OPTIMIZATION OF HEAT TRANSFER IN NOZZLE WITH DIFFERENT VELOCITIES AND FLOW RATES

V.GANGADHAR,
M.Tech Student, Department of Mechanical Engineering, QIS College of Engineering & Technology, Ongole,

R. SATYA MEHER,
Professor, Department of Mechanical Engineering, QIS College of Engineering & Technology, Ongole.

ABSTRACT:

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. CFD analysis has been conducted to analyze flow pattern of supersonic rocket nozzle by using two different fluids Hydrogen Peroxide and methane. Variation in parameters like velocity, static pressure, turbulence intensity and temperature are being analyzed. This work also outlines a method for a numerical specification of a nozzle design and generation of a properly formulated CFD model to analyze it. This work illustrates that using magnification ratio or velocity alone does not result in an optimized design. These factors must be weighted to obtain a design that balances these factors for the geometry of fluid volume to be mixed. Additionally, this work shows that nozzle placement is perhaps more significant than nozzle design for optimum mixing with minimum power consumption.

INTRODUCTION:

Nozzle is the component of a Missile, Rocket or air-breathing Engine that Produces thrust. converting the thermal energy of the hot Chamber gases into Kinetic energy and directing that energy along the Nozzle axis, as illustrated below, accomplish this The propellant is composed of a fuel, typically liquid hydrogen (H2), and an oxidizer, typically liquid oxygen (O2). The propellant is pumped into a combustion chamber at some rate (m) where the fuel and oxidizer are mixed and burned. The exhaust gases from this Process are pushed into the throat region of the nozzle. Since the throat is of less cross-sectional area than the rest of the engine, the gases are compressed to a high pressure. The nozzle itself gradually Increases in cross-sectional area allowing the gases to expand. As the gases do so, they push against the walls of the nozzle creating thrust.

METHODOLOGY:

Total analysis is for reducing the stresses caused the nozzle end dish part due to the high pressure gases. The procedure for a static analysis consists of three main steps:

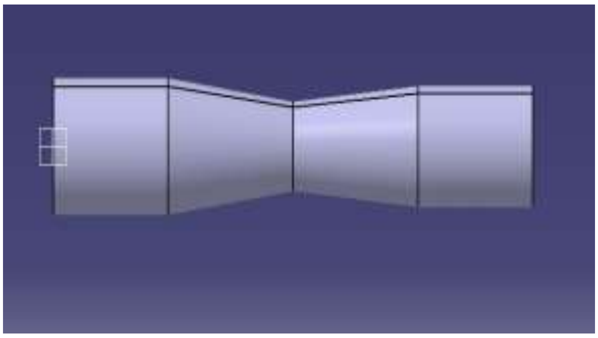
1. Build the model 2. Apply boundary conditions & obtain the solution 3. Review the results

Finite element analysis of nozzle: Finite Element Method (FEM) has become one of the most widely used techniques, for analyzing mechanical loading characteristics in modern engineering components. Traditional analysis techniques can only be satisfactory applied to a range of conventional component shapes and specific loading conditions. Unfortunately, the majority of engineering loading situations are not simple and straight forward therefore traditional techniques often need to be modified and compromised to suit situations for which they were not intended.

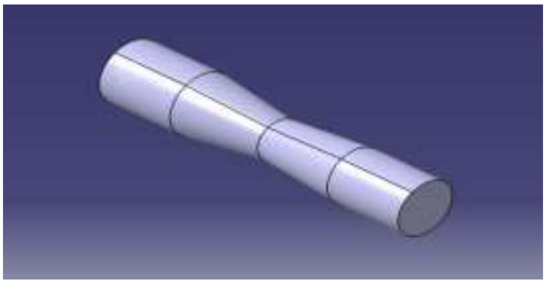
Geometric modelling: Modelling has been carried over in CATIA V5 R 15

Software is very easy modelling critical components. CATIA is very user friendly and parametric.

Nozzle has three components: 1.Nozzle End dish 2.Nozzle Neck 3.Nozzle Cone



Geometry of Simple in CATIA



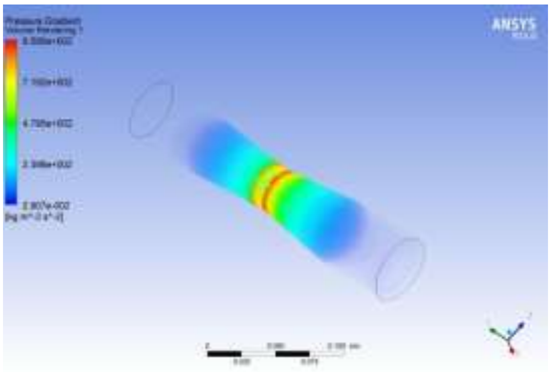
Isometric view

RESULTS AND DISCUSSIONS:

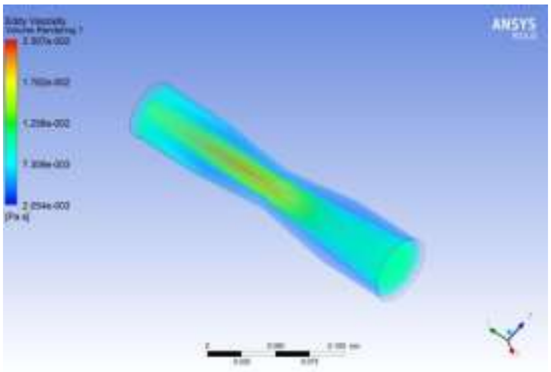
The inlet air was assumed to enter the nozzle at normal temperature and the pressure was taken to be 0.1m/s. The following are results of the analysis of the venture for different angles of the throttle plate.

Fluid type: Hydrogen Peroxide

Pressure: 0.1m/s

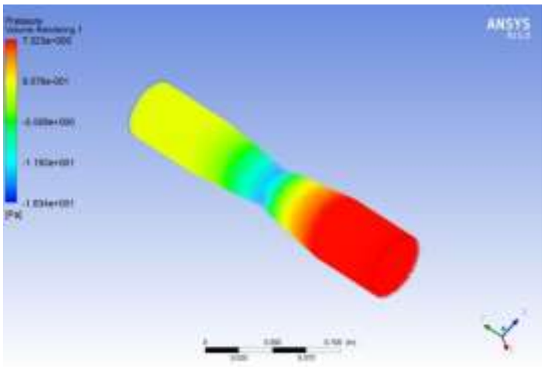


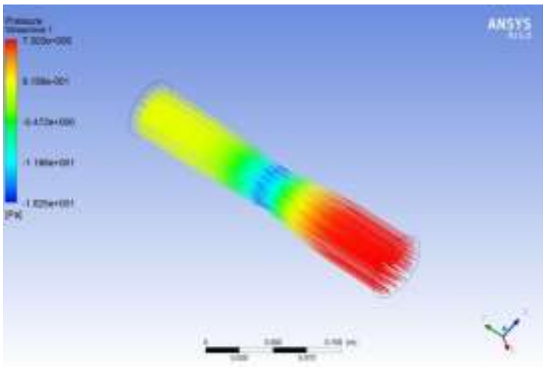
pressure gradient



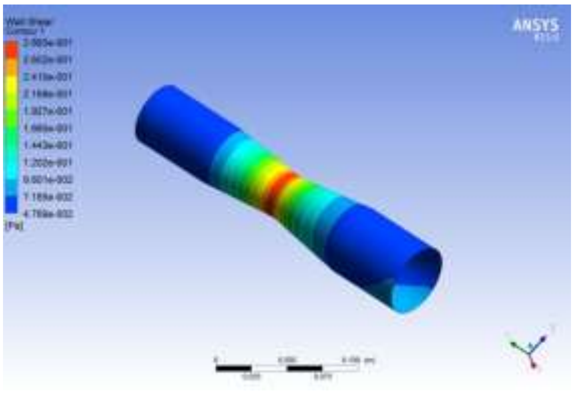
Eddy viscosity

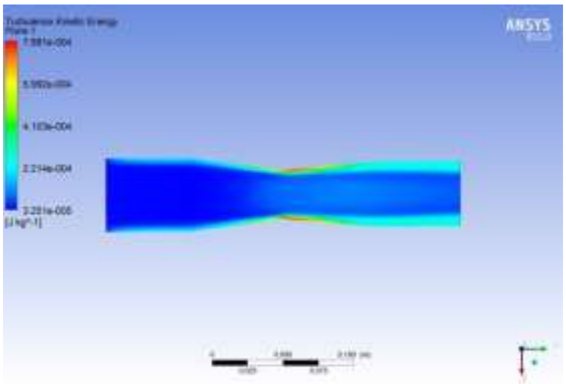
Fig shows that the maximum pressure gradient is $9.59E^{+002}$ and minimum value is $2.907\ E^{-002}$





Pressure volume rendering Pressure streamline



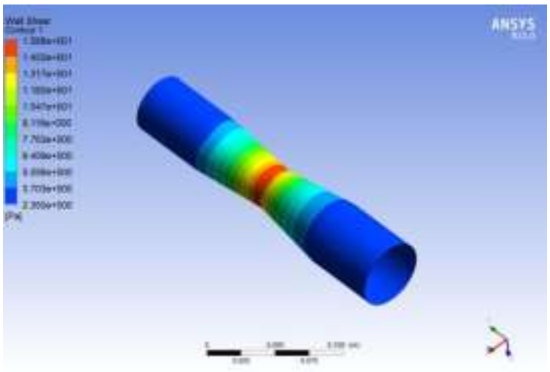


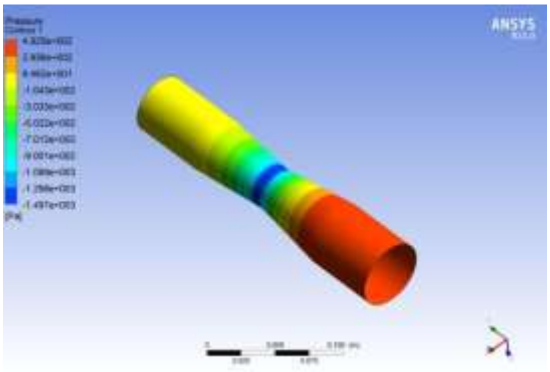
Nozzle wall shear stress

Turbulence kinetic energy

Fluid type: methane

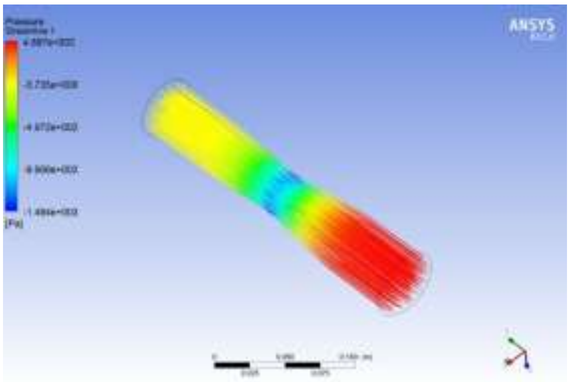
Pressure: 1m/s

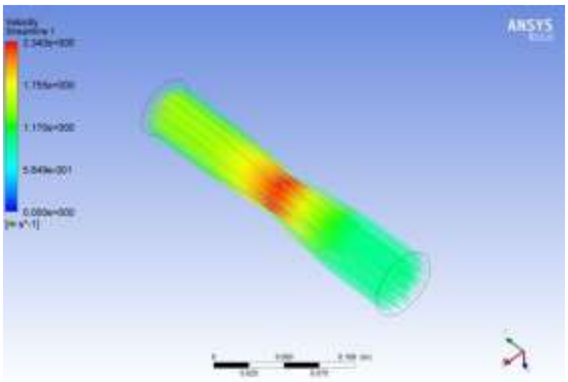




Wall shear

Pressure contor

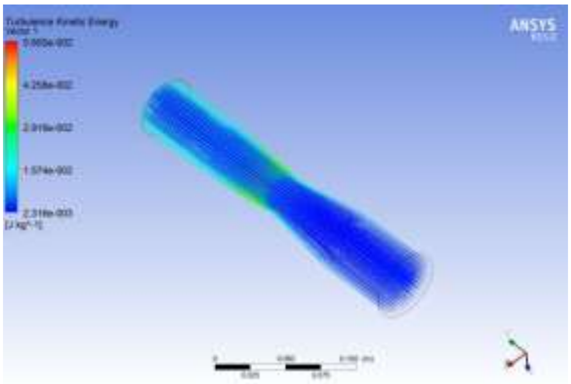




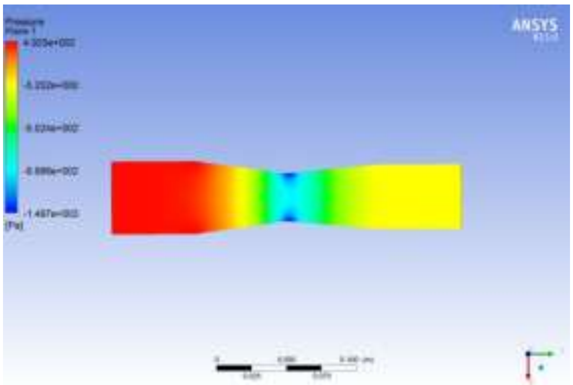
Pressures streamline

Velocity streamline

Fig. shows the statics pressure view for 450 throttle plate From fig. it is clear that when the throttle plate is 450 open, there is less amount of air flow through the inlet valve and hence the mixture is somewhat richer than the other cases. In this case the pressure at the throat of the nozzle is around 4.9200 Pascal. , when the throttle plate is open, the mixture is slightly leaner than in case of 450 opened throttle plate condition. In this case the pressure at the throat of the nozzle is found to be around 9.4600 Pascal.

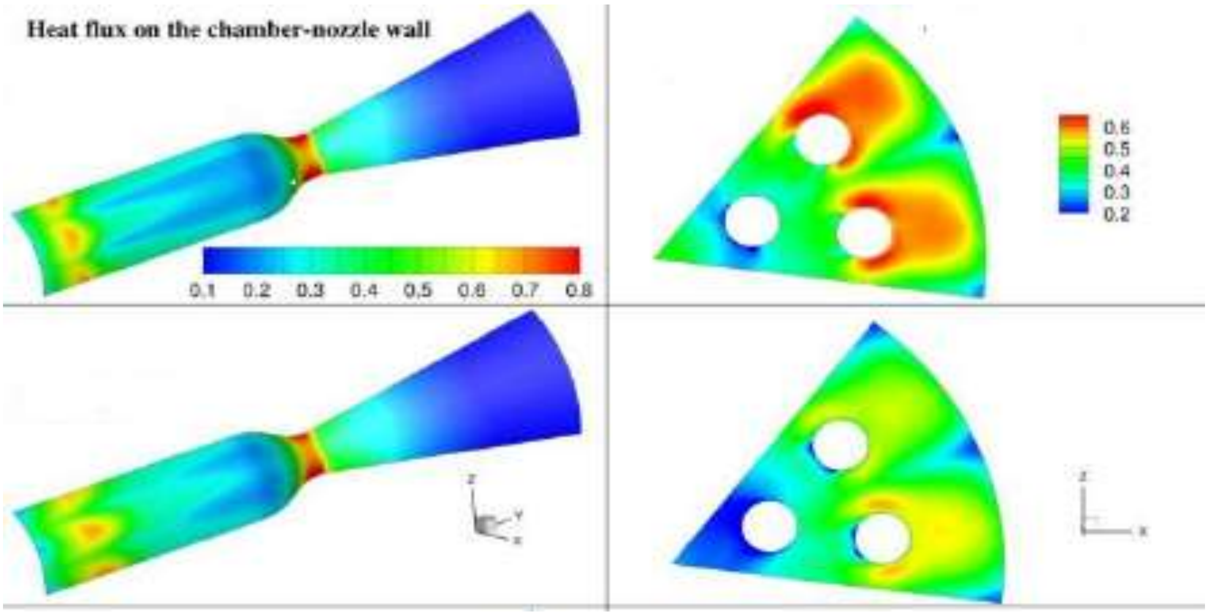


Turbulence kinetic energy

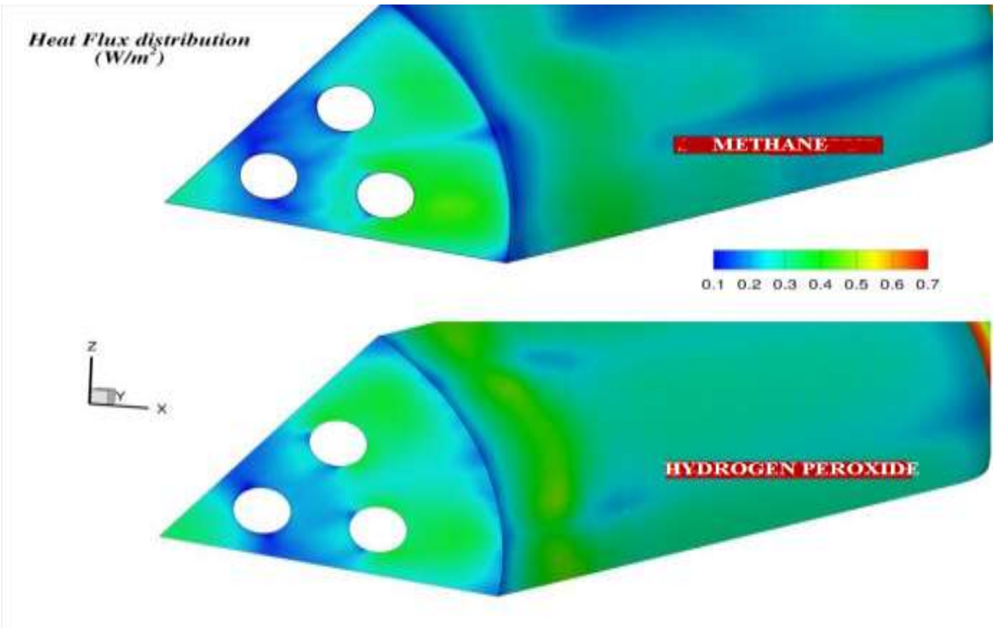


Pressure along y-axis

Calculations were performed on both methane and Hydrogen Peroxide using the same modeling and boundary conditions. Initially, In Fig. the heat flux distribution on the chamber/nozzle (left) and the injection plate (right) walls is shown, in the hypothesis of methane and hydrogen peroxide fluids. The heat fluxes have been normalized to the value given by Bartz correlation. Considering firstly the chamber/nozzle wall, it can be deduced that both the fluids are overestimate the chamber reattachment peak heat flux with respect to that predicted by the mesh, while similar values are detected in the nozzle throat; only a displacement downstream of the peak heat flux on the chamber occurs, even though the intensity does not change. Considering the injection plate wall, a sensible decrease in the predicted heat flux values can be observed switching from methane to Hydrogen Peroxide, approaching the value predicted by the structured mesh. A clearer quantitative comparison is visible in Fig., where a heat flux distribution along the chamber/nozzle wall is reported considering the slice (highlighted in the small frame) along one of the symmetry planes, with L being the full engine length. In the nozzle throat, a very good agreement between the computations using the different fluids can be deduced, with peak heat flux slightly higher than 0.8. Also the heat flux plateau in the second part of the chamber wall differs for some percentage (average value about 0.3). On the contrary, very different predictions can be noticed in the first part of the chamber, with very different re-attachment peak heat flux values; it is worth noting that also the position of the chamber peak value changes, being closer to the plate for the prediction associated with the Hydrogen Peroxide fluid.



Difference of heat flux methane and Hydrogen Peroxide



Heat flux distribution

CONCLUSION: From the above analysis the conclusions obtained are

When the flow inside the nozzle was analyzed with two different fluids, in both cases it was found that the pressure at the throat of the nozzle decreased with the increase in opening of the throttle. Because when the throttle plate opening increases then the flow of air through the nozzle increases. But as obtained from the analysis above the pressure at the throat also decreases with increase in opening of the throttle plate so the flow of air from the float chamber into the throat increases. A numerical CFD campaign was carried out in order to support the demonstrator design. Being a regenerative cooled thrust chamber, both the simulation of the combustion flow and of the flow inside the cooling channel were necessary, considering a weak coupling between them to obtain more accurate results. Different fluids with modeling were implemented to reduce as much as possible the uncertainty on the numerical prediction of the chamber wall heat flux, applied as boundary condition for the simulation of the methane Hydrogen Peroxide and inside the inlet nozzle. Concerning the simulations of the flow inside the chamber, the assumption of gases showed to be inappropriate to describe the solution in the mixing region, leading to a strong overestimation in the wall heat flux in the first part of the chamber. The use of a Hydrogen Peroxide model led to increase the numerical stiffness, thus limiting the simulation to a reduced number of computational grids, those consisting of a smaller number of cells.

The simulation of the fluids inside the cooling channel was performed applying the heat flux predictions by both the methane and the Hydrogen Peroxide fluids, in two different simulations. The results showed the highest values of the hot gas side temperature in the injection head zone for all the considered thermal load profiles, with values in the throat region far away from the allowable limits of the material. The weak thermal coupling was performed by imposing on the bottom channel wall the convective heat transfer profile, obtained on the hot side, and the adiabatic temperature. This approach could be able to properly describe the performance of the cooling system. The effect, as expected, was the decrease of the temperature profiles, in particular in the throat region and in the injection head zone. The CFD simulations were important to provide information about the thermal and fluid-dynamic behaviour of the system in order to verify the design, under the thermo-structural point of view by means of FE simulations.

BIBLIOGRAPHY:

- Olson. A.T. , " Nozzle Discharge Coefficients Compressible flow " *J. fluid eng*
- Weber, H . E. , " Boundary layer Calculation for Analysis and Design " *J. fluids Engg*.
- Keith, T. G., and J.E.A.John, "Calculated Orifice Plate Discharge Coefficients at low Reynolds numbers " *J. fluids Engg*.
- Head ,V.P., "Improved Expansion Factors for Nozzle, Orifices and Variable Area Meters "*J. fluids Engg*.
- Jordon. D, and M.D. Mintz, *Air tables.* McGrew-Hill, New york, 1965.

- Huzel, Dieter K., Huang, David H., 1992, Modern Engineering for Design of Liquid-Propellant Rocket Engines, AIAA v.147, 1563470136.
- Pizzarelli, M., Nasuti, F., Onofri, M., 2008, Flow Analysis of Transcritical Methane in Rectangular Cooling Channels, 44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit 21 - 23 July 2008, Hartford, CT, paper AIAA2008-4456.
- Kreith, F., Bohn, M.S., Principles of Heat Transfer, 6th Ed., Brooks/Cole, 2000, ISBN: 978-0534375966.
- Roncioni P., Cardillo D., Panelli M., Ricci D., Di Clemente M., Battista F., 2013. CFD Modelling and Simulations of the HYPROB Regenerative LOX/CH4 Thrust Chamber. 5th European Conference for Aeronautics and Space Sciences (EUCASS), July 1-5 2013, Munich, Germany.
- Sutton, G.P., Biblarz, O., 2010. Rocket Propulsion Elements. John Wiley & Sons, ISBN-9780470080245.