2nd International Conference on Current Research Trends in Engineering and Technology
© 2018 IJSRSET | Volume 4 | Issue 5 | Print ISSN: 2395-1990 | Online ISSN : 2394-4099
Themed Section: Engineering and Technology

Design and Development of Nozzle to Reduce the Losses

Disha Parmar[1], Dave Kishan[2], Kanojiya Vaishnavi[3], Parghi Jignesh, Mr. Parimal Prajapati[5]

[1-4]U.G. Student, Mechanical Engineering, Sigma Institute of Engineering, Bakrol, Gujarat, India
[5]Assistant Professor, Mechanical Engineering, Sigma Institute of Engineering, Bakrol, Gujarat, India

Abstract

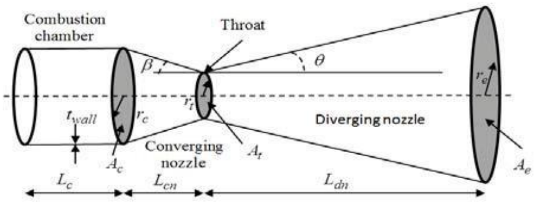
Nozzle is a device designed to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that exhaust from them. Nozzles come in a variety of shapes and sizes depending on the requirement, this is very important for the understanding of the performance characteristics of nozzle. By the proper geometrical design of the nozzle, the exhaust of the propellant gases will be regulated in such a way that maximum effective velocity can be reached. Convergent divergent nozzle is the most commonly used nozzle since in using it the propellant can be heated in combustion chamber. After getting heated the propellant first converges at the throat of the nozzle and then expands under constant temperature in the divergent part. The present project also observes the losses that occur in a rocket or jet engines and how the damage the nozzles. The methodologies to reduce the losses like Divergence losses, low contraction ratios, boundary layer drag, nozzle erosion, non-equilibrium flow, heat losses are also covered.

Keywords: Convergent divergent nozzle, CFD, losses

I. INTRODUCTION

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. Swedish engineer of French descent who, in trying to develop a more efficient steam engine, designed a turbine that was turned by jets of steam. The critical component – the one in which heat energy of the hot high-pressure steam from the boiler was converted into kinetic energy – was the nozzle from which the jet blew onto the wheel. De Laval found that the most efficient conversion occurred when the nozzle first narrowed, increasing the speed of the jet to the speed of sound, and then expanded again. Above the speed of sound (but not below it) this expansion caused a further increase in the speed of the jet and led to a very efficient conversion of heat energy to motion. The theory of air resistance was first proposed by Sir Isaac Newton in 1726. According to him, an aerodynamic force depends on the density and velocity of the fluid, and the shape and the size of the displacing object. Newton" s theory was soon followed by other theoretical solution of fluid motion problems. All these were restricted to flow under idealized conditions, i.e. air was assumed to posses' constant density and to move in response to pressure and inertia. Nowadays steam turbines are the preferred power source of electric power stations

and large ships, although they usually have a different design-to make best use of the fast steam jet, de Laval's turbine had to run at an impractically high speed. But for rockets the de Laval nozzle was just what was needed.



Computational Fluid Dynamics (CFD) is an engineering tool that assists experimentation. Its scope is not limited to fluid dynamics; CFD could be applied to any process which involves transport phenomena with it. To solve an engineering problem we can make use of various

2. Literature

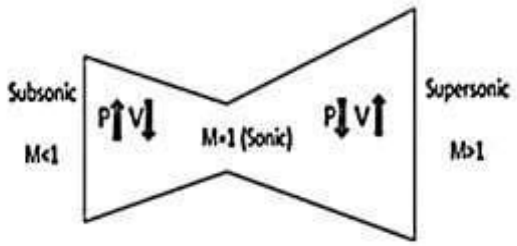
A Convergent-divergent nozzle is designed for attaining speeds that are greater than speed of sound. The design of this nozzle is obtained from the area-velocity relation (dA / dV) = - (A/V)(1-M^2) where M is the Mach number (which means the ratio of local speed of flow to the local speed of sound) A is area and V is velocity.

The decrease in Area results in the increase of pressure and decrease in velocity as seen in the above figure at the entry of the nozzle. The increase in area results in increasing the velocity at the exit of the nozzle by decreasing the pressure. Also we can find out that

3. Basic Isometric relations

The properties i.e. pressure, temperature and velocity at throat are find out by the following methods like the analytical method, experimental methods using prototypes. The analytical method is very complicated and difficult. The experimental methods are very costly. If any errors in the design were detected during the prototype testing, another prototype is to be made clarifying all the errors and again tested. This is a time-consuming as well as a cost-consuming process. The introduction of Computational Fluid Dynamics has overcome this difficulty as well as revolutionized the field of engineering. In CFD a problem is simulated in software and the transport equations associated with the problem is mathematically solved with computer assistance. Thus we would be able to predict the results of a problem before experimentation.

a. M<1 results in subsonic speeds.
b. M=1 results in sonic speeds.
c. M>1 results in supersonic speeds. One important point is that to attain supersonic speeds there is a need to maintain favorable pressure ratios across the nozzle.



$$\frac{P_t}{P_o} = \left[\frac{2}{(\gamma+1)}\right]^{\frac{\gamma}{\gamma-1}}$$

$$\frac{T_t}{T_o} = \left[\frac{2}{(\gamma+1)}\right]$$

$$v_t = \sqrt{\frac{2\gamma}{\gamma+1} RT_0} = \sqrt{\gamma RT_t}$$

Relations Velocity and temperature values at different cross sections are by the following formulae:

The continuity equation is

- **Losses in Nozzle**

1. The usage of small chambers or smaller cross-sections relative to throat area results into pressure losses. Also low nozzle contraction ratios lead to pressure loss and slight reduction in exhaust velocity and thrust. Hence, combustion chambers should have a larger cross-sectional area and nozzle contraction ratio should also be kept high if possible

2. Due to variable cosine function of the divergence angle (like in conical nozzle) creates divergence loss in the exit part of the nozzle. This can be prevented or reduced to some extent by contouring a bell-shaped nozzle.

(3)During transient operations like start, pulsing and stop, the chamber pressures and overall performance are lower.

5. Methodology to reduce losses

Boundary Layer Flow

This loss are created due to viscosity in fluid flow

1.The fluid gets decelerated toward stagnation conditions due to no-slip conditions at the nozzle boundary layer.

2.The thermal energy increases as the kinetic energy decreases leading to energy balance.

3.AxVxpx =pt.AtVt

The steady flow energy equation is as follows

$$\frac{Q-W}{m} = (h + \frac{V^2}{2} + gz)_t - (h + \frac{V^2}{2} + gz)_x$$

(4) Unsteady combustion process and continuous changes in fluid floe can also cause small losses. Hence that should be prevented.

(5)The effective average exhaust velocity is reduced by 0.5 to 1.5% if the wall boundaries have lower velocities. Hence they should possess higher velocities.

(6)Any size of solid particles and/or liquid droplets can cause losses in the nozzle upto 5%.
Hence, the impurities in the gaseous fuel should be avoided.

(7) If any erosion occurs in the throat region, it can increase its diameter from 1-6%

(8) If during the nozzle flow,any chemical reaction occurs then it change gas composition and its properties and also temperature of the gas. This can leads to losses of about0.5%

$$h_0 = C_p T + \frac{v^2}{2}$$

• The viscous portion of the flow (within the BL or 99% of the free jet velocity) is quite smaller when compared to the core flow for large rocket motors.

• Owing to this, the BL losses are normally between 0.5% and 1.5%• However for micro propulsion applications (scale nozzles) the BL losses are much more significant

1.

2. Potential substantial loss for attitude adjustment thrusters as well.

Low Nozzle Contraction Ratio

• Small nozzle contraction ratios cause pressure losses in the chamber

• This loss in pressure will result in reduced thrust and exit velocity

• Increasing the chamber to throat area ratio will improve this loss but at a weight penalty.

Nozzle Throat Erosion

• The interaction of environmental conditions together with the usual requirement that dimensional stability in the nozzle throat be maintained makes the selection of suitable rocket nozzle materials extremely difficult.

• Erosion of the throat (recall throat temperature is generally the highest!)

• Gradual erosion of nozzle throat material changes at over time.

• Ablative cooling may be applied either to the entire combustion chamber liner or to the throat section alone.

Conclusion

The various losses that occur in a rocket nozzle like divergence loss, non-equilibrium g=flow, drag losses, thrust losses, nozzle erosion ,heatTypically all solid rocket boosters have ablative cooled nozzles.

Real Gas Properties

• Also depends on if frozen, chemical equilibrium, or chemical non-equilibrium assumptions are being made as the gases expand and accelerate through the nozzle

Transient Operation

• Lower pressure occurs during transients

These transient processes include motor starting, motor stopping, and motor pulsing

Unsteady Combustion

• Transients in the thrust chamber due to intermittent incomplete combustion events can also cause losses.

• This is different than transient start-up and stopping effects

• Could be a result of bad mixing or fuel/oxidizer ratio injection mass flow variations.

losses and many other losses are covered with a methodology to reduce them.

References

1. Pardha saradhi Natta, V. Ranjith Kumar, Dr. [2]. "Flow Analysis of Rocket Nozzle Using
2. Y V Hanumantha Rao International Journal of Computational Fluid Dynamics (CFD)" . K.M.
3. Engineering Research and Applications (IJERA) PANDEY , Member IACSIT and A.P. Singh ,
4. ISSN: 2248-9622 Vol. 2, Issue 5, September- International Journal of Chemical Engineering
5. October 2012, pp.1226-1235 and Applications, Vol. 1, No. 2, August 2010ISSN: 2010- 0221 "CFD Analysis of Conical