**ROCKET NOZZLE PERFORMANCE OPTIMIZATION USING MODIFIED TURBULENCE**

M.Manoj\*1, Rameez farouk 2, J.Bruce Ralphin Rose3, G.R.Jinu4

\*1Department of Production Technology, MIT Campus, Anna University, Chennai, Tamil Nadu 600 044, India

Technical Manager, Touchstone NDT limited, Port Moresby, Papua New Guinea

3Department of Mechanical Engineering, Anna University Regional Campus- Tirunelveli, Tirunelveli 627 007, India.

4Department of Mechanical Engineering, University College of Engineering, Nagercoil, Kanyakumari, Tamil Nadu 629 004, India.

\* Corresponding Author.

E-mail: royalmanoj88@gmail.com

\*Orcid Id: 0000-0002-3041-8219

**Abstract**

Vortex generators are widely used for enhancing the nozzle thrust output by creating different effects. The Ranque-Hilsh effect is observed in a swirling nozzle flow within a single tube is a spontaneous separation of total temperature. Here the colder stream near the tube centre line and the hotter air near its periphery. Hence the temperature difference due to the presence of disturbed flow can either increase (or) decrease the thrust output. This analysis is made of the velocity, temperature and pressure distribution in a turbulent vortex with radial and axial flow. The controlled flow expansion by modifying the geometry in the downstream of throat region can yield better shock expansion characteristics. The most important factor affecting the total temperature of a fluid element in a compressible vortex is the turbulent shear work done on or by the element. The parameter which is considered for this optimization of the vortex generator is exit nozzle opening area, Efficiency of Energy Separation, Profile of velocity, temperature, Length of the tube. Further the experimental investigation of output mass flow rate by geometry modification also will be studied. Computational Fluid Dynamics (CFD) based optimization procedure using the Parabolized Navier-Strokes (PNS) equation is used to design axisymmetric nozzle. The advantage of this procedure is that it accounts for viscosity during the design process. Remaining process will make an approximated Boundary Layer Correction after the inviscid design is created. The nozzle design begins with concerning control volume and the separation of Aerodynamic and Propulsive forces and moments will be demonstrated. The appropriate control volume can define the propulsive force vector that maximizes cruise efficiency. The different boundary layer thickness must be evaluated at various temperature and pressure conditions. Then the influence of boundary layer thickness in the net output will be evaluated numerically. The nozzle configuration of a truncated perfect nozzle is selected for computations. The starting process of unsteady flow characteristics and flow separation are observed carefully. The reattachment of the flow through nozzle is also visualized using the computational analysis*.*

**KEYWORDS:** Vortex Mixing, CFD, Flow Separation, Wind Tunnel, Shock Separation, Navier-Strokes Equation, Thrust Augmentation, CFD, Numerical Computation.

**1 Introduction**

The man’s thirst of invention has led to many innovative ideas and things that came to existence. One among them in and around 20th century is the rockets. This has led to the high speed travel and data collection and data transfers. The urge for the reliable and robust technique for practical problems is impossible with the analytical process which has led to the experimental and numerical techniques. The numerical method has more advantage over experimental in the aerospace domain such in rockets and spacecraft designs. This has made the Research scholars turn on to CFD, which has helped us to solve the real world problems in numerical scheme with real time visualization of the practical solution for the real world problems.[1]

Today CFDs principal users are working on high speed flow problems in various flow passages which include nozzles of various components. The performance of the rocket engine is highly dependent on the aerodynamic design of the expansion nozzle. Studies show that bell shaped nozzle contour has more advantage than any other shape of the nozzle contour for the upper stage. The bell shaped contour has been optimized using various techniques like Method of Characterization (MOC), Cubic Spline Equation etc., for opt operating conditions with minimum loss. Turbulence effect is generated with the help of a vortex generator which is an aerodynamic surface, consisting of a small obstruction that creates a vortex. [2]

The effect of turbulence and thickness of the boundary layer were one among the factors in determining the efficient thrust output of a nozzle. The amount of turbulence effect produced without loss of any other parameters will help us to achieve much efficient nozzle and this is one of the key features which are evaluated in this paper. The Rocket Nozzle performance is optimized with the help of flow separation control.

**2 Isentropic Flow**

An isentropic flow is a flow process that is both adiabatic and reversible. For an isentropic flow of a perfect gas, several relations can be derived for pressure, density and temperature along a streamline. The simple results obtained with the constant specific heat approximation are useful even for large temperature changes, so long as we use appropriate average values of and γ.

 For constant specific heat

 (1)

which, becomes

 (2)

Since. We can write as

 (3)

From perfect gas law,

 (4)

The mass flow per unit area is

 (5)

The area ratio can written as

 (6)

For a given isentropic flow it is clear that is a constant, so that we can normalize the actual flow area A.

**2.1 Shocks**

 A shock is a discontinuity in a (partly) supersonic flow field. Fluid crossing a stationary shock front rises suddenly and irreversibly in pressure and decreases in velocity. It also changes its direction except when passing through a shock that is perpendicular (normal) to the approaching flow direction. Such plane normal shocks are the easiest to analyze.

**2.1.1 Normal shocks**

For flow through a normal shock, with no direction change, area change, or work done, are:

Continuity: (7)

Momentum: (8)

Energy: (9)

**2.1.2 Oblique shock relation**

An oblique shock wave, unlike a normal shock, will occur when a supersonic flow encounters a corner that effectively turns the flow into itself and compresses. The simple way to produce an oblique shock wave is to place some disturbance into a supersonic, compressible flow.

The oblique shock relations describe the deflection of flow over a shock, see fig.1. The two angles which describe the shock are the wave angle β and flow-deflection angle θ.

****

Fig.1 Oblique shock on free stream flow

Using the continuity equation for the tangential velocity component which does not change across the shock, trigonometric relations eventually lead to the θ-β-M equation which shows θ as a function of M1 and β.

The rise in pressure, density, and temperature after an oblique shock can be calculated as follows:

 (10)

 (11)

 (12)

 (13)

Equation (13) cannot be solved until the flow-deflection angle θ is known. By setting the tangential velocities equal and working on with some trigonometric manipulations the θ-β-M relation can be found:

 (14)

**3 ANALYTICAL METHOD**

 Designing an Ideally operating Liquid Rocket engine nozzle (bell-shaped nozzle) for various operating ambient pressure, as a part of designing, from M. Al-Ajlouni1 and Reid Britton Young2 work it is clear that for start a nozzle design we need to solve Method of Characteristics analytical method. This method is widely used and accepted standard one for ideal nozzle design. From Ref 10, MOC can be evaluated by solving the basic equation which relates Mach angle with Mach number. The equations are as follows:

 (15)

In the above equation the Mach angle being the variable which is to be calculated. But it can’t be calculated as easily, so this is calculated with the aid of little interpolation using the relation

 (16)

Now this relation can be broken with the help of Mach number relation as follows

 (17)

With the aid of computer we can solve this problem the computed output will be the x, y coordinates of a profile which can be used for optimization of the various parameters.



Fig. 2 various views of Nozzle in Catia

As the three dimensional analysis will take much time for computation the model is designed in two dimensional form and meshed. The meshing of the geometry is studied with the help of Ansys Fluent 14. The two dimensional view of the nozzle is shown in the figure from the Ansys 14 workbench geometry modeling window.



Fig. 3 Nozzle with generated points in 2D

**BOUNDARY LAYER**

The boundary layer is the layer of fluid in the immediate vicinity of a bounding surface where the effects of viscosity are significant.



Fig.4 Boundary layer transition

As a consequence of intense mixing a turbulent boundary layer has a steep gradient of velocity at the wall and therefore a large shear stress. In addition to shear stress, heat transfer rates are also high. Typical laminar and turbulent boundary layer profiles are shown in fig.4. For a flat plate it is given by

 (18)

where Rex is the Reynolds Number based on the length of the plate. For a turbulent flow it is given by

 (19)

In actuality there is no sharp “edge” to the boundary layer; that is u→*U* as we get farther. We define the *boundary layer thickness,* δ, as the distance from the plate at which the fluid velocity is within some arbitrary value of the upstream velocity.

δ=y where u=0.99U (20)

To remove this arbitrariness the following definitions are introduced. The velocity profiles for flow past a flat plate-one if there were no viscosity (a uniform profile) and the other if there are viscosity and zero slip at the wall (the boundary layer profile). Because of the velocity deficit, U→u, within the boundary layer. However, if we displace the plate by an appropriate amount δ\*, the boundary layer displacement thickness, the flow rates across each section will be identical. This is true if.

 (21)

where b is the plate width. Thus,

 (22)

The displacement thickness represents the amount that the thickness of the must be increased so that the fictitious uniform inviscid flow has the same mass flow rate properties as the actual viscous flow. It represents the outward displacement of the streamline caused by the viscous effects on the plate. This idea allows us to simulate the presence that the boundary layer has on the flow outside of the boundary layer by adding the displacement thickness to the actual wall and treating the flow over the thickness body as an inviscid flow.

**NOZZLE**

 The upper stage of the space mission is currently using the liquid or cryogenic propellant, for the full expansion of the combustion product the bell shaped nozzle is preferred. The bell shaped nozzle has high expansion ratio and can work effectively in vacuum. The existing bell nozzle contour is studied for the boundary layer growth along the wall. The modification on various parameters for turbulence mixing is studied such as rugged wall, introduction of varying thickness and corner fillet is studied for effective correction of boundary layer. The fig.5 shows the nozzle contour of scaled (1:2) nozzle.

Fig.5 Contour of Bell Shaped Rocket Nozzle

For all flows, ANSYS FLUENT solves conservation equations for mass and momentum. For flows involving heat transfer or compressibility, an additional equation for energy conservation is solved by coupling it with the Navier Strokes equation.

**The Mass Conservation Equation**

The equation for conservation of mass, or continuity equation, can be written as follows:

 (23)

Equation (23) is the general form of the mass conservation equation and is valid for incompressible as well as compressible flows. The source   is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources. For 2D axisymmetric geometries, the continuity equation is given by

 (24)

Where x is the axial coordinate, r is the radial coordinate,   is the axial velocity, and   is the radial velocity.

**Numerical Calculation**

 The numerical calculation is carried out with the help of Ansys software, the initial condition are taken from the standard cryogenic CE-20 rocket nozzle. The inlet pressure of 20 MPa, temperature of 1200 K and outlet of 0 MPa. The results are imported to excel file and the data is plotted below, the calculation shown in the appendix 1.

Fig.6 Pressure Distribution plot for a Bell nozzle Ref Appendix 1.

The pressure distribution of a bell shaped nozzle along the axis of the nozzle is computed. The above plot shows the variation of Cp vs x/c, from the graph we can infer that maximum coefficient of pressure is at the upstream of the nozzle and it gets nearly to zero at the x/c value of around 0.1 percentage of the full nozzle length from the throat.

Fig.7 Reynolds number variation along the axis Ref Appendix 2.

From the plot we can see the rise of Reynolds number as the axial distance grows till the exit. Since Reynolds number is the function of velocity component so it rises as the flow expands. Initially we have assumed the flow is having very low velocity but with very high pressure which is assumed to the combustion chamber pressure.

Fig.8 Reynolds number variation along the axis for the pressure Ref Appendix 2.

The plot shows the variation of the Reynolds number variation for the pressure; the Reynolds number cannot be shown in the graph for its variation since it is variation is too small to plot till throat.

Fig.9 Displacement Thickness Plot for a Bell nozzle Ref Appendix 4.

The typical contour is bell shaped nozzle which has maximum performance in the outer space. The nozzle is evaluated with the boundary layer interaction along the wall. The growth of displacement thickness along the wall has been significantly decreased but when maximum velocity is reached the growth can been seen from the plot. This growth will decrease the thrust, so the region along the wall is modified with small protrusion as per International Standards along with secondary flow injection of 100 m/s at 33 percent of the throat.

Fig.10 Momentum Thickness plot for a Bell nozzle Ref Appendix 4.

The bell shaped rocket nozzle has maximum performance in the outer space based on the complete expansion of the exhaust fumes. The nozzle is evaluated with the boundary layer interaction along the wall for reduction in loss due to wall friction. The growth of momentum thickness along the wall has been significantly decreased but when maximum velocity is reached the growth can been seen from the plot. This growth will decrease the thrust, so the region along the wall is modified with small protrusion as per International Standards along with secondary flow injection of 100 m/s at 33 percent of the throat.

Fig.11 Pressure Distribution plot for a Scaled Bell nozzle tested in wind tunnel Ref Appendix 6.

Scaled nozzle with scale ratio of 1:20 is tested in the subsonic wind tunnel for its expansion ratio. The result obtained is for twelve pressure tapping’s from the throat of the nozzle which are equally spaced. The inlet velocity of the subsonic wind tunnel is 25 m/s with the rpm of 1200. The manometer readings are noted and tabulated in the appendix table 1.

**Nozzle with corrugated wall at the near exit plane**

Fig.12 Pressure Distribution plot for a modified Bell nozzle Ref Appendix .

The nozzle with corrugated wall has significant drop in coefficient of pressure which will enhance the overall exhaust velocity. This is achieved with the reduction in the area near the exhaust along with the expansion as that of normal nozzle.

Fig.13 Pressure distribution for various models Ref Appendix 6.

Plot shows the variation of the three nozzle contour of straight, corrugate and secondary injection for pressure distribution. The modified nozzle has initial high value of coefficient of pressure which is gradually reduced same as other two configuration when it 0.1 percentage of the nozzle.

Fig.14 Displacement thickness for secondary injection of 100 m/s Ref Appendix 5

The nozzle is evaluated with the boundary layer interaction along the wall for reduction in loss due to wall friction. The growth of displacement thickness along the wall has been significantly decreased but when maximum velocity is reached the growth can been seen from the plot. This growth will decrease the thrust, so the region along the wall is modified with small protrusion as per International Standards along with secondary flow injection of 100 m/s at 33 percent of the throat.

Fig.15 Momentum thickness for secondary injection of 100 m/s Ref Appendix 5

The bell shaped rocket nozzle has maximum performance in the outer space based on the complete expansion of the exhaust fumes. The nozzle is evaluated with the boundary layer interaction along the wall for reduction in loss due to wall friction. The growth of momentum thickness along the wall has been significantly decreased but when maximum velocity is reached the growth can been seen from the plot. This growth will decrease the thrust, so the region along the wall is modified with small protrusion as per International Standards along with secondary flow injection of 100 m/s at 33 percent of the throat.

**CONCLUSION**

Thrust enhancement in the nozzle with the correction in boundary layer is one of the current techniques in outer space propulsion. The typical contour is bell shaped nozzle which has maximum performance in the outer space. The nozzle is evaluated with the boundary layer interaction along the wall. The growth of displacement thickness along the wall has been significantly decreased but when maximum velocity is reached the growth is significant. This growth will decrease the thrust, so the region along the wall is modified with small protrusion as per International Standards. The boundary layer growth has linearized initially and then a steep growth is accomplished. Thus thrust has enhanced with sudden growth in the boundary layer by the conservation of mass.

The growth of momentum thickness along the wall has been significantly decreased but when maximum velocity is reached the growth is significant from the results. This growth will decrease the thrust, so the region along the wall is modified with small protrusion as per International Standards along with secondary flow injection of 100 m/s at 33 percent of the throat. This has significantly increased the overall thrust output of the nozzle.

The method of characteristics (MOC) is used to design a nozzle for varying exit Mach number operating condition. The pressure distribution of a bell shaped nozzle along the axis of the nozzle is computed. The maximum coefficient of pressure is at the inlet of the nozzle and it gets nearly to zero at the x/c value of around 0.1 percentage of the full nozzle length.

## REFERENCES

1. M. Al-Ajlouni (2010) “An Automatic Method for Creating the Profile of Supersonic Convergent Divergent Nozzleˮ, Jordan Journal of Mechanical and Industrial Engineering, *4*(3).
2. Vanco, M.R. and Goldman, L.J., 1968. Computer Program for Design of Two-dimensional Suoersonic Nozzle with Sharp-edged Throat. NASA.
3. Young, R. and Hartfield, R., 2012, August. Automated nozzle design through axis-symmetric method of characteristics coupled with chemical kinetics. In 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit (p. 4162).
4. Hancock, P. E., and Bradshaw, P., (Sept. 1983) "The Effect of Free-Stream Turbulence on Turbulent Boundary Layers," Transactions of the ASME, Journal of Fluids Engineering, Vol. 105.
5. Bandyopadhyay, P.R., 1992. Reynolds number dependence of the freestream turbulence effects on turbulent boundary layers. AIAA journal, 30(7), pp.1910-1912.
6. Blair, M. F., and Werle, M. J., (Sept. 1980) "The Influence of Free-Stream Turbulence on the Zero Pressure Gradient Fully Turbulent Boundary Layer," United Technologies Research Center, Rept. R80-914388-12.
7. Sundaram, S. and Yajnik, K.S., 1992. Experimental study on the evolution of a wall layer from a wake. AIAA journal, 30(12), pp.2845-2851.
8. Bandyopadhyay, P.R. and Ahmed, A., 1993. Turbulent boundary layers subjected to multiple curvatures and pressure gradients. Journal of Fluid Mechanics, 246, pp.503-527.
9. Bandyopadhyay, P.R., 1987. Rough-wall turbulent boundary layers in the transition regime. Journal of Fluid Mechanics, 180, pp.231-266.
10. Anders, J.B., 1990. Boundary layer manipulators at high Reynolds numbers. In Structure of Turbulence and Drag Reduction (pp. 475-482). Springer, Berlin, Heidelberg.
11. Gillis, J.C., Johnston, J.P., Kays, W.M. and Moffat, R.J., 1980. Turbulent boundary layer on a convex, curved surface (No. HMT-31).
12. Meadows, K.R., Kumar, A. and Hussaini, M.Y., 1991. Computational study on the interaction between a vortex and a shock wave. AIAA journal, 29(2), pp.174-179.
13. Callis, L.B., 1966. An analysis of supersonic flow phenomena in conical nozzles by a method of characteristics (No. NASA-TN-D-3550).23.
14. Beckwith, I.E. and Moore, J.A., 1955. An accurate and rapid method for the design of supersonic nozzles (No. NACA-TN-3322).
15. Amick, J.L. and Hays, P.B., 1960. Interaction effects of side jets issuing from flat plates and cylinders alined with a supersonic stream (Vol. 60, No. 329). Wright Air Development Division, Air Research and Development Command, United States Air Force.