International Journal of Science and Research (IJSR)
ISSN: 2319-7064
ResearchGate Impact Factor (2018): 0.28 | SJIF (2018): 7.426

Design Optimization of Rocket Thrust Chamber

Dinesh Kumar Bajaj[1], Pandi Siddharth[2]

[1, 2] Assistant Professor, MIT-ADT University, Pune, India

Abstract: ***A rocket engine is a jet engine that uses specific propellant mass for forming high speed propulsive exhaust jet. The reaction of nozzle wall to the gas pressure provides accelerating force, as the gas expands. 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. Computational solutions based on the Navier-Stokes equations, finite rate chemistry, and the k-E turbulence closure are generated with design of experiment techniques, and the response surface method is employed as the optimization tool. In addition to establishing optimum designs by varying emphasis on the individual objectives, better insight into the interplay between design variables and their impact on the design objectives is gained. 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. The design optimization is done using 4 fuel inlets for the rocket thrust chamber.***

Keywords: CFD, Combustion chamber, Design of Experiments, Optimization, Nozzle

1. Introduction

A rocket engine is a jet engine that uses specific propellant mass for forming high speed propulsive exhaust jet. Rocket engines are reaction engines and obtain thrust in accordance with Newton's third law. Since they need no external material to form their jet, rocket engines can be used for spacecraft propulsion as well as terrestrial uses, such as missiles. Most rocket engines are internal combustion engines, although non combusting forms also exist. Rocket engines are in a group have maximum exhaust velocities, are the lightest, and are the least energy efficient of all types of jet engines. The rockets are powered by exothermic chemical reactions of the rocket propellant used. Gas velocities from 2 to 4.5 kilometers per second can be achieved in rocket nozzles. The nozzles which perform this feat are called DE Laval nozzles and consist of a convergent and divergent section. The minimum flow area between the convergent and divergent section is called the nozzle throat. The flow area at the end of the divergent section is called the nozzle exit area from where the gases at their maximum possible releases from the engine.

The basic equation of thrust used is as follows .

$$F = q \times Ve + (Pe - Pa) \times Ae$$

Where
F = Thrust,
q = Propellant mass flow rate,
Ve = Velocity of exhaust gases
Pe = Pressure at nozzle exit,
Pa = Ambient pressure,
Ae = Area of nozzle exit
Under-expanded we have $Pe>Pa$ and Ve is small
Over-expanded nozzle we have $Pe<Pa$ and Ve is large
Ideal expanded nozzle we have $Pe=Pa$

2. Principle of Operation

Rocket engines produce thrust by creating a high-speed fluid exhaust. This fluid is generally always a gas which is created by high pressure (10-200 bar) combustion of solid or liquid propellants, consisting of fuel and oxidizer components, within a combustion chamber. The fluid exhaust is then passed through a supersonic propelling nozzle which uses heat energy of the gas to accelerate the exhaust gases to a very high speed, and from the Newton"s third law the reaction to this pushes the engine in the opposite direction. In rocket engines, high temperatures and pressures are highly desirable for good performance as this permits a longer nozzle to be fitted to the engine, which gives higher exhaust speeds, as well as giving better thermodynamic efficiency.

3. Parts of Rocket Nozzle

A nozzle is used to give the direction to the gases coming out of the combustion chamber. Nozzle is a tube with variable cross-sectional area. Nozzles are generally used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the exhaust stream that emerges from them. The nozzle is used to convert the chemical-thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and temperature. The convergent and divergent (also known as convergent-divergent nozzle) type of nozzle is known as DE-LAVAL nozzle. Throat is the portion with minimum area is a convergent-divergent nozzle.

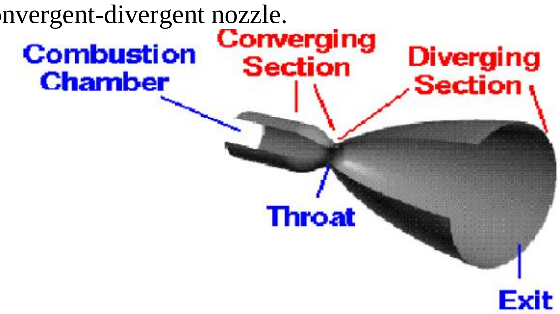


Figure 1: Parts of rocket nozzle

4. Nozzle Geometry

Analysis of nozzle geometry is a very important parameter in determining the performance characteristics of a spacecraft. Proper geometrical nozzle design can regulate the exhaust in such a way that maximum effective velocity can be reached by a spacecraft. Both under and over expansion nozzles can be problematic for flight of a spacecraft.

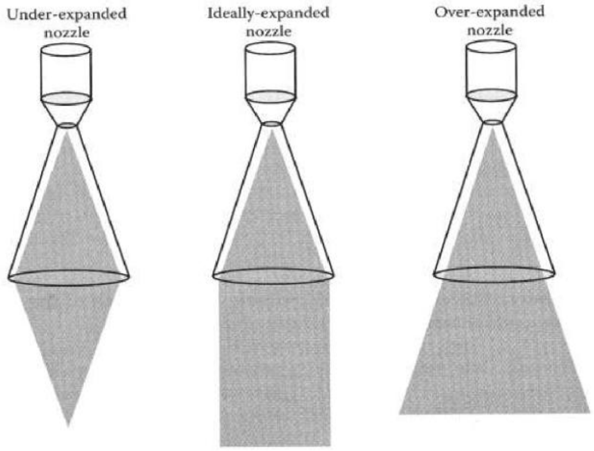


Figure 2: Nozzle Geometry

5. Introduction To CFD

Fluid dynamics deals with the dynamic behavior of fluids and its mathematical interpretation is called as Computational Fluid Dynamics. Fluid dynamics is governed by sets of partial differential equations, which in most cases are difficult or rather impossible to obtain analytical solution. CFD is a computational technology that enables the study of dynamics of things that flow. The Physical aspects of any fluid flow are governed by three fundamental principles: Mass is conserved; Newton's second law; and Energy is conserved.

These fundamental principles can be expressed in terms of mathematical equations, which in their most general form are usually partial differential equations.

Software tools (solvers, pre- and post processing utilities) CFD enables scientists and engineers to perform 'numerical experiments' (i.e. Computer simulations) in a 'virtual flow laboratory' real experiment CFD simulation.

6. Calculations

Assume our rocket engine operates under the following conditions:
q = Propellant mass flow rate = 100 kg/s
k = Specific heat ratio = 1.20
M = Exhaust gas molecular weight = 24
Tc = Combustion chamber temperature = 3600 K
Pc = Combustion chamber pressure = 5 MPa
Pa = Ambient pressure = 0.05 MPa

Case 1: Ideal expanded nozzle we have *Pe=Pa*
The gas pressure and temperature at the nozzle throat is less than in the combustion chamber due to the loss of thermal energy in accelerating the gas to the local speed of sound at the throat. Therefore, we calculate the pressure and temperature at the nozzle throat,
$Pt = Pc \times [1 + (k - 1) / 2]^{-k/(k-1)}$
$Pt = 5 \times [1 + (1.20 - 1) / 2]^{-1.20/(1.20-1)}$
$Pt = 2.82 \text{ MPa} = 2.82\text{x}10^6 \text{ N/m}^2$

$Tt = Tc \times [1 / (1 + (k - 1) / 2)]$
$Tt = 3{,}600 \times [1 / (1 + (1.20 - 1) / 2)]$
$Tt = 3{,}273 \text{ K}$
The area at the nozzle throat is given by
$At = (q / Pt) \times \text{SQRT}[(R' \times Tt) / (M \times k)]$
$At = (100 / 2.82\text{x}10^6) \times \text{SQRT}[(8{,}314 \times 3{,}273) / (24 \times 1.20)]$
$At = 0.0345 \text{ m}^2$

The hot gases must now be expanded in the diverging section of the nozzle to obtain maximum thrust. The Mach number at the nozzle exit is given by
$Nm^2 = (2 / (k - 1)) \times [(Pc / Pa)^{(k-1)/k} - 1]$
$Nm^2 = (2 / (1.20 - 1))\times[(5/0.05)^{(1.20-1)/1.20} - 1]$
$Nm^2 = 11.54$
$Nm = (11.54)^{1/2} = 3.40$

The nozzle exit area corresponding to the exit Mach number is given by
$Ae = (At / Nm) \times [(1 + (k - 1) / 2 \times Nm^2)/((k + 1) / 2)]^{(k+1)/(2(k-1))}$
$Ae = (0.0345 / 3.40) \times [(1 + (1.20 - 1) / 2 \times 11.54)/((1.20 + 1) / 2)]^{(1.20+1)/(2(1.20-1))}$
$Ae = 0.409 \text{ m}^2$

The velocity of the exhaust gases at the nozzle exit is given by
$Ve = \text{SQRT}[(2 \times k / (k - 1)) \times (R' \times Tc / M) \times (1 - (Pe / Pc)^{(k-1)/k})]$
$Ve = \text{SQRT}[(2 \times 1.20 / (1.20 - 1)) \times (8{,}314 \times 3{,}600 / 24) \times (1 - (0.05 / 5)^{(1.20-1)/1.20})]$
$Ve = 2{,}832 \text{ m/s}$

Finally, we calculate the thrust,
$F = q \times Ve + (Pe - Pa) \times Ae$
$F = 100 \times 2{,}832 + (0.05\text{x}10^6 - 0.05\text{x}10^6) \times 0.409$
$F = 283{,}200 \text{ N}$

Case 2: Under-expanded we have *Pe>Pa* and *Ve* is small
Let's now consider what happens when the nozzle is under-expanded, that is *Pe>Pa*. If we assume *Pe=Pa* × 2, we have
$Pe = 0.05 \times 2 = 0.10 \text{ MPa}$
$At = 0.0345 \text{ m}^2$

$Nm^2 = (2/(1.20 - 1))\times[(5 / 0.10)^{(1.20-1)/1.20} - 1]$
$Nm^2 = 9.19$
$Nm = (9.19)^{1/2} = 3.03$

$Ae = (0.0345 / 3.03) \times [(1 + (1.20 - 1) / 2 \times 9.19)/((1.20 + 1) / 2)]^{(1.20+1)/(2(1.20-1))}$
$Ae = 0.243 \text{ m}^2$

$Ve = \text{SQRT}[(2\times1.20 / (1.20 - 1)) \times (8{,}314 \times 3{,}600 / 24) \times (1 - (0.10 / 5)^{(1.20-1)/1.20})]$
$Ve = 2{,}677 \text{ m/s}$

$F=100\times2{,}677+(0.10\text{x}10^6-0.05\text{x}10^6)\times 0.243$
$F = 279{,}850 \text{ N}$

Case 3: Over-expanded nozzle we have $Pe<Pa$ and Ve is large

Now we consider the over-expanded condition, that is $Pe<Pa$. If we assume $Pe=Pa$ / 2, we have

Pe = 0.05 / 2 = 0.025 MPa

At = 0.0345 m^2

$Nm^2 = (2 / (1.20 - 1)) \times [(5 / 0.025)^{(1.20-1)/1.20} - 1]$

$Nm^2 = 14.18$

$Nm = (14.18)^{1/2} = 3.77$

$Ae = (0.0345 / 3.77) \times [(1 + (1.20 - 1) / 2 \times 14.18)/((1.20 + 1) / 2)]^{(1.20+1)/(2(1.20-1))}$

Ae = 0.696 m^2

$Ve = SQRT[(2\times1.20 / (1.20 - 1)) \times (8{,}314 \times 3{,}600 / 24) \times (1 - (0.025 / 5)^{(1.20-1)/1.20})]$

Ve = 2,963 m/s

$F = 100 \times 2{,}963 + (0.025x10^6 - 0.05x10^6) \times 0.696$

F = 278,900 N

We see that both the under-expanded and over-expanded nozzles produce thrusts less than that produced when the condition $Pe=Pa$ is satisfied.

6.1 Parameters for design of the nozzle

1) Length of combustion chamber = 15.6 m
2) Diameter of combustion chamber=21.4 m
3) Diameter of the Throat= 8.5 m
4) Converging length = 16 m
5) Diverging length = 40.44 m.
6) Diameter of fuel inlet = 2.15 m
7) Overall length = 72.54 m

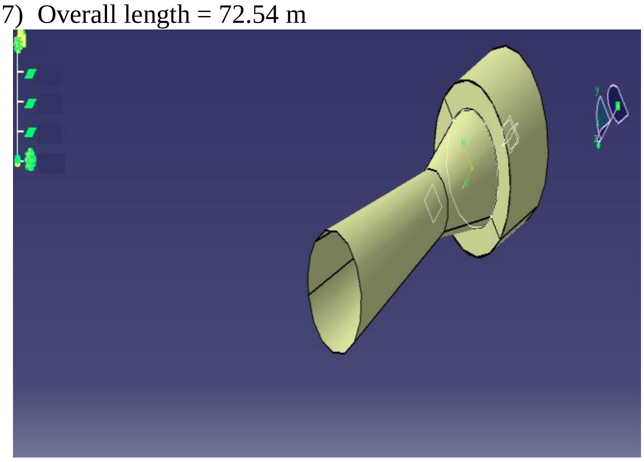


Figure 3: Catia design of rocket nozzle

7. Designing the Thrust Chamber

Aim: To optimize the design by increasing the fuel inlet.
Number of inlets: One, three, four and five.
Software used:

Designing : ICEM CFD cad interface
Pre-Processing : Ansys-Pre
Solver : Ansys-Post

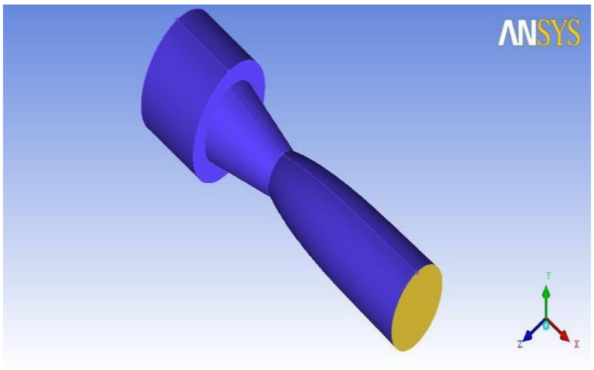


Figure 4: ICEM CFD geometry file

Mesh Information:

Domain	Nodes	Elements
Inlet 1	64896	36125
Inlet 3	65132	362460
Inlet 4	65216	362893
Inlet 5	56315	363433

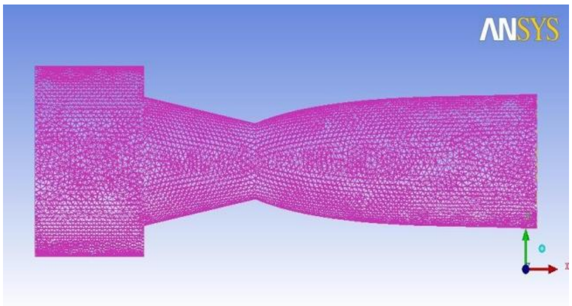
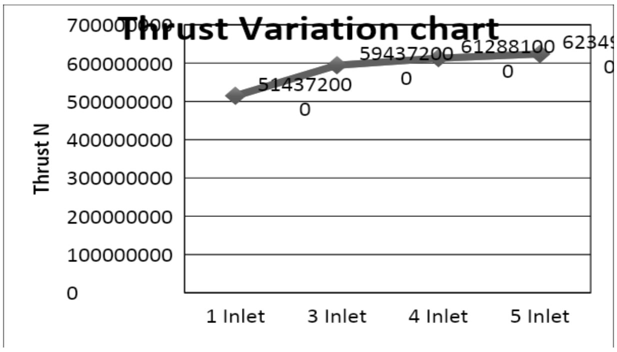
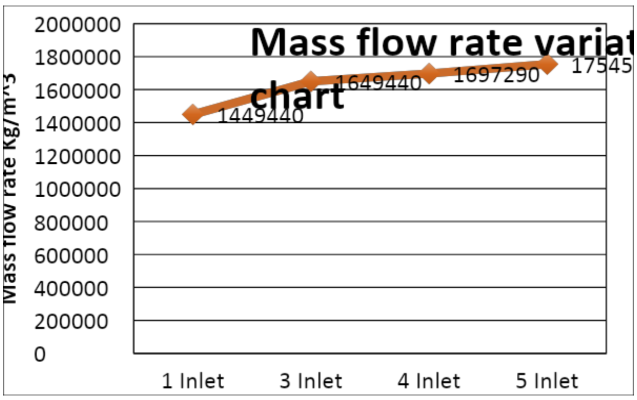


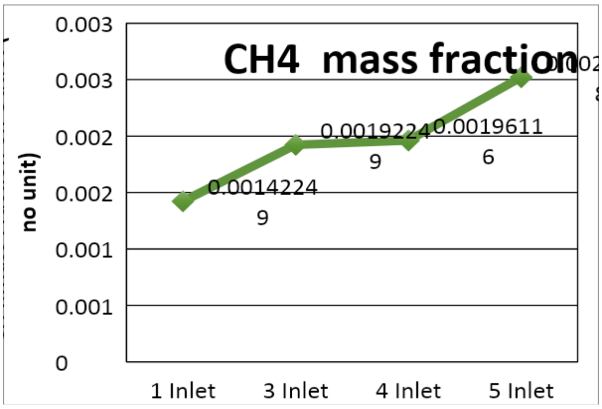
Figure 5: Mesh file of rocket nozzle



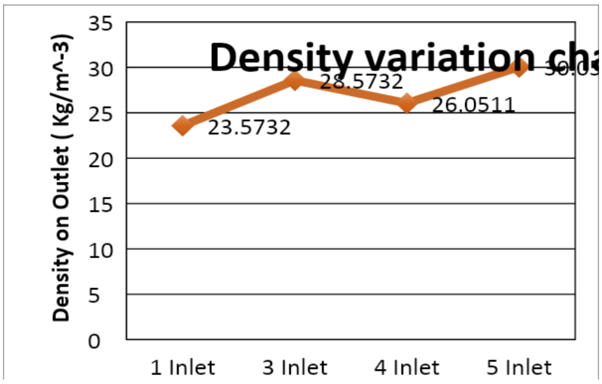
Graph 1: Thrust variation chart



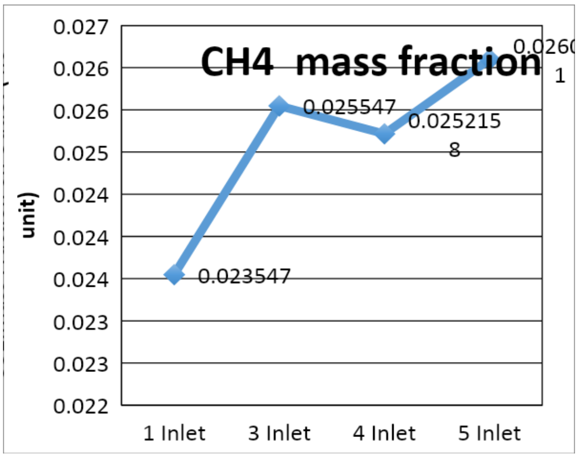
Graph 2: Mass flow rate variation chart



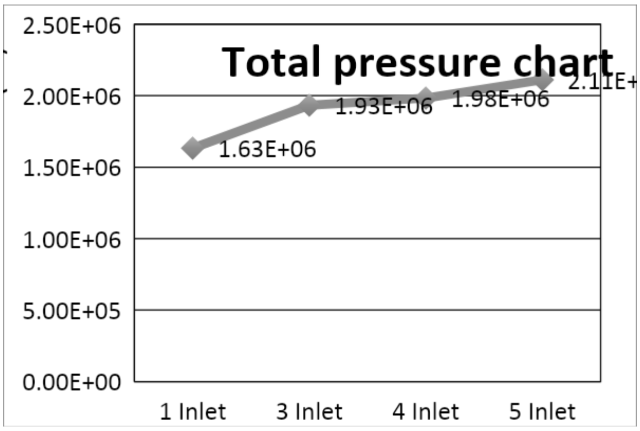
Graph 3: CH4 mass fraction chart



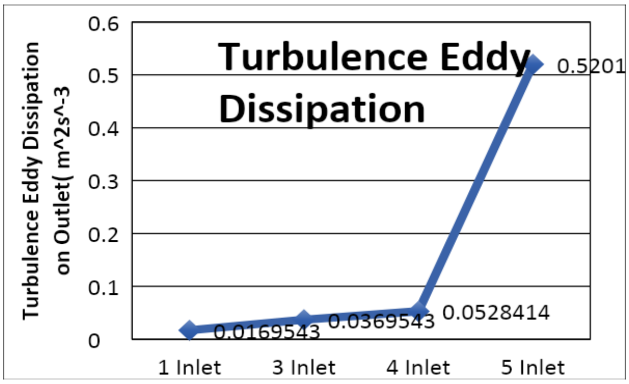
Graph 4: Density variation chart



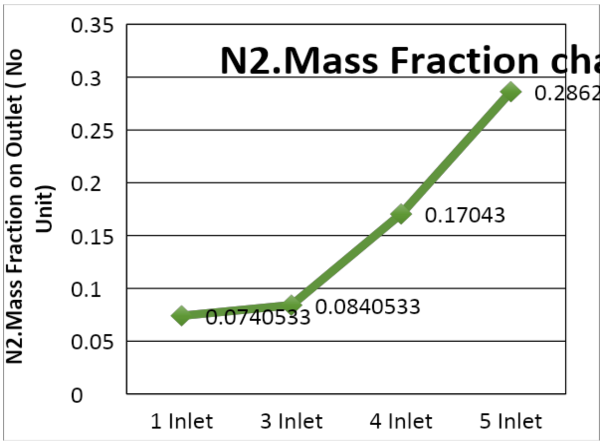
Graph 5: CH4 mass fraction chart



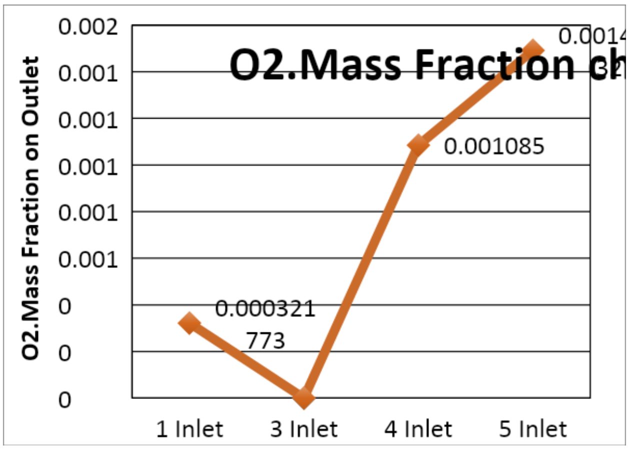
Graph 6: Pressure variation plot



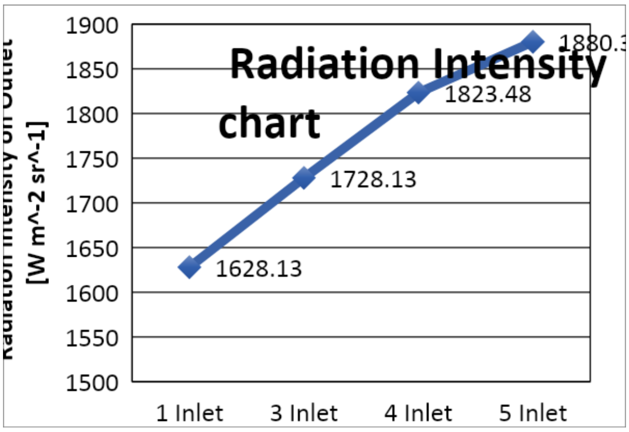
Graph 7: Turbulence eddy dissipation



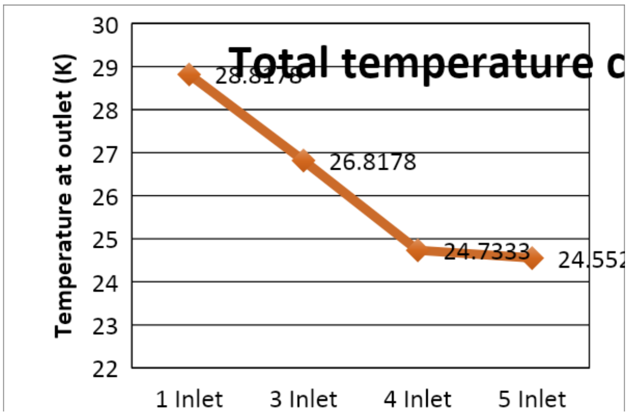
Graph 8: N2 mass fraction chart



Graph 9: O2 mass fraction chart



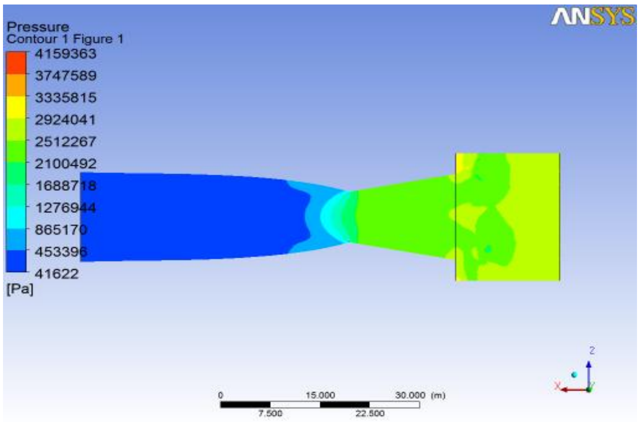
Graph 10: Radiation intensity chart



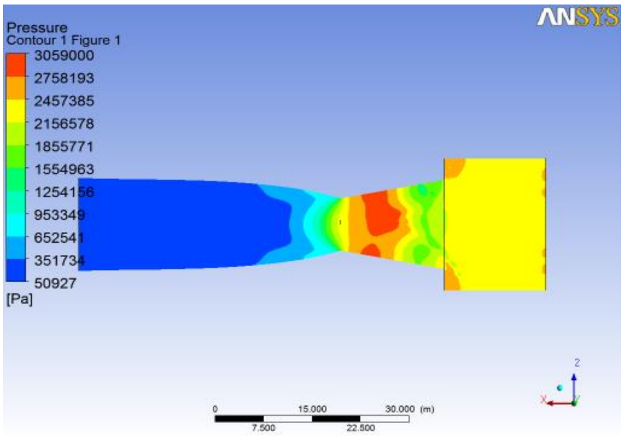
Graph 11: Temperature chart

8. Pressure Contour Plot

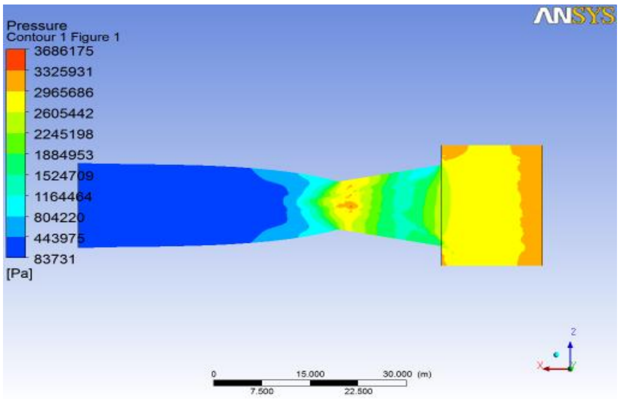
Case <i>



Case<ii>

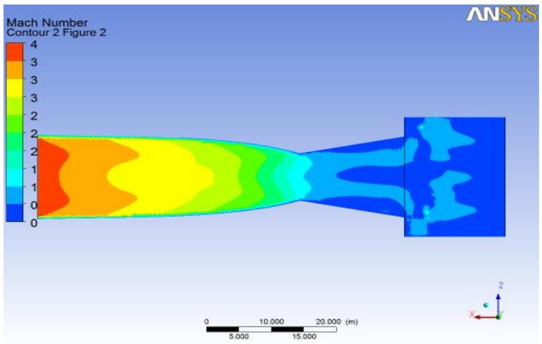


Case <iii>

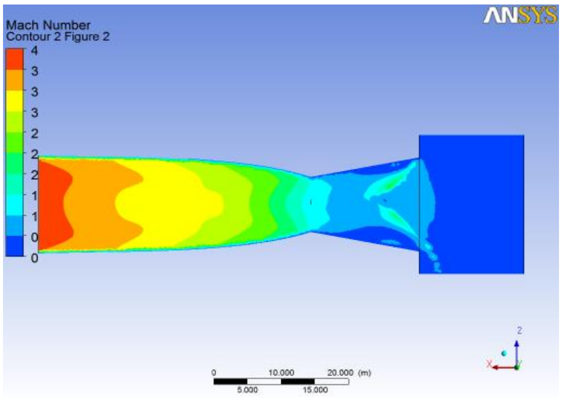


8.1 Mach number plot

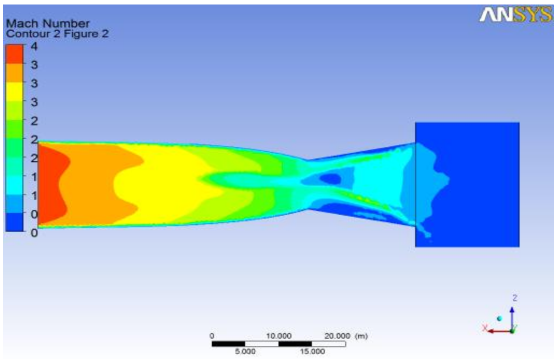
Case<i>



Case<ii>

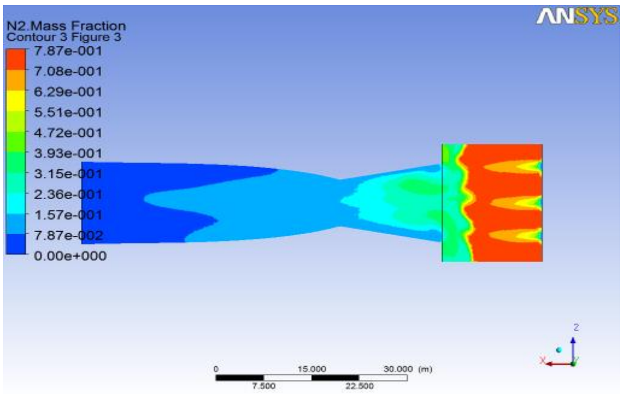


Case<iii>

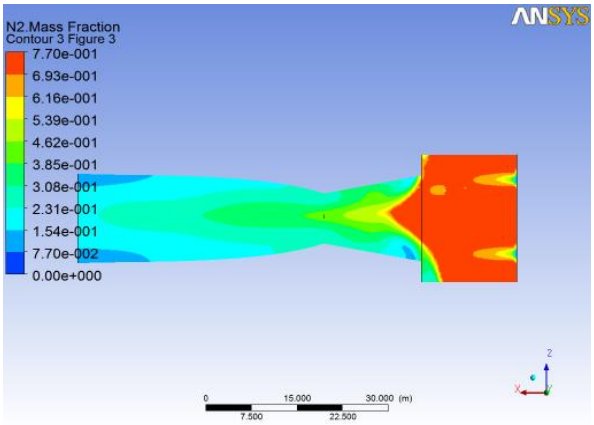


8.2 N_2 mass fraction plot

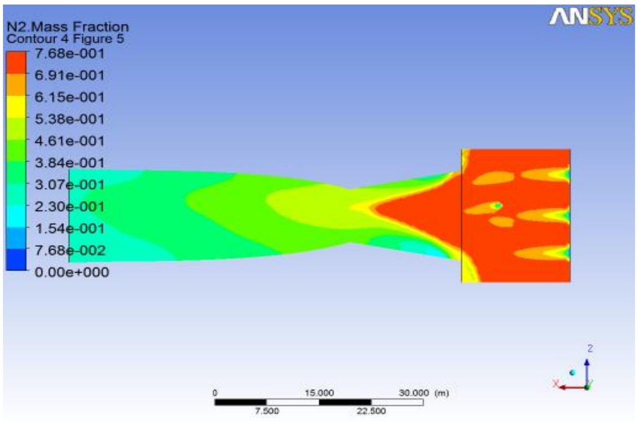
Case <i>



Case <ii>

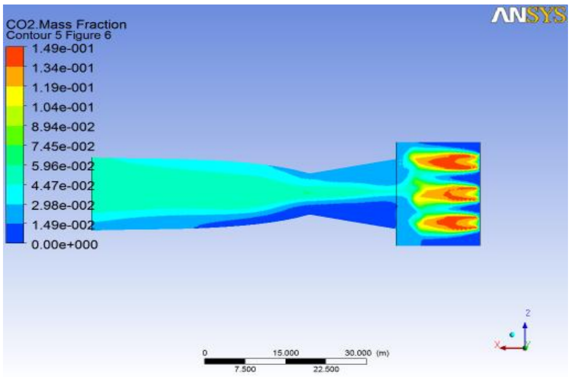


Case <iii>

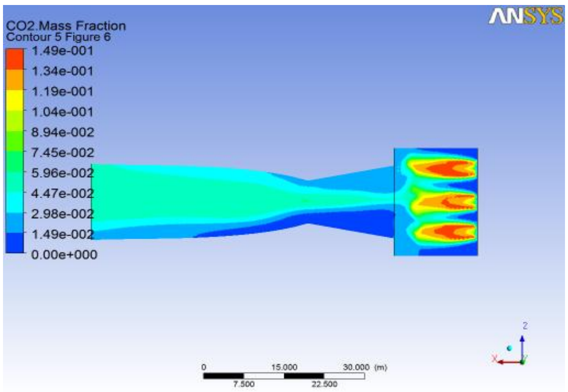


8.3 CO_2 mass fraction distribution

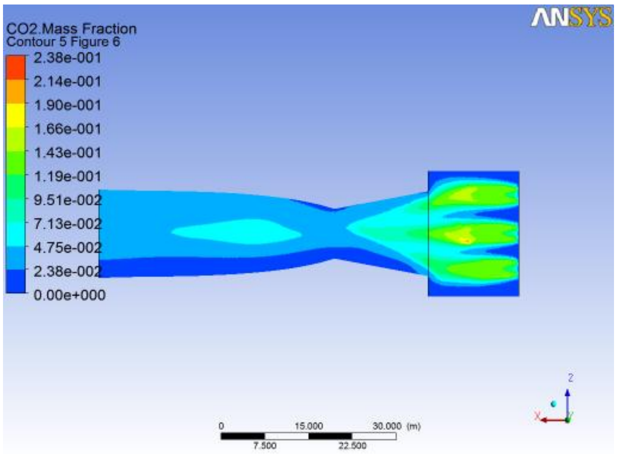
Case <i>



Case < ii>



Case <iii>



9. Conclusion

From the total pressure contour plot, the maximum total pressure and the average total pressure in the combustion chamber in the convergent portion of the nozzle increase. While the average total pressure in combustion chamber first increases and after that pressure goes on decreasing in the convergent portion and at the throat, the total pressure will cover the negative value due to supersonic nozzle. The total pressure in the convergent part is less and velocity increases in this portion. This can be easily visualized form the above figures. There is decrease in stagnation pressure near the nozzle wall due to viscous effect.

From the exit mach number plot maintain and the overall exit temperature decreases across the trust chamber**.**

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 in the combustion chamber and minimum value, and goes on decreasing in part of the nozzle, while near to the nozzle wall mass fraction co_2 has zero.

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. The maximum average mass flow rate total pressure in the combustion chamber is **2.11129E+6** outlet is 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 **24.5527 K** 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. The design optimization is done using 4 fuel inlets for the rocket thrust chamber

References

[1] A. Lyubar and T. Sattelmayer "Numerical investigation of fuel mixing, ignition and flame stabilization by a strut injector in a scramjet combustor."
[2] Gerlinger,P.,Möbus, H., Brüggemann, D. "An Implicit Multigrid Method for Turbulent Combustion", Journal of Computational Physics, 2001, 166.
[3] Ro A. Oman K. M, Foreman, Jo Leng, and H. B. Hopkins "Simulation of hypersonic scramjet exhaust" Grumman aerospace corporation for Langley Research Center CR-2494, March 1975.
[4] K.M.Pandey and T.Sivasakthivel "CFD Analysis of a Hydrogen Fueled Mixture in Scramjet Combustor with a Strut Injector by Using Fluent Software", International Journal of Engineering and Technology, Vol.3, No.2, April 2011.
[5] Hopkins, H., Konopka, W., Leng, J., and Oman, R., "Simulation Experiments
[6] Using Hydrogen/Oxygen Gas Mixtures in a High-Pressure Detonation Tube," NASA CR-128955, 1973.
[7] Cubbage, James M, and Monta, William J."Parametric Experimental Investigation of a Scramjet Nozzle at Mach 6 with Freon and Argon or Air Used for Exhaust Simulation". NASA TP 3048, February 1991.
[8] Kenneth E. Tatum and Lawrence D. Huebner, "Exhaust Gas Modeling Effects on Hypersonic Powered Simulation at Mach 10", AIAA Sixth International Aerospace Planes and Hypersonic Technologies Conference, Chattanooga, TN, AIAA 95-6068, April 3-7, 1995
[9] K.M. Pandey and A.P. Singh "Numerical Analysis of Supersonic Combustion by Strut Flat Duct Length with S-A Turbulence Model" IACSIT International Journal of Engineering and Technology, Vol.3, No.2, April 2011.
[10] Hsu A.T., Anand M.S., Razdan M.K., "An assessment of pdf versus finite-volume methods for turbulent reacting flow calculations", 34th Aerospace Sciences Meeting & Exhibit, January 15–18. Reno, NV: AIAA, 1996.
[11] Pope S.B., "Computationally efficient implementation of combustion chemistry using in situ adaptive tabulation", Comb Theory and Modelling