CFD Analysis of Reverse Flow Multiple Nozzle

Rahul. S
M.Tech Computer Integrated Manufacturing Student
T.K.M. College of Engineering
Kollam, Kerala, India

Harikrishnan. R
Deputy Head, TDD
V.S.S.C
Thiruvanathapuram, India

Nadeera. M
Assistant Professor
T.K.M. College of Engineering
Kollam, India

***Abstract*— The Crew Escape System (CES) is a dedicated system to ensure crew safety in Human Spaceflight Programme and used to quickly separate the crew module from the rest of the rocket in case of emergency during the initial phase of flight. The CES primarily consists of a set of quick acting solid motors which operates in a pre-defined sequence to provide required acceleration to take the crew module along with the crew away from the launch vehicle. The solid motor used in CES are of nonconventional nature. One of these utilizes a differential multiple nozzle system with flow reversal to deliver differential thrust in the desired direction. A Three dimensional (3D) flow field simulation in a reverse flow multiple nozzle motor (RFMN) is carried to study the flow parameters and optimize the nozzle internal aerodynamic profile.**

Keywords—Crew escape module, reverse flow multiple nozzle, cfd

I. INTRODUCTION

The Crew Escape System(CES) is a dedicated system to ensure crew safety in Human Spaceflight Programme and will be called into action in case of exigencies at the launch pad or during the entire duration of the first stage (solid motor) operation of the launch vehicle. The CES primarily consists of a set of quick acting solid motors which operates in a pre-defined sequence to provide required acceleration to take the crew module along with the crew to a safe distance. The crew escape system consists of set of escape motors (solid motor), which is sized and fired such that crew module is pulled away from the launch pad if there is explosion or occurrence of any undesired event at the launch pad. Similarly if the flight is deviated far away from - the nominal trajectory or any contingency occurs during lift off, the escape motor pulls the crew module from the launch vehicle. For this a reverse flow multiple nozzle can be utilized. In this paper 3D flow simulation of Reverse flow multiple nozzle is carried out.

K.P. Singh, J. S. Mathur et.al [1] The role of computational fluid dynamics (CFD) in the design oflaunch vehicle, fighter aircraft, transport aircraft, and missiles in India is explained. 3-D Euler and Navier- Stokes solvers using state-of-the-art numerical techniques and physical models have been described. Applications of these indigenous software's for the prediction of various complex aerodynamic flows over a wide range of angle of attacks,Mach number, are presented. Emergence of CFD methods as an efficient tool for aerospace vehicle design is highlighted.

Advancements in numerical algorithms, computer hardware, and geometric pre-processing have enabled CFD to simulate the flow field around aerospace vehicles with all its geometrical complexities. This matured technology is not only explaining the complex flow physics, but also providing important design inputs for various aerospace vehicle including fighter and transport aircraft, launch vehicles and missiles. The simulation of these complex aero propulsive flows requires enormous computing time and memory. To meet the increasing demands from the designers (one run per day), all the computer codes are parallelised and a number of high performance computing platforms based on multi-node cluster computing have been established. CFD has emerged as an important tool in the design and analysis activities of various aerospace vehicles in India and contributing to a faster, accurate, and less expensive design process.

John Isaac and Suzanne Graham[2]Aerospace companies are under pressure to create the best performing products and design those products on schedule. In the design process CFD analysis plays an important role, by reducing the need for multiple time- and cost-consuming physical prototypes that must be manufactured and tested, redesigned if necessaryand problems discovered. In the development process prototypes will always play an important role, but reducing the time and money expended on them can help a company meet its business goals. It is also apparent that putting CFD in the hands of the design engineer as well as the CFD specialist opens the door to additional and efficient experimentation with design approaches. This results in a much more optimized product. Modern CFD software that enable mechanical engineers to use CFD as an integral part of their design process.

Pardhasaradhi Natta, V.Ranjith Kumar[3]The gases coming out of the combustion chamber is used to give direction to a nozzle. Nozzle is a tube having variable cross-sectional area tocontrol thedirection, mass, rate of flow, speed, shape, and/or the pressure of the exhaust stream that emerges from them, nozzles are generally used. To convert the chemical thermal energy generated in the combustion chamber into kinetic energy for this nozzle is used. The nozzle converts the high pressure,low velocity, high temperature gas in the combustion chamber into high velocity gas of lower pressure and low temperature.

The study is carried using software's like gambit for designing of the nozzle and fluent for analyzing the flows in the nozzle. To understand the air flows in a conical nozzle at different divergence degrees of angle using 2-D axis-symmetric models, which solves the governing equations by a control volume methodfor this numerical study has been conducted.

KunalPansari, S.A.K Jilani et.al[4] A nozzle is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enter) an enclosed chamber or pipe via orifice. The performance and flow characteristics of the convergent-divergent nozzle under various operating pressure ratio and with different nozzle profiles, also determine the location and strength of the normal shock in the divergent portion of the nozzlea numerical study has been carried out to analyze this. Various flow parameters across the normal shock had been obtained by using gas table. A de Laval nozzle is a tube that is pinched in the middle, making an hourglass shape. It is used to accelerate the flow of a gas passing through it to a supersonic speed. It is widely seen in some types of steam turbine and is an essential part of the modern rocket engine and supersonic jet engines. This concludes that the shock strength goes on increasing with decreasing operating pressure ratio and also the shock location move towards exit. Exit Mach number and Mach number ahead of the shock goes on increasing by decreasing the operating pressure ratio.

Maximilian Emans[5] The performance in terms of computing time of different parallel AMG algorithms that are applied within the context of industrial computational fluid dynamics (CFD) problems. An overview of the most important classes of algorithms described in literature, picked four fundamentally different algorithms and perform numerical experiments on up to 16 processors with two benchmarks representing an important class of CFD-problems

When applied as linear solvers within a SIMPLE or SIMPLE-like algorithm the AMG preconditioned conjugate gradient methods considered here perform typically only a few iterations. Since the parallel efficiency of Smoothed Aggregation AMG is satisfying, in particular for larger problems, and the contribution of the setup phase to the total computing time is large, we do not believe that a fully parallel, but more expensive setup will be appropriate to reduce the total computing time.

II. GOVERNING FLUID-FLOWEQUATIONS

Based onfundamental physical principlesthe equationsgoverning fluidmotionare:

Mass : Change of mass = 0 (1)
Momentum: Change of momentum = Force x Time (2)
Energy: Change of Energy = Work + Heat (3)

$$Mass\ balance:\ \frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \qquad (4)$$

Momentum balance equations (x,y,z)

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=\frac{\partial\sigma_{xx}}{\partial x}+\frac{\partial\tau_{xy}}{\partial y}+\frac{\partial\tau_{xz}}{\partial z}+\rho g_x \qquad (5)$$

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=\frac{\partial\tau_{yx}}{\partial x}+\frac{\partial\sigma_{yy}}{\partial y}+\frac{\partial\tau_{yz}}{\partial z}+\rho g_y \qquad (6)$$

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=\frac{\partial\tau_{zx}}{\partial x}+\frac{\partial\tau_{zy}}{\partial y}+\frac{\partial\sigma_{zz}}{\partial z}+\rho g_z \qquad (7)$$

$$\sigma_{xx}=-p+2\mu\frac{\partial u}{\partial x}-\frac{\partial\mu}{3}(\nabla.V) \qquad (8)$$

$$\sigma_{yy}=-p+2\mu\frac{\partial v}{\partial y}-\frac{2\mu}{3}(\nabla.V) \qquad (9)$$

$$\sigma_{zz}=-p+2\mu\frac{\partial w}{\partial z}-\frac{2\mu}{3}(\nabla.V) \qquad (10)$$

$$\tau_{yz}=\tau_{zy}=\mu\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right) \qquad (11)$$

$$\tau_{xy}=\tau_{yx}=\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right) \qquad (12)$$

$$\tau_{xz}=\tau_{zx}=\mu\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right) \qquad (13)$$

Energy balance Equation

$$\frac{\partial(\rho e)}{\partial t}+\frac{\partial(\rho uH)}{\partial x}+\frac{\partial(\rho vH)}{\partial y}+\frac{\partial(\rho wH)}{\partial z}=\nabla.(k\nabla T)+\mu\phi \qquad (14)$$

Øis known as viscous dissipation, Øfor Newtonian fluid in Rectangular Cartesian Co ordinate(x, y, z)

$$\phi=2\left[\left(\frac{\partial u}{\partial x}\right)^2+\left(\frac{\partial v}{\partial y}\right)^2+\left(\frac{\partial w}{\partial z}\right)^2\right]+\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)^2+\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)^2+\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)^2-\frac{2}{3}(\nabla.V)^2 \qquad (15)$$

III. ESCAPE SYSTEM FOR CREW MODULE

A Launch Escape System (LES) is a system connected to the crew module of a crewed spacecraft and used to quickly separate the crew module from the rest of the rocket in case of emergency. If there is an imminent threat to the crew, such as an impending explosion, then LES is designed to use in such situations.

The Crew Escape System is a dedicated system to ensure crew safety in Human Spaceflight Programme and will be called into action in case of exigencies at the launch pad or during the entire duration of the first stage (solid motor) operation of the launch vehicle. The CES primarily consists of a set of solid motors which operates in a pre-defined sequence to provide required acceleration to take the crew module along with the crew to a safe distance.

The crew escape system consists of set of escape motors (solid motor), which is sized and fired such that crew module is pulled away from the launch pad if there is explosion or occurrence of any undesired event at the launch pad. Similarly if the flight is deviated far away from -the nominal trajectory or any contingency occurs during lift off, the escape motor pulls the crew module from the launch vehicle.

IV. SIMULATION PROCEDURE

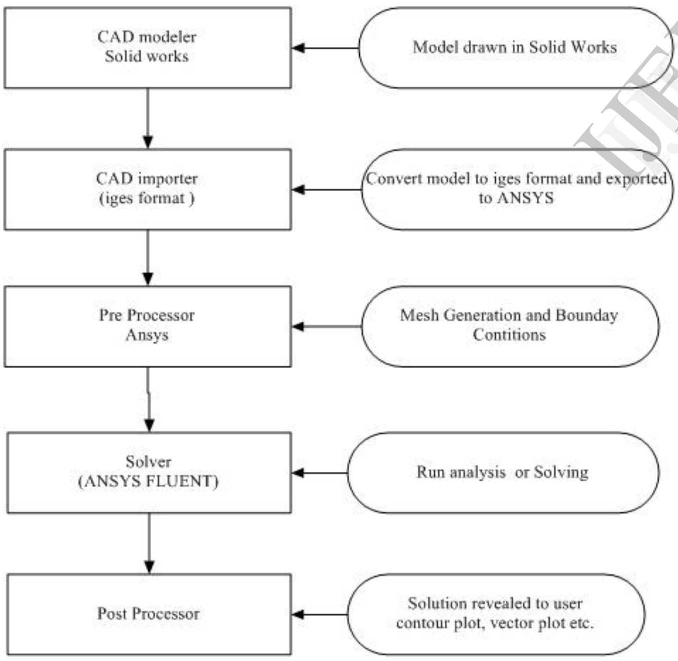

Fig.1 Methodology

Sl.No	Parameter	Nozzle motor
1.	No. of Nozzles	4 Nos.
2.	Cant angle (degrees)	31
3.	Max Pressure (Bars)	94.9
4.	Area ratio	6.2

Table 1nozzle specifications

Space	3D
Time	Steady
Viscous	Standard k-omega turbulence model
Heat Transfer	Enabled

Table 2`Domain specifications

V.RESULTS AND DISCUSSION

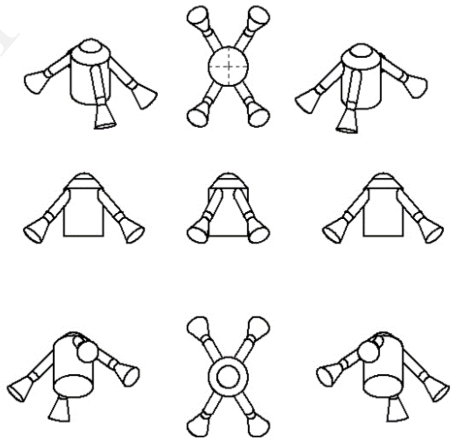

Figure 2 Reverse flow multiple nozzle geometry

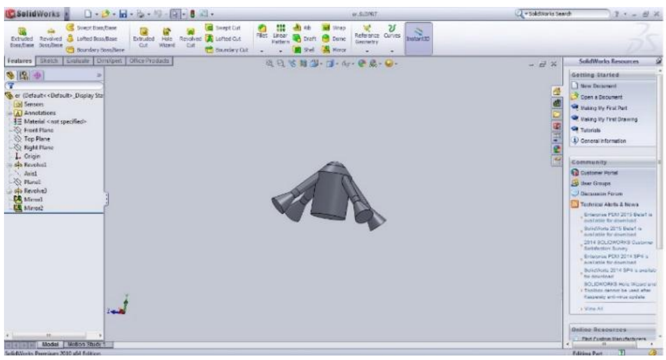

Figure 3 Model drawn in Solid works

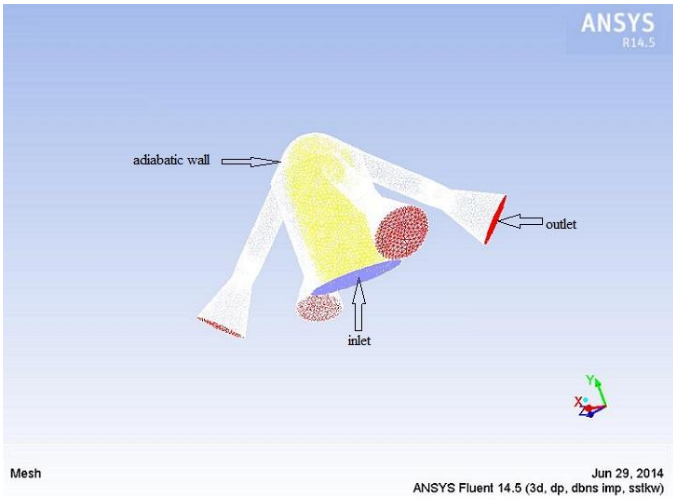

Figure 4 Computational domain with mesh

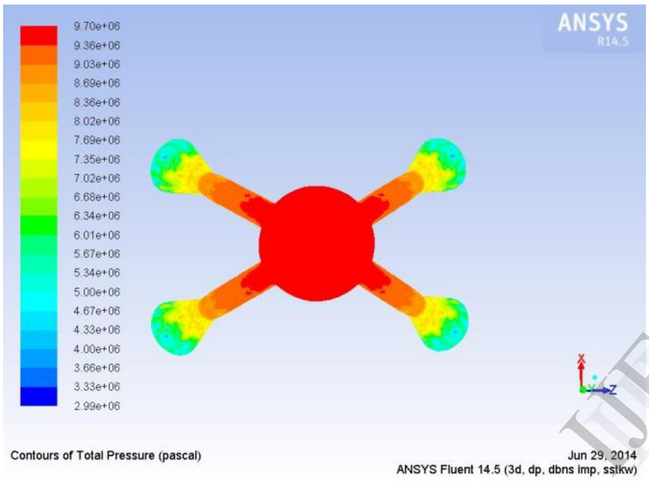

Figure 5 Contours of total pressure

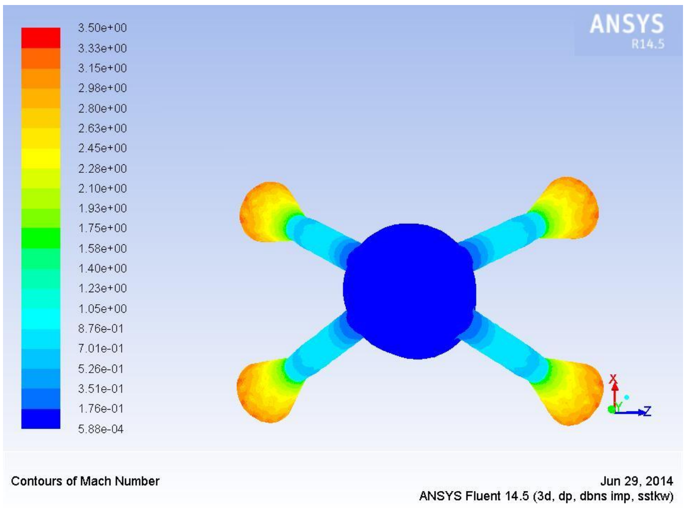

Figure 6 Contours of Mach number

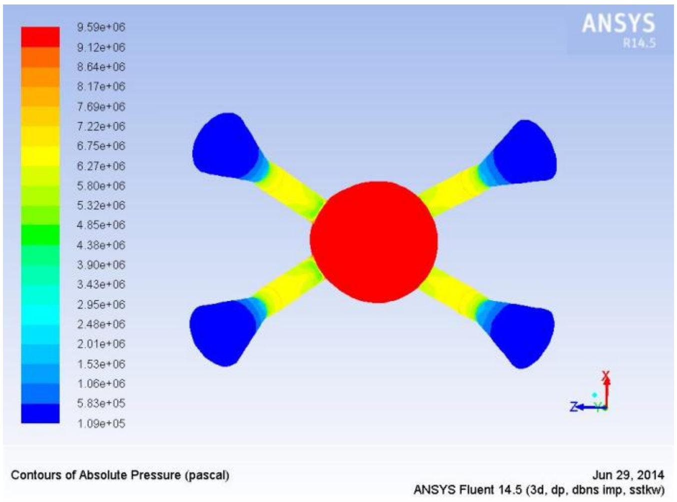

Figure7 Contours of absolute pressure

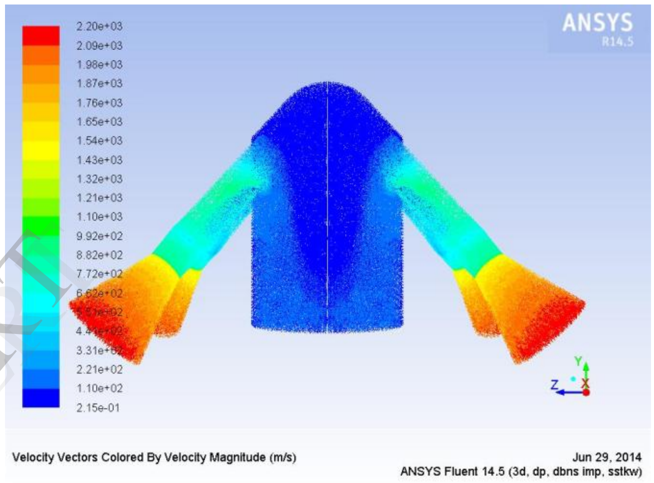

Figure 8 Velocity magnitude

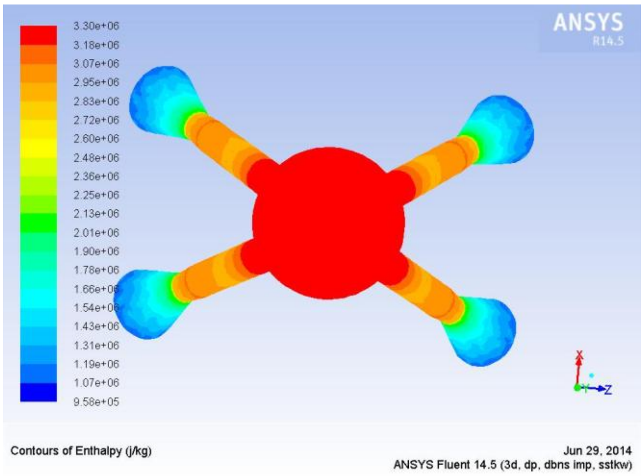

Figure 9 Contours of enthalpy

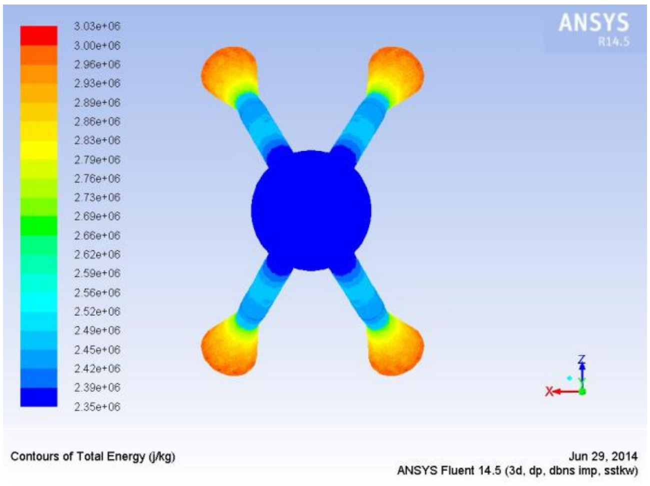

Figure 10 Contours of Total Energy

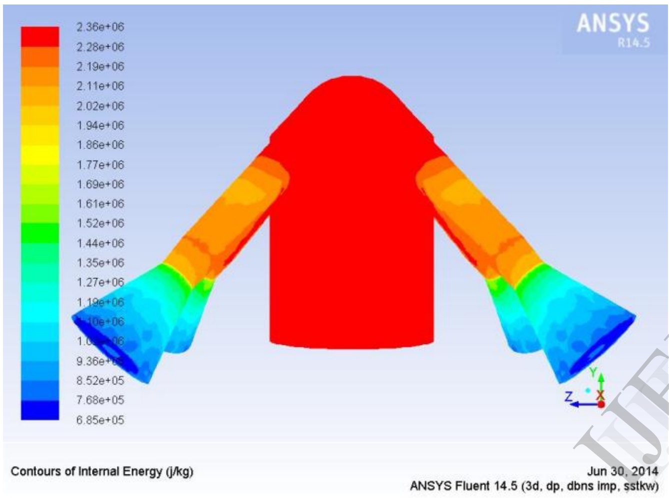

Figure 11 Contours of Internal Energy

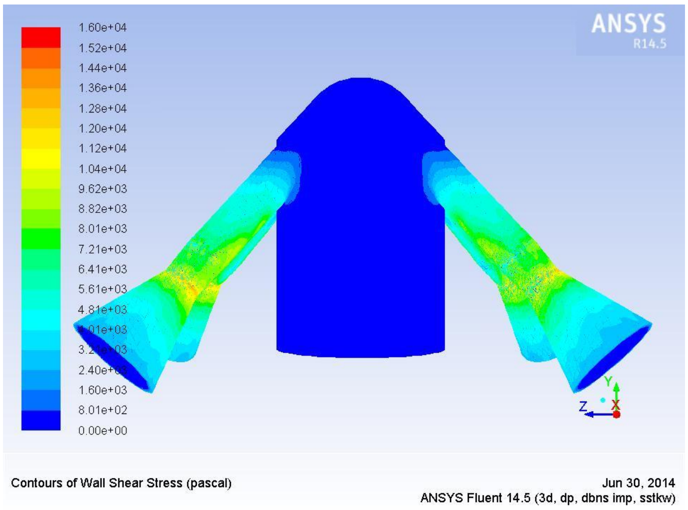

Figure 12 Contours of wall shear stress

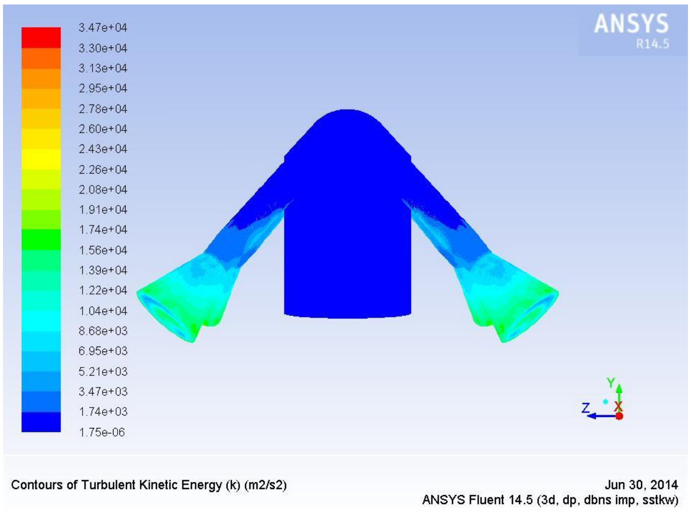

Figure 13 Contours of Turbulent Kinetic Energy

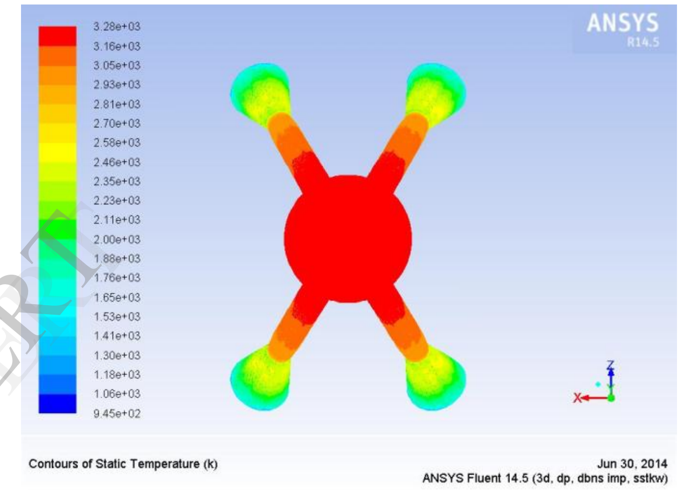

Figure 14 Contours of static temperature

The expected thrust deliverable for the RFN is worked out to be 14.9KN, this is 7% less than for a regular nozzle with identical pressure and simulated geometry. The reason for lower thrust can be attributed to effect of flow turning loss in the RFN.

From the Mach contours (Figure 3.6) it can be concluded that the nozzle dome is acting as a reservoir for the gases before it is getting accelerated at the nozzle entry. The Mach number experienced at the center of the dome is 5.88e-04. The following Figure 15 shows the vector distribution in nozzle entry region.

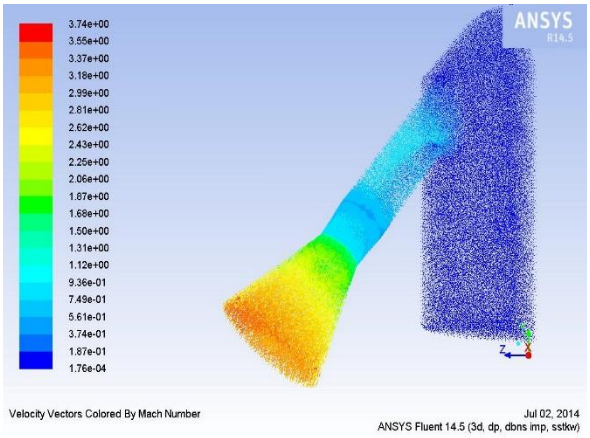

Figure 15 Velocity Vector coloured by Mach number

The exit Mach number is 3.5

At the nozzle exist the total pressure is 5e+06 MPa. This indicates that the nozzle is under expanded, thus there is a scope of further optimizing the internal contour for optimum expansion.

The temperature at the dome is 3280K and the temperature drops to 1060K at the nozzle exit.

VI.CONCLUSION

In the CFD perspective the scope exists in the improvement schemes for hot flow domain can be established only by carrying out, an experimental evaluation of the CFD results. In the development of RFMN also, a proper flow field understanding and visualization is as important as measurement of well shear force, Wall pressure and temperature etc. These have to be a proper mechanism for the aerodynamic characterization of complicated flow field in a less expensive manner.

The establishment of a nozzle testing laboratory (NTL) is an innovative initialize towards the aerodynamics characterization of hot flow domains. The NTL offers the CFD results validations and less expensive experimentations to arrive at optimum aerodynamic profile. Also it possess advantages in generating data in the area where CFD also fails

In CFD simulations it is impossible to bring nonlinearity in temperature for the associated fluids. The presence of separate flow domains mixture of different fluids and incompressible method also beyond the scope of CFD .Modelling of the surface conditions of the surface conditions, surface roughness, absorbing media etc. also extremely difficult.With NTL into picture such limitations in under steady flow and aero characterization of hot flow domains can be overcome.

REFERENCES

[1] K.P. Singh, J. S. Mathur et.al, 2010," Computational Fluid Dynamics in Aerospace Industry in India", Defence Science Journal, Vol. 60, No. 6, November, pp. 639-652.

[2] John Isaac and Suzanne Graham, 2013," CFD in the Aerospace and AeronauticsIndustries",J Aeronaut Aerospace Eng (JAAE), volume2, issue3.

[3] Pardhasaradhi Natta, V.Ranjith Kumar, 2012,"Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd)",International Journal of Engineering Research and Applications (IJERA),Vol. 2, Issue 5, September- October, pp.1226-1235.

[4] Kunal Pansari, S.A.K Jilani et.al, 2013,"Analysis of the Performance and Flow Characteristics of Convergent Divergent (C-D) Nozzle, International Journal of Advances in Engineering & Technology, Vol. 6, Issue 3, pp. 1313-1318.

[5] Maximilian Emans, 2010,"Performance of parallel AMG-preconditioners in CFD-codes for weaklycompressible flows", science direct journal of Parallel Computing, volume 36, pp 26–338.