中國機械工程學刊第四十二卷第一期第 121~130 頁(民國一百一十年)
Journal of the Chinese Society of Mechanical Engineers, Vol.42, No.1, pp 121~130 (2021)

Study of Factors Influencing the Thrust Response of Convergent Nozzle Block

Nayyer Nayab Malik*, M.Javed Hyder **

Keywords：Convergent Nozzle block, Optimal Thrust, CFD Analysis, Isentropic Flow, ANSYS Fluent.

ABSTRACT

This paper investigates the effect of different parameters on thrust response of convergent nozzle block. Computational Fluid Dynamics (CFD) based simulations have been carried out and the results are experimentally verified through model testing setup. The scope of this research is to develop understanding about the thrust generation phenomenon by single/multi- nozzle blocks and to utilize this knowledge in related applications. Convergent nozzles are usually employed in air jet propulsion system where it is used as thruster providing sufficient reaction force for lifting platform. It is therefore required to select and design a nozzle that can displace more nearby air with a steady flow to get the stable thrust output. CFD analysis has been done on convergent nozzle block's to determine the optimal fluid dynamic parameters and for studying the effect of flow characteristics on thrust output. For the simulations, ANSYS Fluent has been used as software platform. The density based solver has been adopted with utilization of k-ω SST transport model equation. Structured mesh is applied on the nozzle domain to get precise and consistent results. Optimal solution is obtained by achieving mesh grid independency based on y^+ criterion, reducing the simulation time. It is established that high area ratio, high inlet pressure, longer converging length of nozzle with S3 shape gives the higher thrust value while increasing the number of nozzles in a block provides stable fluid flow with greater thrust per unit

Paper Received May, 2019. Revised November, 2019, Accepted November, 2019, Author for Correspondence: Nayyer Nayab Malik.

* *PhD Scholar , Department of Mechanical Engineering, Pakistan Institute of Engineering & Applied Sciences (PIEAS), P.O. Nilore, Islamabad, Pakistan..*

** *Professor, Department of Mechanical Engineering, Pakistan Institute of Engineering & Applied Sciences (PIEAS), P.O. Nilore, Islamabad, Pakistan..*

INTRODUCTION

Nozzles are the crucial components of the jet propulsion system that convert the kinetic energy into required thrust for lifting the platform in accordance with New-ton's third law of motion (Khaleel et al, 2014; Patel, 2014). The aerodynamic system depends on the thrust generated by ejection of fluid jet from the exhaust nozzle. Various types of nozzles have been employed for different applications. Based on the geometry, some of the commonly used nozzles are convergent, Convergent-Divergent (C-D), divergent (diffuser), aero spike bell nozzle, de laval type. Convergent conical nozzles have simple construction and were widely used in early jet engines to convert pressure energy into kinetic energy (Bumataria et al, 2015). The knowledge of the pressure, area ratio and the geometry is important in the estimation of performance of the jet engine (Hagiwara et al, 2009; Tellez et al, 2012; Kowsik et al, 2013; Bumataria et al, 2015; Alam et al, 2016;). The selection of nozzles is made carefully to get the required thrust and reduction in the noise level. The quality of the thrust produced depends on the performance of the exhaust nozzle. The factors affecting the nozzle thrust include the geometrical and fluid dynamic parameters. The change in the size and shape directly affects turbulent velocity profile and sudden transition may cause the unpredictable pressure distribution generating high turbulence effect that could damage the nozzle (Alam et al, 2016). Careful design of nozzle geometry reduces wall friction viscous losses in the internal boundary layer and helps in attaining the optimum thrust. Inlet pressure, velocity and the density of the flowing fluid are the dynamic parameters used for adjusting and regulating the thrust for a given nozzle. The thrust performance of the nozzle can be observed by amount of thrust generated and the quality of the fluid flow jet field (Tellez et al, 2012; Pansari et al, 2013). These parameters are varied, establishing the suitable configuration to efficiently meet the thrust requirement.

Computational Fluid Dynamics (CFD) is a numerical iterative technique based on the governing equations representing the physical phenomenon of fluid flow characteristics. Presently CFD techniques play a vital role in the component development as they give reasonably reliable results. Since fewer resources are needed in carrying out the simulations, this approach is gaining its role in the preliminary design process and prototype development. The observations obtained from the simulations provide appropriate results that could be correlated with the real time system (Sherma et al, 2012; Patel et al, 2014).

Profound research has been carried out to achieve the optimal exhaust nozzle thrust based on experimental and CFD techniques. The testing setup although gives more precise results but it not only adds complexity, cost and time constraints to the system but also requires time to time adjustments. On the other hand CFD based technique provides a nominal approach giving acceptable results proving to be more feasible and require less operational expertise. Most of the previous studies gave emphasis on the design of nozzle by varying the geometry (Pandey et al, 2010; Satyanarayana et al, 2011; Sun et al, 2013; Khaleel et al, 2014) to obtain the maximum thrust for the given input fluid dynamic parameters whereas with the exception of few (Khaleel et al, 2014; Bumataria et al, 2015, Barale et al, 2016) not much emphasis is given on the change in pressure ratio to obtain desired results. Considerable research is carried out to analyze the fluid flow field generated at the nozzle exit (Shimizu et al, 2008; Tellez et al, 2012; Pansari et al, 2013; Lee et al, 2015; Barale et al, 2016;). The improvement in the analysis results can be made by combining all the above strategies giving suitable solution that could be validated with the experimental methodology.

This paper explores the thrust characteristics of single/multi-nozzle block carefully designed to get stable thrust response. The CFD analysis has been conducted on 2D axisymmetric convergent nozzle block followed by 3D simulations of Block of multiple nozzles utilizing ANSYS FLUENT platform. ANSYS software has been used for quite a while for the bench mark applications, providing reasonable solution close to real time systems. The investigations have been carried out to study the dynamic characteristics of the air flowing through convergent nozzle block under isentropic conditions. The main goal of the present research is to analyze the thrust performance for the achieving viable results by varying the parameters including inlet pressure, area ratio, converging length, shape and number of the convergent nozzles in a block. The parameters are modified individually keeping the rest of the factors constant. The compressible air flowing through the nozzle provides almost negligible friction. For this reason isentropic model has been adopted as a preliminary design analysis and to further simplify the problem, steady state flow dynamics is employed for the 2D analysis. Structured mesh is applied to the structure to improve the quality of solution and to get consistent results. The computation involves the k-ω SST transport equation. The results are then compared with the experimental results to validate the results.

NOZZLE THEORETICAL MODEL

The design and the fluid flow characteristics are dependent on the physical phenomenon represented by the governing equations. The flow through the nozzle is based on the Bernoullis equation while the thrust generated dependent on Newtons third law of motion. The isentropic relations for the subsonic flow through the convergent nozzle, as mentioned in [Deshpande et al, 2014; Balega et al, 2015; Alam et al,2016; Malik et al, 2016), are given by:

Mass Flow Rate of Fluid at Nozzle Exit:

$$\dot{m} = \frac{Ap_i}{\sqrt{T}}\sqrt{\frac{\gamma}{R}}\left(\frac{2}{\gamma+1}\right)^{\frac{\gamma+1}{2(\gamma-1)}} \tag{1}$$

Nozzle Pressure Ratio:

$$\frac{P_e}{P_i} = \left(1+\frac{\gamma-1}{2}M_e^2\right)^{\frac{\gamma}{\gamma-1}} \tag{2}$$

Density of Fluid:

$$\rho = \frac{P}{RT} \tag{3}$$

Nozzle Exit Velocity:

$$V_e = M_e\sqrt{(\gamma RT)} \tag{4}$$

Nozzle Area Ratio:

$$\frac{A_e}{A_i} = \frac{1}{M_e}\left[\left(\frac{2}{\gamma+1}\right)\left(1+\frac{\gamma-1}{2}M_e^2\right)^{\frac{\gamma+1}{2(\gamma-1)}}\right] \tag{5}$$

Thrust Generated by Nozzle:

$$F = \dot{m}V_e - \left(p_e - p_i\right)A_e \tag{6}$$

In the above equations,

F = Nozzle Output Thrust
$\dot{m}$ = M ass Flow Rate
p_i = Nozzle Inlet Pressure
p_e = Nozzle Exit Pressure
p_o = Free Stream Pressure
ρ = Density of Air at a Point
R = Real Gas Constant
T = Total Temperature of Nozzle Domain
A_i = Inlet Area
A_e = Exit Area
M_e = M ach Number of Fluid at Nozzle Exit
γ = Specific Heat Ratio

Based on the above equations, the standard isentropic table is formulated which is utilized to design a nozzle. For a given set of parameters table are tabulated to obtain the desired output. The length of the nozzle is adjusted to improve flow characteristics and to get the optimal performance.

SIMULATION PROCEDURE

Methodology

The procedure involves the steady state analysis of 2D axisymmetric and 3D nozzle jet block. The basic structure of convergent nozzle is shown in Fig 1. Air is used as a fluid flow medium under subsonic isentropic conditions. Density based solver has been adopted utilizing k-ω SST model having ability to predict a free shear flow giving reliable measurement of wakes, mixing of layer, round & radial jets making it the suitable choice for wall bounded and free flows (Ruangtrakoon et al,2012). All values are taken in absolute terms. The variable settings and the boundary conditions employed in the simulations are given in Table 1.

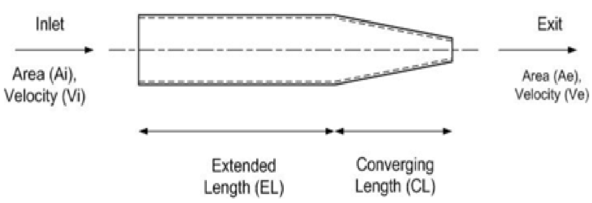

Fig. 1. Convergent Nozzle.

Table 1: Settings for CFD Analysis

Inlet Pressure Range (MPa)	0.1 - 0.2 (absolute)
Outlet Pressure Range (MPa)	0.101325 (absolute)
Operating Pressure (MPa)	0.101325 (absolute)
Solver	Density Based
Model Adopted	k – ω SST Based
Fluid Medium	Air (as ideal gas)

Geometry Selection

To begin with the preliminary design, a reasonable size of the nozzle has been selected from the isentropic tables. The 2D axisymmetric and 3D convergent nozzle block geometry has been made in ANSYS Workbench Design Module. Each section is labeled under named selection to identify the parts of the working domain. The modification in the geometry domain is made in accordance to Fig 2.

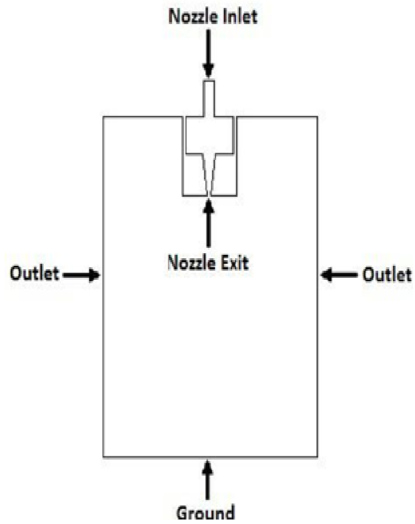

Fig. 2. Domain for CFD Analysis

Mesh Generation

Meshing is one of the critical part of the simulation procedure that directly affects the integrity of the results. It may affect the convergence of solution or gives vague results, failing to determine the physical phenomena. The meshing has been applied on the geometrical domain using ANSYS Workbench Meshing Module. To achieve the reliable results the quality of mesh has to be improved. This requires proper grid size selection, aspect ratio, skewness, elemental and orthogonal quality. The procedure to select the minimum grid size is based on y^+ criterion which shows the perpendicular distance of the grid point from the wall boundary. The y^+ value, as in (Blockena et al, 2007; Ariff et al, 2009), can be defined by the following expression given below:

$$y^+ = \frac{\mu}{\rho u_t} \tag{7}$$

It determines whether the wall adjustment cells are laminar or turbulent. According to (Ariff et al, 2009; Biswas et al, 2015), the subdivisions of the near wall region in a turbulent boundary layer can be summarized as follows:

- $y^+ < 5$: in the viscous sub layer region (velocity profiles is assumed to be laminar and viscous stress dominates the wall shear)
- $5 < y^+ < 30$: boundary layer region (both viscous and turbulent shear dominates)
- $30 < y^+ < 300$: Fully turbulent portion or log-law region (corresponds to the region where turbulent shear predominates)

In the analysis y^+ considerations have been made in accordance to (Ariff et al, 2009; Biswas et al, 2015) for the acceptable range and the grid size has been modified until the mesh independent solution is obtained. The structured mesh and reduced grid size has been used to achieve mesh independent solutions to get consistent, accurate and reliable results of the CFD analysis. Since the area of interest is the flow inside the nozzle, finer mesh is applied at interior then the rest of the domain. The mesh generated in the software is shown in Fig 3.

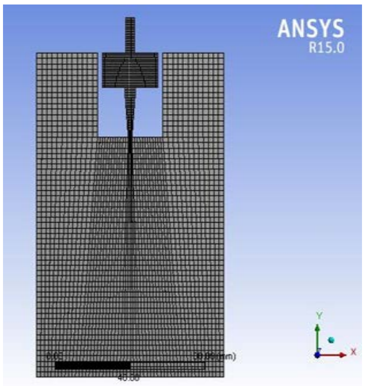

Fig. 3. Meshing Domain.

SIMULATION RESULTS & ANALYSIS

The simulation results are obtained on the basis of fluid flow characteristics and the thrust generated from the exit of convergent nozzle. The effect of parameters including inlet pressure, cone length and the area ratio is thoroughly investigated to check the thrust performance to achieve optimal solution. The effect of each parameter is observed by changing one of the variables while keeping the others constant.

A brief analysis is carried out, employing the simulations representing overall results to show the influence of the modifications on the effective thrust.

Effect of Changing Area Ratio

The area ratio is modified while keeping the inlet area constant. The area ratio is changed from 0.250 to 0.500. The inlet pressure of 0.2 MPa and the cone length of 25 mm are selected.

The static pressure with greater the area ratio show less variation from nozzle inlet to outlet reducing the turbulence factor and raising the static pressure. The comparison of the results in Fig 4 show increasing trend of nozzle exit static pressure with increase in area ratio.

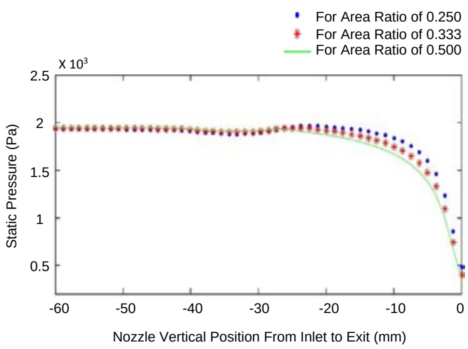

Fig. 4. Variation of Static Pressure of Different Area Ratio's

Increase in the mach number is observed at nozzle exit with the decrease in area ratio. However less variation in mach number is seen due to reduced gradient effects. Fig 5 demonstrates the comparison of mach number for different area ratio's.

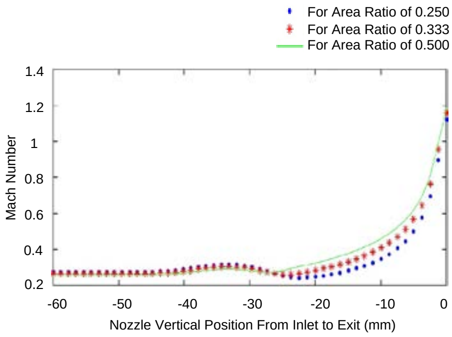

Fig. 5. Variation of Mach Number for Different Area Ratio's

The fluid releases the air with less force from nozzle exhaust as the area ratio decreases. With small area ratio, the expansion in flow field jet profile spread is noticed and venacontracta (portion of fluid jet flow where it has minimum cross sectional area) is formed closer to the nozzle exit resulting in larger no stream line space (given as no flow velocity lines). Fig 6, shows the path lines for the fluid flow domain.

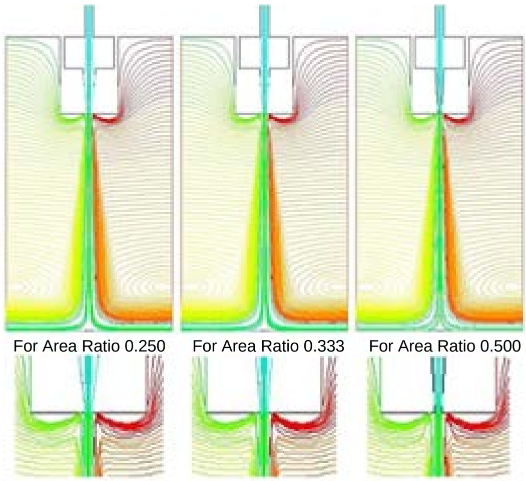

Fig. 6. Flow Pattern for Different Area Ratios

Effect of Pressure Variation

The pressure is varied from 0.1 MPa to 0.2 MPa with the fixed area ratio of 0.25 and cone length 25mm. With greater inlet pressure, the results show increase in the static pressure exerting force causing compactness between the molecules. The observations reveal that greater inlet pressure gives higher static pressure at nozzle exhaust as indicated in Fig 7.

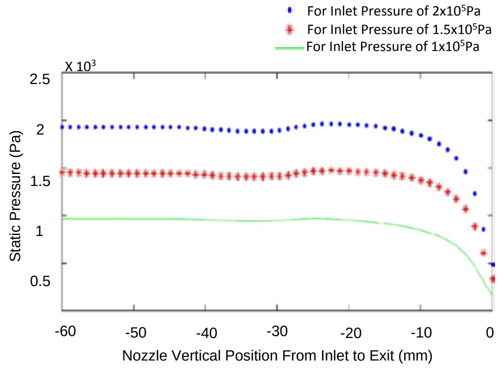

Fig. 7. Static Pressure Profile for Different Inlet Pressures.

The velocity variation reduces from inlet to outlet with increase in inlet pressure. Furthermore it is depicted that higher inlet pressure results in lower mach number at nozzle exit. Fig 8 gives the comparison of mach number at different inlet pressures.

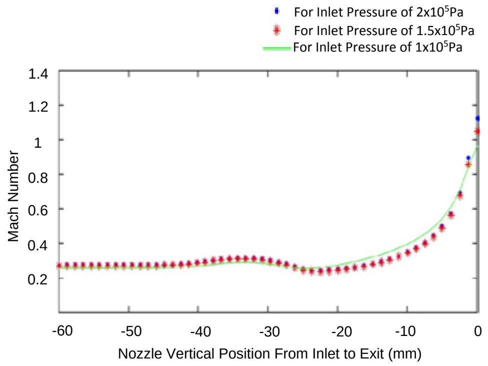

Fig. 8. Mach Number Profile for Different Inlet Pressures

The higher the pressure, more force is applied on the fluid that urges the flow to converge earlier to form venacontracta and greater flow spread is observed. The no flow space expands with increase in inlet pressure as shown in Fig 9.

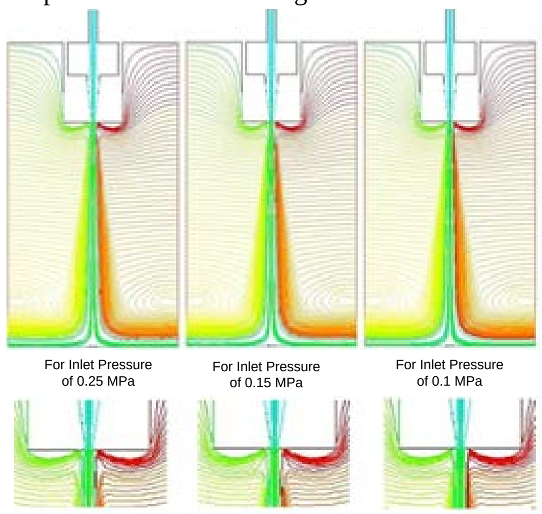

Fig. 9. Flow Pattern for Different Inlet Pressures

Effect of Modifying Nozzle Cone Length

The nozzle convergent length is modified from 15 mm to 25mm. For the pressure of 0.2 MPa and area ratio of 0.250. It is inferred that longer converging length reduces the turbulence effect resulting in slight decrease in nozzle exit static pressure and the change occurring from inlet to outlet is very small as indicated in Fig 10.

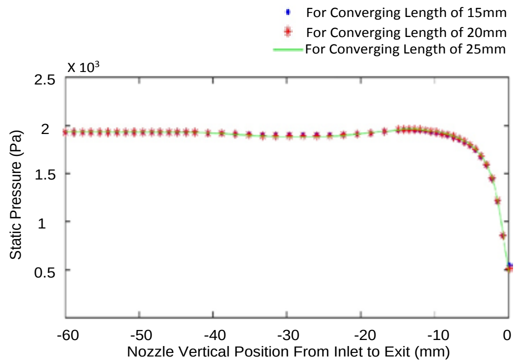

Fig. 10. Static Pressure Profile for Different Converging Length's

Increase in mach number is observed with longer converging length. There is negligible change in mach number from inlet to outlet as pointed out in Fig 11.

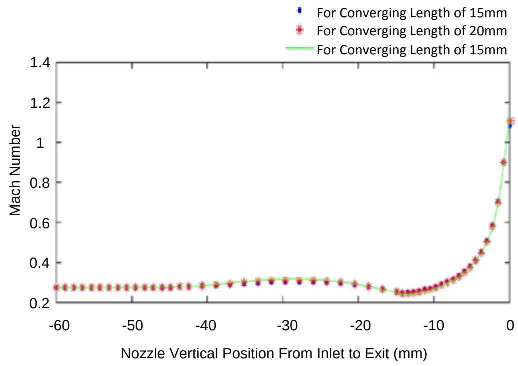

Fig. 11. Mach Number Profile for Different Converging Length's

For longer nozzle converging length, there is smooth transition of flow causing it to converge farther away from the exit. There is considerable change in the jet diameter as we increase the converging length. For shorter length nozzle venacontracta is formed closer to nozzle exit and no flow region is significantly decreased as specified in Fig 12.

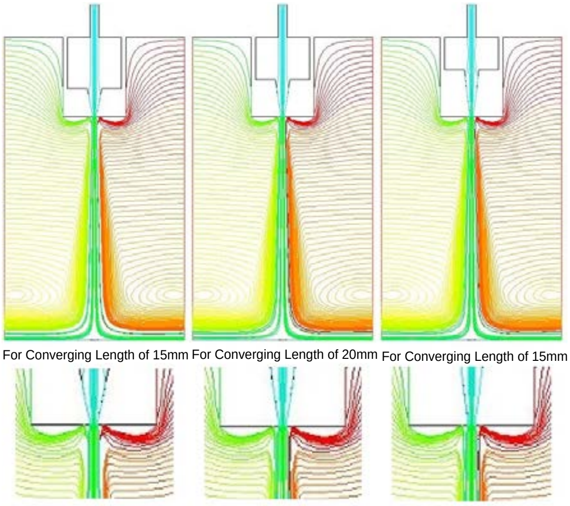

Fig. 12. Flow Pattern for Different Converging Lengths

Effect of Modifying Nozzle Shape

The variation in shape parameter is one of the significant factors affecting the output thrust of nozzle. Three different nozzle shapes that have been analyzed are shown in Fig 13.

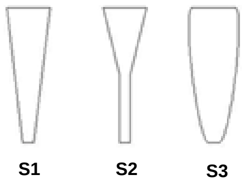

Fig. 13. Nozzle Shapes Used for Analysis

It is observed that the static pressure profile for nozzle S2 is abruptly changing from inlet to outlet due to to sudden change in the geometry while smooth transition is achieved by S2 and S3. The nozzle block with S2 shape produces less output static pressure at nozzle exit whereas the one with S3 shape has the maximum static pressure value for the same inlet pressure as given in Fig 14.

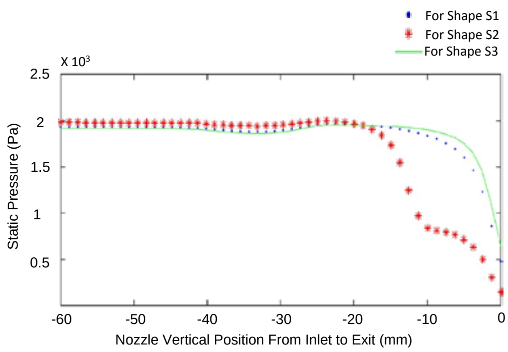

Fig. 14. Static Pressure Profile for Different Nozzle Shapes

Similar vivid variation is seen in mach number from inlet to outlet of nozzle S2. The mach number gained by the S2 is highest whereas its lowest value is obtained by nozzle S3.

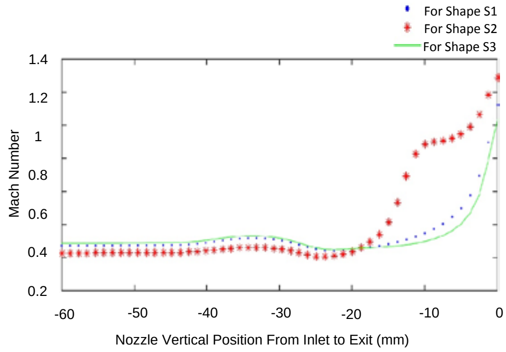

Fig. 15. Mach Number Profile with Different Nozzle Shapes

The fluid flow through the nozzles S1,S2 and S3 have been compared in Fig 16. In nozzle S1, there is no sudden change and fluid exits to converge at a certain point with noticeable no flow region. There is sudden change in flow path for nozzle S2 that initially decelerates the fluid and then attains the constant rate, producing venacontracta farther away from the nozzle exit with considerably reduced no flow zone. For S3 nozzle, the fluid has been held encapsulated inside due to bulging path and starts to spread outwards soon after leaving outlet resulting in earlier formation of venacontracta. This maximizes the no flow space (greater than S1 & S2) just below the nozzle exit. Also nozzle shapes S1 and S2 have slight difference in jet size while S3 jet diameter has significantly increased.

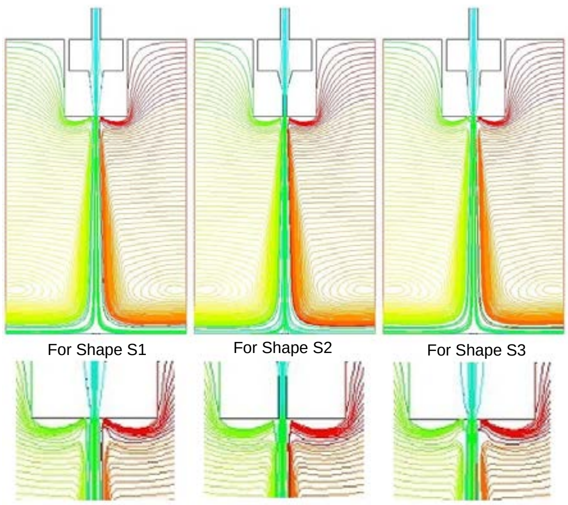

Fig. 16. Flow Pattern for Different Nozzle Shapes

Effect of Adding Number of Nozzles

The 3D analysis has been carried out to determine the effect of adding the number of nozzles in a symmetric sequence on the thrust response of nozzle block. The multi-nozzle blocks are mainly utilized where spread of air flow is required especially in achieving the stability of hovering platforms. The comparison of single nozzle and multi-nozzle blocks is made to check its suitability for the selective applications.

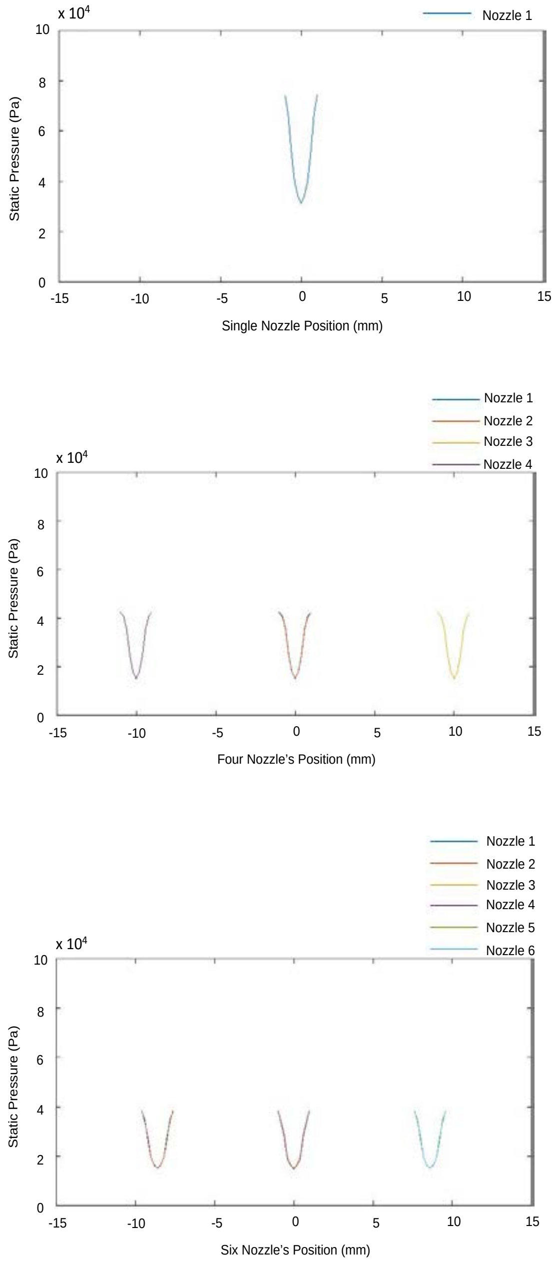

Fig. 17. Static Pressure for Different Nozzle Blocks.

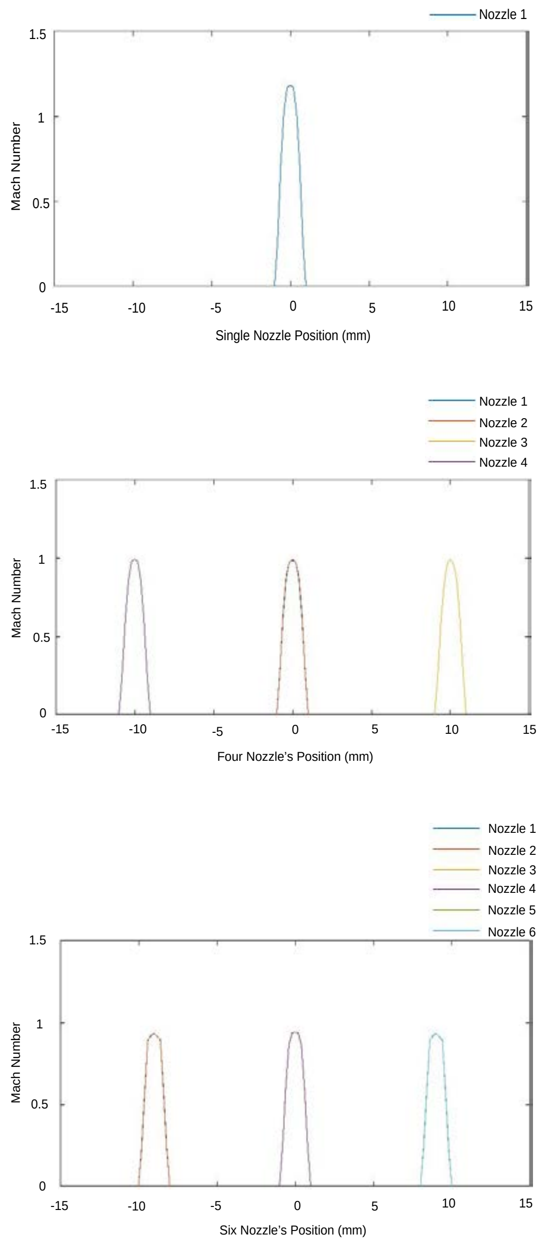

Fig. 18. Mach Number Achieved by Different Nozzle Blocks.

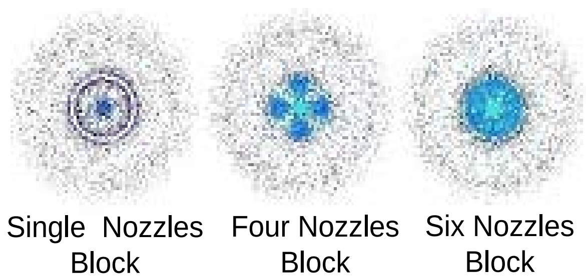

Fig. 19. Flow Through 3D Nozzle Blocks

The resulting plots of 3D nozzle blocks reveal that static pressure and the velocity for the single nozzle is higher than the multi-nozzle blocks. The fluid flows directly to the single nozzle with slight swirling effect due to the empty space around it while in multi-nozzle blocks there is complex turbulent effect to get even flow distribution to each nozzle in the block. It is further noticed that the height pointed thrust is produced for a solo nozzle block but reducing its value per unit area. In contrast to this, multi-nozzle blocks tends to produce expanded periphery of flow path to get better reaction force per unit area against the generated thrust. The solution to the analysis is demonstrated in Fig17-19.

EXPERIMENTAL VARIFICATION

Validation of the results obtained from the CFD simulations requires experimental verification. To carry out the investigation, trial setup has been established. It consists of a test stand having fixed rectangular board at its top with convergent nozzle block joined at the center. Nozzles of with variable parameters are examined to study the thrust variation effects. Nozzle block is coupled with the pressure regulating tank (electromechanically regulated) that is connected to the compressor through a soft tube. The compressor has a mass flow rate of 0.0042kg/s.

Just below the nozzle, load sensor is present which directly measures the real time output thrust .Calibrated weights are used for tuning the load sensor to get the actual reading. The measured signal goes to the Data Acquisition and Signal Conditioning Unit. From here the filtered signal is sent to the computer where it is analyzed and processed to display the data.

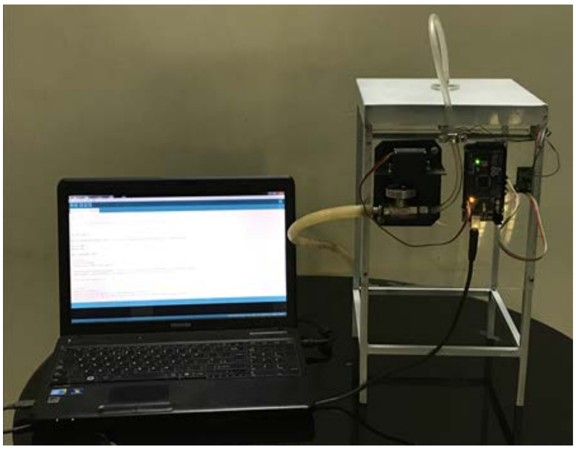

Fig. 20. Experimental Setup (Front View).

Fig. 21. Experimental Setup (Side View).

Knowing the value of mass flow of air, static and dynamic pressure at nozzle exit, the thrust output response can be recorded utilizing equation (6).The overall results obtained are given in the Table 2.

Table 2 : Overall Results

Area Ratio (AR)	Inlet Pressure(IP) [MPa]	Converging Length [mm]	Shape Type (S)	No of Nozzles (X)	Simulated Thrust (T_s) [N]	Actual Thrust (T_a) [N]
0.5	0.20	25	S1	1	1.107	1.045
0.33	0.20	25	S1	1	0.984	0.932
0.25	0.20	25	S1	1	0.952	0.874
0.25	0.15	25	S1	1	0.921	0.903
0.25	0.10	25	S1	1	0.875	0.841
0.25	0.20	20	S1	1	0.944	0.927
0.25	0.20	15	S1	1	0.939	0.909
0.25	0.20	25	S2	1	0.903	0.872
0.25	0.20	25	S3	1	0.961	0.948
0.25	0.20	25	S1	1	0.916	0.874
0.25	0.20	25	S1	4	0.897	0.859
0.25	0.20	25	S1	6	0.859	0.857

The results show some difference in the simulated and the experimental values. This is mainly due to the unaccounted frictional losses and conservative selection of the model with limitations. Apart from that, close correlation between both the results is observed, verifying the true thrust output response.

CONCLUSION

The CFD based analysis has been carried out on the 2D/3D convergent nozzle blocks with the compressed air flowing through it under isentropic condition. The variable parameters including the nozzle area ratio, inlet pressure, converging length, shape and number of nozzles in a block are analyzed to investigate the performance of the thrust generation process. The results show that variation in area ratio has significant effect on the thrust output due to the change in mass flow rate. So higher the area ratio higher would be the thrust at output. The modification in cone length not only contributes in enhancing the efficiency of convergent nozzle but also improves the performance by attenuating the turbulence effect on the jet flow field. High input pressure considerably reduces the velocity but large thrust produced by the nozzle. The geometrical shape greatly affects dynamic properties due change in fluid flow path. It is noted that S1 type nozzle block produces maximum thrust as compared to the S2 & S3 type. Furthermore more number of nozzles in a block reduces the turbulence intensity that assists in attaining steady flow condition. The maximum estimated thrust that is obtained from the nozzle exit is approximately about 1.107 N and its actual value obtained is 1.045N. Simulations reveal that the parameters are interdependent on each other, thus changing the overall combined effect. From the above observations it is concluded that reasonably high area ratio, high inlet pressure, longer converging length of nozzle with S3 shape gives the higher thrust value while increasing the number nozzles in a block provides stable fluid flow with greater thrust per unit area. The experimental data conforms closely with the simulated results, thus authenticating the adopted CFD technique.

REFERENCES

Alam, M.M.A., Setoguchi, T. and Kim, H.D., Matsuo, S., "Nozzle Geometry Variations on the Discharge Coefficient", J. of Prop. and Pwr. Res., vol.5, pp.22-33, (2016).

Ariff, M., Salim, S.M. and Cheah, S.C., "Wall Y+ Approach for Dealing with Turbulent Flow Over a Surface Mounted Cube: Part 1 Low Reynolds Number", Intl. Conf. on CFD in the Minls. and Proc. Indst., CSIRO, Melbourne, Australia, (2009).

Barale, D.D., Limbadri, C. and Arakerimath, R.R., "Modelling and Parametric Fluid Flow Analysis (CFD) of convergent nozzle used in pelton turbine", Intl. Eng. Res. Jrnl, pp.25-34, (2016).

Belega, B.A. and Nguyen, T.D., "Analysis of Flow in Convergent-Divergent Rocket Engine Nozzle Using Computational Fluid Dynamics", Intl. Conf. of Sci. Paper (AFASES), pp.1-6, (2015).

Biswas, K., Sinha, P.K., Mullick, A.N. and Majumdar, B., "A Computational Analysis of Flow Development through a Constant Area C- Duct", J. of Eng. Res. and App., vol.5, pp.69-73, (2015).

Blockena, B., Stathopoulos, T. and Carmeliet, J.,"CFD Simulation of the Atmospheric Boundary Layer: Wall Function Problems", Atmos. Env., ELSEVIER, vol.41 (2), pp. 238-252, (2007).

Bumataria, R.K., Patel, D.V. and Shah, S.D., "Experimental Analysis of Convergent, Convergent Divergent Nozzles at Various Mass Flow Rates for Pressure Ratio and Pressure Along the Length of Nozzle", Intl. J. of Adv Res in Eng, Sci and Tech, (2015).

Deshpande, N.D., Vidwans, S.S., Mahale, P.R., Joshi, R.S. and Jagtap, K.R., "Theoretical CFD Analysis of De Laval Nozzle", Intl. J. of Mech. and Prod. Eng, vol.2, pp.33-36, (2014).

Hagiwara, S., Sawada, F., Horisawa, H., Funaki, I., "Experimental and Numerical Investigation on Thrust Performance Improvement of Micro-Single-Nozzle Thrusters", Trans. of Jpan Soc of Aero and Space Sci, vol.7, pp.17-22, (2009).

Khaleel, MD., Kishore, N.P. and Kumar, D.S., “Design and CFD Analysis of Conical Nozzle for 2.75 Inch Rocket System”, IJESRT, Vehicles,” pp.465-470, (2014).

Kowsik, V.G.D., Joseph, C., Justin, M.P.A., Sainath, R., Murugan, Ilakkiya, S., "Thrust Force Analysiss of Spike Bell Nozzle", Intl. J. of App. Res. in Mech Eng, vol.3, pp.54-59, (2013).

Lee, J.W., Kim, I.H. and Kwo, S., "Effect of Unsteadiness and Nozzle Asymmetry on Thrust of a Micro Thruster" Proc. of PwrMEMS and microEMS, Sendai, Japan, (2008).

Malik, N.N. and Hyder, M.J., "CFD Based Thrust Analysis of Convergent Conical Nozzle with Different Area Ratio's" Symp. on Adv. in Mech. Eng., PIEAS, Pakistan, pp.1-7, (2016).

Pandey, K.M. and Singh, A.P., "CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software", Intl. J. of Chem. Eng. and App., pp. 279-285, (2010).

Pansari, K. and Jilani, S.A.K., "Analysis of the Performance and Flow Characteristics of Convergent Divergent (C-D) Nozzle", Intl. J. of Adv. in Eng Tech, vol. 6, pp.1313-1318, (2013).

Patel, K.S., "Flow Analysis and Optimization of

Supersonic Rocket Engine Nozzle at Various Divergent Angle using Computational Fluid Dynamics (CFD)", IOSR, J. Mech and Civil Eng., vol.11, pp.1-10, (2014).

Ruangtrakoon, N., Thongtip, T, Aphornratana, S., Sriveerakul, T., "CFD Simulation on the Effect of Primary Nozzle Geometries for a Steam Ejector in Refrigeration Cycle", Intl. J. of Therm. Sci., pp.1-13, (2012).

Satyanarayana, G., Varun, C. and Naidu, S.S., "CFD Analysis of Convergent-Divergent Nozzle", Acta Tech. Corv. BOE, (2011).

Sharma, A., Sharma, P. and Kothari, A., "Numerical Simulation for Pressure Distribution in Pelton Turbine Nozzle for the Different Shapes of Spear", Intl. J. of Innov. in Eng. and Tech, vol.1, pp.95-107, (2012).

Sharma, A., Sharma, P. and Kothari, A., "Numerical Simulation for Pressure Distribution in Pelton Turbine Nozzle for the Different Shapes of Spear", Intl. J. of Innov. in Eng. and Tech, vol.1, pp.95-107, (2012).

Shimizu, T., Kodera, M. and Tsuboi, N., "Internal and External Flow of Rocket Nozzle", J. Earth Sim., vol.9, pp.19-26, (2008).

Sun, P., Zhao, X.X. and Gao, J., "Influencing Analysis to Nozzles Inner Flow Field of Different Pintle Radius and Shapes", J. of App. Sci., vol.13 (14), pp.2744-2751, (2013).

Tellez, J.L.G., Velazquez, M.T., Montes, S.A., Herrera, J.A.L.O., Leon, A.R., "Computational Analysis of Flow Field on the Propulsion Nozzle of a Micro-Turbojet Engine", J. of Sci Res, vol.4, pp.1-5, (2012).