NUMERICAL ANALYSIS OF CONE TYPE ROCKET NOZZLE

[1]BEENA D. BALONI; [2]SONU P. KUMAR; [3]SALIM A. CHANNIWALA

[1, 2, 3] Mechanical Department, SVNIT- Surat, India
E-mail: [1]pbr@med.svnit.ac.in; [2]sonupkumar007@gmail.com; [3]sac@med.svnit.ac.in

Abstract- The basic principle of a rocket engine is based on the Newtonian third principle that "to every action there is an equal and opposite reaction". A rocket engine is throwing mass in one direction and benefiting from the reaction that occurs in the other direction. The hot pressurized gas passing through the nozzle converted to a supersonic speed as the expansion takes place so that the heat energy of propelling the flow is maximally converted into directed kinetic energy. Because of this, it is an essential part of the modern rocket engine. Cone type nozzle is simple and easy to construct amongst the various type of rocket nozzles. The present work incorporates 2D axisymmetric flow analysis within the cone type nozzle, at design and off-design condition, by using computational fluid dynamic software GAMBIT 2.4.6 and FLUENT 6.3.26.Shear Stress Transport k-ω turbulence model has been chosen for flow analysis. The result shows the distribution of the Mach number, pressure, temperature and velocity inside the cone type rocket nozzle. FLUENT predictions are used to verify the isentropic flow assumption, and that the working fluid reached the user defined the desired exit Mach number. Good agreement in the area ratio and the exit Mach number were achieved.

Keywords - Rocket Nozzle, Cone type Nozzle, Numerical Analysis, Compressive Waves (Shock Wave)

I. INTRODUCTION

The rocket is a device that stores its own propellant mass and expels this mass at high velocity to provide a reaction force, the thrust. It mainly comprises of combustion chamber and nozzle. The combustion chamber is a constant duct with sufficient length to allow complete combustion of propellants before the nozzle accelerates the gas products. A rocket nozzle is a propelling nozzle (usually of the de Laval type) used in a rocket engine to expand and accelerate the combustion gases produced by burning propellants. The exhaust gases exit the nozzle at hypersonic velocities. It is used to give the direction to the gases coming out of the combustion chamber.

Generally,a nozzle is a tube with a variable cross-sectional area and is used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the exhaust stream that emerges from them. Gas flows from the chamber into the converging portion of a nozzle, past the throat, through the diverging portion and then exhausts into the ambient as a jet. Pressure of ambient is generally referred to as design back pressure. To accelerate the air, a favorable pressure gradient must be exerted across the nozzle. The nozzle is over-expanded ($P_e < P_a$) at lower altitudes than design altitude and under-expanded ($P_e > P_a$) at higher altitudes than design altitude. When the rocket engine operates in off-design conditions, compression and expansion waves are formed around the exhaust jet, with consequent density discontinuities, which gradually achieve a match between the pressure in the jet and the ambient pressure.

Conical nozzle walls diverge at a constant angle (α). A small angle produces greater thrust, because it maximizes the axial component of exit velocity and produces a high specific impulse. The main penalty of this type of nozzle is a longer and heavier nozzle that is more complex to build. At the other extreme, size and weight are minimized by a large nozzle wall angle. However, large cone angles reduce performance at low altitude because high ambient pressure causes overexpansion and flow separation. There is an optimum conical nozzle shape and length (typically between 12° and 18° half angle) and it is usually a compromise which depends on the specific application and flight path [1].

Sutton G P [1], described that, for a rocket nozzle with a divergence cone angle of 30° (half angle of 15°), the exit momentum and therefore the exhaust velocity will be 98.3% of the velocity.An investigation was conducted to obtain nozzle performance data with relatively large-scale models at pressure ratios as high as 120 by O. E. Campbell et al. [2].Conical convergent-divergent nozzles with divergence angles of 15°, 25°and 29° were each tested at area ratios of approximately 10, 25 and 40.They concluded that, at a given surface area ratio, decrease in α from 29° to 15° increases the thrust ratio about 3% (for vacuum), 6% (for the pressure ratio of 200).

Also stated that the design thrust ratio decreased with an increase in nozzle divergence angle according to the parameter $(1 + cos\alpha)/2$.The efficiency of the conical nozzle increased as we increase the divergence angle of the nozzle up to certain extent and then decrease [3].Literature suggests that in rocket application sharp corners downstream the throat was generally avoided due to chemical kinetics effects and a wall contour, having a radius of curvature equal to 0.5 times the throat radius, were widely used [4]. Investigation of the performance of different turbulence models in numerical simulations

of two dimensional convergent divergent nozzle flow fields at over expanded conditions suggested that an algebraic turbulence models predicted the shock location downstream of experimental position and one equation model predicted upstream. Thus, thetwo equation model gave the best overall agreement with experimental pressure distribution at over expanded condition [5]. However, Ostlund et al. [6] had shown that without corrections, the standard two-equation turbulence model fails to predict the measured separation characteristics in the nozzle. In order to increase the reliability of design results, viscous losses should be induced by using the Navier-Stokes equations. The design code based on the Navier-Stokes equations may produce more accurate design result [7].Xin H. zou et al. [8], had done comparative analysis of the two-dimensional (2-D) nozzle and an axisymmetric nozzle on the aerodynamic characteristic by numerical simulation. He concluded that the thrust vector angles decreased with an increase in the nozzle pressure ratio (NPR).

Numerical study had been conducted to understand the air flows in a conical type rocket nozzle at the corresponding inlet condition using two-dimensional axi-symmetric models in present paper. The governing equations were solved by a control volume method. The method of characteristics, which usually designed for the De-Laval nozzle, were used to take the nozzle geometry co-ordinates. The flow is simulated using fluent software. The flow parameters like pressure, temperature and area of a nozzle at the exit are defined prior to the simulation. The design and off design simulations are done with the cone type rocket nozzle for both cold and hot flow process.

II. NUMERICAL SIMULATION

To clearly understand the cone type rocket nozzle performance under different flow conditions the numerical simulation is carried out using computational fluid dynamic software GAMBIT 2.4.6 and FLUENT 6.3.26. For the analysis purpose,the dimensional specifications for the cone nozzle is taken from the literature [3] and details for the same is represented in Table I.

Details	Dimension	Details	Dimension
Inlet Diameter	0.22 m	Divergent length	2.02 m
Throat Diameter	0.128 m	Divergent angle	15°
Exit Diameter	1.12 m	Nozzle length	2.66 m
Convergent length	0.64 m		

Table I Specifications of cone type rocket nozzle

The computational domain is modeled in GAMBIT according to the dimension's geometrical specifications discussed in TableI. Bottom- up approach is applied to create the fluid domain of nozzle geometry. Model geometry is meshed with structured grid consists of 25600 mesh faces. To have a good quality mesh it is required to split the computational domain in to the number of parts (faces), which allows a better mesh with low skewness. The nozzle geometry is divided into the number of zones and boundary conditions are applied. The inlet condition is given as a pressure inlet and the nozzle outlet is given as pressure outlet. Nozzle throat is given as an interior and nozzle surfaces are given as a wall. Figure 1 shows the meshed nozzle geometries with boundary conditions (specified in brackets).

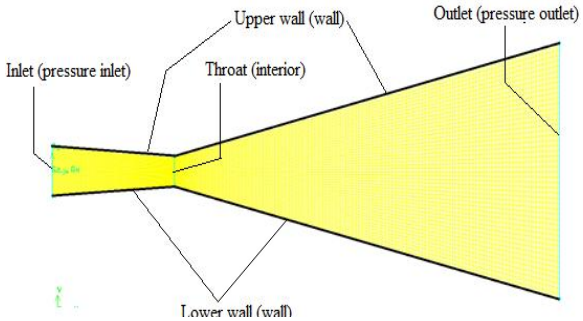


Figure 1Meshed fluid domain with boundary conditions of a cone nozzle

Meshed file created in the GAMBIT 2.4.6 modeler is imported in commercial CFD package FLUENT 6.3.26 and flow simulation is carried out. This CFD packages workon the basis of finite volume approach. The code Fluent has been used to solve 2-D steady state Reynolds-averaged Navier-Stokes (RANS) equations in a cone type nozzle for different flow conditions. The Shear Stress Transport k-ω (SST k-ω) model is used to simulate flow through these nozzles. From accuracy and convergence consideration, the second order upwind scheme was used for all the conservation equations after getting the convergence with the first order upwind scheme. Grid Adaption based on the pressure gradient criterion was also invoked in order to capture the flow discontinuities accurately. With grid Adaption, final solutions in most of the cases were obtained on grids that had 2 to 3 times the original number of cells. The convergence criteria for the residuals are set to e^{-4}. Once the solution gets converged, the contours of pressure, temperature, velocity and the Mach number are plotted.

III. RESULTS AND DISCUSSION

Numerical simulation of 2D cone type nozzle is done at design pressure and off-design conditions for cold as well as hot flow conditions. The term design pressure condition indicates that the nozzle is operating at its design exit pressure, which gives a shock free isentropic flow. For the cold flow analysis,

the air is isentropically expands to a design pressure of 1 atm with a stagnation temperature of 300K. Whereas; in the hot flow analysis, the Hydrogen gas at 1000K is expanded to 1 atm pressure. Off-design condition is considering, the flow through the nozzle at the over-expanded condition with a shock wave inside the diverging section of the nozzle. In the off design analysis, the nozzle is operating at the back pressure value of 15 bar (for both cold and hot flow). Based on numerical simulations at design and off-design conditions, results are plotted in terms of variations in fluid properties like pressure, temperature, velocity and Mach number throughout the nozzle length. The results of simulation are compared with the analytical calculation based on 1D theory of compressible flow.

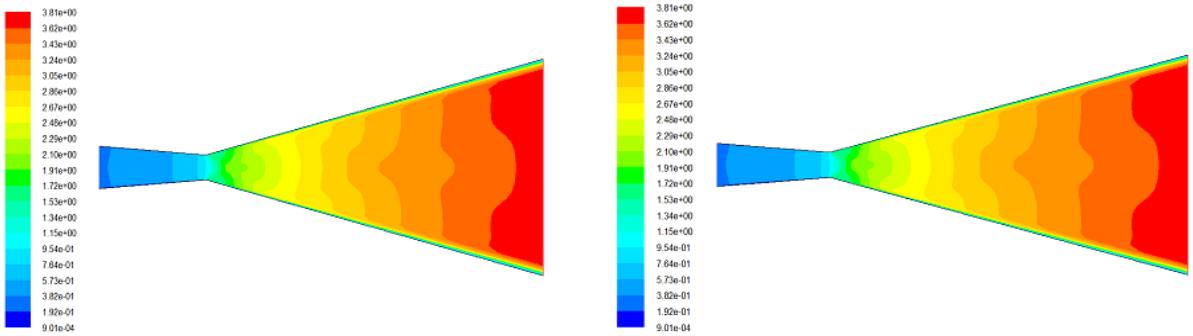
At first, the numerical and analytical analysis of a cone nozzle is carried out at design conditions for cold and hot flow. Figure 2 represents the Mach number contours,at the design pressure, for cold and hot flow. The contours reveal that; the flow is subsonic in the convergent portion having the value of Mach number 0.361. At the throat a Mach number of 1.0 and in the divergent portion the flow becomes supersonic with the Mach number of 3.65 and 3.64 at an exit for cold and hot flow respectively. This represents the complete expansion of flow. Details of flow properties, at different locations of nozzle, are represented in Table 2 and Table 3. From the Table 2, it is clear that the value of static pressure from numerical analysis is 3.8% and 3% higher than the analytical result of the inlet and outlet respectively. This is due to the ideal condition, i.e. isentropic flow (the assumption of constant stagnation pressure throughout the nozzle) in case of analytical analysis whereas; the simulation results based on the adiabatic flow and shows loss of stagnation pressure throughout the nozzle. Simulation results suggest, 4.3% of stagnation pressure loss in the nozzle. The exit Mach number shows a 3.23% lower value with a Mach number of 3.65 in case of simulation compared to the analytical value of 3.8. In case of hot flow, the Mach number of 3.64 is achieved at an exit, which is 4.31% lower than the designed value. The numerical results of hot flow analysis suggest an acceleration of flow to a velocity of 4484.04 m/s with an outlet temperature of 267.55 K and 5.41% of stagnation pressure loss in the nozzle. Results reveals that the flow is continuously accelerating throughout the nozzle, therefore the pressure gets reduced from an inlet to an outlet of the nozzle. In both cold and hot flow analysis, a good agreement is observed between analytical and numerical results.

The off-design performance nozzle is also analyzed in both analytical and numerical way. In this analysis, the nozzle is operating at 15 bar back pressure (for cold flow, NPR = 7.565 and for hot flow, NPR = 7.677). The off design performance of the nozzle will create a shock inside the nozzle, and due to its presence, variations of flow properties are observed. With the assumption of steady one dimensional flow through the nozzle, the detail calculations are carried out based on the theory of shocks. From the theoretical point of view to find the location and properties at the upstream and downstream of shock, considering that flow is choked. Therefore, the mass flow rate before and after the shocks are same (constant). The Numerical simulation of the cone nozzle at off design condition is carried out with back pressure value 15 bar. This change in exit pressure from the design exit pressure (i.e. atm pressure), create an overexpansion flow through the nozzle. As a result, there is a shock wave occurs in the diverging section of the nozzle.

Figure 3 shows the Mach number contours in the nozzles for the cold and hot flow at 15 bar back pressure. One can observe clear shock formation, at the diverging section of the nozzle. The variation of Mach number indicates continuous increase in Mach number up to the shock formation. A maximum of Mach number 3.59 is reached upstream of the shock wave. At this location, due to the shock formation supersonic Mach number is converted from 3.59 to subsonic mach number 0.45. The mach number contour clearly describe the location of the shock wave at x = 2.375 m at diameter 0.98 m and x = 2.3 m at diameter 0.943 m in cold and hot flow respectively (Refer: Fig. 4). This suggests that there is minor change occurs in the shock position due to change of fluid and temperature in the present case of cone nozzle. The shock is the combination of oblique and normal shocks, flow near the nozzle walls interacted with an oblique shock whereas; the center flow is affected with normal shock. The flow near the wall gets separated due to the shock wave boundary layer interaction. The detached flow pattern is shown in Fig. 3.

Figure 5-a) represents the contours of static pressure along the nozzle length. One can observe an acceleration of flow due to the reduction in static pressure up to the shock formation. However, the energy transformation rate is very less after one fourth part of the divergent section till the shock location. Due to high exit pressure than design one, the flow is not able to accelerate the further and to cop up with the exit condition it adjusted within itself by mean of shock formation. The shock is converted high velocity supersonic flow to the subsonic one and in this process flow compresses. After a normal shock the flow converted into subsonic, and a nozzle behaves as a diffuser which increases the pressure in the rest of the nozzle length. Static pressure is 1.15 bar before the shock and boost around18 bar aftershock (Refer:Fig.5-a) in case of cold flow. Whereas; static pressure of 1.22 bar is reached just in front of the shock, across the shock the pressure is increased to a value around 19 bar (Refer: Fig.5-b).

To further validation of simulation results, the simulation results are compared with Analytical results as given in Table 4. One can observe that for 15 bar back pressure, there is a shock wave is developed at an upstream Mach number of 3.85 (analytical), from numerical simulation a Mach number of 3.59 is achieved, which is 6.75 % lower than the theoretical value in case of cold flow. Due to the difference in the Mach number, the location of shock is also observed different in both analysis, the shock wave is captured at 1.198 m^2 in analytical analysis, whereas; simulation results locate the shock at 0.98 m^2, an 18.1% difference is observed for cold flow. In case of hot flow analysis, flow proceeds to the nozzle isentropically to a Mach number of 3.59, where a shock is developed which diffuses the flow to a 0.45 Mach number. But, analytical analysis shows a 3.88 Mach number upstream and a 0.44 Mach number downstream of shock, which is 7.4% lower (upstream) and 2.2 % higher than the theoretical value. There is a 21 % difference is observed between numerical and analytical values of the area at which shock is captured. The difference between numerical and theoretical analysis is because of absence of flow interaction between the boundary layer and shock effect in theoretical analysis.



a) For cold flow **b) For hot flow**

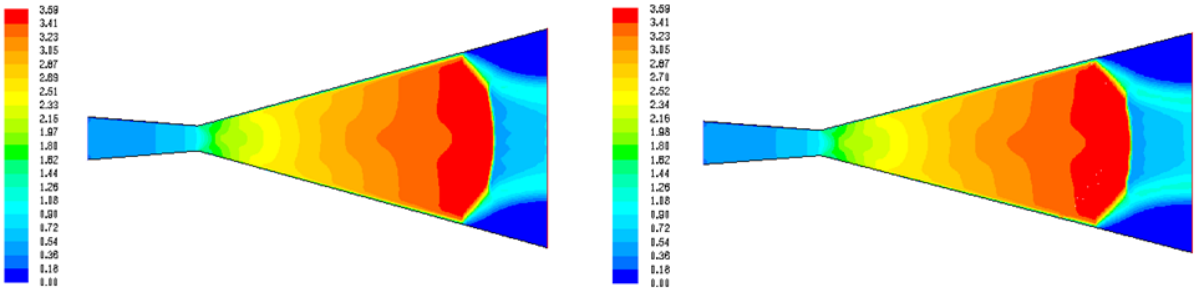
Fig. 2 Contours of the Mach number inside fluid domain at design condition

Properties	Inlet		Throat		Outlet	
	Analytical	Numerical	Analytical	Numerical	Analytical	Numerical
Static Pressure, P (bar)	103.55	103.59	59.95	59.84	1.013	1.045
Stagnation Pressure, P_o (bar)	113.49	113.49	113.49	113.31	113.49	108.52
Static Temperature, T (K)	292.24	292.321	250	249.61	77.92	82.23
Velocity, V (m/s)	124.8	123.95	316.93	317.32	667.95	652.94
Mach number, M	0.364	0.3618	1.0	1.0	3.8	3.65

Table II The flow properties of a Cone nozzle at the different section for cold flow (T_0 = 300 K)

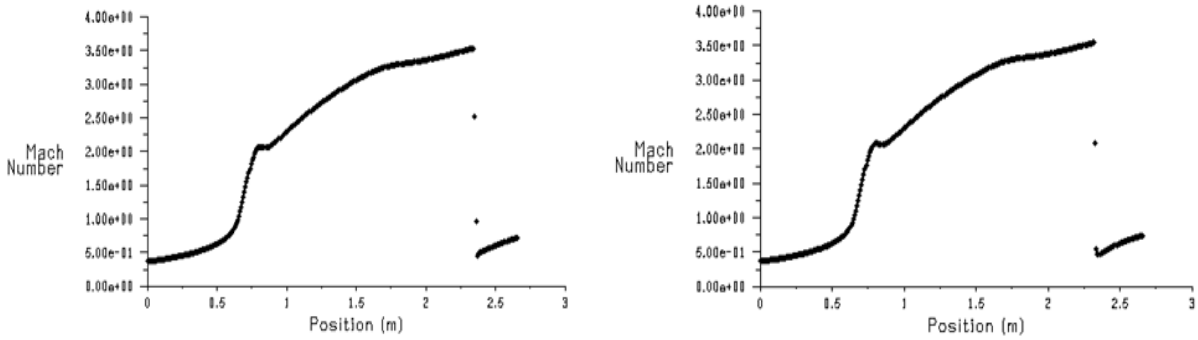
Properties	Inlet		Throat		Outlet	
	Analytical	Numerical	Analytical	Numerical	Analytical	Numerical
Static Pressure, P (bar)	105.03	105.08	60.64	60.60	1.013	1.063
Stagnation Pressure, P_o (bar)	115.15	115.15	115.15	114.86	115.15	108.92
Static Temperature, T (K)	973.58	974.02	829.875	829.69	252.51	267.55
Velocity, V (m/s)	869.17	858.026	2205.78	2197.465	4623.59	4484.043
Mach number, M	0.363	0.361	1.0	1.0	3.8	3.64

Table III Flow properties of a Cone nozzle at the different section for hot flow (T_0 = 1000 K)



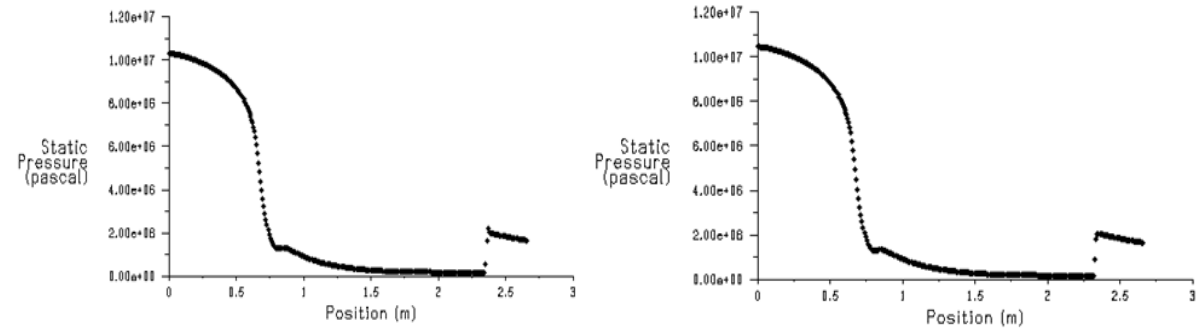
a) For cold flow **b) For hot flow**

Fig. 3 Contours of the Mach number inside fluid domain at off-design condition



a) For cold flow **b) For hot flow**

Fig. 4 Variations in the Mach number inside the fluid domain at the off-design condition



a) For cold flow **b) For hot flow**

Fig. 5 Variations in static pressure inside the fluid domain at the off-design condition

Properties	For cold flow Air at 300K ($\gamma = 1.4$)		For hot flow Hydrogen at 1000K ($\gamma = 1.44$)	
	Analytical	Numerical	Analytical	Numerical
Exit Mach number, M_2	0.49	0.454	0.52	0.454
Upstream Mach number of shock, M_x	3.85	3.59	3.88	3.59
Downstream Mach number of shock, M_y	0.44	0.45	0.44	0.45
Upstream stagnation pressure, bar	113.49	113.87	115.15	115.4
Downstream stagnation pressure, bar	17.68	18.1	18.06	18.35
Shock location A_x, m^2	1.198	0.98	1.194	0.943

Table 4Flow properties of a cone nozzle at the different section at off design condition

IV. CONCLUSION

The numerical and analytical analysis of a cone type rocket nozzle is carried out at design and off-design condition for cold and hot flow. The following conclusions are made based on the analysis:

1. At design pressure, the cone nozzle gives a good agreement with the theoretical one dimensional analysis. The exit Mach number results from the numerical analysis shows only a 3.23 % and 4.31 % difference from the analytical result for the cold flow and hot flow analyses respectively.
2. In the off design analysis of the cone nozzle, a combination of oblique and normal shock wave is captured in the diverging section under theexit pressure value of 15 bar.
3. There is a significant variation is observed between the theoretical and simulation results in case of location of shock wave. The shock wave predicted sits at smaller diameter than that estimated from the analytical study, because the numerical analysis is carried out for the two dimensional flow, but in the analytical analysis is based on the assumption of one dimensional adiabatic frictionless steady flow.

REFERENCES

[1] George P. Sutton and Oscar Biblarz, "Rocket Propulsion Elements", John Wiley & Sons, Eighth Edition,2011, Ch. 3.

[2] O.E.Campbell and J. M. Farley, “Performance of Several Conical Convergent-Divergent Rocket Type Exhaust Nozzles”, NASA, 1960.

[3] Pardhasaradhi Natta, V.Ranjith Kumar and Y.V.Hanumantha Rao, "Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics", International Journal of Engineering Research and Applications (IJERA), Vol. 2, Issue 5, 2012.

[4] Jan Ostlund, "Flow Process in Rocket Engine Nozzles with Focus on Flow Separation and Side-Loads", Licentiate Thesis, Stockholm, 2002.

[5] A Hamed and C Vogiatzi, "Assessment of turbulence models in over expanded 2D CD nozzle flow simulations", AIAA, Joint Propulsion Conference and Exhibit, 1995.

[6] Jan Ostlund and Jaran M, "Assessment of turbulence models in over expanded Rocket Nozzle Flow Simulations", Presented at 35th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, AIAA Paper 99-2583, 1999.

[7] M.Yumusak and S. Eyi, "Design Optimization of Rocket Nozzles in Chemically Reacting Flows", Computers & Fluids, 2012.

[8] Xin H. Zou, Qiang Wang, "The Comparative Analysis of Two Typical Fluidic Thrust Vectoring Exhaust Nozzles on Aerodynamic Characteristics", 2011.

★★★