ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Design and Optimization of Scrafed Nozzle

Madhavi Thanniru
M.Tech- Aerospace Engineering
Department of Aeronautical Engineering
MLR Institute of Technology

Ms. S Alka
Assistant Professor.
Department of Aeronautical Engineering
MLR Institute of Technology

Dr. M. Satyanarayana Gupta
Professor & HoD
Department of Aeronautical Engineering
MLR Institute of Technology

ABSTRACT

The increasing demand for higher performance in rocket launchers promotes the development of nozzles with higher performance, which is basically achieved by increasing the expansion ratio. However, this may lead to flow separation or over expanded flow and ensuing unstationary, asymmetric forces, so-called side-loads, which may present life-limiting constraints on both the nozzle itself and other engine components.

At ground, these types of engines operate in an over expanded flow condition with an ambient pressure higher than the nozzle exit pressure. As the ambient pressure decreases during ascent, the initially over expanded exhaust flow, passes through a stage where it is adapted i.e. the ambient pressure is equal to the nozzle exit pressure, and then finally becomes under expanded. At high altitudes, the under expansion of the flow results in a further expansion of the exhaust gases behind the rocket.

The present thesis elaborates a comprehensive, up-to-date review of External and Internal supersonic flow over conical and scarfed nozzle. The purpose of the report herein is to investigate the interaction effects which occur between the nozzle exhaust flow and the external flow field associated at transonic and supersonic Mach number range.A physical description of turbulent shock wave in the exhaustplume is given, based on theoretical concepts, computational results and experimental observation. This is followed by an in-depth discussion at different conditions for different nozzles. The internal flow characterizes the development of flow through nozzle and external flow describes the exhaust plume characteristics. So, totally the present thesis studies effectiveness of nozzle by using commercial software ANSYS CFX.

INTRODUCTION

NOZZLES: The nozzle uses the pressure generated in the combustion chamber to increase thrust by accelerating the combustion gas to a high supersonic velocity. The velocity that can be achieved is governed by the nozzle area ratio (i.e., the nozzle exit area, divided by the throat area) which in turn is determined by the design ambient pressure-the atmosphere into which the nozzle discharges. Low ambient pressure (encountered at high altitudes) leads to a high nozzle exit area, higher gas exit velocity, and hence, more thrust. High thrust efficiency is achieved as a result of careful design of the nozzle shape or contour.

The amount of thrust produced by the engine depends:

1. On the mass flow rate through the engine
2. the exit velocity of the flow
3. The pressure at the exit of the engine.

The value of these three flow variables are all determined by the rocket nozzle design.

The performance of rocket engines is highly dependent on the internal and external flow of nozzle and aerodynamic design of the expansion nozzle, the main design parameters being the contour shape and the area ratio. The optimal design of traditional bell-type nozzles for given operating conditions (i.e. chamber and ambient pressures) is already supported by accurate and validated tools. However, during operation at chamber pressures below design pressure, the flow (exhaust plume) will not be fully attached, but separated. The separation line will move towards the nozzle exit as the chamber pressure increases (during start-up) or when the ambient pressure decreases

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

(during the vehicles ascent). Different kinds of dynamic loads occur in the nozzle when the flow is separated, the most well known of these being the so called side-load, that has attracted the attention of many researchers. This occurs during testing at sea level condition or during the first phase of the actual flight. The increasing demand for higher performance in rocket launchers promotes the development of nozzles with higher performance and hence larger area ratio, where the problem of flow separation and side-loads is present during a substantial part of the ascent.

One cause of the adverse pressure gradient during ascent is the expansion of the exhaust plume as the rocket gains altitude. In a low ambient pressure environment, the high pressure at the nozzle exit rapidly expands the exhaust jet in both downstream and radial directions. This produces an obstruction to the free-stream flow which forms an adverse pressure gradient near the aft section of the rocket. Ultimately, the flow separates and recirculation from the base of the vehicle to the upstream separation point allows convective transport of hot exhaust gas along the surface of the vehicle.

Figure: Exhaust Plume from Rocket Nozzle

LITERATURE REVIEW

Plume Structure

As mentioned before, the plume structure is complex. Due to this, it is worthwhile to provide an introduction into this subject. The plume structure itself does not depend on the fuel and oxidant, whilst the emissions of course do. According to Simmons [2], the core flow from the nozzle exit can be divided into two sections: the in viscid inner core and the viscous outer core (mixing layer). Within the in viscid core, it is assumed that no chemical reactions occur [2].Figure below presents a schematic of the near field plume structure. In this figure, it can also be seen that the mixing layer thickness increases with increasing distance from the nozzle exit. However, in the in viscid core, shocks and expansion waves occurs and thus, the flow structure is dominated by these gas dynamic characteristics.

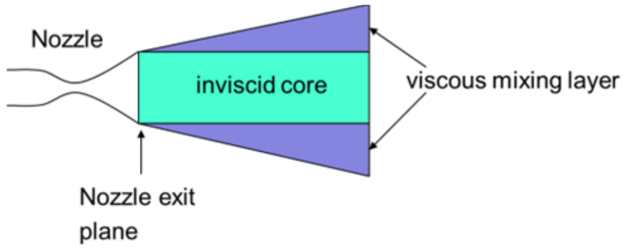

Figure: Plume structure at near field

The plume can be divided into the two parts mentioned above. The flow in the in viscid core is non-reacting in contrast to the mixing layer. In the mixing layer the flow is viscid and surrounds the core flow. The reactions take place by mixing with the ambient environment [2].

Near Field

Lohn et al. [4] produced a schematic view of the plume structure as shown in figure 1.5. In this figure, the flow exiting the nozzle generates at the nozzle lip expansion waves which are reflected at the plume boundary. Expansion waves only occur when the flow in the nozzle has a higher pressure than the ambient environment. When the pressure at the nozzle exit is smaller than the outside pressure, a shock occurs.

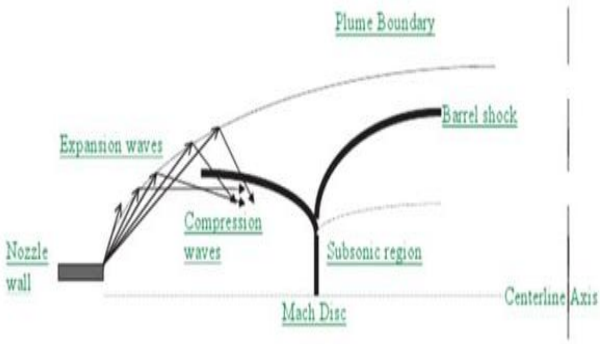

Figure: Flow structure of a supersonic single nozzle plume

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

However, in most exhaust plumes the gas pressure at the nozzle exit is higher than the ambient pressure. This leads to an under-expanded jet. In order to attempt to match the ambient pressure, the plume will expand after the nozzle exit so that there will be expansion waves at the nozzle exit. These waves will propagate downstream to a point where the expansion waves intercept the ambient boundary, which has a constant pressure.

Alternatively, Denison et al. [8] developed a model to investigate the reactions of the plume with the ambient atmosphere. The authors conclude that afterburning can be a source of local ozone depletion. They also state that afterburning can convert HCl into Cl and other active chlorine containing species. Already in the late 1970s, numerical simulations were conducted by Gomberg and Stewart [8]. Knowledge about afterburning in SRM plumes and the resulting chemical reactions were obtained through their simulations.

Nozzle Fundamentals

The main system used for space propulsion is the rocket – a device that stores its own propellant mass and expels this mass at high velocity to provide force. This thrust is produced by the rocket engine, by accelerating the propellant mass particles to the desired velocity and direction, and the nozzle is that part of the rocket engine extending beyond the combustion chamber. Typically, the combustion chamber is a constant diameter duct into which propellants are injected, mixed and burned. Its length insufficient to allow complete combustion of the propellants before the nozzle accelerates the gas products. The nozzle is said to begin at the point where the chamber diameter begins to decrease. The flow area is first reduced giving a subsonic (Mach number < 1) acceleration of the gas. The area decreases until the minimum or throat area is reached. Here the gas velocity corresponds to a Mach number of one. Then the nozzle accelerates the flow supersonically (Mach number > 1) by providing a path of increasing flow area.

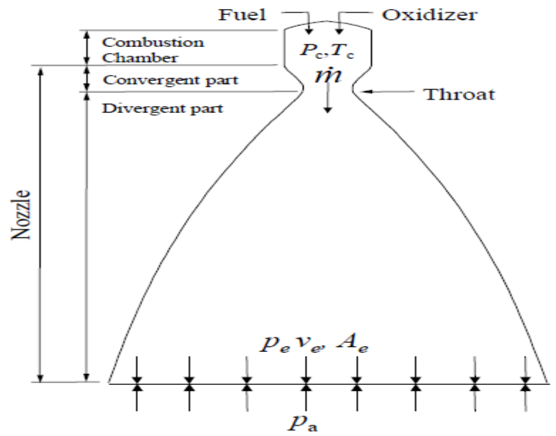

Fig. Nozzle configuration

Nozzle Contour Design And Flow Field

Different types of conventional convergent-divergent rocket nozzles exist, each producing their own specific internal flow field. Before analyzing separation and side-loads in rocket nozzles, it is essential to understand the features of the different contour types, as the internal flow field determines the characteristics of the nozzle separation behaviour. Figure below shows examples of the Mach number distribution in some of the most common nozzle types. Methods to generate these nozzle contours will be discussed in the following.

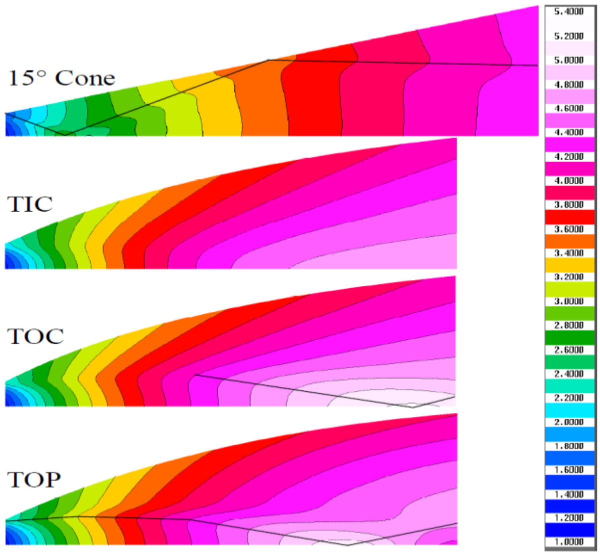

Figure: Mach number distribution in a 15^{0}conical (ε =43.4, *L*=20.9), TIC (*MD*=4.67, ε=43.4, *L*=17.7), TOC(ε=43.4, *L*=17.7) and TOP ε=43.4, *L*=17.7) nozzle (From top to bottom). The thick line indicates the approximate position of the internal shock.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

IDEAL NOZZLE

As mentioned above, the ideal nozzle is a nozzle that produces uniform exit flow conditions. The nozzle contour, which achieves this, can be designed with MOC. An outline of an ideal nozzle flow is shown in Figure below.

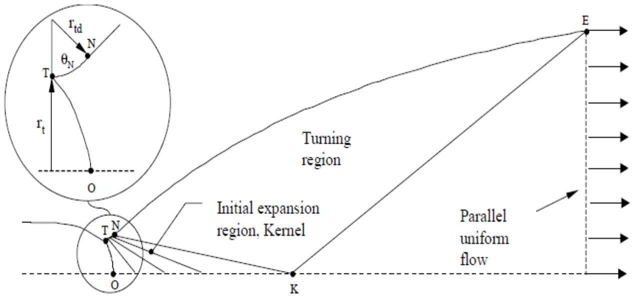

Figure. Basic flow structures in an ideal nozzle

Contour TNE is the diverging portion of the nozzle. After the initial expansion TN, the contour NE turns the flow over to axial direction. TN also defines the Mach number at K, which is equal to the design Machnumber obtained at the exit. With the Mach line NK defined it is possible to construct the streamline between N and E with the use of MOC which patches the flow to become uniform and parallel at the exit and thus complete the nozzle design. In Figure below the left and right running characteristics are shown for an ideal nozzle. The design Mach number is M=4.6 and the gas properties are γ=1,2 with a molecular mass =13.63g/mole. The TDK program was used to generate the starting line TO.

DIRECTLY OPTIMISED NOZZLES

The classical design methods described above rely on an in viscid design. After an in viscid design has been completed, a boundary layer correction is added to compensate for the viscous effects. The main reason for calculating the in viscid and viscous flows separately was that the computational capability in the past was such that the Navier-Stokes (N-S) equations could not be used in the design of contours. Advances in the computational technology since the 1950's allow scientists nowadays to use N-S solvers in parallell with direct optimisation techniques in the design loop. A typical design or an optimisation may include the following steps:

1. The design requirements are specified.
2. An objective function is constructed. The minimum or maximum of which yields the design requirements, e.g. max. performance and min. nozzle weight etc..
3. The set of design parameters or variables is specified.
4. An initial value for each of the design parameters is estimated.
5. An initial solution is computed by using the estimated design parameters.
6. The objective function is computed from the difference between the design requirements and the computed solution.
7. The sensitivity of the objective function to the design parameters is calculated.
8. An optimisation problem is solved to generate a new set of design variables.
9. A new solution is computed and compared with the design requirements.
10. If the design requirements are met or a minimum or maximum is reached, then the procedure stops, otherwise the process is repeated from step 6 onward.

PROBLEM DESCRIPTION

Nozzles of high performance rocket engines in use for first- or main stage propulsion, e.g. the AmericanSSME, the European Vulcain, or the Japanese LE-7, operate from sea-level with one bar ambient pressure upto near vacuum. At ground, these types of engines operate in an over expanded flow condition with anambient pressure higher than the nozzle exit pressure. As the ambient pressure decreases during ascent, the initially over expanded exhaust flow, passes through a stage where it is adapted i.e. the ambient pressure is equal to the nozzle exit pressure, and then finally becomes under expanded. This is because a nozzle flow with a small overexpansion can adapt to the ambient without forming a strong shock system, i.e. the Mach disc. Innozzles featuring an internal shock, e.g. TOC, TOP and CTIC nozzles, the cap-shock pattern can be observed. This is the pattern first observed at the nozzle exit during start up. By increasing the combustion chamber pressure, the flow becomes less over expanded. At some point the internal shock

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

intersects the centre line and a transition to a Mach disc pattern takes place, see Figure below a) and Figure below.

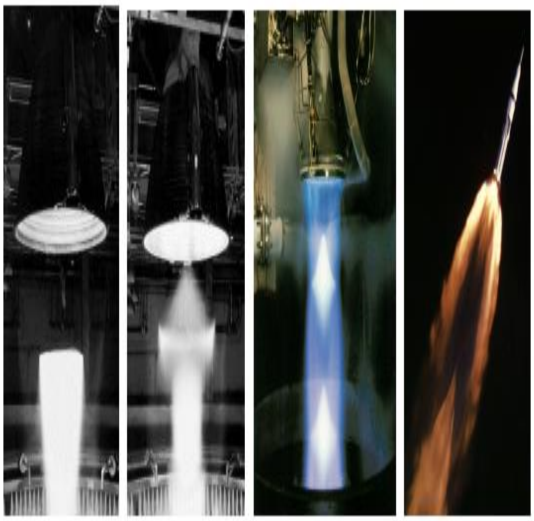

Figure: Exhaust plume pattern: a) Vulcain, overexpanded flow with classical Mach disk, b) Vulcain,overexpanded flow with cap-shock pattern, c) RL10-A5, overexpanded flow with apparent regular reflection, and d) under expanded flow, photographed during launch of Saturn 1-B[13].

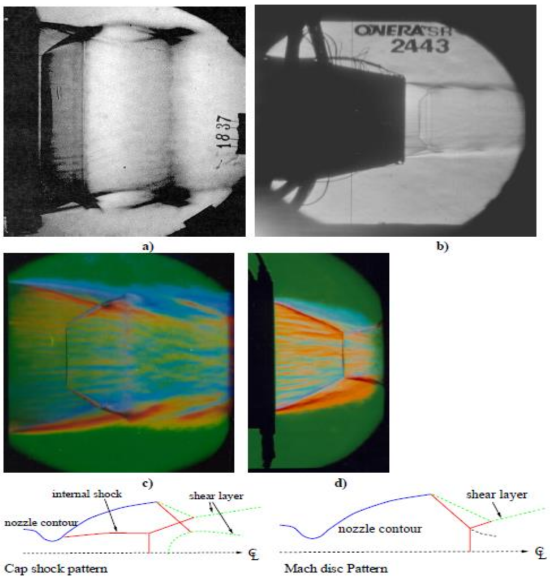

Figure:. Exhaust plume patterns for parabolic subscale nozzles, with cap-shock pattern, a) S1 VAC FFA,b) TOP ONERA, c) P6 TOP DLR, and d) for a truncated ideal nozzle, with Mach disk, P6 TICDLR [13].

Figure: a-c) show Schlieren images of the exhaust plume of parabolic sub-scale nozzles tested at DLR,ONERA, and FFA. For comparison, the exhaust plume of a truncated ideal nozzle is also shown where the classical Mach disk is clearly visible.

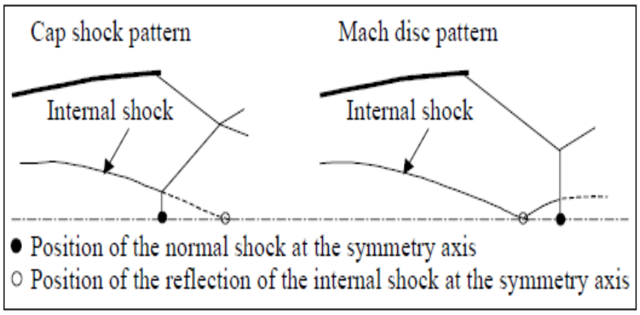

Figure:. Illustration of transition between cap shock and Mach disc pattern: The transition occurs when the normal shock hits the reflection point of the internal shock at the symmetry axis.

The above described shock patterns are not only an exhaust plume phenomenon. They also exist inside the nozzle at highly overexpanded flow conditions, when the jet is separated from the nozzle wall.

COMPUTATIONAL FLUID DYNAMICS

CFD or computational fluid dynamics is predicting what will happen, quantitatively, when fluids flow, often with the complications of simultaneous flow of heat, mass transfer (eg perspiration, dissolution), phase change (eg melting, freezing, boiling), chemical reaction (eg combustion, rusting), mechanical movement (eg of pistons, fans, rudders), stresses in and displacement of immersed or surrounding solids.

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to simulate the interaction of fluids and gases with the complex surfaces used in engineering. Even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. Ongoing research, however, may yield software that improves the accuracy and speed of complex

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

simulation scenarios such as transonic or turbulent flows. Initial validation of such software is often performed using a wind tunnel with the final validation coming in flight test.

The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for in viscid and Navier- Stokes equations for viscous flow).

NOZZLE DESIGN

NOZZLES: The nozzle uses the pressure generated in the combustion chamber to increase thrust by accelerating the combustion gas to a high supersonic velocity.

The design of the nozzle must trade off:

1. Nozzle size (needed to get better performance) against nozzle weight penalty.

2. Complexity of the shape for shock-free performance vs. cost of Fabrication.

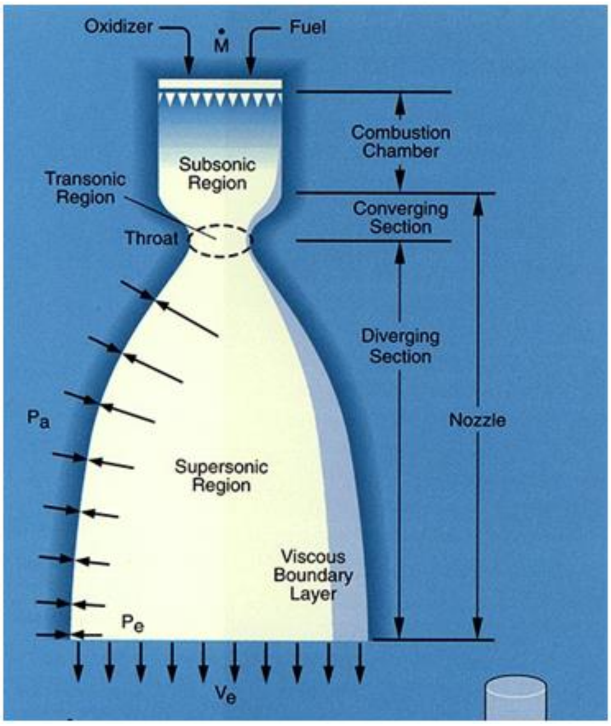

Figure. 6.1 Nozzle Regions

Explanation: Convergent-divergent: convergent section followed by a fixed divergent section for the design of the nozzle. This nozzle configuration is called a **convergent-divergent**, or **CD**, nozzle. In a CD rocket nozzle, the hot exhaust leaves the combustion chamber and converges down to the minimum area, or **throat,** of the nozzle. The throat size is chosen to **choke** the flow and set the mass flow rate through the system. The flow in the throat is sonic which means the Mach number is equal to one in the throat. Downstream of the throat, the geometry diverges and the flow is is entropically expanded to a supersonic Mach number that depends on the area ratio of the exit to the throat. The expansion of a supersonic flow causes the static pressure and temperature to decrease from the throat to the exit, so the amount of the expansion also determines the exit pressure and temperature. The exit temperature determines the exit speed of sound, which determines the exit velocity. The exit velocity, pressure, and mass flow through the nozzle determine the amount of thrust produced by the nozzle. Decreasing flow area results in subsonic (Mach number < 1) acceleration of the gas. The area decreases until the minimum or "throat" area is reached. Here the gas velocity corresponds to a Mach number of one, denoting that the flow velocity is equal to the local speed of sound of the particular gas. Then the nozzle accelerates the flow supersonically (Mach number > 1) by providing a path of increasing flow area.

The major objective of the designer of the nozzle is to maximize the thrust coefficient and minimize the weight of the nozzle. One way to maximize the thrust is to continuously adopt the nozzle such that the pressure thrust remains zero at all altitudes. The weight of the nozzle depends on the material of construction, wall thickness and the length of the nozzle which depends on the expansion ratio.

COMPUTATIONAL STUDY: Computational study was conducted on two nozzle configurations to compare a conical nozzle with scarfed nozzle optimize for maximum thrust using Rao method.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

In this work both conical nozzle and scarfed nozzle are designed for operating with a solid propellant booster carrying propellant combination as NC (Notrocellulose) and NG (Nitroglycerine).

- With the dimensions given in the two Dimensional sectional view the conical, elliptical and scarfed nozzle configurations are modelled in ICEM CFD.
- Pre processing of the model that is meshing of the models is done in the ICEM CFD.
- Then the computational analysis of conical and scarfed nozzles has been performed to estimate the temperature, velocity, pressure and other flow parameters in CFX. The results are presented in the form of contour plots and graphs obtained from the simulation in CFX.

Conical Nozzle: A conical nozzle of length 123.80 mm, cone angle 16˚ and half angle 8˚ is designed with the coordinate points.

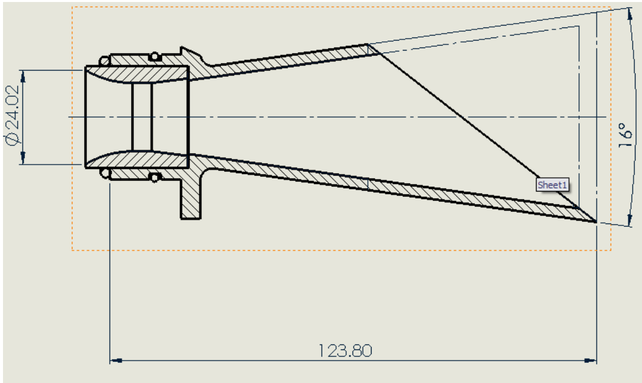

Figure. Two Dimensional Sectional View

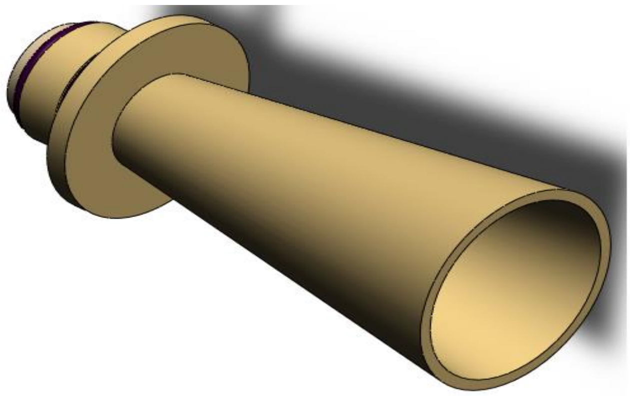

Figure. Conical Nozzle

Scarfed Nozzle: For certain tactical missile designs it is desirable to employ several starp-on solid rocket boosters attached circumferentially around a centre body. However, this propulsion system concept creates significant problems for the missile designer. The performance of each individual booster will be subject to the motor-to-motor variability in thrust level and burn time that is characteristic of solid rocket motors. During this booster operation, the thrust level differences between the individual rocket motors will result in a moment about the centre of gravity. The differences in burn time between various motors will also generate significant moments as each motor will cease operation at a different time. Any moments generated by the propulsion systems will have an adverse impact on the design of the guidance and control system for the system for the missile. Consequently, it is desirable to eliminate these moments.

The scarfed nozzle is composed of three distinct sections:

Throat: Throat section is constructed from two circular arcs intersect smoothly at the throat wall point, T. the throat section is completely specified by the upstream arc radius ρ_{tu}, the downstream arc radius ρ_{td}, and the throat wall radius Y_t.

Basic nozzle: Downstream the throat is the basic nozzle is the unscarfed supersonic expansion contour that attaches smoothly to the throat wall at point E. In case of conical contour, the basic nozzle is completely specified by the half angle, α and expansion ratio at point E,

Nozzle extension: the nozzle extension is a cylindrical wall of radius, Ye, that attaches to the basic nozzle at point E forming discontinuity in the slope of the nozzle wall. The nozzle extension is truncated (scarfed) by a plane which passes through point E at an angle, β to the nozzle axis and normal to the x-y plane. The end of the nozzle extension is denoted as wall point F. The nozzle extension is completely specified by the scarf angle β

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

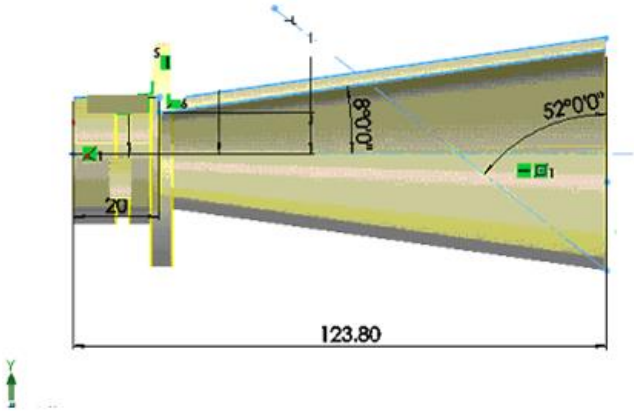

Figure. Scarf angle of nozzle

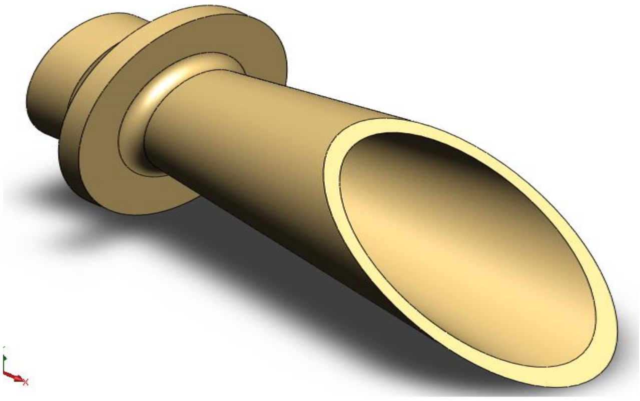

Figure. Scarfed Nozzle

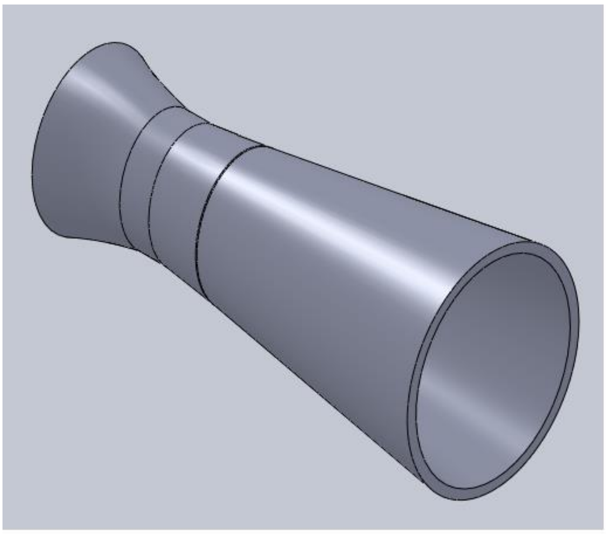

Figure 6.6 Full length Nozzle

MESHING:

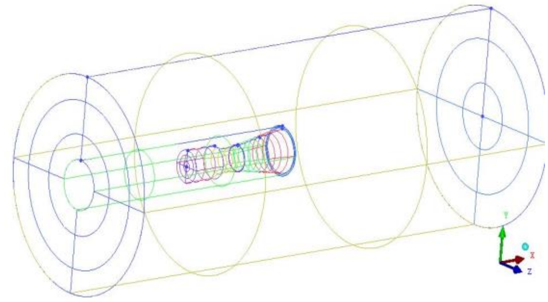

Figure. Conical Nozzle geometry in ICEM

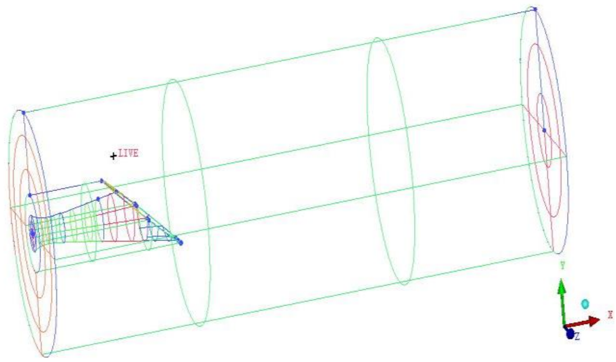

Figure. Scarfed Nozzle with domain geometry in ICEM

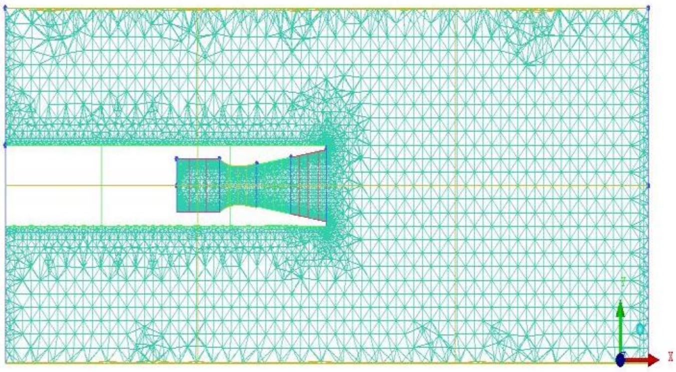

Figure. Conical Nozzle and domain with Mesh in ICEM

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

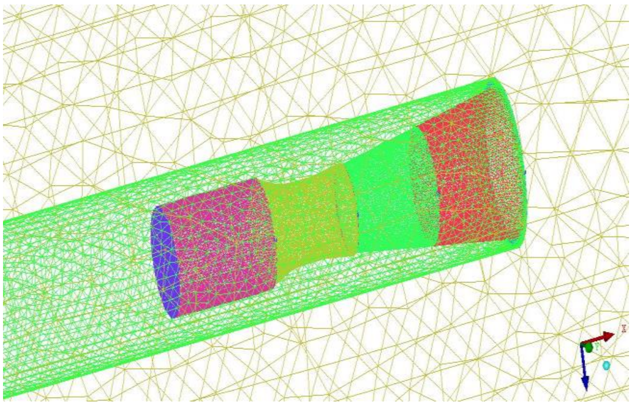

Figure. ConicalNozzle with Mesh in ICEM

CFD MODEL

The computational domain size is selected as AxB (five times the diameter of nozzle x three times the diameter of the nozzle) for covering all major effects of the flow. The nozzle of given dimensions is at the centre of the inlet (left) portion. A proper boundary condition at inlet permits flow through the nozzle inlet only.

BOUNDARY CONDITIONS

A chamber pressure of 49 bar and velocity 65.41 m/s is applied to the inlet of the nozzle designated as the pressure inlet. The temperature of 3000 K obtained according to computation is also applied at the inlet. Ambient conditions were assumed at the outlet (The gauge pressure was taken as zero and temperature as 300 K). The nozzle wall was assumed to be perfectly adiabatic (By taking conduction of the wall as zero while computing). The boundary at the inlet end is also taken to be adiabatic in a similar way.

NUMERICAL ANALYSIS

Solution procedure

A total energy method is applied by taking into consideration the viscous flow effects. The solution was done in both implicit and explicit formulation (with small time step). Better convergence of solution was obtained while using the explicit formulation. The material type was specified as ideal gas in the case of fluid with density, thermal conductivity and viscosity based on kinetic theory and specific heat as a polynomial function of temperature. Energy equation was used and the viscous model adapted was standard Shear Stress Transport (SST) for turbulence having standard wall functions. This gave the required convergence to all the residuals.

Table. Input Data

	Chamber	Throat	Exit
Temperature	2953	2704	1494
Molecular weight	26.22	26.31	26.40
Pc	50 bar		

RESULTS & DISCUSSIONS

Computational analysis is performed on the conical nozzle of full and short length and Scarfed nozzle to get the contour plots of pressure, velocity, Mach number, velocity at different operating conditions.

Full Nozzle

At M=0

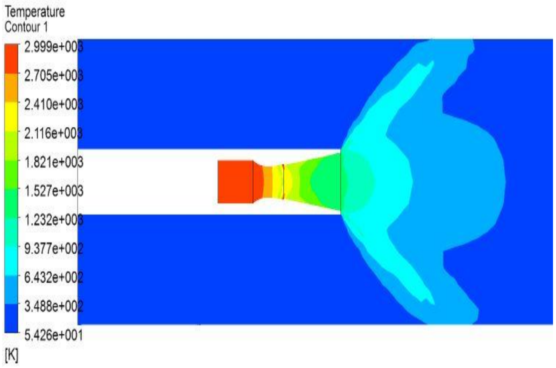

Figure. Temperature contour of Full Nozzle at M=0

Above temperature contour shows the drop to a very low value of 1800 K from 3000 K at inlet to the immediate exit leading to the correct design of nozzle for given conditions.

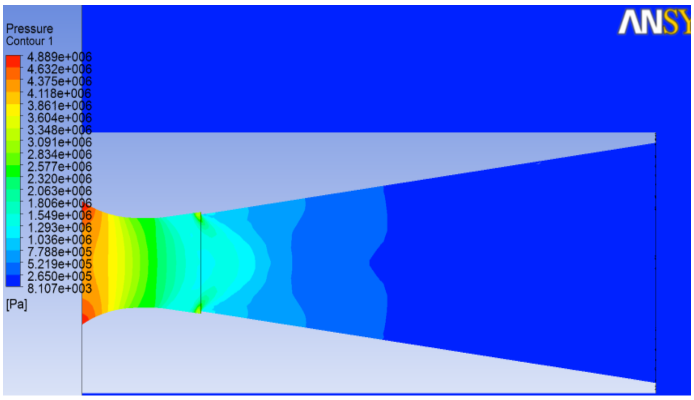

Figure 7.2 Pressure contour of Full Nozzle at M=0

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

The contour plots of pressure shows a variation from 49 bar at inlet of the nozzle to free steam condition at the outlet.

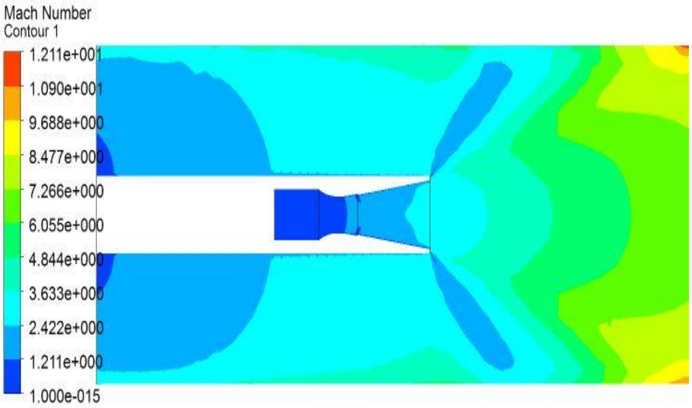

Figure. Mach number contour of Full Nozzle at M=0

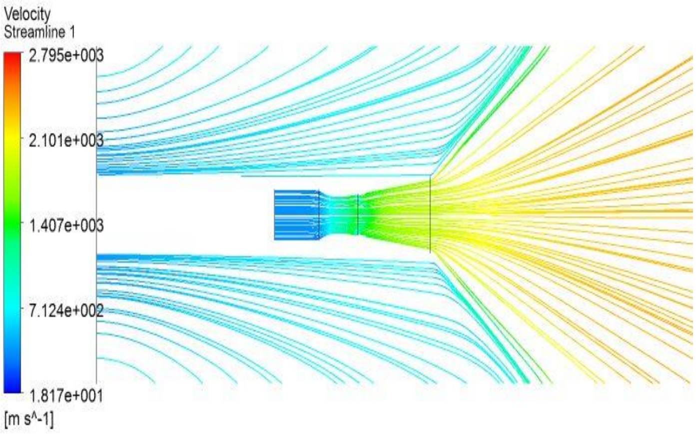

Figure. Velocity streamline of Full Nozzle at M=0

M=2

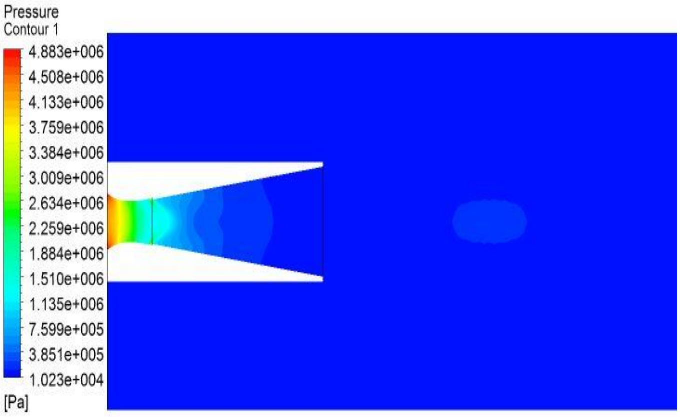

Figure. Pressure contour of Full Nozzle at M=2

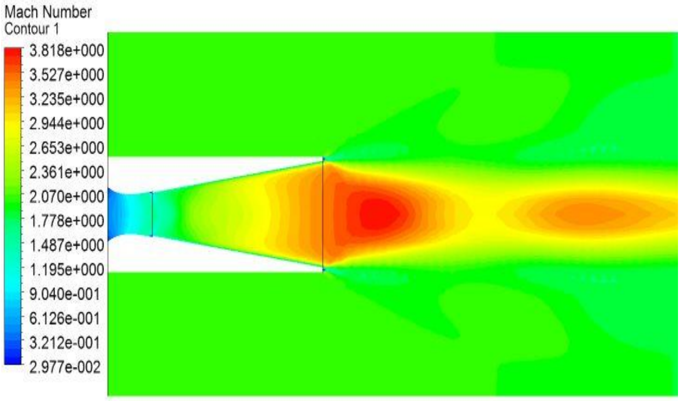

Figure. Mach number contour of Full Nozzle at M=2

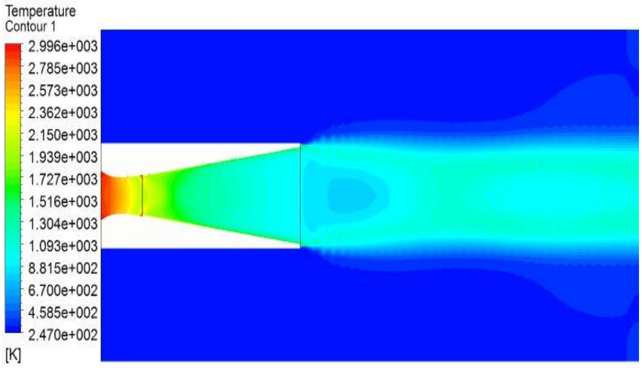

Figure. Pressure contour of Full Nozzle at M=2

M=3

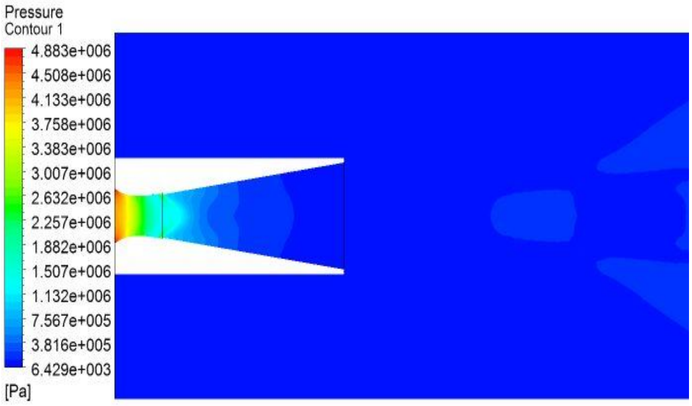

Figure. Pressure contour of Full Nozzle at M= 3

ISSN No: 2348-4845
International Journal & Magazine of Engineering,
Technology, Management and Research
A Peer Reviewed Open Access International Journal

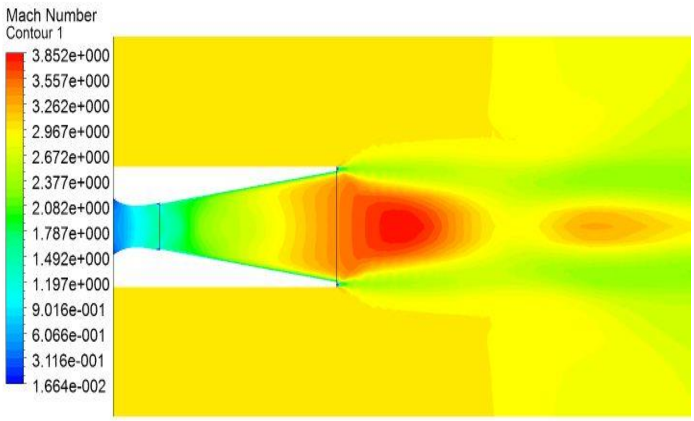

Figure. Mach number contour of Full Nozzle at M= 3

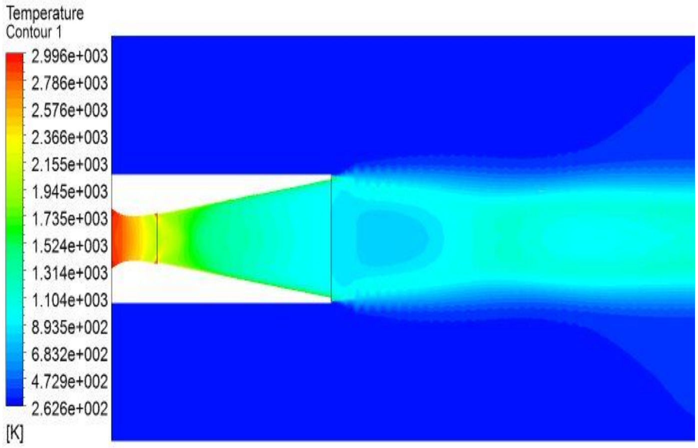

Figure. Temperature contour of Full Nozzle at M= 3

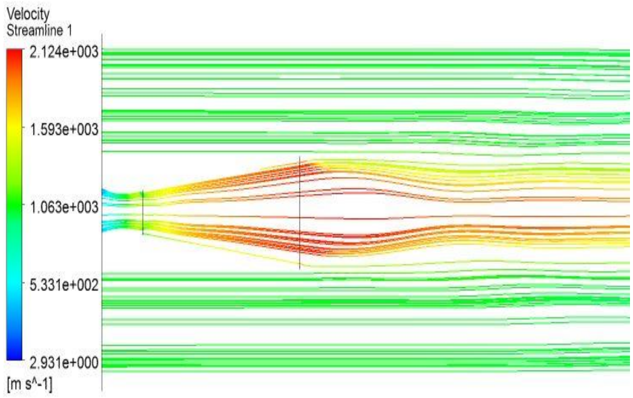

Figure 7.11 Velocity Streamlines of Full Nozzle at M=0

Compared to Mach 0 conditions at Mach 2 the exhaust plume is disturebed which can be seen from figure Mach contour. Even though the internal flow is fully developed without shocks but external flow i.e. exhaust plume has a diamond shock pattern due to variation in ambient pressures. This is an over expanded conditions leading to formation of shock waves to eualth ambient pressure with exit pressure of nozzle.

Short Nozzle

In this case the length of the nozzle has been decreased to find its effect on nozzle performance. The main aim of shortening the length of nozzle is to reduce the effective length offree exhaust gas plume and shock pattern and also in view of weight limitations.

But in general the nozzle length is designed according to the requirement of thrust and to attain shock free flow. Reduction in length of the nozzle leads to minimum pressure drag or boattail drag.

M=0

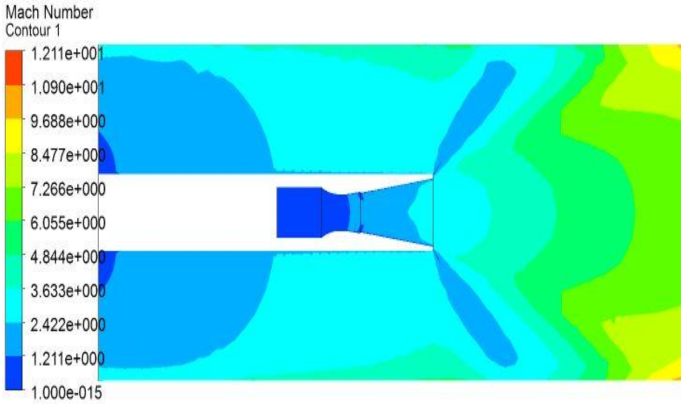

Figure. Mach number contour of Short Nozzle at M= 0

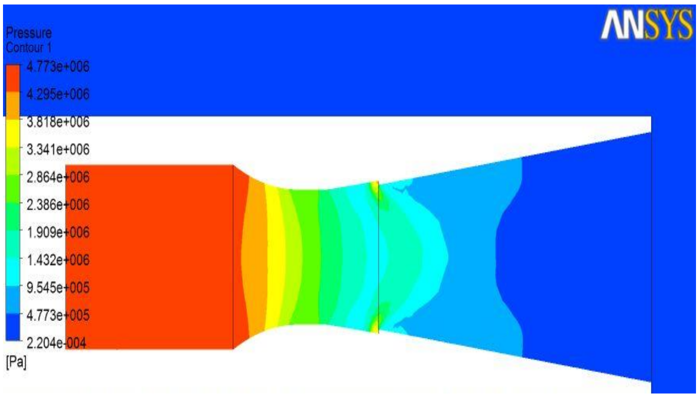

Figure. Pressure contour of Short Nozzle at M=0

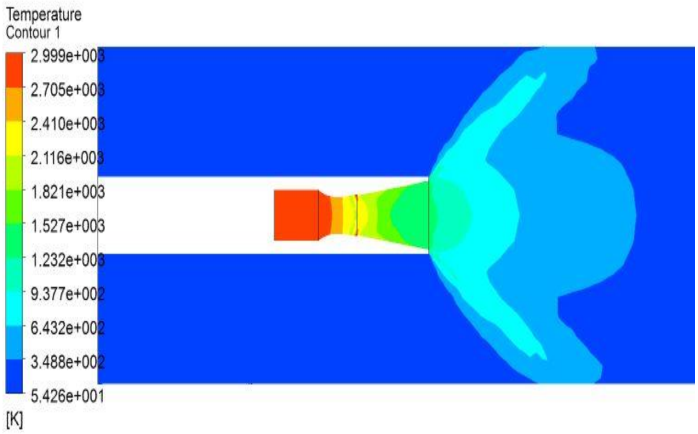

Figure . Temperature contour of Short Nozzle at M=0

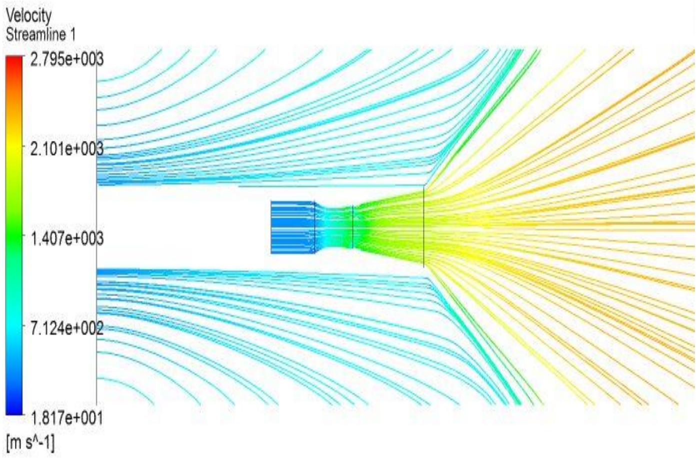

Figure. Velocity streamline of Short Nozzle at M= 0

M = 1.2

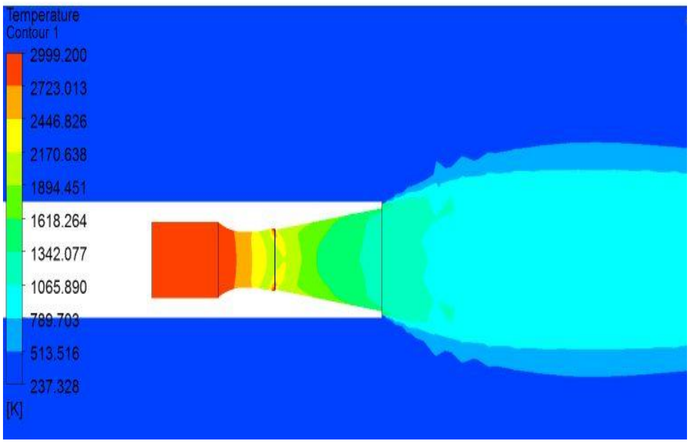

Figure. Temperature contour of Short Nozzle at M= 1.2

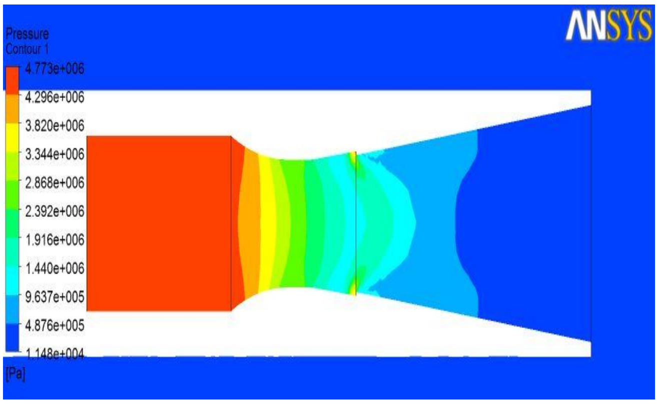

Figure. Pressure contour of Short Nozzle at M= 1.2

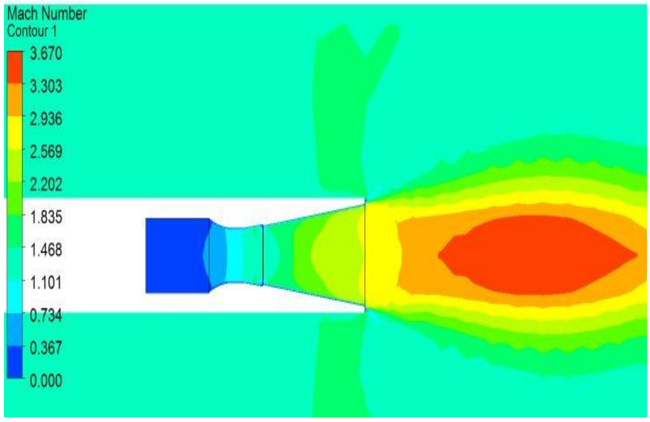

Figure 7.18 Mach number contour of Short Nozzle at M= 1.2

M= 2

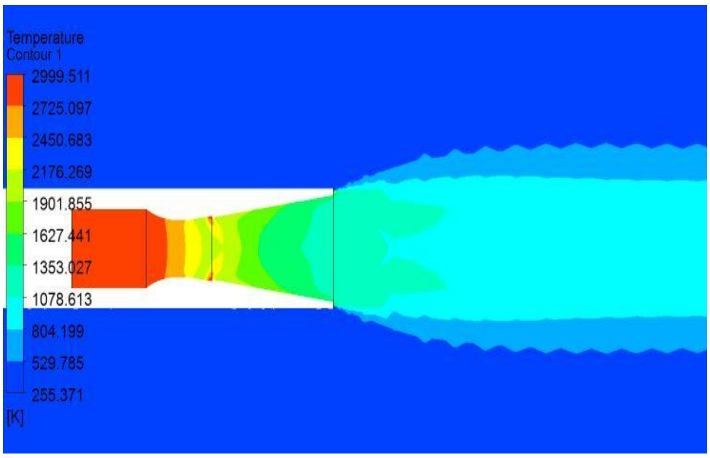

Figure 7.19 Temperature contour of Short Nozzle at M= 2

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

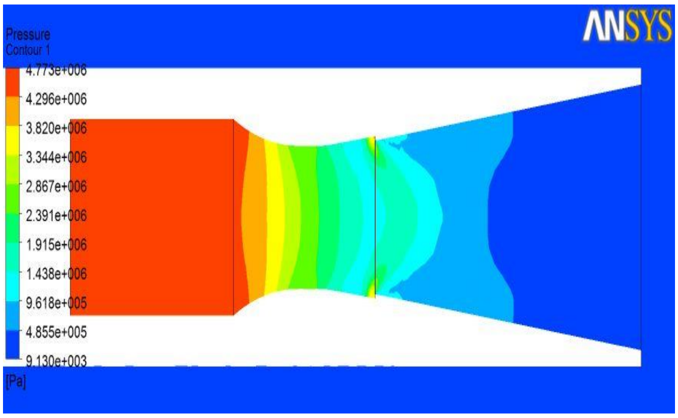

Figure. Pressure contour of Short Nozzle at M= 2

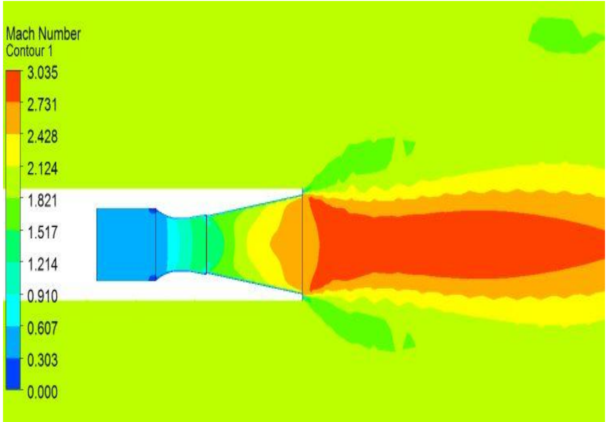

Figure. Mach number contour of Short Nozzle at M= 2

The flow has fully developed to Mach 2 at exit of nozzle but still the exhaust plume has developed it to Mach 3 due to ambient pressure. Shock is of very weak resembling a mach wave such that shock pattern is of more length compared to previous cases.

M=3

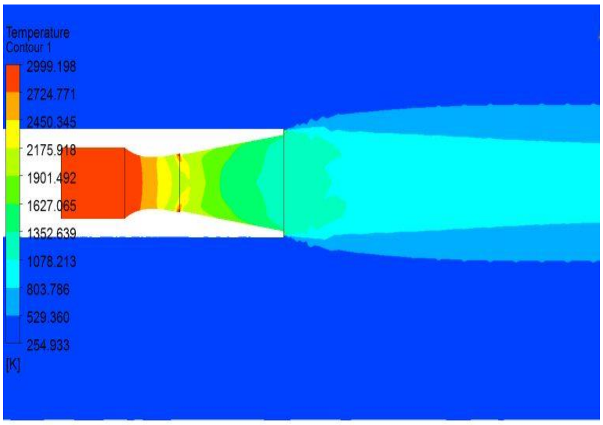

Figure. Temperature contour of Short Nozzle at M= 3

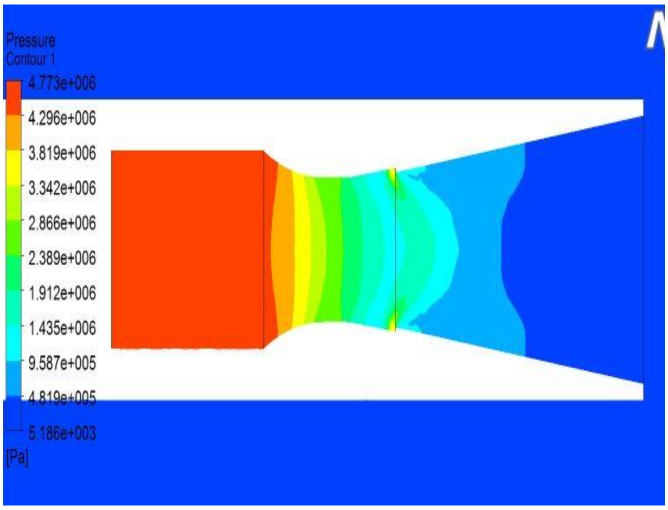

Figure. Pressure contour of Short Nozzle at M= 3

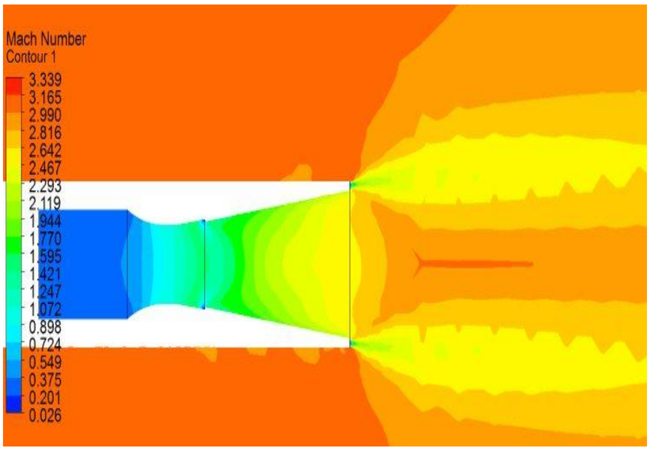

Figure. Mach number contour of Short Nozzle at M= 3

As the length of the body and passage are sized and shaped to provide a flow area for the exhaust gases less than the cross-sectional area of the exhaust plume at the position of the body. As a consequence of the reduced flow area caused by the body, the motor exhaust gases may pass around and through the body passage only after undergoing a normal shock wave upstream of the body. The normal shock wave results in subsonic exhaust gas flow velocity downstream of the shock wave, greatly reducing exhaust temperature and pressure impingement effects farther downstream.

Scarfed Nozzle

The main idea behind choosing Scarfed nozzle is to minimize weight and the effect of exhaust gas

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

temperature on the nozzle. The main engine and its plume could get extremely hot and will radiatesignificant heat to its adjacent components. The stowed solar panel is next to theengine nozzle and becomes the major concern.

M=1.2

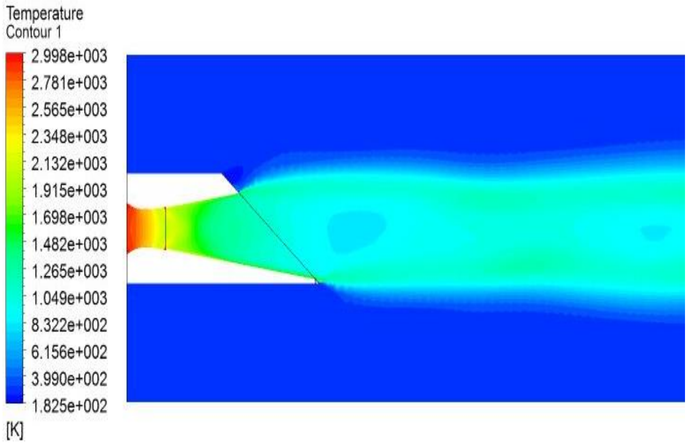

Fig. Temperature contour of Scarfed Nozzle at M= 1.2

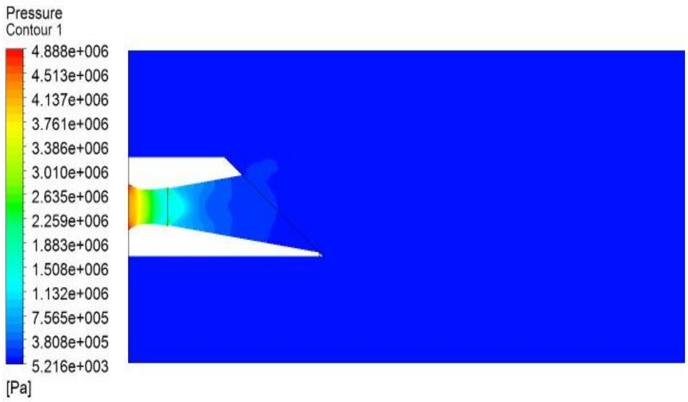

Figure.Pressure contour of Scarfed Nozzle at M= 1.2

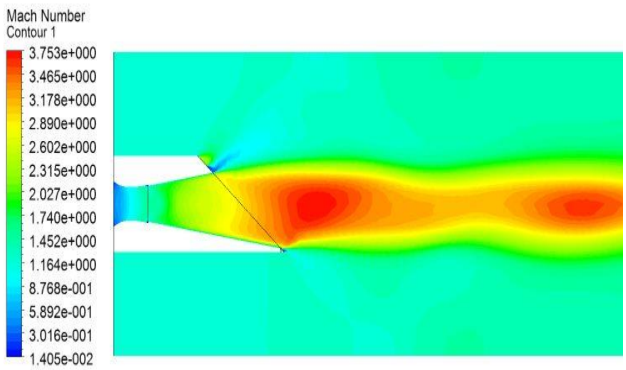

Fig. Mach number contour of Scarfed Nozzle at M=1.2

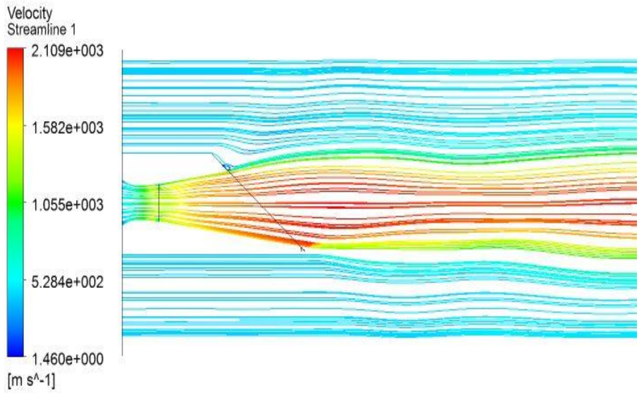

Fig.Velocity streamlines of Scarfed Nozzle at M= 1.2

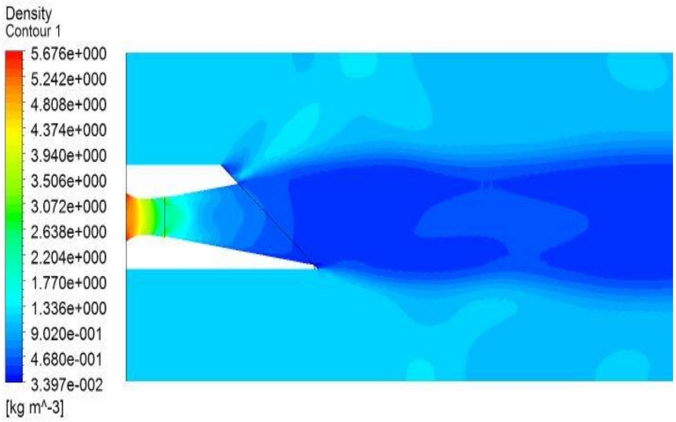

Fig. Density contour of Scarfed Nozzle at M= 1.2

M=2

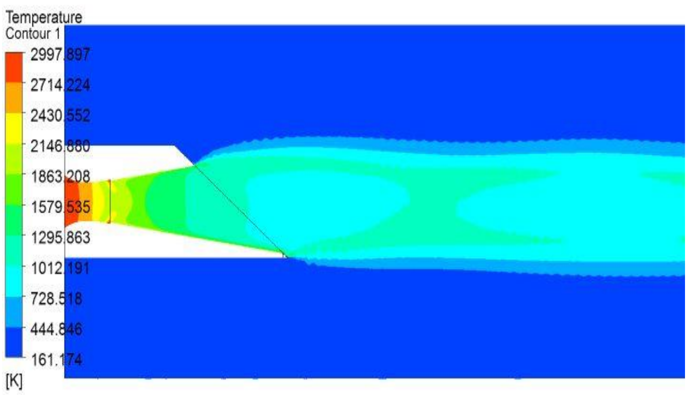

Fig. Temperature contour of Scarfed Nozzle at M= 2

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

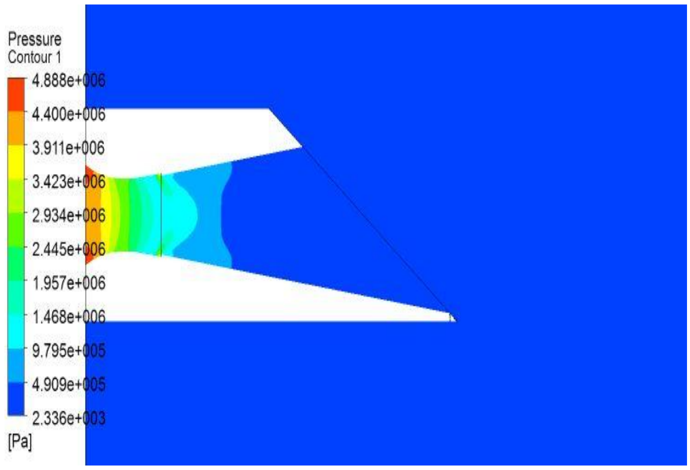

Fig. Pressure contour of Scarfed Nozzle at M= 2

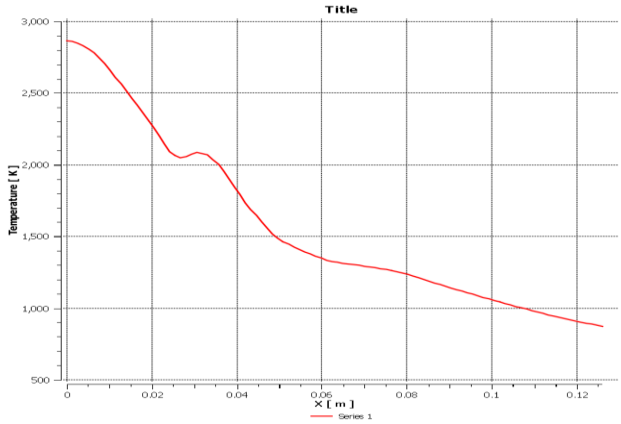

Fig. Temperature plot along axis of Scarfed Nozzle at M= 2

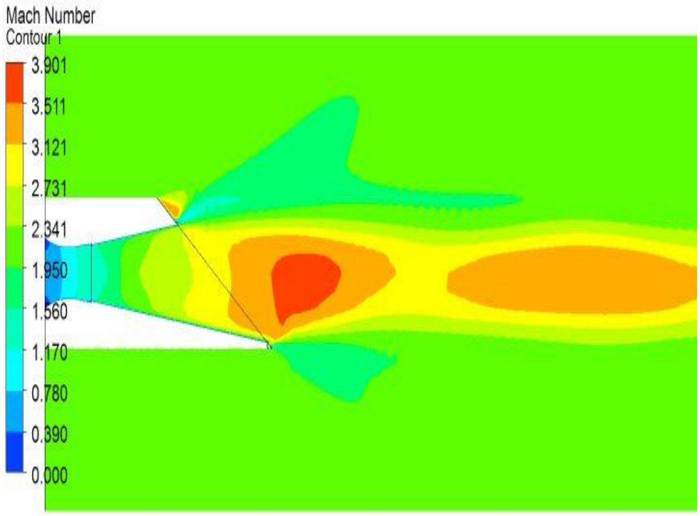

Fig. Mach number contour of Scarfed Nozzle at M= 2

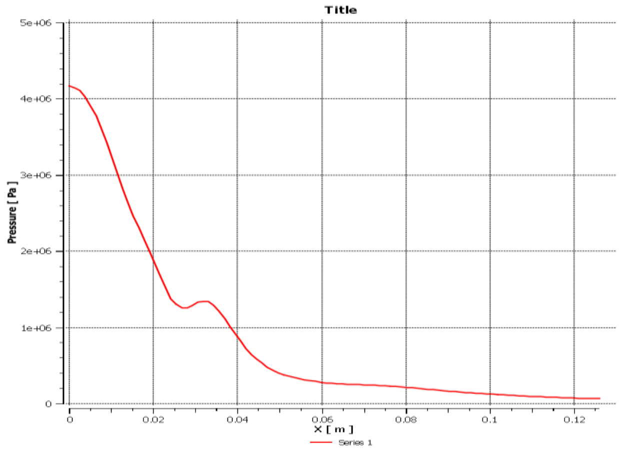

Fig. Pressure plot along axis of Scarfed Nozzle at M=2

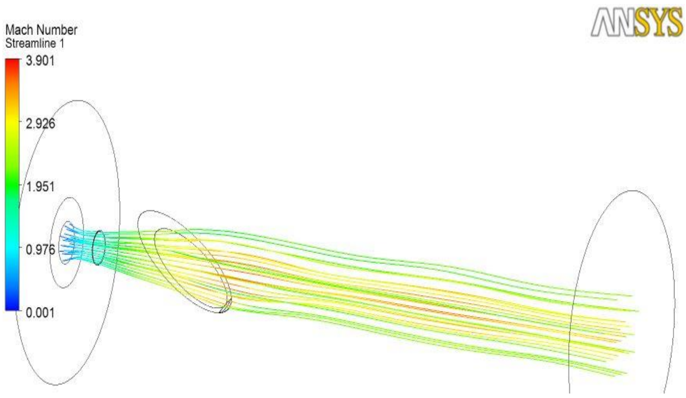

Fig. Mach number contour of Scarfed Nozzle at M= 2

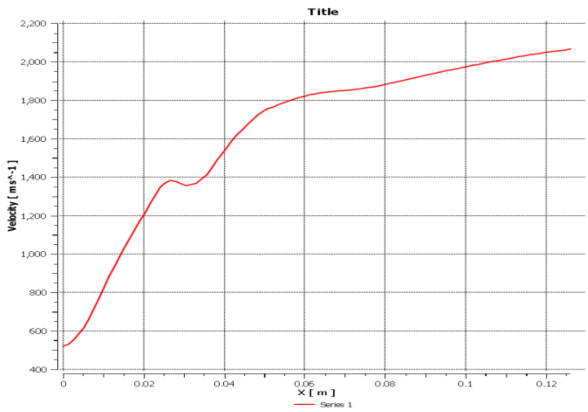

Fig. Velocity plot along axis of Scarfed Nozzle at M=2

Plots in figures above shows the internal variation of flow properties temperature,pressure and velocity respectively along the axis of nozzle. Even though nozzle has been scarfed the flow has been fully developed as same as full length nozzle. Apart from

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

weight optimization scarfed nozzle has been efficient as that of normal nozzle.

M=3

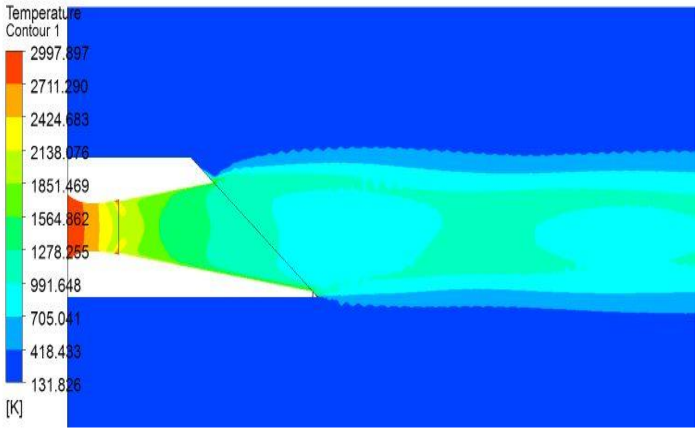

Fig. Temperature contour of Scarfed Nozzle at M= 3

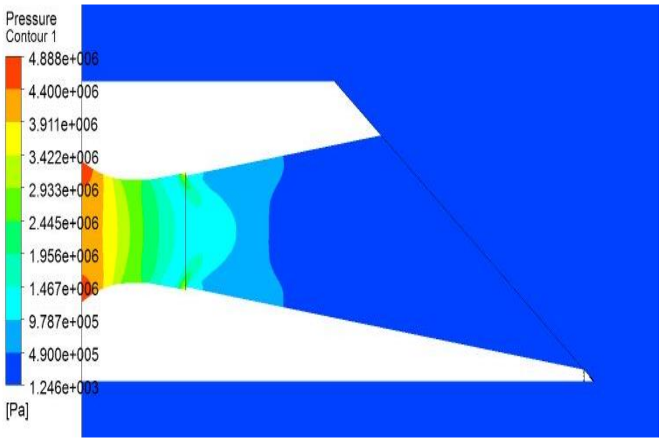

Fig. Pressure contour of Scarfed Nozzle at M= 3

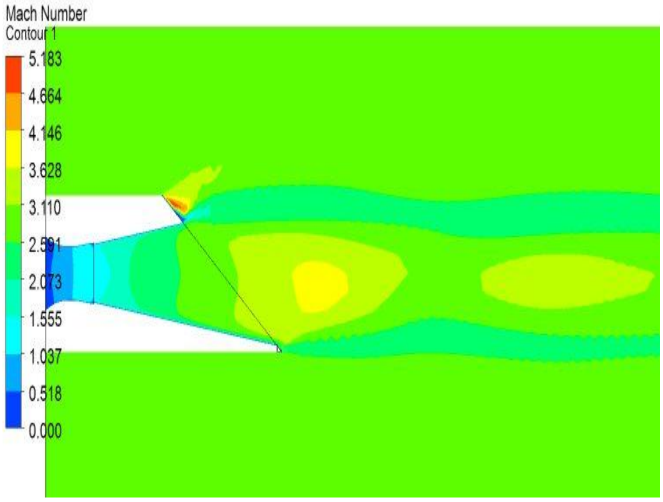

Fig. Mach number contour of Scarfed Nozzle at M= 3

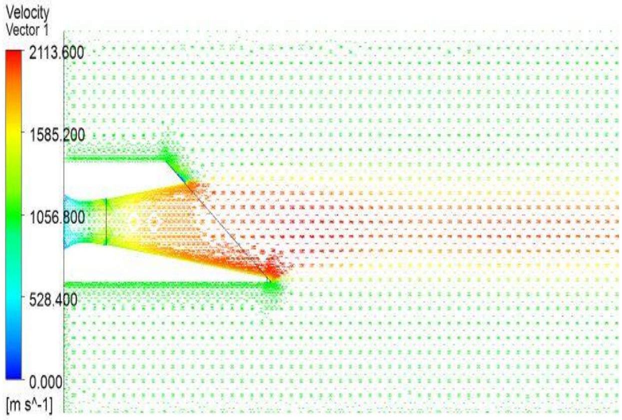

Fig. Velocity vectors of Scarfed Nozzle at M= 3

Computational analysis is performed on the Scarfed nozzle of scarf angle 52˚ to get the contour plots of pressure, velocity, Mach number, velocity etc.The contour plots of pressure shows a variation from 49 bar at inlet of the nozzle to free steam condition at the outlet.

The temperature also drops to a very low value of 900 K from 3000 K at inlet as observed from figure 7.37 at the immediate exitThe drop in pressure and temperature adds to the increase in velocity at the exit. Velocity varies from 65.41 m/s to 1718.148 m/s at the immediate exit which provides sufficient thrust to the rocket. Contours of Mach number are shown in figure which varies from 3.189 to 0.126 which implies design is correct. It is observed that at the throat the mach number varies from 0.8 to 1.00

CONCLUSIONS & FUTRE SCOPE OF WORK

Computational analysis was conducted to compare the performance of a conical nozzle of half angle 8˚ and scarfed nozzle with scarf angle 52˚ of same size 123.80 mm length and diameter of 24.02 mm with same conditions of inlet pressure 50 bar and temperature 3000 K.

SST turbulence model can able to capture the exhaust plume characteristics and shock pattern simialr to that of experimental results very effectively. Decreasing the length of the nozzle extends the length of shock pattern in exhaust gases leading to increase in boattail drag.

The asymmetry of the plume profile is low outgoing from the tested scarfed nozzle configuration. The corresponding change of the thrust angle is about θ_T = 3˚-5˚. the velocity vectors contour plots shows how the vectors in case of scarfed are inclined when compared to straight vector lines in case of conical nozzle. The mach number of the conical nozzle is more when compared with the scarfed nozzle. The scarfed nozzle is preferred due to the configuration constraints.

By this external and internal flow analysis over different nozzle at different conditions the effectiveness of nozzle has been analysed. Out of these nozzle Scarfed nozzle has shown same performance to that of full length nozzle but with weight optimization, thermal protection and exhaust plume characteristics scarfed nozzle is better than Conical nozzle.

Generally rocket which travels at different altitude conditions(pressure conditions) leading to different exit mach the exhaust plume shock pattern varies effecting thrust vectoring operation.

With these results it is concluded that CFD is an efficient and economical research tool to predict the physics of flow through nozzle and to characterize exhaust plume giving a direction for the development efficient nozzle.

SCOPE FOR FUTURE WORK

In the next stage the computational analysis of elliptical can be tabulated. Then the comparison of experimental analysis results with the computational analysis results of all the three configurations (Conical nozzle, scarfed nozzle and elliptical nozzle).

Rockets at high-altitude are subject to a fluid dynamics phenomenon known as Plume-Induced FlowSeparation (PIFS).The distance between the end of the vehicle and the separation point of the surface is denoted as the PIFSdistance. Accurate prediction of the PIFS distance is critical to the design of the thermal protection system.

The ability to change the sonic boom signature of rockets or missiles during launch by manipulating the shape of the exhaust plumes from the engines has to be studied computational fluid dynamics (CFD) codes, several exhaust nozzles can be designed with varied internal geometry in order to produce different shapes of exhaust plumes.

This analysis can be extnded for Afterburn of exhaust plume. An additional set of reactions can occur in the viscousmixing layer, which is known as afterburning. The occurrenceof afterburning, however, depends on the altitude. Thechemical reaction which leads to afterburning occurs onlywhen the oxygen concentration is high enough. Afterburningleads to higher temperatures in the mixing zone and thus, toinfrared emissions, which can be observed.

REFERENCES

1. Dash, S. M., Wolf, D. E., *Fully-Coupled Analysis of Jet Mixing Problems Part I: Shock-Capturing Model, SCIPVIS*, NASA Contractor Report 3761, 1984
2. Simmons, F. S., *Rocket Exhaust Plume Phenomenology*, The Aerospace Press, El Segundo, California, USA, 2000
3. Dao, P. D., Farley, R., Soletsky, P., Gelbwachs, J., *LIDAR Measurements of the Stratospheric Exhaust Plume of Launch Vehicles*, 35th Aerospace Sciences Meeting & Exhibit, AIAA 970526. Aerospace Sciences Meeting, Reno, NV, USA, January 6-9, 1997
4. Lohn, P. D., Wong, E. P., Smith Jr., T. W., *Rocket Exhaust Impact on Stratospheric Ozone*, TWR Space Electronic Group, 1999
5. Smith, L. D., Edwards, J. R., Pilson, D., *Summary of the Impact of Launch Vehicle Exhaust and Deorbiting Space and Meteorite Debris on Stratospheric Ozone*, TRW Space and electronics Group, 1999
6. Burke, M. L., Zittel, P. F., *Laboratory Generation of Free Chlorine HCl under*

Stratospheric Afterburning Conditions, Combustion and Flame, Vol. 112, No. 1-2, 1998

7. Brady, B. B., Martin, L. R., Lang, V. I., *Effects of Launch Vehicle Emissions in the Stratosphere*, Journal of Spacecraft and Rockets, Vol. 34 (6), pp. 774–779.
8. Denison, M. R.; Lamb, J. J.; Bjorndahl, E. Y.; Lohn, P. D.,*Solid Rocket Exhaust in the Stratosphere: Plume Diffusion and Chemical Reactions*, Journal of Spacecraft and Rocket, Vol. 31, No. 3, pp. 435– 442, 1994.
9. Rickey. J. Shyne, "Analysis and design of optimized truncated scarfed nozzles subject to external flow effects", NASALEWISRESERCHCENTRE, cleveland, ohio
10. Trinks.H, "Exhaust plumes of scarfed nozzles", Technical University of Harburg, Harburg, West Germany - AIAA 90-1733
11. Jay s. Lilly, "The application of scarfed nozzle for thrust vector adjustmen", - AIAA 90-2210
12. Jay s. Lilly' "Scarfed perfect nozzles for thrust vector adjustment in tactical strap-on boosters", - AIAA 90-1733.