Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2019
(ICMERE2019) 11 – 13 December, 2019, Chittagong, Bangladesh

ICMERE2019-PI-110

Investigating the Effect of Various Angles of Divergence of A Conical Rocket Nozzle During Boost Phase

T. Islam[1], M. S. B. Zaman [2], M. E. Hoque [1], F. Rashid[1]
[1]Department of Mechanical Engineering, RUET, Rajshahi-6204, Bangladesh
[2]Department of Mechanical and Chemical Engineering, IUT, Gazipur-1704, Bangladesh

tariqul132110@gmail.com

Abstract *- : The space industry has made progress beyond leaps and bounds in the last 70 years. For decades, engineers and rocket scientists have worked tirelessly to design more efficient and high-quality rockets. One significant component of rockets still under development and innovation is the nozzle itself. The nozzle has undergone significant changes from having a convergent shape to a converging-diverging shape we see today. In designing nozzles, one crucial aspect of the design is the divergence angle. This angle occurs at the diverging section of the nozzle just beyond the throat. This angle has to be carefully selected to ensure maximum Mach number hence maximum thrust, ensuring the rocket lifts off during its boost phase. The study undertaken investigates how the various parameters related to thrust vary as the divergence angle changes by conducting a series of CFD simulations. The best angle is selected based on the optimum thrust obtained.*

Keywords: Mach number, CFD, converging-diverging nozzle

1. INTRODUCTION

The first nozzle had been developed in the year 1883, by the engineer Carl G.P de Laval [1]. This is also known as the de Laval nozzle and is one of the most widely and commonly used nozzles. Bell nozzles, conical nozzles, converging-diverging nozzle (also called the de Laval nozzle), and the aerospike are some of the nozzles that have been studied and investigated over the years [2].
NASA spent over 500 million dollars on nozzle experiments from the year 1950-1970 [2]. In recent times, more work has been done to improve the shape and design of nozzles, in particular the converging-diverging shape of the nozzle and its angle of divergence [3]. Computational fluid dynamics (CFD) plays a vital role in visualizing processes where transport occurs with an exchange of heat and energy particularly in the performance of nozzles [4]. In literature, extensive work has been done by various researchers, who have investigated the effect of divergence angle on a number of nozzle performance indicators, such as Mach number, outlet pressure, maximum outlet velocity etc. Ande et al. [5] studied four different divergent angles (9^o,12^o,15^o and 18^o) and their effect on Mach number and static pressure on a converging-diverging nozzle. The results showed an ideal divergent angle of 15o based on highest Mach number. Bhagat et al. [3] investigated the effect of nozzle divergence angle on under-expanded plumes in the rarefied slip flow regime i.e., at an altitude of 80 km using the OPEN FOAM software. Using the ANSYS FLUENT module, Kuttan et al. [4] studied the formation of oblique shocks on nozzles with divergent angles of 4°,7°, 10°, 13°, 15°. A similar study was carried out by Patel [6], where he studied how the angle of divergence influences formation of shock and attainment of maximum velocity, hence thrust. Vishnu et al. [7] explained the importance of nozzle design and investigated the shock structure at different divergence angles at two different pressure ratios. Mason et al. [8] conducted experimental studies conducted experiments to determine the effect of throat contouring on the nozzle internal performance. The focus of this paper will be the study of different angles of divergence and subsequently the effect that different divergence angles have on certain properties of the flow using ANSYS FLUENT. These properties include total thrust and Mach number. With our simulation results, we will predict the optimum angle of divergence based on the maximum thrust obtained.

2 MODEL DEVELOPMENT

For numerical analysis, CFD technique coupled with heat transfer was applied. The entire analysis was conducted using ANSYS FLUENT software. The turbulent model used was shear-stress transport $k-\omega$. The mathematical models are given below. The SST $k-\omega$ model also known as Shear stress Transport $k-\omega$ model works on the following governing equations where k represents the turbulence kinetic energy and ω gives the specific dissipation rate.

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\Gamma_k\frac{\partial k}{\partial x_j}\right]+G_k-Y_k \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (1)$$

$$\frac{\partial}{\partial t}(\rho\omega)+\frac{\partial}{\partial x_j}(\rho\omega u_j)=\frac{\partial}{\partial x_j}\left[\Gamma_\omega\frac{\partial \omega}{\partial x_j}\right]+G_\omega-Y_\omega+D_\omega \ldots\ldots\ldots\ldots\ldots\ldots (2)$$

In these equations, the term G_k represents the production of turbulence kinetic energy. G_ω represents the generation of ω. Γ_k and Γ_ω represent the effective diffusivity of k and ω respectively. Y_k and Y_ω represent the dissipation of k and ω due to turbulence. D_ω represents the cross-diffusion term. The SST $k-\omega$ model is based on both the standard $k-\omega$ model and the standard $k-\varepsilon$ model. To blend these two models together, the standard $k-\omega$ model has been transformed into equations based on k and ω, which leads to the introduction of a cross-diffusion term D_ω, defined as

$$D_\omega=2(1-F_1)\rho\frac{1}{\sigma_{\omega,2}}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (3)$$

ρ is defined as the density of the fluid. u_i and u_j are the velocity components composed of mean and fluctuating velocity components ($i=1,2,3,\ldots$).
The effective diffusivities for the $k-\omega$ model is given by

$$\Gamma_k=\mu+\frac{\mu_t}{\sigma_k}\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (4)$$

$$\Gamma_\omega=\mu+\frac{\mu_t}{\sigma_\omega}\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (5)$$

Where σ_k and σ_ω are the turbulent Prandtl numbers for k and ω respectively. The turbulent viscosity, μ_t, is computed by combining k and ω as follows

$$\mu_t=\frac{\rho k}{\omega}\frac{1}{max\left[\frac{1}{\alpha^*},\frac{SF_2}{\alpha_1\omega}\right]}\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (6)$$

Production of k
The term G_k represents the production of turbulence kinetic energy. From the exact equation for the transport of, this term may be defined as

$$G_k=-\rho\overline{u_i'u_j'}\frac{\partial u_j}{\partial x_i}\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (7)$$

Where, u' term represents the fluctuating velocity components.

Production of ω
The term G_ω represents the production of ω and is given by

$$G_\omega=\frac{\alpha}{\nu_t}G_k\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (8)$$

Dissipation of k
The dissipation of k is given by

$$Y\_k=\rho\beta\wedge *f\_(\beta\wedge *)k\omega\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (9)$$

Dissipation of ω
The dissipation of ω is given by

$$Y_\omega=\rho\beta\omega^2\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (10)$$

The concept of conservation of energy is used to determine the heat transfer for the system. The governing equation is as follows.

$$\frac{\partial}{\partial t}(\rho E)+\nabla.(\vec{v}(\rho E+p))=\nabla.\left(k_{eff}\nabla T-\sum_j h_j\vec{J_j}+(\overline{\overline{\tau_{eff}}}.\vec{v})\right)\ldots\ldots\ldots\ldots (11)$$

Here k_{eff} is the effective conductivity $(k+k_t)$. Where k_t is the turbulent thermal conductivity, defined according to the turbulence model being used, and $\vec{J_j}$ is the diffusion flux of species j. The first three terms on the right-hand side of Equation represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively.

3 BOUNDARY CONDITIONS

For boundary conditions, a pressure inlet was considered and flow was considered to be for an ideal gas, i.e. ideal gas conditions were used. The viscosity used was Sutherland viscosity and smooth internal conditions were considered at the inner wall.

4 RESULTS AND DISCUSSION

The Mach number is defined as the ratio of the actual velocity of the fluid (or object in still fluid) to the speed of sound in the same fluid at the same state [9]. Mathematically, it is defined as

Ma=V/c

Where, Ma represents Mach number, V represents speed of object and c represents speed of sound in same fluid as object. Mach number is often used to describe fluid flow regimes. For a Mach number of unity, the flow is termed "sonic", for less than unity as "subsonic", slightly higher than unity "supersonic" and much higher than unity as "hypersonic" [9]. In our study, we investigated how Mach number varies for 10 different divergence angles starting form 10o and ending at 20o. The results were plotted on a graph showing Mach number against angle of divergence

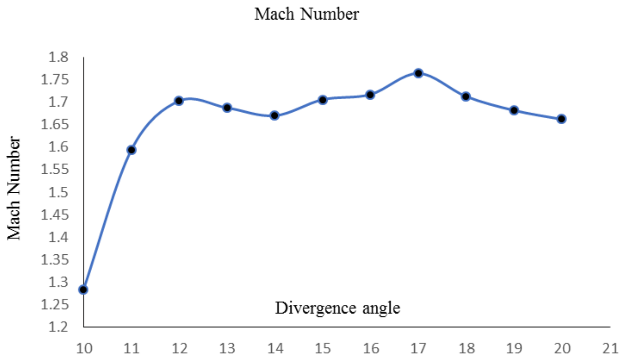

Fig 1: Graph of Mach number against divergence angle

The graph above illustrates how the Mach number varies with the divergence angle of a converging-diverging nozzle. The Mach number follows a trend of increasing and decreasing at regular intervals. However, there is a significant increase in the Mach number as the divergence angle increases, proving that extremely low divergence angles are unsuitable for practical purposes and hence, this design consideration must be kept in mind when designing nozzles for rocketry and related applications. The Mach number is seen to peak at 12^{o} and 17^{o} with values of 1.7 and 1.76 respectively. Between 13^{o} and 17^{o} there is a decrease in Mach number that could be due to the formation of shock. However, it should be noted that after 17^{o}, there is another steep fall in the Mach number with a corresponding increase in divergence angle. This shows that further increasing the divergence angle will be unsuitable for design and application purposes and thus a suitable angle to adopt would be 17^{o}, which yields the highest Mach number.

The second aspect of our study deals with the variation of total thrust of the nozzle with the corresponding variation of divergence angle. The thrust of a rocket nozzle is the force exerted by the exhaust gases exiting the nozzle on the launch pad. The total thrust is comprised of two separate thrusts: the momentum thrust and the pressure thrust [10]. Estakhrsar et al. [10] provided the following equation for the total thrust:

$$F=\dot{m}V_e+(P_e-P_0)A_e$$

Here, F is the total thrust, $\dot{m}$ is the mass flow rate of the exhaust gases, Ve is the exhaust velocity, Pe is the exit pressure, Po is the back pressure and Ae is the exit area. Similar to our investigation of Mach number, we studied how the thrust varies with the divergence angle.

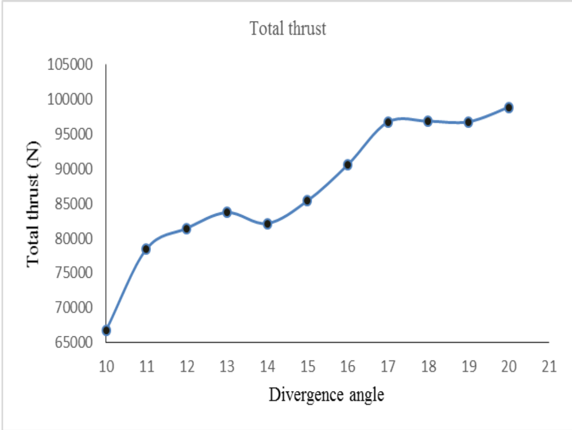

Fig 2: Graph of total thrust against divergence angle

From the graph above, the total thrust is seen to increase with divergence angle and then decrease after a certain point. From 10^{o} to 13^{o} the thrust increases steadily from approximately 66000 N to 85000 N and then sharply falls to 80000 N. However, from then on, there is a steady rise in the total thrust and at 17^{o} the thrust is at its maximum. Beyond 17^{o}, there is another decrease in total thrust, followed by a rise at 20^{o}. However, it must be noted that even though the total thrust is high at 20^{0}, the corresponding Mach number is quite low (1.65), hence, it is not a suitable value overall.

5 CONCLUSION

The design of nozzles has been a topic of interest for the last 60 years, since we stepped into the space age. With the passage of time, more advanced and efficient methods have been developed to create and test new designs of nozzles. However, the basic principle behind rocket propulsion remains the same and this ensures that rocket engineering and rocketry follow the necessary guidelines established by experimentation and live testing validation to ensure proper and accurate simulation. Nowadays, the initial testing is conducted with the help of powerful simulation software and the simulation with the best results are taken as a benchmark to proceed to experimental phase. Without accurate simulation results, millions of dollars would otherwise be wasted to construct and experimentally test new designs or operating conditions. Hence, this work is a small step towards the continual improvement and development of rocket science and rocket engineering.

6 REFERENCE

[1] R. Stark, "Flow Separation in Rocket Nozzles – An Overview," in *American Institute of Aeronautics and Astronautics*, San Jose, CA, 2013.

[2] "Aerospaceweb.org," August 1997. [Online]. Available: http://www.aerospaceweb.org/design/aerospike/shapes.shtml. [Accessed 2019].

[3] D. R. M. N. D. V. K. S. A Bhagat, "EFFECT OF NOZZLE DIVERGENCE ANGLE ON PLUME EXPANSION IN OUTER-SPACE CONDITIONS," in *1st International ISHMT-ASTFE Heat and Mass Transfer Conference*, Thiruvananthapuram, India, 2015.

[4] M. S. Biju Kuttan P, "Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics," *The International Journal Of Engineering And Science (Ijes),* vol. 2, pp. 196-207, 2013.

[5] V. N. K. Y. Raghu Ande, "umerical investigation on effect of divergent angle in convergent-divergent rocket engine nozzle," *CHEMICAL ENGINEERING TRANSACTIONS,* pp. 787-792, 2018.

[6] K. S. Patel, "Flow Analysis and Optimization of Supersonic Rocket Engine Nozzle at Various Divergent Angle using Computational Fluid Dynamics (CFD)," *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE),* vol. 11, no. 6, pp. 01-10, 2014.

[7] M. A. K. ,. G. S. Vishnu K. G, "Analysis of Flow Structure by Varying Divergence Angle and Contour of Supersonic C-D Nozzles," *International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE),* vol. 2, no. 7, pp. 28-34, 2017.

[8] P. L. R. J. Mason ML, "The effect of throat contouring on two dimensional converging-diverging nozzles at static condition.," *NASA technical paper 1704,* pp. 1-70, 1980.

[9] J. M. C. Yunus A Cengel, Fluid Mechanics Fundamentals and Applications, McGraw Hill, 2004.

[10] H. M.-M. J. M. H. Hamedi-Estakhrsar, "Investigation of effects of convergence and divergence half-angles on the performance of a nozzle for different operating conditions," *Journal of the Brazilian Society of Mechanical Sciences and Engineering,* vol. 40, pp. 1-12, 2018.