Computational Fluid Dynamics Analysis of Compressible Flow Through a Converging-Diverging Nozzle using the k-ε Turbulence Model

Muhammad Waqas Khalid
School of Chemical and Materials Engineering
National University of Sciences & Technology
Islamabad, Pakistan
mkhalid.eme12@scme.nust.edu.pk

Muhammad Ahsan
School of Chemical and Materials Engineering
National University of Sciences & Technology
Islamabad, Pakistan
ahsan@scme.nust.edu.pk

Abstract—**The thrust produced by a rocket motor is mainly dependent upon the expansion of the product gases through a nozzle. The nozzle is used to accelerate the gases produced in the combustion chamber and convert the chemical-potential energy into kinetic energy so that the gases exit the nozzle at very high velocity. It converts the high pressure, high temperature, and low-velocity gas in the combustion chamber into high-velocity gas of lower pressure and low temperature. The design of a nozzle has particular importance in determining the thrust and performance of a rocket. In recent years, it has received considerable attention as it directly impacts the overall performance of the rocket. This paper aims to analyze the variation of flow parameters like pressure, Mach number, and velocity using Finite Volume Method (FVM) solver with the standard k-ε turbulence model in Computational Fluid Dynamics (CFD). The simulation of shockwave inside the divergent nozzle section through CFD is also investigated. In this regard, a nozzle has been designed using Design Modeler, and CFD analysis of flow through the nozzle has been carried out using ANSYS Fluent. The model results are compared with theoretically calculated results, and the difference is negligible.**

Keywords-Converging-Diverging (C-D) nozzle; CFD; ANSYS Fluent; shockwave; k-ε turbulence model

I. INTRODUCTION

The nozzle is widely used in various areas, from rocket propulsion to fuel sprayer. It has been applied in industrial, aerospace, automobile, and other sectors. The nozzle is a major part of any high-performance engine or rocket motor. It is used to control the velocity, direction, and required parameters of the flow. Nozzles are designed to operate in all flow regions like subsonic, sonic, supersonic, and hypersonic. The design of the supersonic nozzle remains a challenging task in fluid mechanics. In a supersonic nozzle, not only the physical parameters of the nozzle play an essential role, but the thermodynamic parameters of the flow also play a crucial role in defining the design of a nozzle. The Converging-Diverging Nozzle known as de Laval nozzle is the most common and efficient design in rocketry. The chemical potential energy produced in the combustion chamber (rocket motor) is converted into kinetic energy by the nozzle. The nozzle converts high pressure, high temperature, and low velocity (subsonic) gases into low pressure, low temperature, and high velocity (supersonic) gases, hence producing high thrust [1]. To achieve the desired objectives, de Laval found that the most efficient conversion occurred when the nozzle area converges until the throat area, where the flow travels at sonic velocity, followed by a divergent section of the nozzle which accelerates the gases to supersonic or hypersonic velocities based on the design [2]. The exit velocity achieved in a converging-diverging nozzle is governed by the area ratios and pressure ratios [3]. The parameters which affect the performance or design of the nozzle include area ratios, pressure ratios, ratio specific heat constants, divergence nozzle angle, and nozzle length.

Studies have been carried out by keeping the Mach number constant and varying the divergence angle to observe its effect on various parameters like velocity, temperature, and pressure [4, 5]. Authors in [6] designed a nozzle by varying the exit Mach number and observed its effect on the length of the nozzle, variation in pressure and velocity while keeping the throat area constant in all cases. Authors in [7, 8] varied the divergence angles and observed their effect on Mach number, pressure, and exit velocity. It was shown that the divergence angles up to a specific limit provide better results. In supersonic nozzles, the sudden expansion of gases can produce shock inside the nozzle due to flow separation. A good nozzle design must be optimized to achieve maximum thrust without producing shock due to flow separation [9]. Hence, various simulations have been carried out to choose the best nozzle design for maximum thrust based on the same input conditions [10-12].

II. CFD MODELING

A. Basic Equations

The primary governing equations are the equation of conservation of mass or continuity equation and the equation of conservation of momentum. The continuity equation or mass conservation equation in differential form is:

Corresponding author: Muhammad Ahsan

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}) = S_m \quad (1)$$

Conservation of momentum equation is:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \quad (2)$$

B. *Standard k-ε (SKE) Model*

The standard k-ε turbulence model in ANSYS Fluent has become the workhorse of practical engineering flow calculations [13, 14]. The standard k-ε model is a model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). In this model, it is assumed that the flow is turbulent, and other effects like molecular viscosity are negligible. Therefore, it is only valid for turbulent flows.

C. *Transport Equations for the Standard k-ε Model*

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]$$
$$+G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (3)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma}\right)\frac{\partial \varepsilon}{\partial x_j}\right]$$
$$+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (4)$$

D. *Modeling the Turbulent Viscosity*

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (5)$$

where $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ, σ_k, and σ_ε are the constants in the abovementioned equations and their default values are [15]: $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, C_μ=0.09, σ_k=1.0, σ_ε=1.3

E. *Nozzle Parameters*

There are few parameters which affect the nozzle performance regardless of the type of fuel used inside the combustion chamber. The throat area and performance parameters like pressure, temperature, and velocity at the throat are the most significant among them. The following are the basic formulas to calculate the values of pressure, temperature, and velocity at the throat:

$$P_t = P_1\left\{\frac{2}{(k+1)}\right\}^{\frac{k}{k-1}} \quad (6)$$

$$T_t = \frac{2T_1}{(k+1)} \quad (7)$$

$$v_t = \sqrt{\frac{2k}{k+1}RT_1} \quad (8)$$

F. *Nozzle Dimensions*

Designing, modeling, and mesh generation of the Convergent-Divergent Nozzle were carried out in Design Modeler and Ansys Mesh. The basic nozzle dimensions are given in Table I.

TABLE I. NOZZLE DIMENSIONS

Parameter	Dimensions (m)
Nozzle length (L)	0.6
Nozzle exit radius (r_2)	0.12
Nozzle inlet radius (r_1)	0.1
Area ratio	0.5625

G. *Simulation Setup*

The Design Modeler application provides a persistent and parametric feature-based modeling environment that is a perfect tool for design optimization. Its modeling standard is to outline a 2D sketched profiles and use them to generate features. The advantages of Design Modeler over other modeling tools are that it is easy to use, has user-friendly interface, and it has a function to divide a model into small separate surfaces during meshing for improved mesh generation. The geometry of the nozzle is shown in Figure 1. The flow of high-pressure gases enters a nozzle from a combustion chamber with relatively low velocity. The total length of nozzle is 0.6m with variable diameter along the axis. The pressure of fluid flow at the inlet is varied from 300kPa to 240kPa (the pressure produced inside the chamber is dependent upon the type, size, and configuration of the propellant). At the outlet of the nozzle, the boundary condition was set to pressure outlet to observe the expansion of gases at sea level conditions.

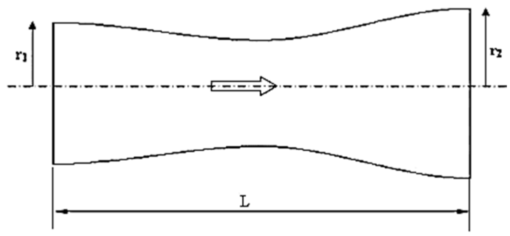

Fig. 1. Sketch of the proposed nozzle

A design modeler has the provision of producing finer mesh at the desired area. Therefore the mesh grid adjacent to the wall is finer as compared to the center region. The purpose is for the fine mesh is to capture the small gradients near the walls efficiently (Figure 2). CFD is widely used for analyzing the fluid flow by solving the governing equations of fluid dynamics. CFD can be used to explain and analyze complex numerical problems involving multiphase flow and interaction. It has excellent potential in simulating in order to obtain a better design and optimize it rather than wasting time and cost on prototypes [16]. CFD is widely used in engineering, and it gives better understanding and visualization of the problem. It is involved with physical laws in the form of partial differential equations called Navier-stroke equations [17]. CFD analysis consists of three steps called pre-processing, solving, and post-processing. Here, ANSYS Fluent is used for computational analysis of the fluid flow inside the nozzle. This analysis includes various performance factors like inlet pressure, inlet temperature, inlet velocity/Mach number, outlet pressure outlet temperature, and outlet velocity/Mach number [18]. After designing and meshing the nozzle, the ANSYS Fluent was launched. 2D, double precision options were selected. The case file was imported from an appropriate directory to pre-process the nozzle design (Figure 3). Pre-processing involves the transformation of physical problem statements into software. The type of material, fluid, and boundary conditions are defined as shown in Table II. The desired energy and flow models are selected, and assumptions were made concerning the nature of the flow, whether it's viscous or inviscid, compressible or non-compressible, steady, or non-steady [19].

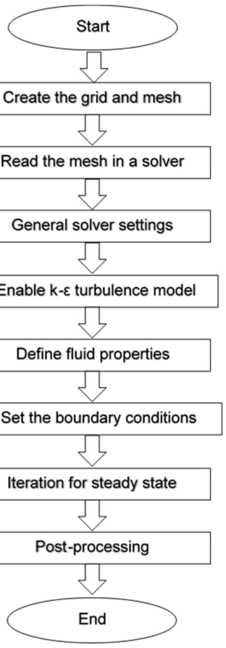

Fig. 2. Simulation procedure for CFD model

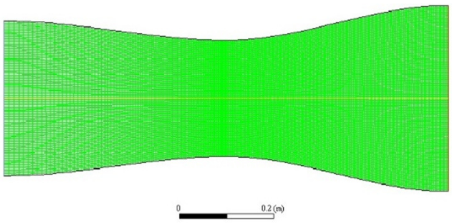

Fig. 3. Mesh imported in ANSYS Fluent

TABLE II. GENERAL SETUP

Setup	
Solver type	Pressure-based
2D space	Axisymmetric
Time	Steady
Boundary conditions for Case 1	
Pressure at inlet	300 kPa
Temperature at inlet	300K
Pressure at outlet	100 kPa
Temperature at iutlet	300K
Boundary conditions for Case 2	
Pressure at inlet	240 kPa
Temperature at inlet	300K
Pressure at outlet	100 kPa
Temperature at outlet	300K
Boundary conditions for Case 3 (inviscid flow)	
Pressure at inlet	240 kPa
Temperature at inlet	300K
Pressure at iutlet	100 kPa
Temperature at iutlet	300K

III. RESULTS AND ANALYSIS

The nozzle plays a vital role in producing thrust and controlling the speed and direction in different applications like aerospace and rocketry. In this study, simulations were performed to carry out an analysis of fluid flow inside the nozzle, shock simulation inside the nozzle using the k-ε model, and then comparing it with the inviscid flow.

A. Case 1: Fluid Flow in Converging-Diverging Nozzle (SKE Model)

In Figure 4, the red color shows the highest value of pressure, and blue is showing the lowest. The pressure at the inlet is close to 280kPa, and it is decreasing along the axis. The decrease in pressure is steady. The pressure at the outlet is close to 15kPa. The temperature inside the chamber is generally around 3000K, but we are studying the flow properties under the k-ε turbulence model and then flow properties of inviscid flow, therefore, the temperature at the inlet is kept at 300K, which is reduced during the expansion of gases in the divergent section of a nozzle to 125K, as shown in Figure 5. In Figure 6, the velocity of flow at the inlet is low as the pressure of the gases is higher. As the pressure decreases, the velocity increases along the axis of the nozzle and attains its highest value of close to 600m/s at the exit of the nozzle. The Mach number is a key parameter used for the calculation of thrust of the nozzle and it segregates whether a nozzle is subsonic, sonic, or supersonic. The Mach number is close to 1 at the throat of the nozzle, and the maximum achieved is at the exit, i.e., 2.64, as shown in Figure 7.

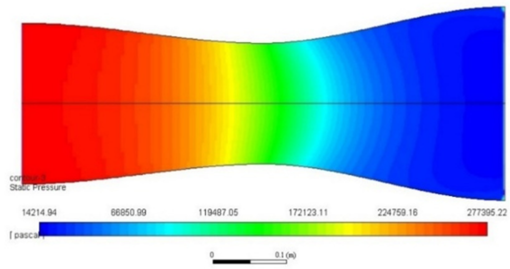

Fig. 4. Pressure contours

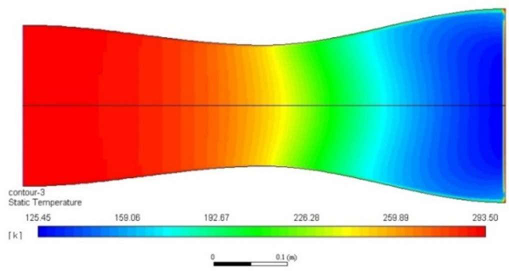

Fig. 5. Temperature contours

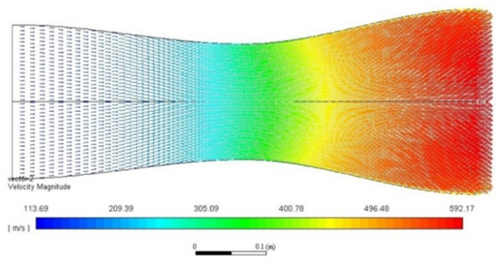

Fig. 6. Velocity vectors

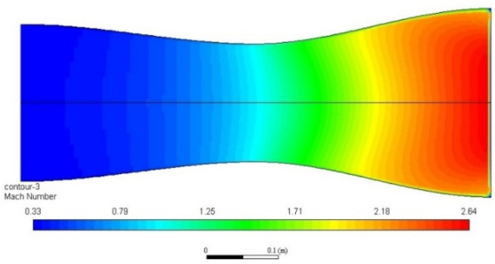

Fig. 7. Mach number contours

B. Case 2: Shockwave of Converging Diverging Nozzle (SKE Model)

There is a phenomenon called over-expanded nozzle, in which the exhaust pressure of the product gases falls below the atmospheric pressure, and expansion of the gases occurs very rapidly inside the nozzle producing a discontinuity in the flow. This sudden discontinuity is called a shockwave. This type of shockwave generally occurs where the area ratio gradient is higher, and pressure drops rapidly, often close to the exhaust end of the supersonic nozzle. The speed of the flow just before the shock has a significant impact on the shock. The higher the Mach number just before the shock, the more significant would be the effect of shock on the nozzle. This can cause a reduction in thrust and sometimes physical damage to the nozzle. The second case is simulating a shock inside the nozzle in viscous flow using the standard k-ε model. The pressure at the inlet is close to 222kPa, as can be seen in Figure 8. As flow passes through the throat, the expansion of the gases happens suddenly, unlike in Case 1 where this expansion was steady, hence, producing a shock inside the nozzle. At that point, the pressure is measured close to 22kPa, much lesser than the atmospheric pressure.

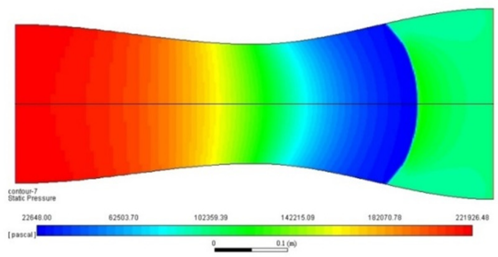

Fig. 8. Pressure contours

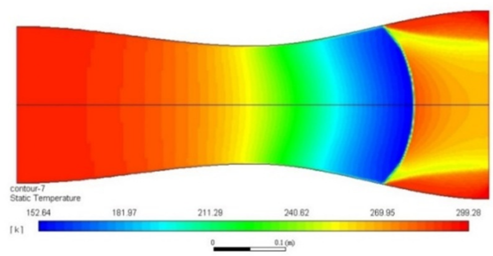

Fig. 9. Temperature contours

Similar with Case 1, the inlet temperature is kept at 300K. As soon as flow passes through the throat of the nozzle, a sudden decrease in temperature is observed to 152K with the sudden expansion of gases, shown in Figure 9. In Figure 10, the velocity vectors of flow show a rapid increase in the velocity of the divergent section of the nozzle. Figure 10 also shows the formation of a vortex close to the wall of the nozzle just after the shock because the flow separation has occurred close to the outlet of the nozzle, and backpressure is generated. In Figure 11, the Mach number is approximately one at the throat and highest at the shock front, i.e. 2.20. The walls' frictions are higher close to the wall of the nozzle and lesser at the center. Hence, velocity is lower close to the wall, thus producing shock earlier close to the wall than it takes place in the main flow. It makes the shock shape as curved.

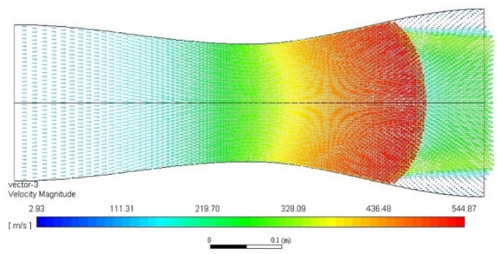

Fig. 10. Velocity vectors

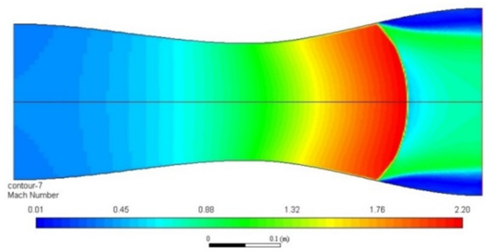

Fig. 11. Mach number contours

C. Case 3: Shockwave of Converging Diverging Nozzle (Inviscid Flow)

Inviscid flow analysis assumes the flow to be laminar and disregards the effect of viscosity on the flow. It is only applicable for high Reynolds number applications where inertial forces dominate the viscous forces. The inviscid theory predicts a simple shock structure consisting of a normal shock followed by a smooth recovery to exit pressure in the divergent part. But, viscous effects like wall boundary layer and flow separation alter drastically the flow in a converging-diverging nozzle. The velocity of the flow is high in this simulation. Therefore, we can assume the flow to be inviscid so that we can have a comparison of shock and flow characteristics with the viscous flow simulations. The third case is simulating the shock inside the nozzle in an inviscid flow. The pressure at the inlet is 222kPa. In this case, the pressure decreases in the divergent section more steadily as compared to Case 2. The wall frictions and viscosity of the flow are assumed to be zero. In this case, the shock location is shifted close to the nozzle exit, shown in Figure 12. As in Cases 1 and 2, the inlet temperature is kept at 300K. As soon as the flow passes through the throat of the nozzle, a gradual decrease in temperature is observed up to 120K where the shock is formed as shown in Figure 13. In Figure 14, The velocity vectors of flow showing increase in the velocity of the divergent section of the nozzle. There is no flow separation at the outlet. Hence, no vortex is formed, and no backflow is observed.

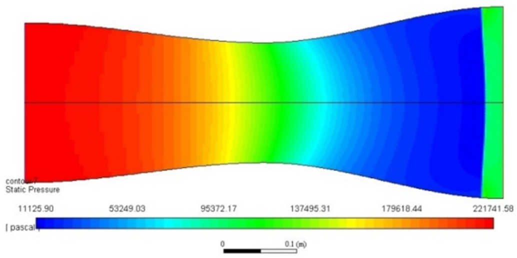

Fig. 12. Pressure contours

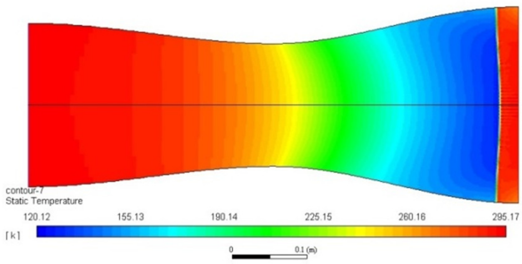

Fig. 13. Temperature contours

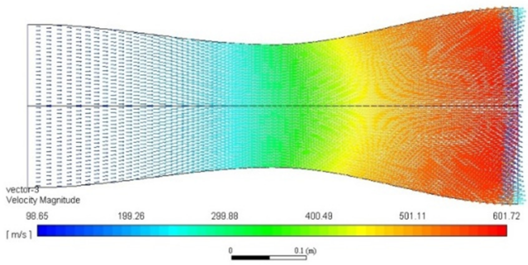

Fig. 14. Velocity vectors

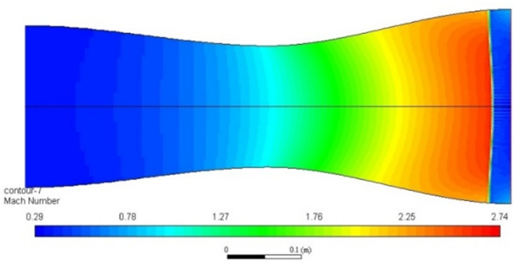

Fig. 15. Mach number contours

In Figure 15, it is shown that the Mach number is approximately one at the throat and highest at the shock front, i.e. 2.74. The viscosity causes a reduction in the momentum of the flow, as this loss is not considered in inviscid simulations. Therefore, the achieved Mach number is higher as compared to the standard k-ε turbulence model simulations with the same boundary conditions in both cases. The shock shape is straight in this case, as there are no wall frictions.

D. Comparison with Analytical Results

Table III shows the comparison of values of pressure, temperature, and velocity at the throat obtained from these simulations and the analytical results calculated by using (6), (7), and (8). The simulation results obtained from this model showed good agreement with the analytical results.

TABLE III. ANALYTICAL AND SIMULATED RESULTS COMPARISON

Chamber pressure (kPa)	Pressure at throat (kPa)		% Difference
	Analytical	This model	
300	164.15	168.10	2.35
240	131.30	132.80	1.13
	Temperature at throat (K)		
300	260.87	252.50	3.20
240	260.87	249.23	4.46
	Velocity at throat (m/s)		
300	342.05	335.60	1.88
240	342.05	328.75	3.88

IV. CONCLUSION

A detailed study was conducted on the effect of inlet pressure variation on the Mach number and shock produced inside the nozzle using the standard k-ε turbulence model. The pressure values were varied to check the position of the shock and the expansion of the product gases in the divergent portion of the nozzle. The outlet pressure was kept constant to 100kPa, which is close to the atmospheric pressure at sea level. The initial inlet pressure was set to 300kPa, and simulation was performed. No shock was observed inside the nozzle, and the expansion of the gases was steady inside the nozzle. The initial inlet pressure was gradually reduced. Simulations were carried out to study and analyze the behavior of exhaust flow inside the nozzle. In the second case, the inlet pressure value was set to 240kPa and shock was observed inside the nozzle. In this case, the expansion of the gasses was sudden, shock was produced, and over-expanded nozzle behavior was observed. In the third case, the flow was assumed to be inviscid. A shock was observed inside the nozzle but close to the exit. It is evident that there are no wall frictions as the shape of shock is straight. Hence, the simulations using the standard k-ε turbulence model give more realistic values than the simulation carried out using inviscid flow conditions.

REFERENCES

[1] V. Jayakumar, S. Madhu, A. Muniappan, A. H. Ansari, S. N. Kumar, "Investigation of thermal characteristics in solid rocket nozzle with insulate using Cad/Cae", International Journal of Pure and Applied Mathematics, Vol. 119, No. 7, pp. 443-456, 2018

[2] S. Sofyan, V. Wuwung, "RX-320 rocket static pressure combustion chamber prediction and validation by using inverse method", Jurnal Teknologi Dirgantara, Vol. 16, No. 1, pp. 45-58, 2018

[3] N. D. Deshpande, S. S. Vidwans, P. R. Mahale, R. S. Joshi, K. Jagtap, "Theoretical and CFD analysis of de-Laval nozzle", International Journal of Mechanical and Production Engineering, Vol. 2, No. 4, pp. 33-36, 2014

[4] P. Natta, V. R. Kumar, Y. H. Rao, "Flow analysis of rocket nozzle using computational fluid dynamics (CFD)", International Journal of Engineering Research and Applications, Vol. 2, No. 5, pp. 1226-1235, 2012

[5] K. M. Pandey, A. P. Singh, "CFD analysis of conical nozzle for mach 3 at various angles of divergence with fluent software", International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, pp. 179-185, 2010

[6] V. Ramji, R. Mukesh, I. Hasan, "Design and numerical simulation of convergent divergent nozzle", Applied Mechanics and Materials, Vol. 852, pp. 617-624, 2016

[7] M. S. Hossain, M. F. Raiyan, N. H. Jony, "Comparative study of supersonic nozzles", International Journal of Research in Engineering and Technology, Vol. 3, No. 10, pp. 351-357, 2014

[8] P. Biju Kuttan, M. Sajesh, "Optimization of divergent angle of a rocket engine nozzle using computational fluid dynamics", The International Journal Of Engineering and Science, Vol. 2, No. 2, pp. 196-207, 2013

[9] G. Mohan Kumar, D. X. Fernando, R. M. Kumar, "Design and optimization of de Lavel nozzle to prevent shock induced flow separation", Advances in Aerospace Science and Applications, Vol. 3, No. 2, pp. 119-124, 2013

[10] G. Satyanarayana, C. Varun, S. Naidu, "CFD analysis of convergent-divergent nozzle", Acta Technica Corviniensis-Bulletin of Engineering, Vol. 6, No. 3, pp. 139, 2013

[11] B. V. V. Naga Sudhakar, B. P. C. Sekhar, P. N. Mohan, M. D. Touseef Ahmad, "Modeling and simulation of convergent-divergent nozzle using computational fluid dynamics", International Research Journal of Engineering and Technology, Vol. 3, No. 8, pp. 346-350, 2016

[12] A. S. Swaroopini, M. G. Kumar, T. N. Kumar, "Numerical simulation and optimization of high-performance supersonic nozzle at different conical angles", International Journal of Research in Engineering and Technology, Vol. 4, No. 9, pp. 268-273, 2015

[13] M. Ahsan, "Numerical analysis of friction factor for a fully developed turbulent flow using k–ε turbulence model with enhanced wall treatment", Beni-Suef University Journal of Basic and Applied Sciences, Vol. 3, No. 4, pp. 269-277, 2014

[14] N. A. Najar, D. Dandotiya, F. A. Najar, "Comparative analysis of k-ε and spalart-allmaras turbulence models for compressible flow through a convergent-divergent nozzle", The International Journal of Engineering and Science, Vol. 2, No. 8, pp. 8-17, 2013

[15] N. Rolander, J. Rambo, Y. Joshi, J. K. Allen, F. Mistree, "An approach to robust design of turbulent convective systems", Journal of Mechanical Design, Vol. 128, No. 4, pp. 844-855, 2006

[16] M. Elashmawy, "3D-CFD simulation of confined cross-flow injection process using single piston pump", Engineering, Technology & Applied Science Research, Vol. 7, No. 6, pp. 2308-2312, 2017

[17] K. Pansari, S. A. K. Jilani, "Numerical investigation of the performance of convergent divergent nozzle", International Journal of Modern Engineering Research, Vol. 3, No. 5, pp. 2662-2666, 2013

[18] A. K. Reji, G. Kumaresan, A. S. Menon, J. Parappadi, A. P. Harikrishna, A. Mukundan, "Simulation and validation of supersonic flow through a convergent-divergent nozzle", International Journal of Pure and Applied Mathematics, Vol. 119, No. 12, pp. 2135-2142, 2018

[19] M. Elashmawy, A. Alghamdi, I. Badawi, "Investigation of the effect of pipeline size on the cross flow injection process", Engineering, Technology & Applied Science Research, Vol. 6, No. 3, pp. 1023-1028, 2016

[20] A. A. Khan, T. R. Shembharkar, "Viscous flow analysis in a convergent-divergent nozzle", International Journal of Computational Engineering Research, Vol. 3, No. 5, pp. 5-15, 2008