CFD Analysis of a Rocket Nozzle with Four Inlets at Mach 2.1

K.M.Pandey, *Member, IACSIT* and S.K.Yadav

Abstract—**In this paper CFD analysis of pressure and temperature for a rocket nozzle with four inlets at Mach 2.1 is analyzed with the help of fluent software. when the fuel and air enter in the combustion chamber according to the x and y plot, it is burning due to high velocity and temperature and then temperature increases rapidly in combustion chamber and convergent part of the nozzle and after that temperature decreases in the exit part of the nozzle. it is concluded in this paper that Four inlet rocket nozzle is having better performance than single inlet and two inlet as seen from the previous research work done.**

Index Terms- **Four inlets, Mach number, Rocket nozzle, Total temperature, Total pressure, Static temperature.**

I. INTRODUCTION

The world today relies heavily on combustion of fossil fuels for its energy needs, as the major portion of energy used worldwide is contributed by combustion. Hence the scope of the combustion is very wide. Even in day to day life, we use combustion appliances directly or indirectly. The processes involved in combustion are governed by several different phenomena. This includes transport of mass, momentum and energy. We know that the driving forces such as concentration, pressure and temperature gradient can cause species transport, momentum transport and energy transport respectively. Boris M. Kiforenko, Zoya V. Pasechnik and IgorYu. Vasil'ev[1] worked on "Comparison of the rocket engines efficiency in the case of low thrust orbit-to-orbit transfers" and their findings are the following: The main task of this paper is to compare two types of low thrust rocket engines: constant thrust vs. variable-thrust engines. They are concerned with efficiency, where efficiency is evaluated in the case of the orbit-to-orbit transfer with maximum payload mass in the central Newtonian gravity field. The launch mass of the space vehicle is supposed to be fixed. The traditional solution is the decomposition of the problem into parametric and dynamical parts. The corresponding variation problems differ for two rocket thruster types under consideration. They proposed change of variables, which makes it possible to reduce averaged equations of optimal motion of a spacecraft with the mentioned engines to the unified form. Using this unified form comparison of the performance of constant- and variable-thrust engines is conducted.

Zhigang Feng and QiWang[2] worked on "Research on health evaluation system of liquid-propellant rocket engine ground-testing bed based on fuzzy theory" and their findings are the following: In this paper, the theory based on multi-sensor information fusion is studied, which is used to evaluate the health condition of liquid-propellant rocket engine ground-testing bed. The concept of health degree is defined. It is used as a quantitative index for evaluating the health condition of the ground-testing bed. In order to evaluate the health condition of the ground-testing bed on different levels, health degrees of a single parameter, of a sub-system and of a system are defined. They accordingly measure the health conditions of single parameter, sub-system and system of the ground-testing bed. The method of fuzzy data fusion is used to calculate the health degree. In this method the weight of each monitoring sensor is calculated by analytic hierarchy process (AHP), then, the multi-sensor data are fused by the fuzzy comprehensive evaluation method, next, the sequence data are fused by two-grade index evaluation method, finally, the health degree is calculated by defuzzification method. Based on these, the health condition evaluation system of ground-testing bed is set up, which can evaluate the health condition of the ground-testing bed properly and quantitatively. At last, the application of the health evaluation system in fault detection and health evaluation of the ground-testing bed are discussed.

Vadim Zakirov and Vladimir Pavshook[3] worked on "Russian Nuclear Rocket Engine Design for Mars Exploration" and their findings are described here. This paper is to promote investigation into the nuclear rocket engine (NRE) propulsion option that is considered as a key technology for manned Mars exploration. Russian NRE developed since the 1950s in the former Soviet Union to a full-scale prototype by the 1990s is viewed as advantageous and the most suitable starting point concept for manned Mars mission application study. The main features of Russian heterogeneous core NRE design are described and the most valuable experimental performance results are summarized. These results have demonstrated the significant specific impulse performance advantage of the NRE over conventional liquid rocket engine (LRE) propulsion technologies. Based on past experience, the recent developments in the field of high-temperature nuclear fuels, and the latest conceptual studies, the developed NRE concept is suggested to be upgraded to the nuclear power and propulsion system (NPPS), more suitable for future manned Mars missions. Although the NRE still needs development for space application, the problems are solvable with additional effort and funding. Johannes Lux, Dmitry Suslov and Oskar Haidn[4] worked on "porous liquid propellant rocket engine injectors" and their findings are as followings. A novel injection concept for cryogenic liquid propellant rocket engines has been tested and verified successfully using an optically accessible combustion chamber. The injector

Manuscript Received on 3 December, 2010, accepted on 14 December ,2010.

K.M.Pandey, Member *IACSIT*, Department of Mechanical Engineering, N.I.T Silchar, Assam, India(email: kmpandey2001@yahoo.com).

S.K.Yadav, Asst. Professor, Department of Mechanical Engineering, Gurgaon Engineering College, Gurgaon.

consists of a porous faceplate made from sinter metal and five LOX posts arranged in a classical parallel showerhead configuration. While liquid oxygen is injected at similar velocities known from typical coaxial injector elements, the fuel enters the combustion chamber at much lower velocities. The flame stabilization for both LOX/H2 and LOX/CH4 propellant combination has been investigated at sub, near, and supercritical pressures. Alexander Potapov, YuriyShtefan and Elena Lichman[5] worked on “Research of material for un cooled nozzle extensions of liquid rocket engines” and their findings are the following. One of the ways to increase the performance of liquid rocket engines (LRE) is to use nozzle extensions (NE) made of carbon–carbon composites (CCC). The following works were performed during development of the NE: a fabrication method was verified, physio mechanical properties of the material were defined, effects of high temperature on a CCC oxidizing resistance were studied, a method to protect CCC against oxidation was developed. Prototype NE were manufactured and tested in a LRE test chamber. The test firings demonstrated good performance characteristics of CCC. Lacaze, B. Cuenot, T. Poinsot and M. Oschwald[6] worked on “Large eddy simulation of laser ignition and compressible reacting flow in a rocket-like configuration” and their findings are described here. The control of ignition in a rocket engine is a critical problem for combustion chamber design. Delayed ignition may lead to high-amplitude pressure fluctuations that can damage the burner (strong ignition), whereas early ignition may fail. This paper describes a numerical study of a strong ignition sequence observed in a laboratory-scale single-injector rocket chamber ignited by a laser and fueled with gaseous oxygen and hydrogen. OH-emission images, Schlieren pictures, and pressure measurements make it possible to follow the flame propagation experimentally. The present large eddy simulation (LES) approach includes shock treatment, a six species–seven reaction chemical scheme for H2–O2, and a model for the energy deposition by a laser. Flame/turbulence interaction is modeled with the thickened flame concept. LES is used to compute both the filling phase (during which the gaseous hydrogen and oxygen mix) and the ignition phase. The flame location and structure, as well as the temporal evolution of the chamber pressure obtained numerically, are in good agreement with the experiment. The use of complex chemistry in the computation also allows the comparison of LES data with experimental OH-images and shows that the sensitivity of the CCD camera used to record the spontaneous emission of the OH∗ radical is not high enough to properly locate the flame front in rich regions. The combined experimental and numerical results lead to a more detailed analysis of the ignition processes and its coupling with flow rate oscillations in the H2 and O2 feeding lines. ValeriyI.Timoshenkoa,, IgorS.Belotserkovetsa and VjacheslavP.Gusininb[7] worked on “Problems of providing completeness of the methane-containing block-jet combustion in a rocket-ramjet engine’s combustion chamber” and their findings are the following. Some problems of methane-containing hydrocarbon fuel combustion are discussed. It seems that reduction of methane burnout zone length is one from main problems of designing new type engine. It is very important at the creation of combustion chambers of a rocket-ramjet engine for prospective space shuttle launch vehicles. M. Masquelet, S. Menona and Y. Jinb, R. Friedrich[8] worked on “Simulation of unsteady combustion in a LOX-GH2 fueled rocket engine” and their findings are the following. This paper presents results from an investigation of unsteady combustion inside a small-scale, multi injector liquid rocket engine. A time-accurate approach in an ax symmetric geometry is employed to capture the unsteady flow features, as well as the unsteady heat transfer to the walls of the combustion chamber. Both thermally perfect gas (TPG) and real gas (RG) formulations are evaluated for this LOX-GH2 system. The Peng–Robinson cubic equation of state (EoS) is used to account for real gas effects associated with the injection of oxygen. Realistic transport properties are computed but simplified chemistry is used in order to achieve a reasonable turnaround time. Results show the importance of the unsteady dynamics of the flow, especially the interaction between the different injectors. The RG EoS, despite a limited zone of influence, is shown to govern the overall chamber behavior. The sensitivity of the results to changes in the system parameters is studied and some general trends are discussed. Although several features of the simulations agree well with past experimental observations, prediction of heat flux using a simplified flux boundary condition is not completely satisfactory. Reasons for this discrepancy are discussed in the context of the current ax symmetric approach.

David R. Greatrix[9] worked on “Regression rate estimation for standard-flow hybrid rocket engines” and their main results are cited here. The present effort is towards predicting with some accuracy hybrid rocket engine fuel regression rates under standard flow conditions. A convective heat feedback modeling approach is applied in tying the mass-flux-dependent heat flux directed into the regressing fuel surface, to the subsequent solid fuel grain regression rate. Factors such as transpiration, hydraulic port diameter, and effective fuel surface roughness are incorporated into the phenomenological surface regression rate model. A number of comparisons between the model’s predicted results and corresponding experimental data are made, in illustrating the efficacy of the present approach for a classical head-end-injection engine. Where substantial differences between theory and experiment exist, this might be due to one of several identifiable factors related to non-standard flow, such as the presence of radiant heating, swirl or flow impingement in or at the boundaries of the experimental core flow. K.M.Pandey and S.K.Yadav[10] worked on “CFD Analysis of a Rocket Nozzle with Two Inlets at Mach 2.1” and their findings are, pressure and temperature for a rocket nozzle with two inlets at Mach 2.1 is analyzed with the help of fluent software. When the fuel and air enter in the combustion chamber according to the x and y plot, it is burning due to high velocity and temperature and then temperature increases rapidly in combustion chamber and convergent part of the nozzle and after that temperature decreases in the exit part of the nozzle. It is concluded in this paper that two inlet rocket nozzle is having better performance than single inlet.

II. Objective of Present work

The objective of the present work is to simulate supersonic flow through rocket nozzle with combustion chamber to precisely understand the flow dynamics and variation of flow properties in combustion chamber with the nozzle. This simulation is carried out using fluent and gambit software. Gambit is used for generating the required mesh and simulation done using fluent.

III. Numerical Methodology

A numerical method adopted to approximate the governing equations, along with the relevant boundary conditions, by a system of linear algebraic equations is known as a discretization method. Thus, a problem involving calculus is transformed into an algebraic problem which can then be solved on a computer by using a solution methodology. A discretization technique and a solution methodology constitute the numerical methodology used to solve a heat transfer and fluid flow problem. There are many discretization methods, but the most commonly used are the Finite difference method (FDM), the Finite volume method (FVM) and the Finite element method (FEM). During the early days of Computational fluid dynamics (CFD) finite-difference methods were the most popular. They are algorithmically simple, efficient, and accurate. However, they are best used on uniform grids and hence on regular computational domains. With advances of CFD, and its application to industrial problems, there is a need for methods for computing flows in complex geometries. To adapt the finite difference method to such geometries, we can map the complex domain into simple domains, either globally or locally, and solve the equations there. However, such transformation makes the governing equations take quite complicated forms and may lead to a loss of computational efficiency and accuracy. Alternatively, one can use schemes based on the finite volume methods directly on the physical domain (i.e. without transformation). Finite volume methods are essentially a generalization of the finite-difference method, but use the integral form of the governing equations of flow rather than their differential form. This gives greater flexibility in handling complex domains, as the finite volumes need not be regular. The FLUENT code, which is used to simulate the flow field is based on the finite volume discrtization scheme and is one of the best application software for this purpose.

IV. Computational Model

This problem analyses the flow analysis of air and combustible gaseous mixture along the inlet passage and exhaust passage of a turbojet engine. A divergent type of inlet is taken and the exhaust nozzle is convergent type. The mole fractions of each of the species are also shown in the analysis.

Enable physical models, define material properties

1. Define the domain as two dimensional, and keep the default (segregated) solver.
2. Enable the k-epsilon turbulence model.
3. Enable heat transfer by activating the energy equation.
4. Enable chemical species transport and reaction.
5. Initialize the field variable.
6. Turn on residual plotting during the calculation.

V. Geometry and Grid Arrangement

A 2D axi- symmetric computational domain was considered, the initial design parameters for De Laval nozzle with combustion chamber for Mach number 2.1 with combustion chamber.

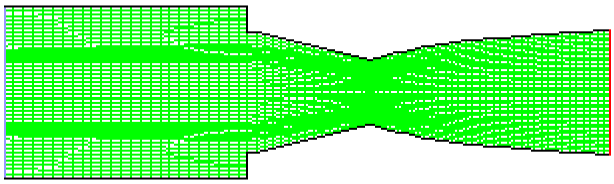


Figure-1Grid arrangement for axi-symmetric Rocket nozzle with combustion chamber

A. *Problem Description*

The liquid fuel combustion system considered here is depicted in Figure 1. A liquid spray of pentane fuel enters a 2D combustion chamber in which air is flowing at 650 K and 100 m/s. The combustion chamber walls are held at a constant temperature of 1200 K. The Reynolds number, based on inlet conditions, is roughly 100,000 and the flow is turbulent. As the pentane Evaporates, it enters the gas phase and reacts. The combustion is modelled using the mixture-fraction approach, with the equilibrium mixture consisting of 11 chemical species ($C5H12$, $CH4$,CO, $CO2$, $H2$, $H2O(g)$, $H2O(l)$, $O2$, OH, C(s) and $N2$).The spray is assumed to consist of 100-micron diameter liquid droplets injected at 300 K over a filled spray half-angle of 30 degrees on the duct centreline. The mass flow rate of liquid fuel is 0.004 kg/s, corresponding to very fuel-lean conditions in the flow.

VI. Results and Discussion

This CFD numerical experiment allows us to study in details the physical difference between the laminar and turbulent flows, all other parameter being equal in a fashion impossible to obtain in an actual levorotatory experiment. A numerical study has been conducted to understand the gas flow in a nozzles using 2D continuum axi symmetric model, which solves the properties of combustion by the control volume method. The numerical model was validated with exiting experimental data employing slip and no-slip boundary condition at the wall. The numerical results showed good agreement with experimental data on exit.

A. *Rocket Nozzle with Combustion Chamber*

1) *Total Pressure*

The maximum total pressure in the combustion chamber is 1.17e+05, and the average total pressure in the combustion chamber is 1.10e+05, the average total pressure in the conversion portion of the nozzle is 1.40e+05,while the average total pressure in combustion chamber ,pressure increase in combustion chamber and after that pressure goes on decrease in the convergent portion and at the throat total pressure is cover the negative value, due to supersonic nozzle total pressure in the convergent part is less and velocity increase in this portion. You can easily visualize the figure 2 that. There is decrease in stagnation pressure near the nozzle wall due to viscous effect.

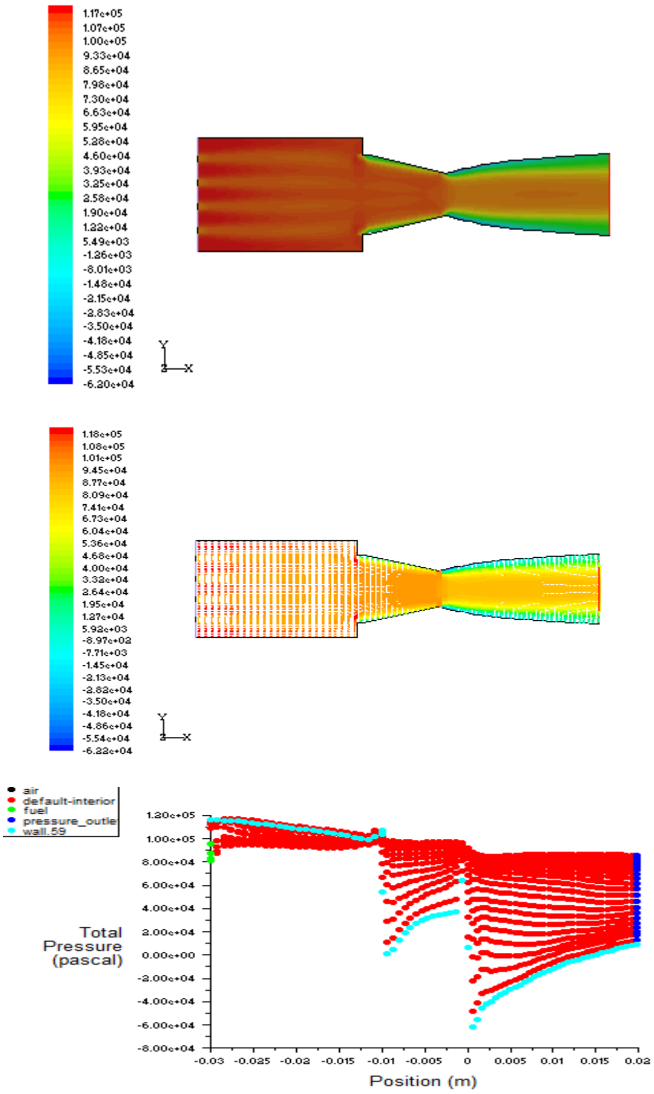


Figure-2 Total Pressure

2) Total Temperature

The average temperature in combustion chamber is 2.54e+03, and maximum temperature is 3.67e+03, the total temperature decrease in the divergent part of the nozzle compared the combustion chamber and convergent part of the nozzle. When the fuel and air is enter in the combustion chamber according to the x and y plot, its burn due to high velocity and temperature and then temperature increase rapidly in combustion chamber and convergent part of the nozzle and after that temperature decrease in the exit part of the nozzle. A maximum of 3.67e+03 is attained and beyond which the temperature steadily decreases.

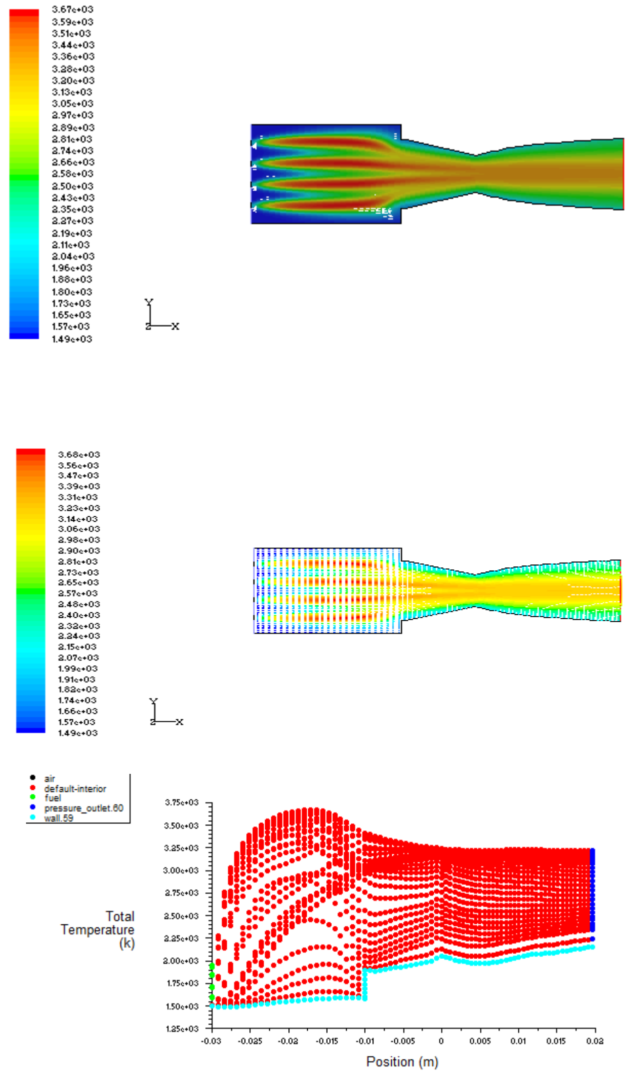
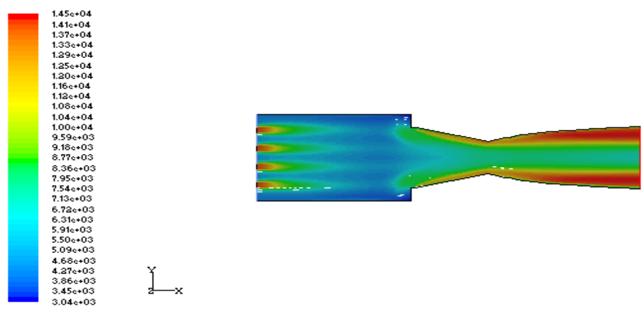
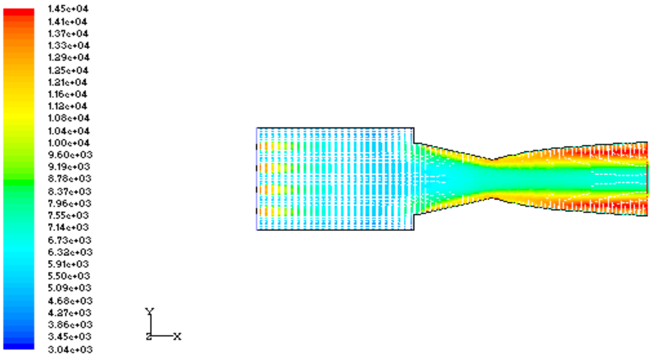


Figure – 3 Total Temperatures

3) Turbulence Intensity

The nozzle is designed for stream line flow and hence the intensity has high of turbulence is less inside the combustion chamber compared to divergent part of the nozzle. The turbulence intensity has high value about 1.45e+04 at fuel inlet and nozzle exit. Further downstream as flow gets stabilizes, the turbulence intensity also reduces. Turbulence intensity at the fuel inlet is 1.45e+04. A maximum of 1.45e+04(%) is attained and beyond which the turbulent intensity steadily decreases.





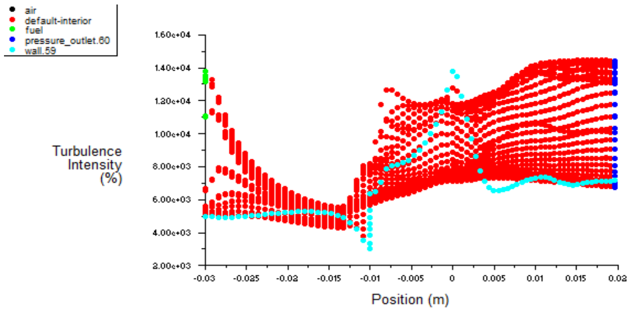
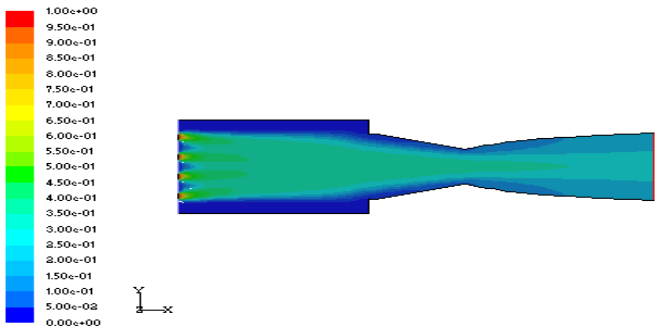


Figure- 4 Turbulence Intensity

4) *Mass fraction of C_5H_{12}*

The maximum mass fraction of pentane is 1.00e-01 at the fuel inlet is attained beyond which the mass fraction steadily decrease, near to the wall mass fraction of pentane is zero. When the chemical reaction occurs, the bond within molecules of the reactance is broken, and electrons rearrange to form product. in combustion reaction rapid oxidation of combustible element of the fuel results in energy release as combustion product are formed .When the fuel enter in the combustion chamber is one .while as well as fuel enter in combustion chamber it cover higher value 1and after that it goes on decrease in straight throw nozzle, and it is less in the part of the nozzle, near to the wall of combustion chamber is zero. When the using four jet for same amount of air and fuel the value increase and decreases.



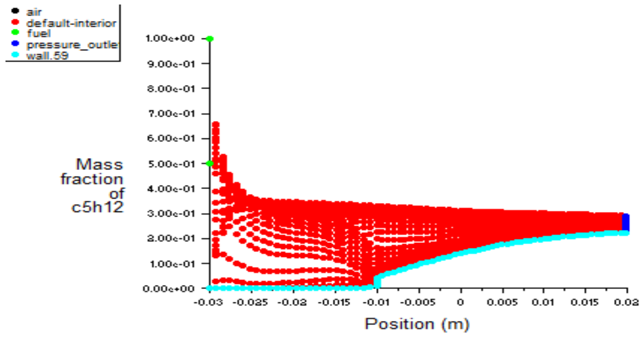
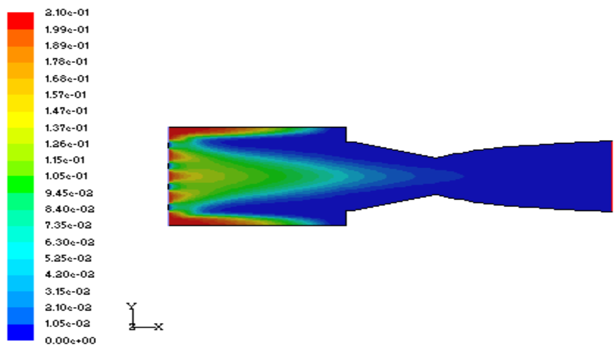


Figure-5 Mass fraction ofC_5H_{12}

5) *Mass Fraction of O2*

For the same value of air and fuel if increase the fuel inlet the amount of air is decrease compared to fuel amount. When the air is enter in the combustion chamber and burned with fuel , then mass fraction has high value 2.10e-01 in the combustion chamber and minimum value2.10e-02,and goes on decreasing in part of the nozzle, while near to the nozzle wall mass fraction of oxygen has zero. Due to the stream line flow mass fraction of O_2 flow straight in the nozzle. According to x and y Plot mass fraction value of oxygen increase in combustion chamber while it enters in the divergent part of the nozzle.



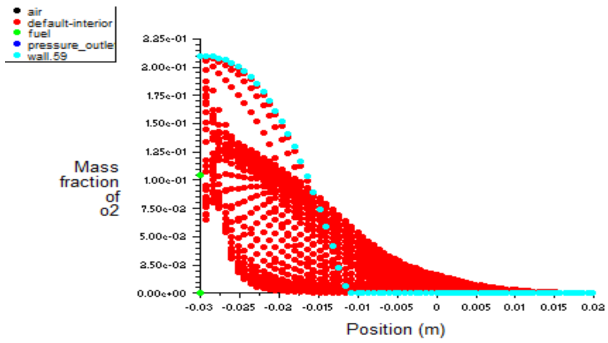


Figure-6 Mass fraction of O_2

6) *Mass fraction of Co2*

For the same value of air and fuel if increase the fuel inlet the amount of air is decrease compared to fuel amount. When the air is enter in the combustion chamber and burned with fuel , then mass fraction has high value 1.69e-01 in the combustion chamber and minimum value.0791,and goes on decreasing in part of the nozzle, while near to the nozzle wall mass fraction co_2 has zero. Due to the stream line flow mass fraction of co_2 flow straight in the nozzle. According to x and y Plot mass fraction value of oxygen increase in combustion chamber while it enters in the divergent part of the nozzle.

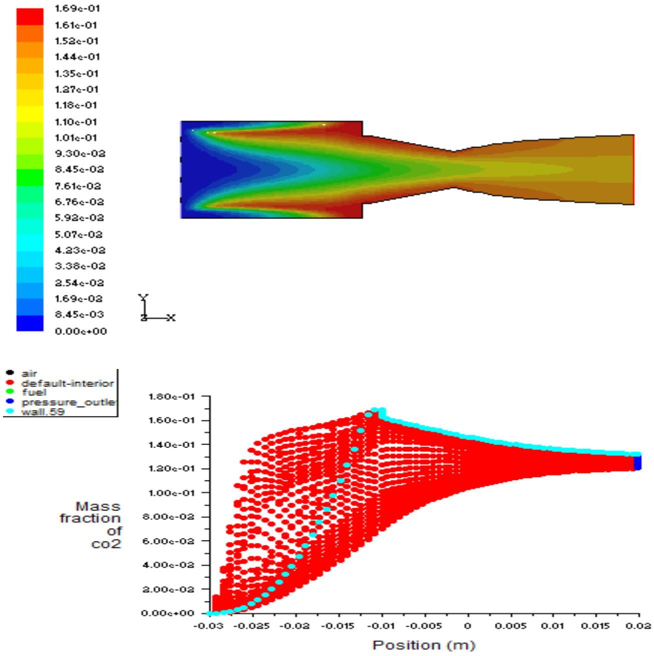


Figure-7 Mass fraction of Co_2

7) Mass fraction of H_2o

In combustion reaction rapid oxidation of combustible element of the fuel results in energy release as combustion products formed. The three major combustible chemical elements in most common fuel are carbon, hydrogen and sulphur ,combustion is complete when all the carbon present in the fuel is burned, all hydrogen burned in to water, and other combustible element are fully oxidized. Here the carbon dioxide is has the higher vale 8.30-02 and minimum value is zero. In the combustion chamber Carbon dioxide is zero means fuel is completely burned. Carbon dioxide is maximum at the exit part of the nozzle.

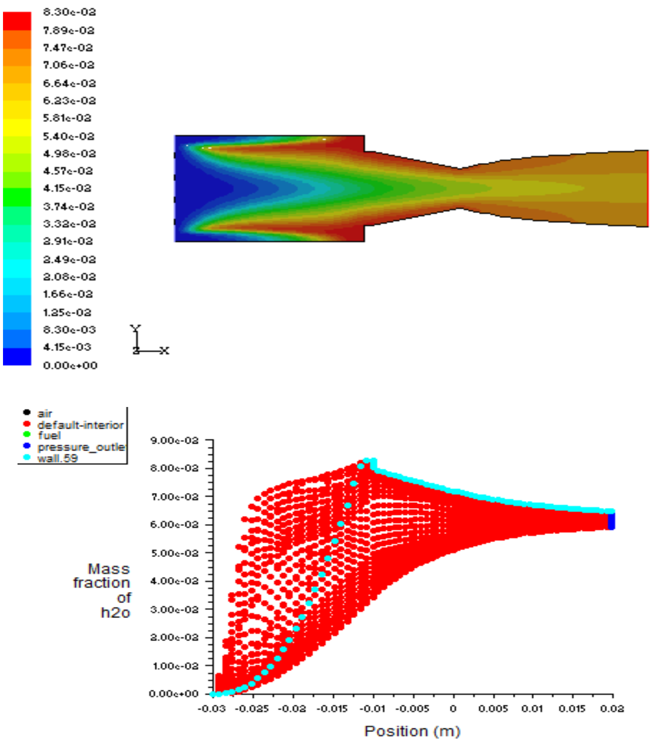


Figure-8 Mass fraction of H_2o

8) Mass fraction of N2

Mass fraction near to the combustion chamber wall has higher value is 0.79, and average valve is 0.474and minimum value near to the fuel inlet, while as long as air cover the distance in the combustion chamber the value goes on decrease.

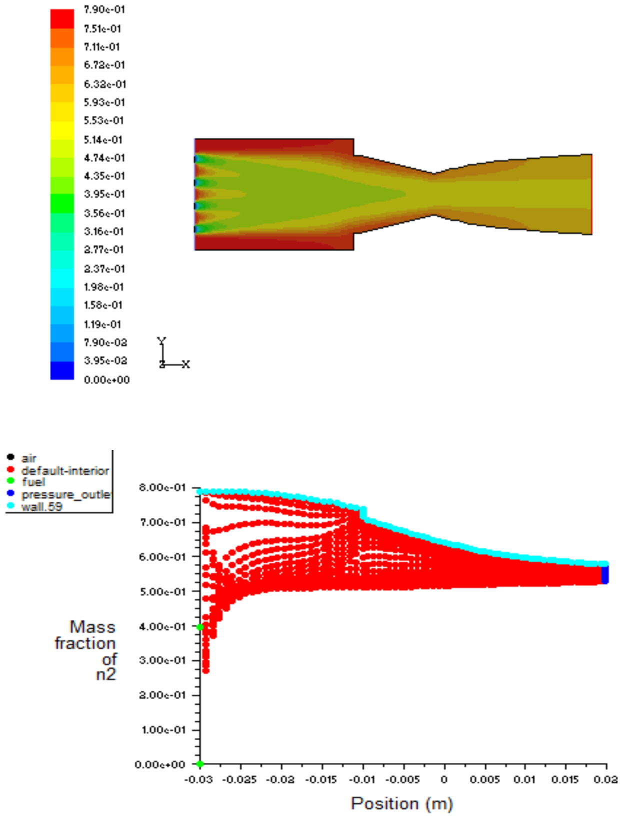
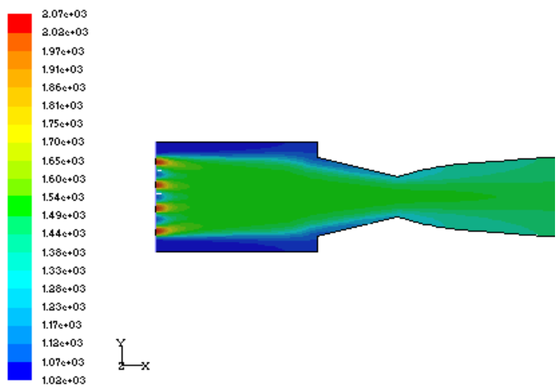


Figure –9 Mass fraction of N_2

9) Specific Heat

The contours are shown in figure 10.the specific heat is largest where the pentane is concentrated, near the fuel inlet and the where the temperature and combustion product concentration are large. Due to the four fuel inlet heat capacity increase. Relative to the constant value used before, substantially lowers the peak flame.



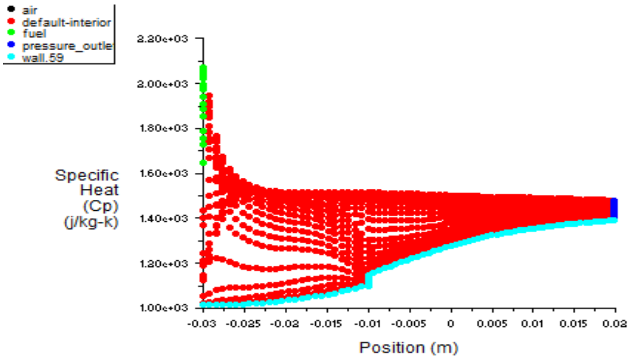


Figure-10 Specific Heat

VII. Conclusion

A model was developed to determine the pressure, temperature and flow distribution in the combustion chamber region. The model includes various parameters of the jet- and ambient gas and can therefore be used for hot gases. Several steps of the model were validated with good agreement with experimental data and numerical results found in the literature. The maximum total pressure in the combustion chamber is 1.17e+05, and the average total pressure in the combustion chamber is 1.10e+05, the average total pressure in the conversion portion of the nozzle is 1.40e+05,while the average total pressure in combustion chamber, pressure increase in combustion chamber and after that pressure goes on decrease in the convergent portion and at the throat total pressure is cover the negative value, due to supersonic nozzle total pressure in the convergent part is less and velocity increase. The average temperature in combustion chamber is 2.54e+03, and maximum temperature is 3.67e+03, the total temperature decrease in the divergent part of the nozzle compared the combustion chamber and convergent part of the nozzle. When the fuel and air is enter in the combustion chamber according to the x and y plot, its burn due to high velocity and temperature and then temperature increase rapidly in combustion chamber and convergent part of the nozzle and after that temperature decrease in the exit part of the nozzle. A maximum of 3.67e+03 is attained and beyond which the temperature steadily decreases. The maximum mass fraction of pentane is 1.00e-01 at the fuel inlet is attained beyond which the mass fraction steadily decrease, near to the wall mass fraction of pentane is zero.

ACKNOWLEDGEMENT

The authors acknowledge the valuable suggestions from Prof. E. Rathakrishnan, Professor, Department of aerospace engineering, IIT Kanpur , India. The authors acknowledge the financial help provided by AICTE from the project AICTE: 8023/RID/BOIII/NCP(21) 2007-2008 .The Project id at IIT Guwahati is ME/P/USD/4.

References

[1] Boris M. Kiforenko, Zoya V. Pasechnik and IgorYu. Vasil'ev, "Comparison of the rocket engines efficiency in the case of low thrust orbit-to-orbit transfers", *Acta Astronautica* Vol 60, 2007,pp- 801-809.

[2] Zhigang Feng and QiWang "Research on health evaluation system of liquid-propellant rocket engine ground-testing bed based on fuzzy theory", *Acta Astronautica* Vol 61,2007, pp-840-853.

[3] Vadim Zakirov and Vladimir Pavshook; "Russian Nuclear Rocket Engine Design for Mars Exploration", *Tsinghua Science And Technology.*, Vol 12, 2007, pp-256-260.

[4] Johannes Lux , Dmitry Suslov and Oskar Haidn, "porous liquid propellant rocket engine injectors", *Aerospace Science and Technology*, Vol 12, 2007, pp-469-477.

[5] Alexander Potapov, YuriyShtefan and Elena Lichman, "Research of material for un cooled nozzle extensions of liquid rocket engines", *Acta Astronautica*, Vol 64, 22-27.

[6] G. Lacaze, B. Cuenot, T. Poinsot and M. Oschwald, "Large eddy simulation of laser ignition and compressible reacting flow in a rocket-like configuration", *Combustion and Flame*, Vol 156, 2007,PP-1166-1180.

[7] ValeriyI.Timoshenkoa,IgorS.Belotserkovetsa,Vjacheslav,P.Gusininb, "Problems of providing completeness of the methane-containing block-jet combustion in a rocket-ramjet engine's combustion chamber", *Acta Astronautica*, Vol 65, 2009, pp-231-1237.

[8] M. Masquelet, S. Menona, Y. Jinb and R. Friedrich, "Simulation of unsteady combustion in a LOX-GH2 fueled rocket engine", *Aerospace Science and Technology* Vol 13, 2009, pp-466-474.

[9] David R. Greatrix, Regression rate estimation for standard-flow hybrid rocket engines, *Aerospace Science and Technology* 13, 358-363, (2009).

[10] K.M.Pandey and S.K.Yadav, "CFD Analysis of a Rocket Nozzle with Two Inlets at Mach 2.1", *Journal of Environmental Research and Development*, Vol 5, No 2, 2010, pp- 308-321,

Dr. K.M.Pandey did his PhD in Mechanical Engineering in 1994 from IIT Kanpur. He has published and presented 170 papers in International & National Conferences and Journals. Currently he is working as Professor of Mechanical .Engineering Department, National Institute of Technology, Silchar, Assam, India..He also served the department in the capacity of head from July 07 to 13 July 2010. He has also worked as a faculty consultant in Colombo Plan Staff College, Manila, Philippines as seconded faculty from Government of India. His research interest areas are the following; Combustion, High Speed Flows, Technical Education, Fuzzy Logic and Neural Networks , Heat Transfer, Internal Combustion Engines, Human Resource Management, Gas Dynamics and Numerical Simulations in CFD area from Commercial Softwares.
Email:kmpandey2001@yahoo.com.

S.K.Yadav did his M.Tech in Thermal Engineering from Mechanical Engineering Department of NIT Silchar. Currently he is working as a Asst. Professor of the Mechanical Engineering Department, Gurgaon Engineering College, Gurgaon.