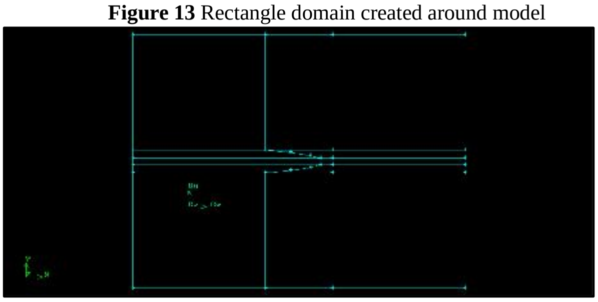


Figure 14 Rectangle domain created around axisymmetric inlet

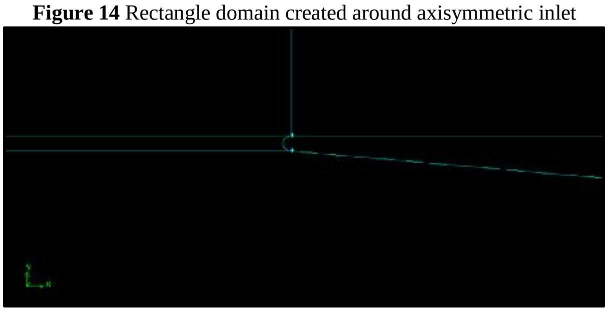


Figure 15 Rounded Leading edge separations

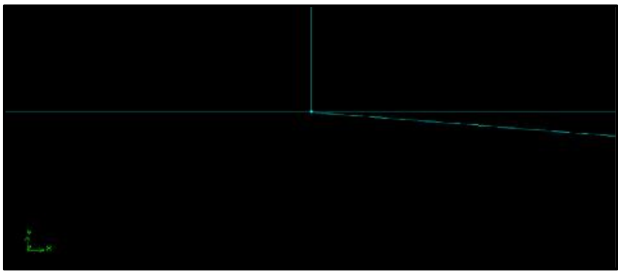


Figure 16sharp Leading edge separations

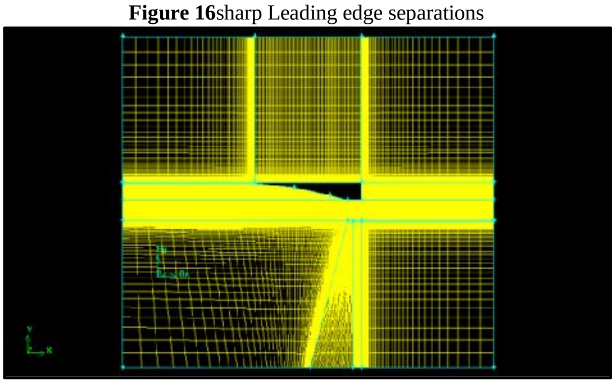


Figure 17 Two Ramped Inlet model without deflection

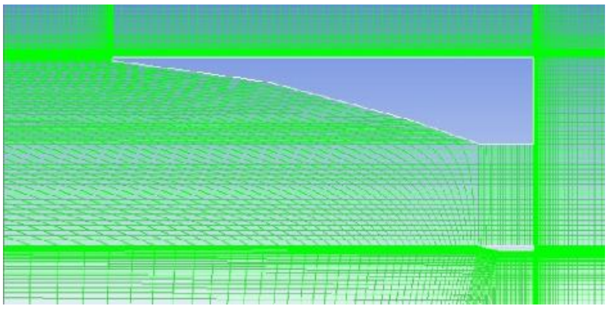


Figure 18 Mesh of scramjet inlet

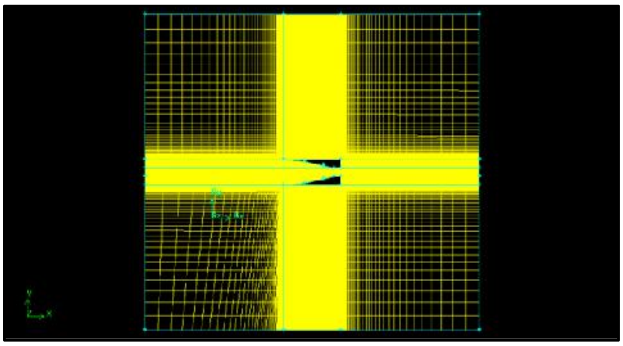


Figure 19 Axisymmetric Inlet model with rounded leading edge

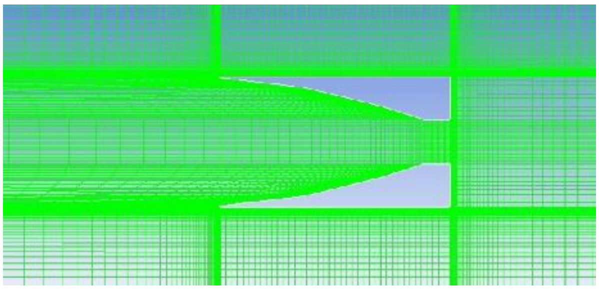


Figure 20 Mesh of axisymmetric scramjet inlet

The grid independence test is done which involves transforming the generated physical model into a mesh with number of node points depending on the fineness of the mesh. The various flow properties were evaluated at these node points.

The extent of accuracy of result depended to a great extent on the fact that how fine the physical domain was meshed. After a particular refining limit the results changes no more. At this point it is said that grid independence is achieved. The results obtained for this mesh is considered to be the best. This mesh formation was done with GAMBIT

B. Boundary conditions

For two dimensional computations over the model a structured grid consists of quadrilateral calls are made. The overall rectangular domain is made of several iterations were chosen for all models. Inlet exit was the part of the outlet boundary face whereas the model base was situated on the boundary which was assigned as wall boundary. The grid generation scheme is quad/tri type cells of volume meshing. Grid with approximately 30000 cells is made for every inlet models. The initialize boundary condition for all the scramjet inlet models after the meshing can be done.

Table 7 Boundary conditions for all models

Name	Type
Outlet	Pressure outlet
Upper boundary	Wall
Lower boundary	Wall
Mode 1	Wall
Mode 2	Wall
Fluid	Air

The grid for the scramjet inlet 2D models generated using the software GAMBIT and the other specification discussed. Grid independence study results in formation of fine grids to obtained desired results. Separated domains was selected based on several iterations were chosen. The initialize boundary condition for all the scramjet inlet models is given been chosen.

RESULTS AND ANALYSIS

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

A. Analysis of scramjet inlet in fluent

Table 8 Inlet Boundary Conditions for Mach 5

Parameter	values
Mach number	5
Reference temperature	221.65
Turbulent Viscosity	0.01
Turbulent Ratio	10

Model 1:Rounded leading edge with three ramps and without deflection

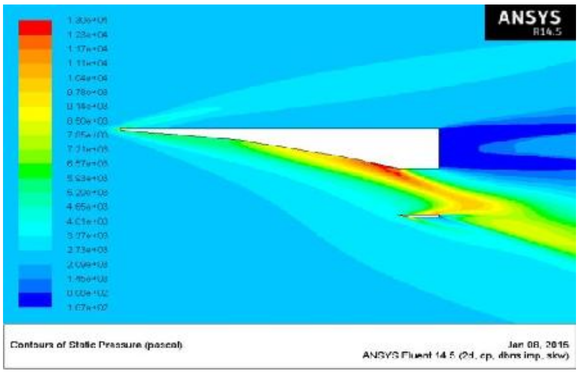


Figure 21 Pressure Contour

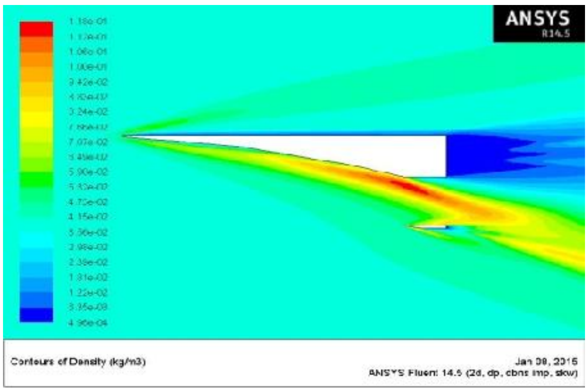


Figure 22 Density Contour

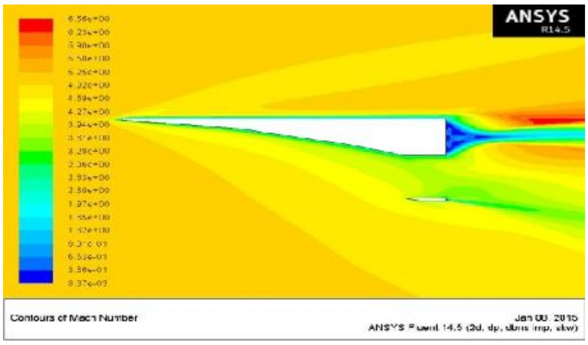


Figure 23 Mach Contour

Mode 2:Sharp leading edge with three ramps and without deflection

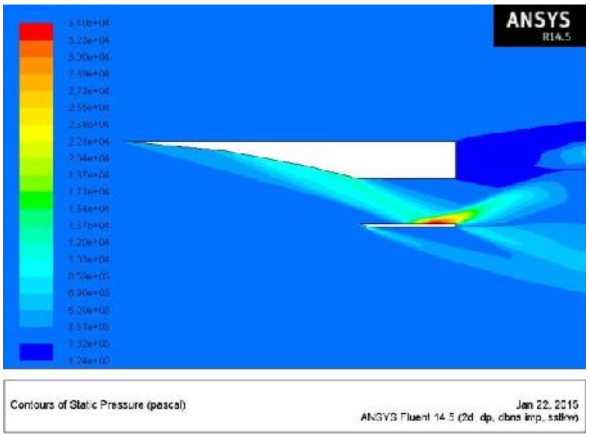


Figure 24 Pressure Contour

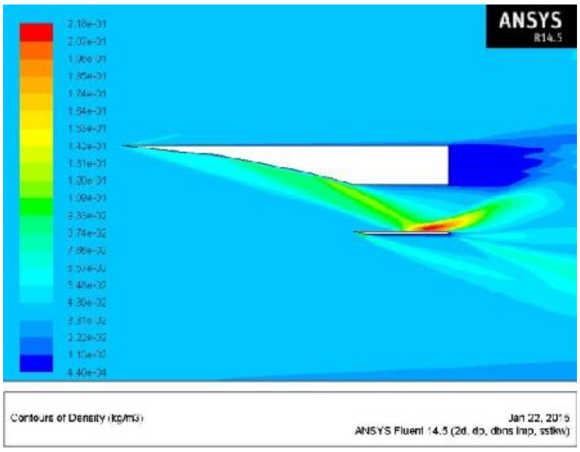


Figure 25 Density Contour

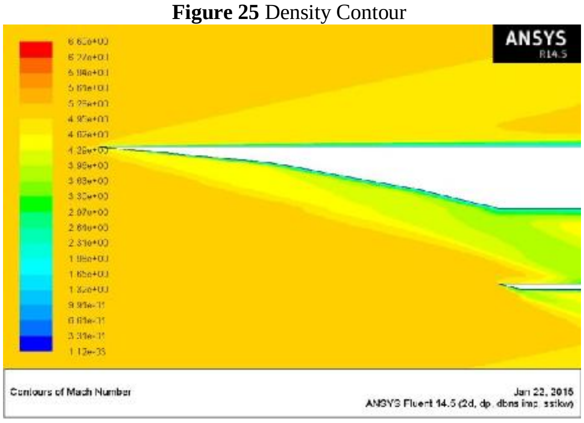


Figure 26 Mach Contour

Model 3:Four Ramped Inlet model with deflection

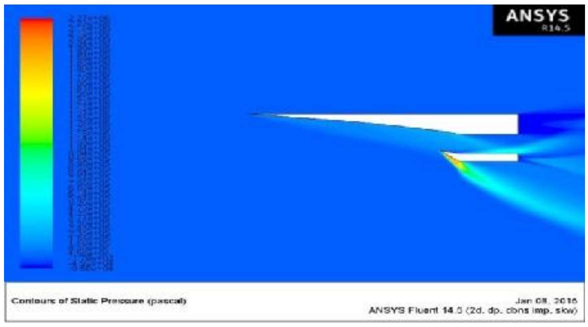


Figure 27 Pressure Contour

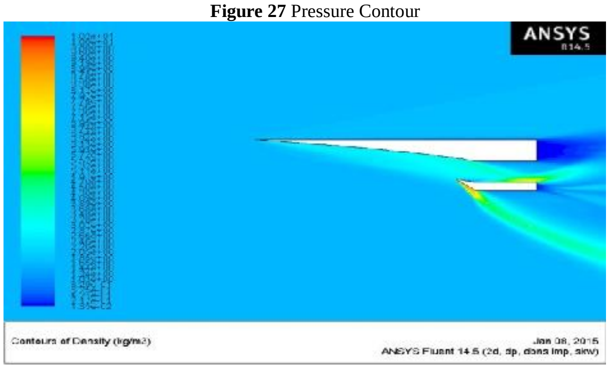


Figure 28 Density Contour

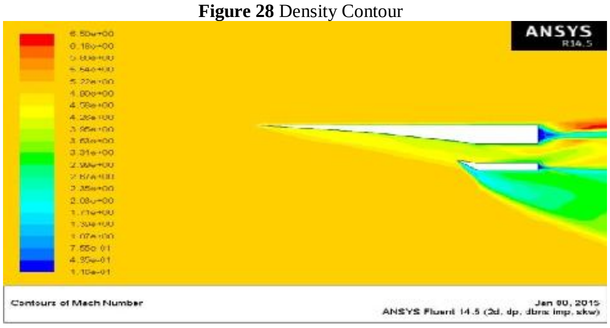


Figure 29 Mach Contour

Model 4: Two Ramped Inlet model with deflection

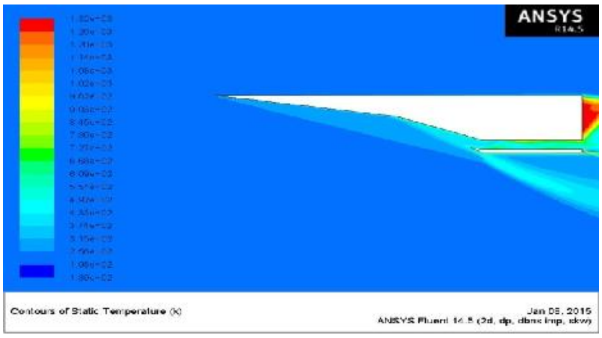


Figure 30 Pressure Contour

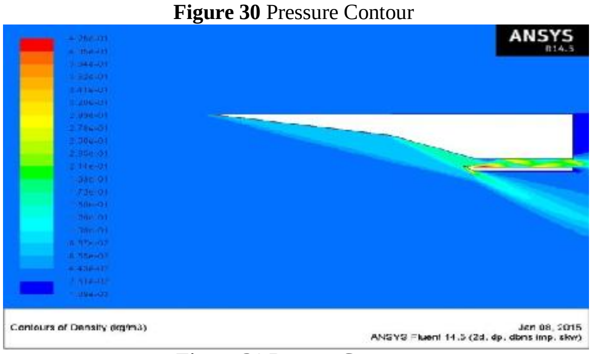


Figure 31 Density Contour

Figure 32 Mach Contour

Model 5: Axisymmetric Inlet model with rounded leading edge

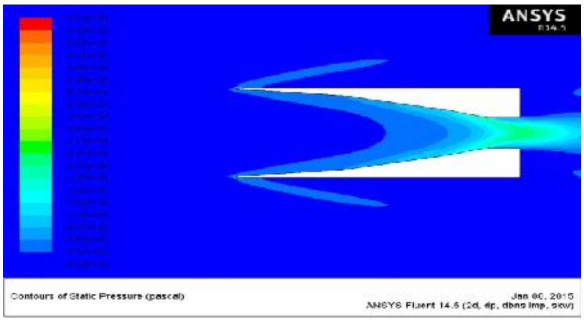


Figure 33 Pressure Contour

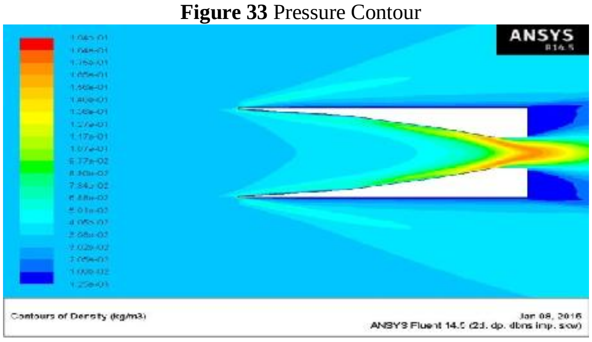


Figure 34 Density Contour

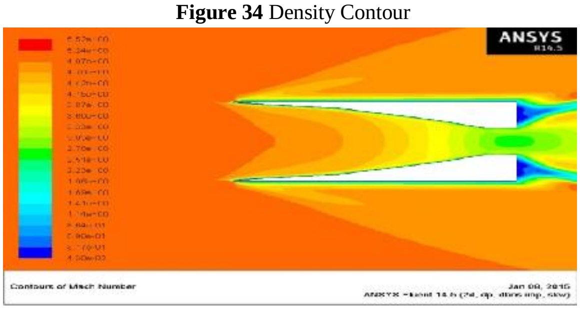


Figure 35 Mach Contour

Model 6: Axisymmetric Inlet model with sharp leading edge

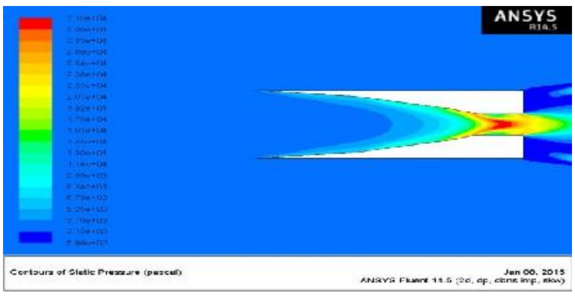


Figure 36 Pressure Contour

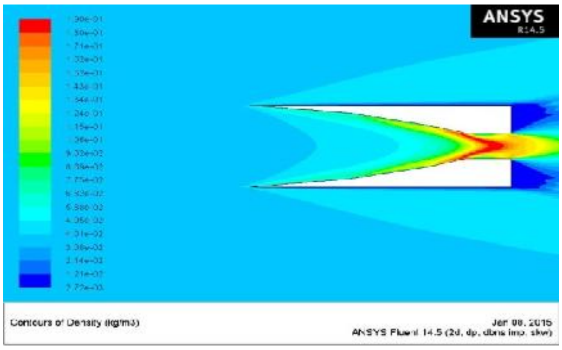


Figure 37 Density Contour

Figure 38 Mach Contour

For Mach number 8

This is analysis carried out for Mach number 8 for all the scramjet inlet models. Table 7.2 gives the boundary condition.

Table 9 Inlet Boundary Conditions for Mach 8

parameter	values
Mach number	8
Reference temperature	226.5 k
Turbulent Viscosity	0.01
Turbulent Ratio	10
Altitude	30 km

Model 1: Rounded leading edge with three ramps and without deflection

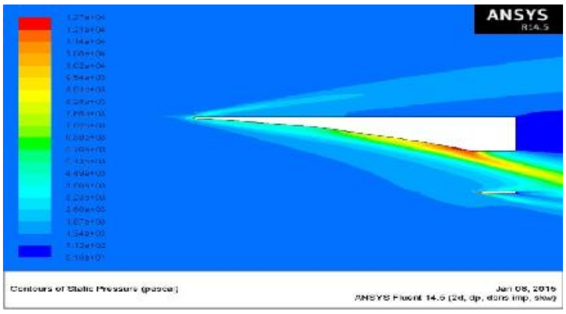


Figure 39 Pressure Contour

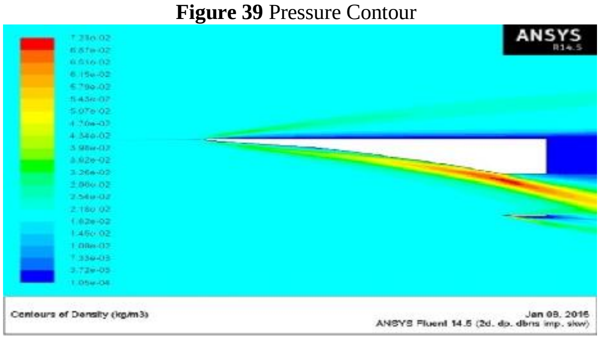


Figure 40 Density Contour

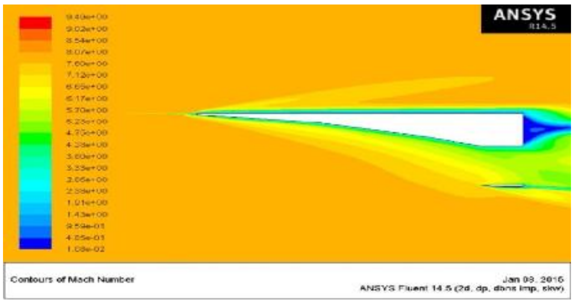


Figure 41 Mach Contour

Model 2:Sharp leading edge with three ramps and without deflection

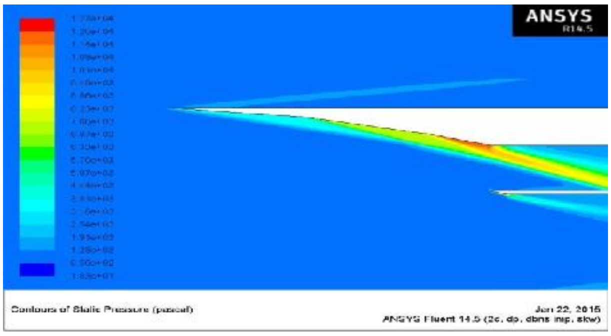


Figure 42 Pressure Contour

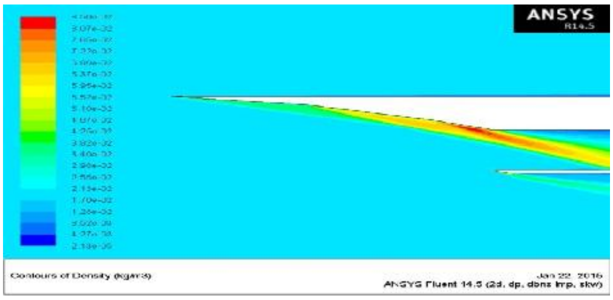


Figure 43 Density Contour

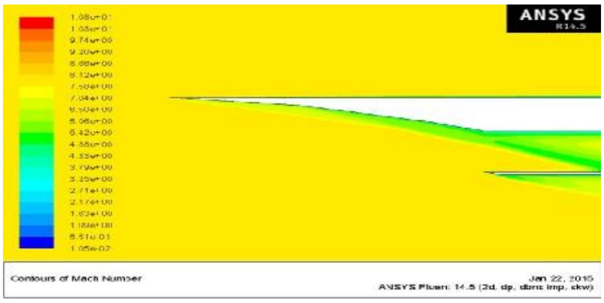


Figure 44 Mach Contour

Model 3:Four Ramped Inlet model with deflection

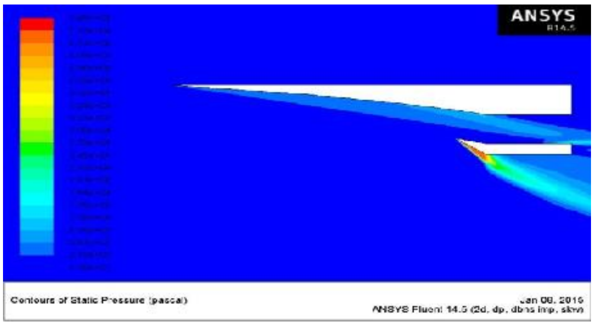


Figure 45 Pressure Contour

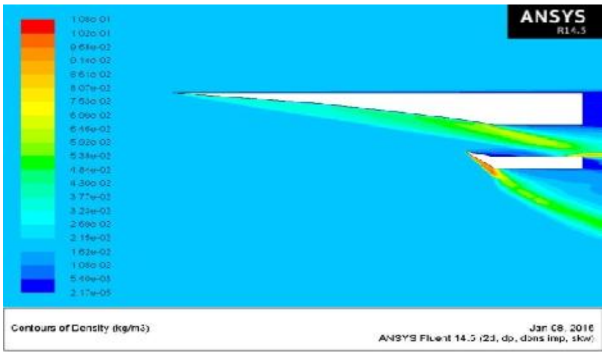


Figure 46 Density Contour

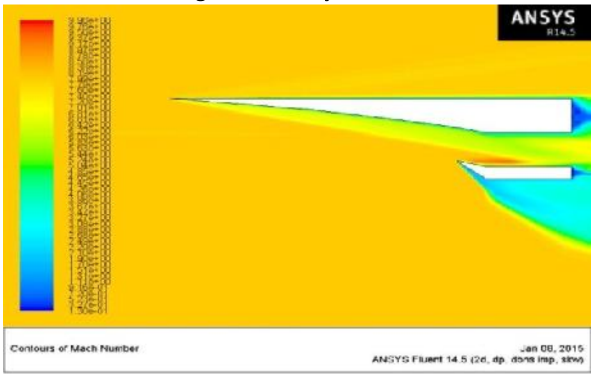


Figure 47 Mach Contour

Model 4: Two Ramped Inlet model with deflection

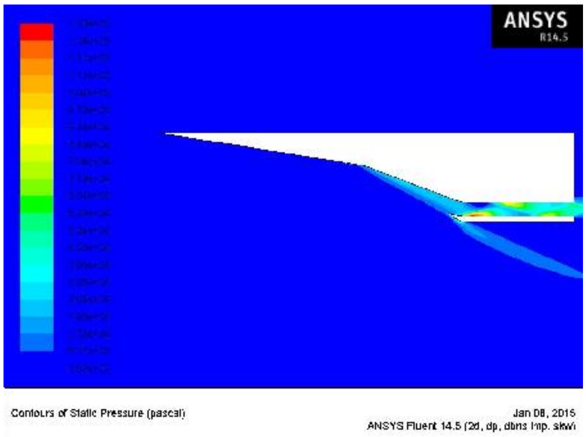


Figure 48 Pressure Contour

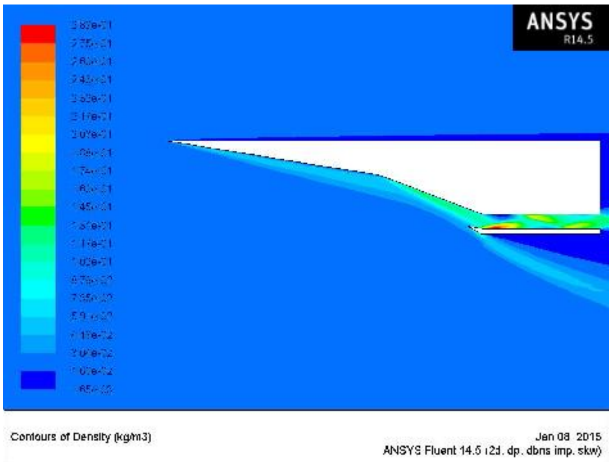


Figure 49 Density Contour

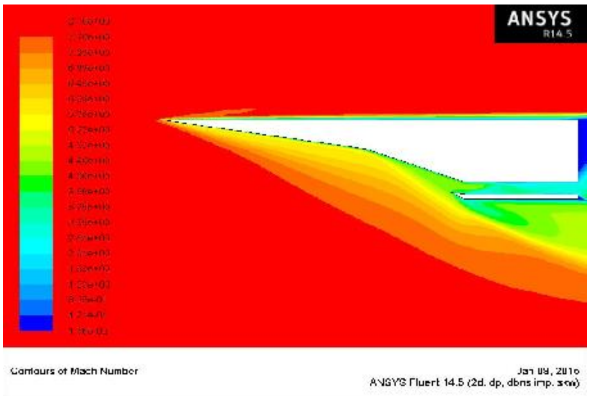


Figure 50 Mach Contour

Model 5:Axisymmetric Inlet model with rounded leading edge

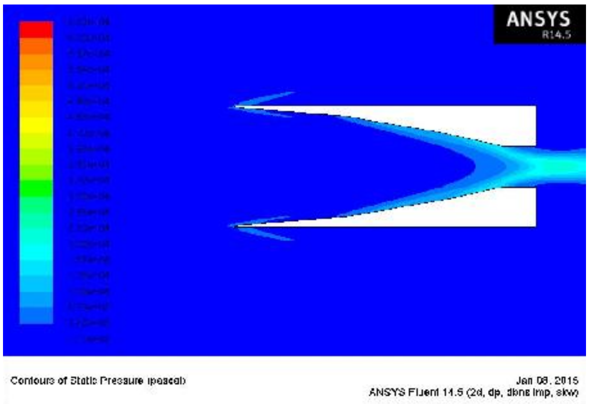


Figure 51 Pressure Contour

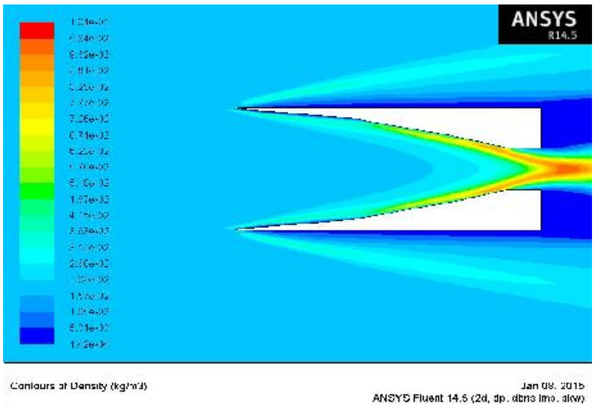


Figure 52 Density Contour

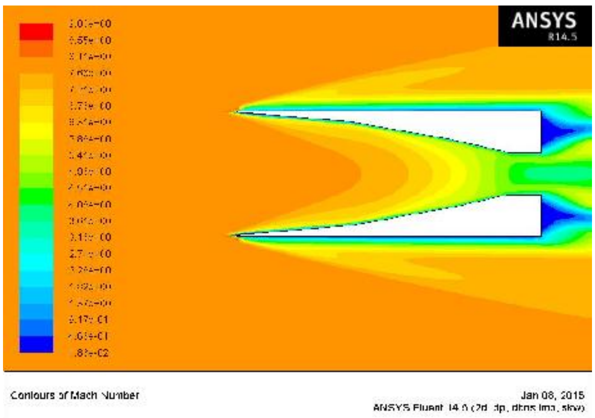


Figure 53 Mach Contour

Model 6:Axisymmetric Inlet model with sharp leading edge

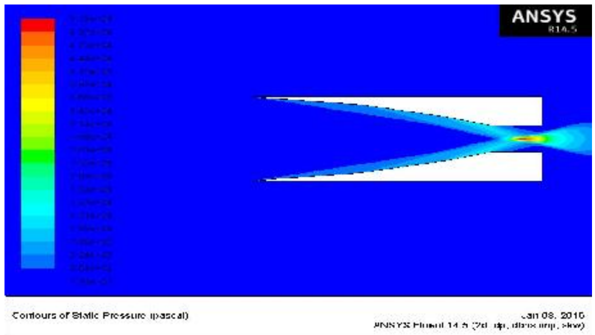


Figure 54 Pressure Contour

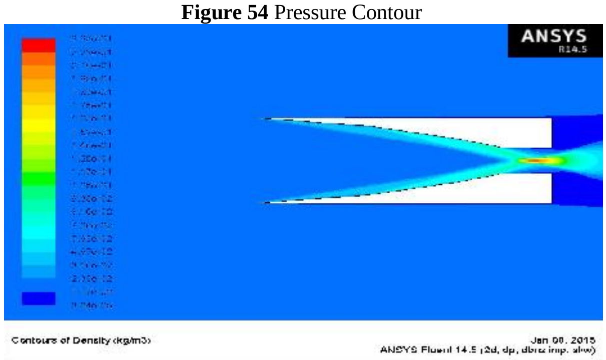


Figure 55 Density Contour

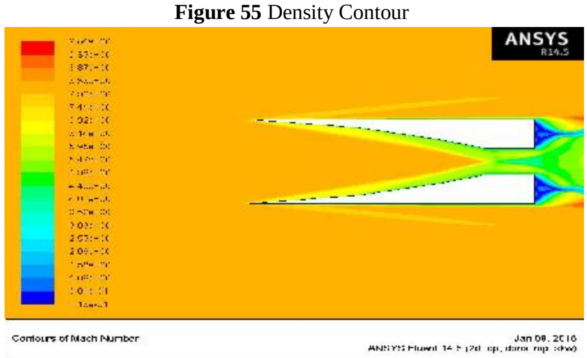
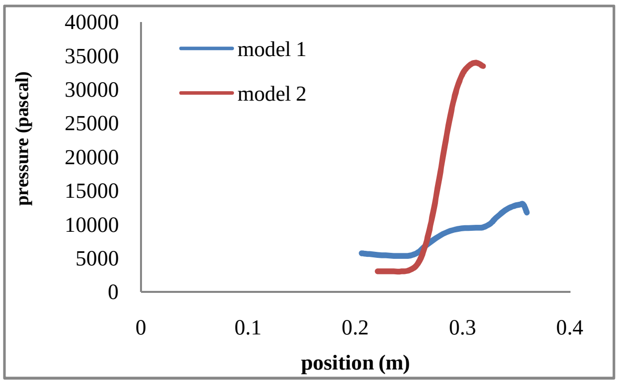


Figure 56 Mach Contour

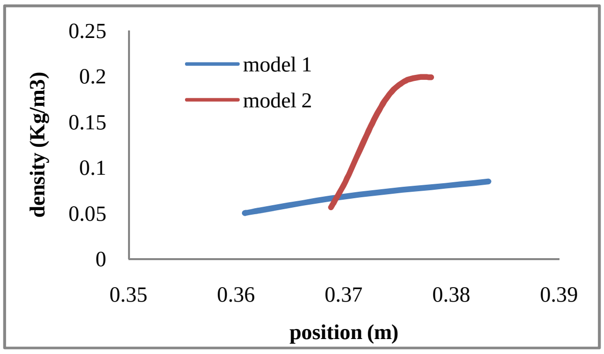
RESULTS AND DISSCUSION

The simulation contours obeys the flow pattern which analysed here as plots to compare the performance of the models with respect these designs. Here, is to compare the standard parameters such as Pressure, density and Mach number between the model 1&2, 3&4, 5&6 in two Mach numbers.

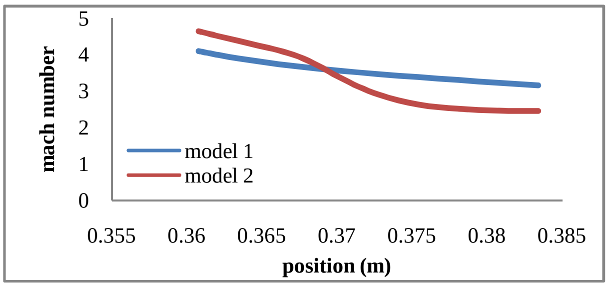
For Mach Number 5



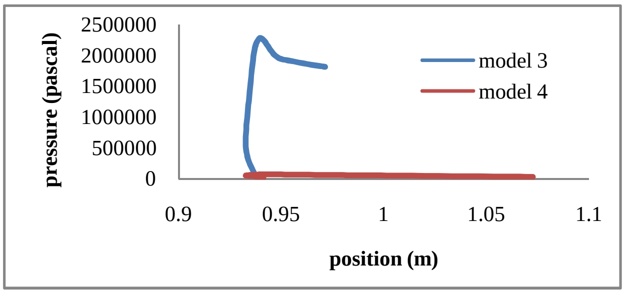
Graph 1 pressure difference between model 1&2



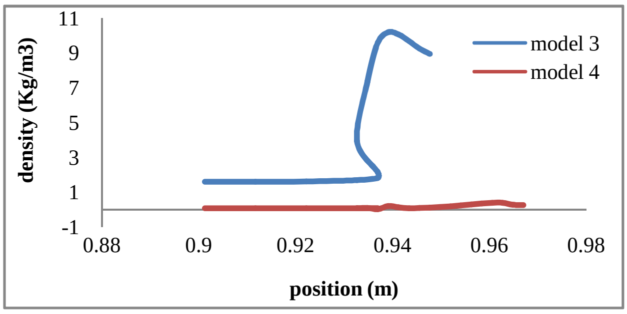
Graph2 Density difference between model 1&2



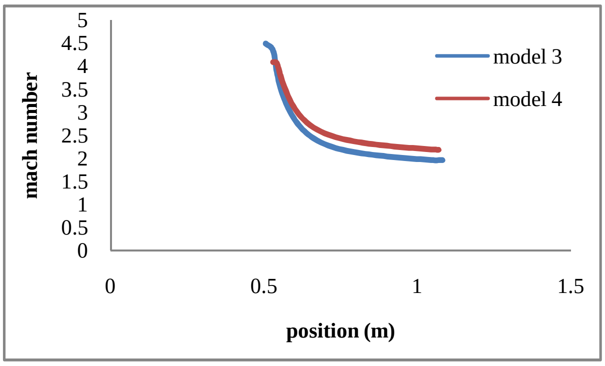
Graph3 Mach number between model 1&2



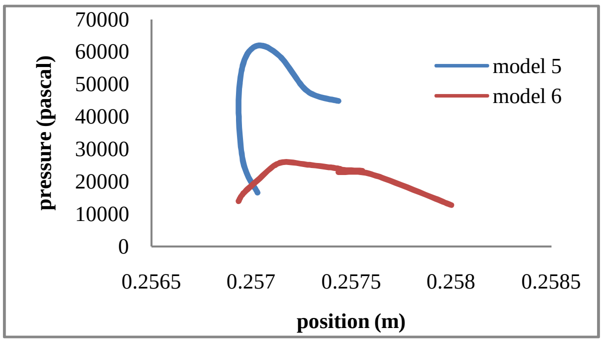
Graph4 Pressure difference between model 3&4



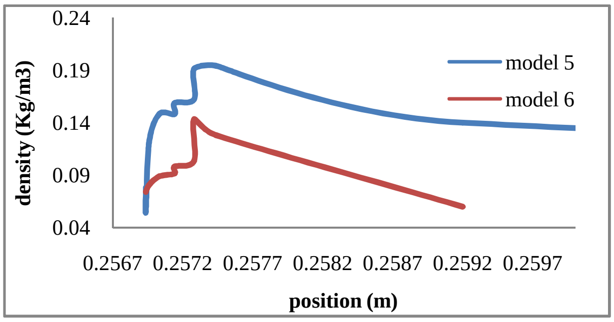
Graph5 Density difference between model 3&4



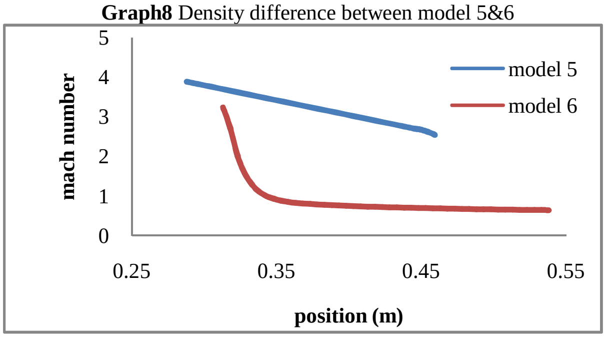
Graph6 Mach number between model 3&4



Graph7pressure differences between model 5&6



Graph8 Density difference between model 5&6

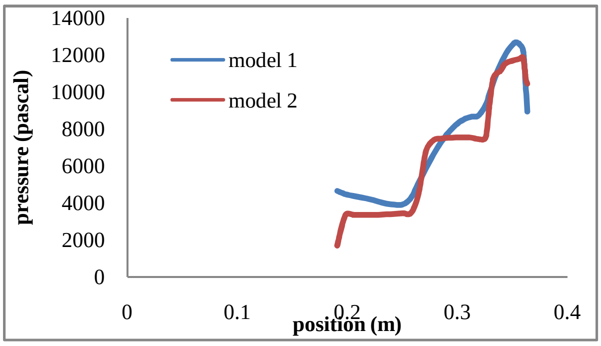


Graph9 Mach number between model 5&6

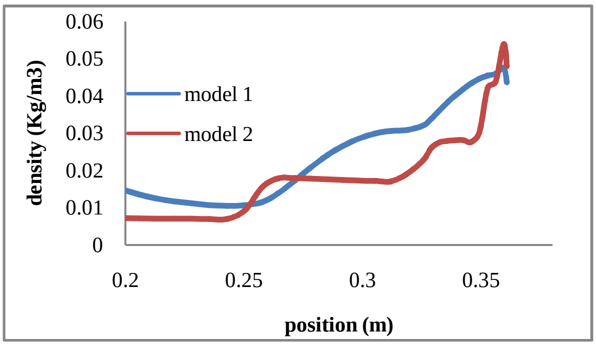
And finally concluded from above comparison graphs of all the scramjet inlet model given below

Model 1&2 – Sharp leading edge (model 2) gives higher performance than the blunted leading edge (model 1).
Model 3&4 – Four ramped inlet (model 3) gives greater performance than the two ramped inlet model (model 4)
Model 5&6 – But in blunted leading edge (model 5) gives better performance when compared to sharp leading edge (model 6) axisymmetric inlet models

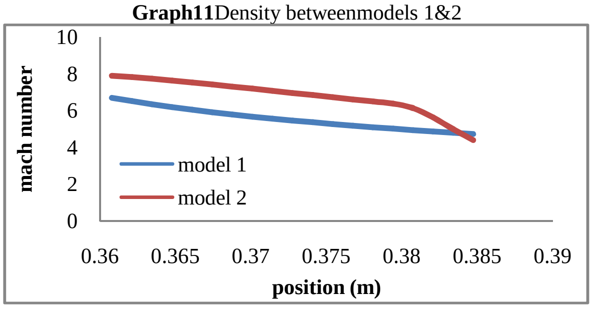
For Mach Number 8



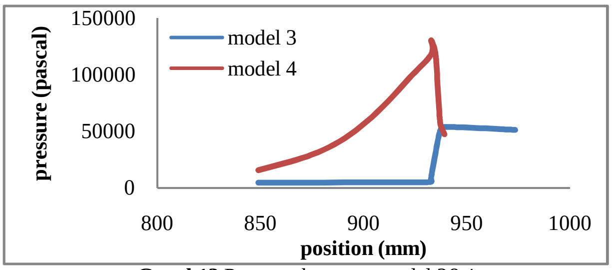
Graph10 Pressure between model 1&2



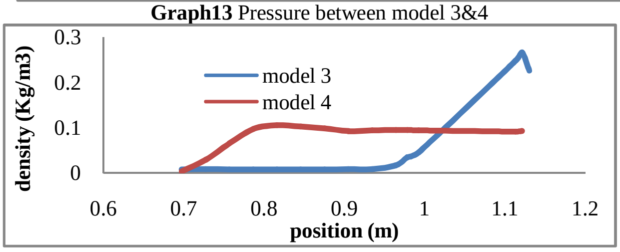
Graph11 Density betweenmodels 1&2



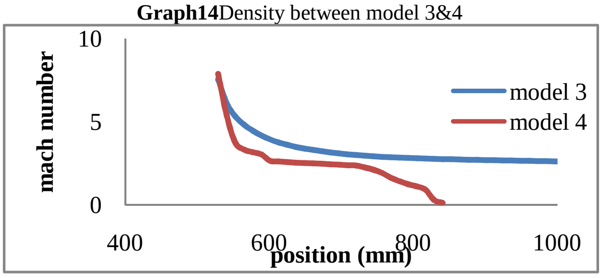
Graph12 Mach number between model 1&2



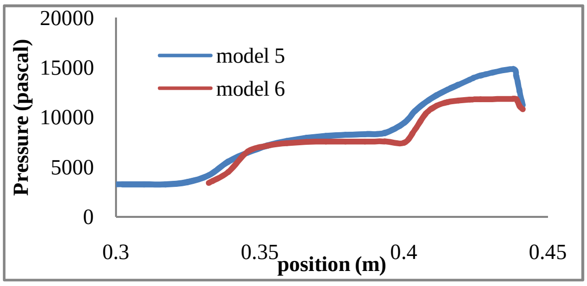
Graph13 Pressure between model 3&4



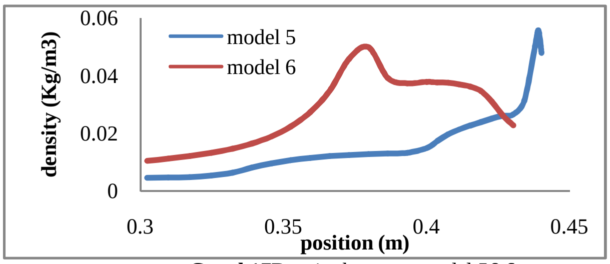
Graph14 Density between model 3&4



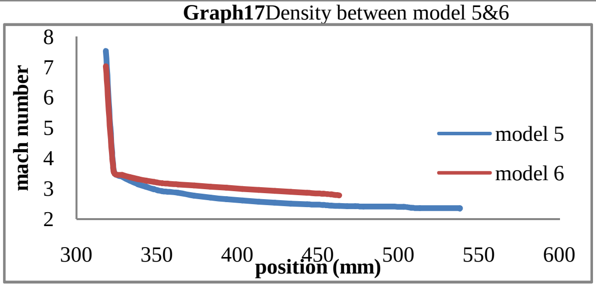
Graph15 Mach number between model 3&4



Graph16 Pressure between model 5&6



Graph17 Density between model 5&6



Graph18 Mach number between model 5&6

And finally concluded from above comparison graphs of all the scramjet inlet model given below

Model 1&2 – Blunted (model 1) and sharp leading edge (model 2) are almost same values in standard parameters.

Model 3&4 –But two ramped inlet model (model 4) gives better results when compared to four ramped model.

Model 5&6 – But blunted leading edge (model 5) gives better higher performance when compared to sharp leading edge (model 6) axisymmetric inlet models

CONCLUSION

The purpose of this paper was to determine which model is best when comparing to other models with two Mach number. Hence, a Scramjet engine was then modeled in GAMBIT and analysis was carried out in FLUENT for the same with different design models.

Table 10 maximum values of various models

Model	1	2	3	4	5	6
Static pressure(pa)	12979.64	33985.88	2272689	83145.63	61942.8	31581.39
Density(kg/m^3)	0.1175706	0.2179502	10.19968	0.4261046	0.1941202	0.1897528
Static temperature (k)	1261.69	1403.429	1337.236	1315.088	1317.955	1257.355

Table 11 maximum values of various models

Model	1	2	3	4	5	6
Static pressure(pa)	12690.87	12650.36	53676.96	130485.1	66405.25	52168.04
Density(kg/m^3)	0.0723200	0.0849673	0.105547	0.2890632	0.1036289	0.2324728
Static temperature(k)	3241.732	3094.486	3095.757	3057.646	3243.61	2846.257

Amongst all designs, a design with four ramps yielded better results than the other designs. By this Analysis we can conclude the “K-omega turbulence model exactly simulates the flow field characteristics in supersonic and hypersonic conditions” in capturing shocks at leading edges. The result obtained in the present study and its analysis is applicable only to a similar or a congruent geometry to the geometry that has been proposed in this work. Thus the vital performance parameters obtained from the FEM numerical simulation are compared and analysed by parameterizing various inlet ramp contour, Mach number and cowl angle at hypersonic limits. Table below Approximate values of maximum increase of parameters inMach 5 and 8 from contour

References

1. A Numerical Investigation Of Flowfield Modification In High-Speed Airbreathing Inlets Using Energy Deposition by Matthew Flynn Rohweder ,missouri university of science and technology
2. Analysis Of A Three-Dimensional, High Pressure Ratio Scramjet Inlet With Variable Internal Contraction, Oliver M. Hohn1 and Ali Gülhan2,German Aerospace Center (DLR), 51147 Cologne, Germany
3. Design Of Three-Dimensional Hypersonic Inlets With Rectangular-To-Elliptical Shape Transition,M. K. Smart, NASA Langley Research Center, Hampton, Virginia 23681
5. Engineering and Engineering Mechanics, The University of Texas at Austin,210 E. 24th St. WRW 305A, Austin, Texas, 78712,
6. Experimental And Computational Investigation Of A Dynamic Starting Method For Supersonic/Hypersonic Inlets, Ryan Throckmorton1 and Joseph A. Schetz2 ,Virginia Tech, Blacksburg, VA, 24061-0203, USA
7. Numerical Computations For Designing A Scramjet Intake, M. Krause, B. Reinartz, J. Ballmann, Department of Mechanics, RWTH Aachen University
8. Performance Analysis Of Variable-Geometry Scramjet Inlets Using A Low-Order Model, Derek J. Dalle, Sean M. Torrezy, and James F. Driscollz,University of Michigan, Ann Arbor, MI 48109
9. Scramjet engines enabling the seamless integration of air& space operations, usaPratt & Whitney Space Propulsion, Hypersonics, West Palm Beach, FL, 33410-9600
10. Scramjet Inlets, Professor Michael K. Smart, Chair of Hypersonic PropulsionCentre for Hypersonic, the University of Queensland, Brisbane 4072, AUSTRALIA
11. Shock Induced Separating Flows In Scramjet Intakes,Yufeng Yao and Daniel Rincon, School of Aerospace and Aircraft Engineering, Kingston UniversityRoehampton Vale, Friars Avenue London sw15 3dw, uk

How to cite this article:

Murugesan S *et al*., Computational Analysis Of Scramjet Inlet. *International Journal of Recent Scientific Research Vol. 6, Issue, 4, pp.3391-3403, April, 2015*

****