International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN(P): 2249-6890; ISSN(E): 2249-8001
Vol. 8, Issue 6, Dec 2018, 373-382

NUMERICAL ANALYSIS OF CONVERGENT-DIVERGENT NOZZLE USING FINITE ELEMENT METHOD

FHARUKH AHMED. G. M[1, 2], ABDULREHMAN A. ALROBAIAN[3], ABDUL AABID[4] & S. A. KHAN[4]

[1]*Research Scholar, Department of Mechanical Engineering, Bearys institute of Technology Mangalore, Karnataka, India*

[2]*Assistant Professor, Department of Mechanical Engineering, Government Engineering College, Huvinahadagali, Karnataka, India*

[3] *Department of Mechanical Engineering, Qassim University, Buraidah, Saudi Arabia*

[4]*Department of Mechanical Engineering, International Islamic University Malaysia, Kuala Lumpur, Malaysia*

ABSTRACT

In this paper, Finite element method (FEM) were used to simulate the different flow configuration. Convergent divergent (CD) nozzle was considered with extended divergent duct. 1 mm of microjets orifice diameter were arranged at ninety degrees at PCD 13 mm to control base pressure in a suddenly expanded flow. The designed Mach number of CD nozzleis 1.87 and area ratio 3.24 was considered. The different L/D of the duct was used from 2 to 10. The nozzle pressure simulated for 3, 5, 7, 9 and 11. In this case. Two-dimensional planar model was designed using ANSYS fluent analysis. The total wall pressure distribution and Mach number from inlet to the outlet was observed. From the results, it is detected that the microjets control the loss of pressure and decreases the drag at the suddenly expanded region. The results also show, we can fix the flow parameter which will result in the maximum gain in the base pressure and velocity. In present study, the CD nozzle designed and modelled using available ANSYS fluent database: K-ε standard wall function turbulence model has been used and validated with the commercial computational fluid dynamics (CFD).

KEYWORDS: CFD, C-D nozzle, ANSYS, Pressure & Mach Number

Received: Sep 05, 2018; **Accepted:** Sep 27, 2018; **Published:** Nov 07, 2018; **Paper Id.:** IJMPERDDEC201842

Original Article

INTRODUCTION

A sudden expansion of the flow in supersonic regimes is a leading problem in many fluid applications. In sudden expansion problem is an important field in the numerous engineering problems role in several fluid application. In rocket and jet engine test-cells noticed that systems have been used to simulate the upper atmosphere flow field; a jet discharging yields an effective pressure which is sub atmospheric pressure. However, CFD analysis have its own advantage to simulate the flow field's plays an important role in current technologies of a design optimization by providing the improvised solutions for a given problem with different commercially available tools. Recently, Khan et al., identified the velocity and pressure effect in suddenly expanded CD nozzle and the effect of micro-jets control using CFD analysis [1], [2].

In early attempts, Anderson et al., (1968) explained base pressure and hence the noise generated by the sudden enlargement of air in a circular duct. Depending on the area ratio and the nozzle's geometry it was noticed that attached flow of base pressure was having a least value. Khan et al., (2002-2012) experimentally investigated the base pressure control using micro-jets and explained the benefits of the flow regulation to control base pressure in a suddenly expanded CD nozzle considering the variables such as the Mach number, area ratio, NPR, and L/D ratio for the cases when control present or absent for the computation of the pressures. However, it is seen that very limited research has been conducted out in the literature to regulate the flow at the base area from a CD nozzle using micro-jets as the control mechanism.

From last two decades the work has been done using CFD simulation by different available design and analysis tools. Some of them are specified here, the CD nozzle used to for convert pressure energy to kinetic energy in order to produce thrust using CFD for various performance parameters [13]. CFD is used to analyses fluid flow of supersonic rocket nozzle at various degree of divergent angle by using two-dimensional axis-symmetric model [14]. De-Laval nozzles was used to convert the thermal and pressure energy into useful kinetic energy providing theoretical formulae and validated with numerical results obtained by using the CFD software ANSYS Fluent [15]. Supersonic converging-diverging nozzle has been modelled and analyses the flow after throat at a certain point using ANSYS FLUENT. The simulations were carried for 2D and 3D model to provide better comparative platform [16]. Numerical method was used to simulate the flow by convergent-divergent nozzle with Mach number M = 2.6 at nozzle exit using the RANS equations with k-ω SST turbulent model [17]. The study was carried out using computer based CFD simulation by ANSYS code with designed tool Solid works CAD for capable of modelling. ANSYS is capable of simulating and analysing thermo-fluid characteristics for convergent-divergent aircraft nozzle [18].

CFD simulation has been carried to study the velocity and pressure effects for different designed Mach number, area ratio and length to diameter ratio of the CD nozzle [19]–[21]. CFD simulation was carried to Box wing configuration and simulated the unconventional non-planar configuration using ICEM CFD and ANSYS CFX solver [22]. Invested the flow-field by numerical approach using CFD simulation to investigate the efficacy of the supersonic Mach numbers due to the flow from supersonic nozzle exhausted in a larger circular duct [23].

The objective of this paper is to investigate the flow field through the convergent-divergent nozzle. Considered two-dimensional planar model with k-ε turbulence model. We have identified the velocity and pressure for different L/D ration and NPR. The model was designed with and without microjets control. It also been identified the flow field of pressure and velocity and compared with control of microjets. Moreover, experimental results of a CD nozzle with the configuration of suddenly expanded flow by Sher Afghan Khan & Rathakrishnan, 2003 compared with the present CFD (ANSYS FLUENT software) numerical results.

DEFINITION OF THE PROBLEM

The CD nozzle is modelled based on the designed Mach number 1.87 shown in Fig. 1. Micro-jets are designed 1mm of diameter and located at the pitch distance of 1.3 mm from the divergent diameter. The main aim of this study is to analyze the flows past a CD nozzle and the computation of the flow parameters such as pressure, and Mach number with the effect of different L/D and NPR using 2D CFD simulation with and without micro-jets.

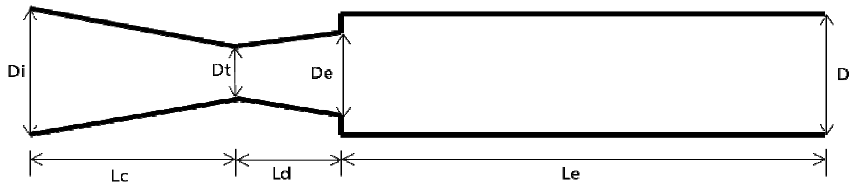


Figure 1: CD Nozzle with Enlarged Duct

The dimensions of CD nozzle with suddenly expanded duct are mentioned in table 1.

Table 1: Dimension of CD Nozzle

Mach Number	1.87
Inletdiameter (Di)	28.72 mm
Throatdiameter (Dt)	8.648 mm
Exit diameter (De)	10 mm
Extended diameter (D)	18 mm
Convergent length (Lc)	35 mm
Divergent length (Ld)	12.926 mm
Extended length (Le)	180 mm
Micro-jets diameter (Dm)	1 mm

FINITE ELEMENT METHOD

The 2D planar model of CD nozzle is shown in Figure 2(a) and closed form of finite element meshing is shown in Figure 2(b). ANSYS workbench has been used and created structural mesh, number of elements has been used high to create fine mesh in closed area at edge of the planar body. Total, 38,368 binary nodes, were generated for 2D planar model.

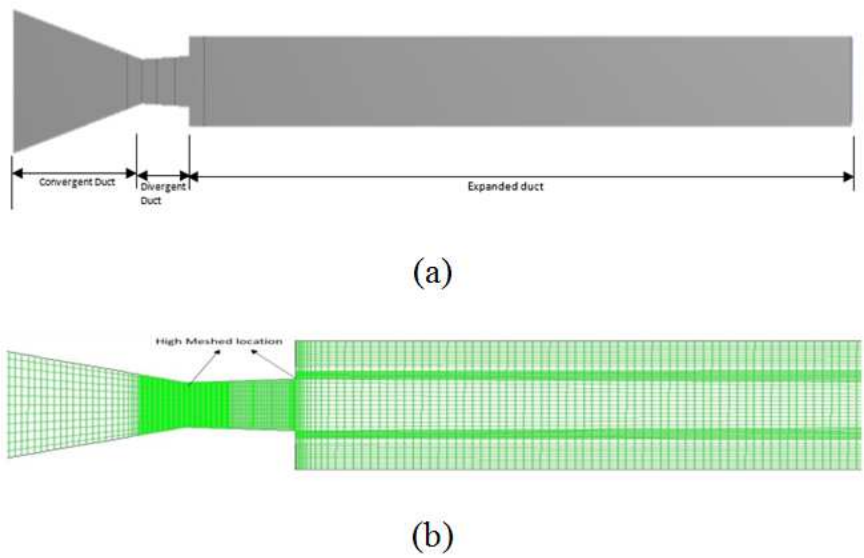


Figure 2: 2D Planar Fluid Body (a) Finite Element Model (b) Meshing

Fluent Step for Solution Initialisation

Computations of the flow field inside the control volume were done using RANS (Reynolds-Averaged Navier-Stokes) equations with k-ε standard turbulent model [24], [25]. The table 2 shows the most important setting which has been used for simulating the results in the present study.

Table 2: Solution Set up

Solution Method	
Solver	Absolute 2D planar Pressure-Based
Turbulence Model	k-ε standard Wall function
Fluid	Idealgas, Viscosity by Sutherland law
Boundary Condition	Inlet: Pressure Inlet, Outlet: Pressure Outlet, Wall: Wall
Solution Method	Second orderupwind
Solution Initialization	Standard fromInlet
Reference Value	Inlet (Solid surface)
Solution iteration	Until solution Converged

RESULTS AND DISCUSSIONS

This papers continues the recent work done by Khan et al., [1]. In this we computed the all cases of NPR and L/D ratios to identify flow phenomenon through the CD nozzle.

Validation of Finite Element Model

In order to validate the finite element (FE) model, the CD nozzle with micro-jets control at the base and expanded duct placed on the divergent portion of the CD nozzle shown in figure 1of Khan et al., (2002) is considered. By comparing the results obtained from Khan et al., (2003) and the present FE results, the agreement as presented in table 3 for the case of L/D=10.

Table 3: Verification of Present FE Model

Pressure (Case: L/D=10)	Khan et al., (2003) (Experimental)	Present Results (Numerical)	Relative Error
Without control at base (Pb) (NPR 3)	-14.53	-14.96	2.8%
With control at base (Pb) (NPR 3)	-14.29	-13.98	2.1%

Pressure Flow

The pressure for a nozzle or CD nozzle is a very important consideration for high speed jets. Figure 3 illustrate the phenomenon of pressure flow for L/D = 10 and NPR = 11, at the base reason the expansion wave has been generated due to sudden expansion of CD nozzle. At the corner of the base have recirculation flow which is generating drag to reduce this small micro jets have been employed at the base area which results the recirculation becomes low and increases the forward flow to the outlet portion of the nozzle which may give high thrust performance shown in figure 3(b).

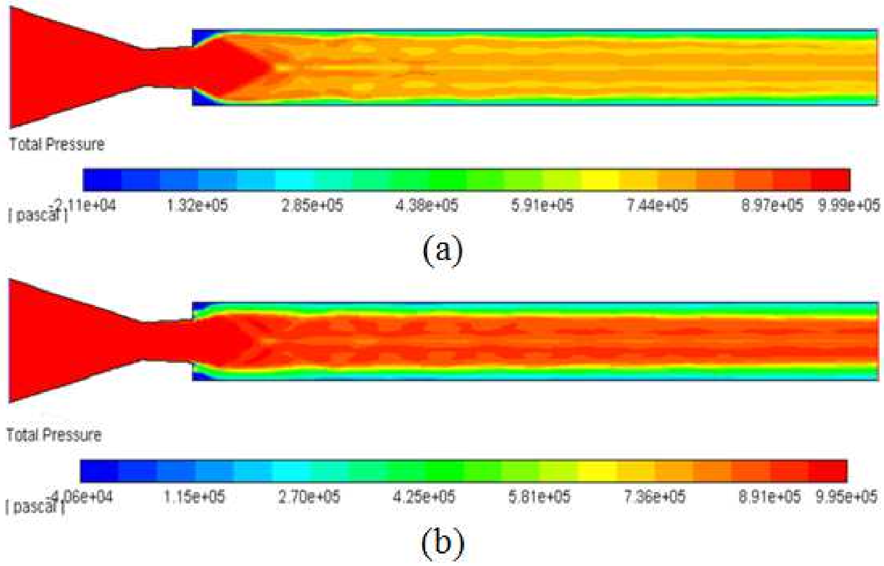


Figure 3: Close View of CD Nozzle with Pressure Flow for L/D = 10 and NPR = 11
(a) With Control (b) Without Control

Figure 4 explains the total pressure flow from inlet to the outlet with and without control of micro jets and considered only one case of flow process for L/D = 10 and NPR = 11. The phenomenon which shows in this figure 4, the pressure decreases when it reaches at the end of the enlarged duct it means that the pressure losses at the exit region. When micro jets were employed flow-process slightly will change and control the pressure at the suddenly expanded duct. However, the vertical portion shows the position of enlarged duct at the divergent region therefore, the wall position found constant at this region.

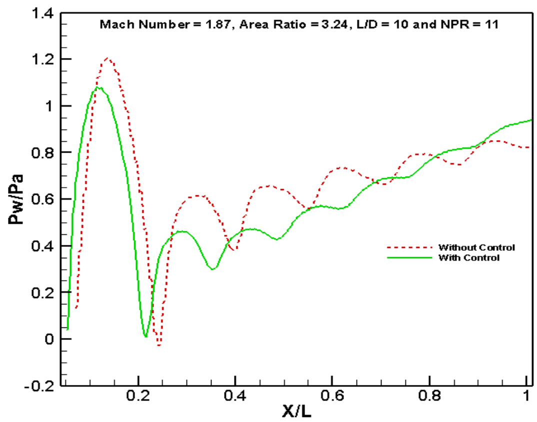


Figure 4: Wall Pressure for L/D=10 and NPR=11

To identify different cases of the CD nozzle, selected the location at the base as well as at the wall for different L/D where L represents the length and D is the diameter of the expanded duct. To obtain several results, the length of the duct has been changed and recorded the pressure flow which says that the pressure variation is different for different NPR.

Figure 5(a) shows the value of NPR is will become decreased at 11 for base pressure as well as wall pressure which shows in figure 5(c) and NPR slightly to be changes for the case of L/D as shown in figure (c) and (d). The base and wall pressure was divided with the atmospheric pressure 98273 Pascals which is obtained from the Sher Afghan Khan & Rathakrishnan (2003). Finally, the plots represent base pressure versus NPR, base pressure versus L/D.

Similarly, wall pressure versus NPR, wall pressure versus L/D with and without micro-jets as depicted in figure 5. The outcome base on CFD show that the loss of pressure is minimum when micro-jets installed at the base and flow of rotation also become control.

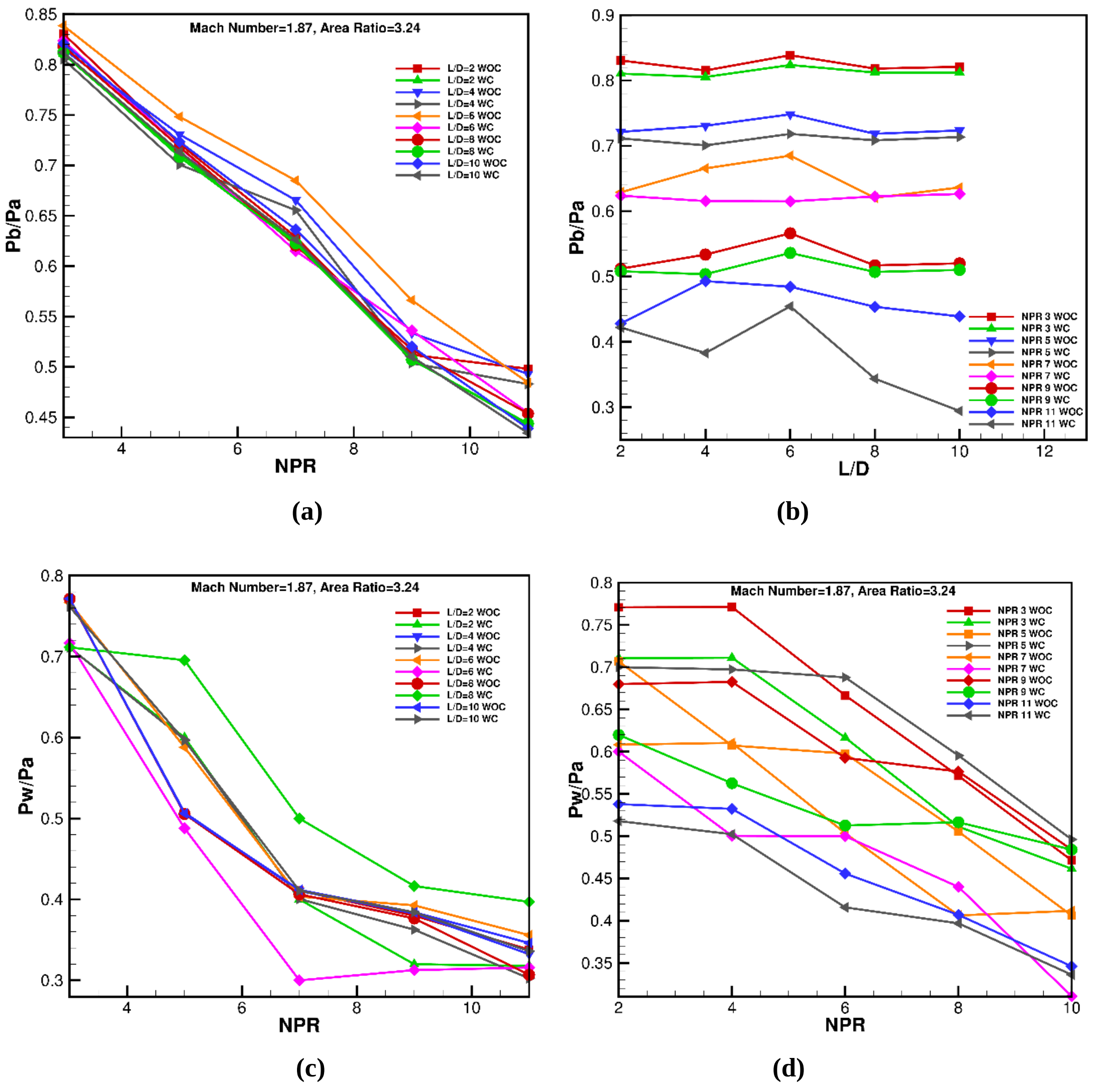


Figure 5: Pressure Plots-(a) and (b) for Base Pressure, (c) and (d) for Wall Pressure

Mach Number

On the other hand, the Mach number also considered for better understand. In this case, the value of Mach number is increased at the outlet of the CD nozzle. Therefore, to obtain all results the selected the one location of the base and wall to identify all L/D and NPR values. Figure 6 illustrate the contours for the case of L/D = 10 and NPR 11 which shows from the different color variation lowest to the highest values at the CD nozzle. From the figure 6(b) it is clearly showing that at the recirculation zone Mach number becomes very low this means reduce the drag and the forwarded the flow to the exit of the nozzle which may results increases the Mach Number.

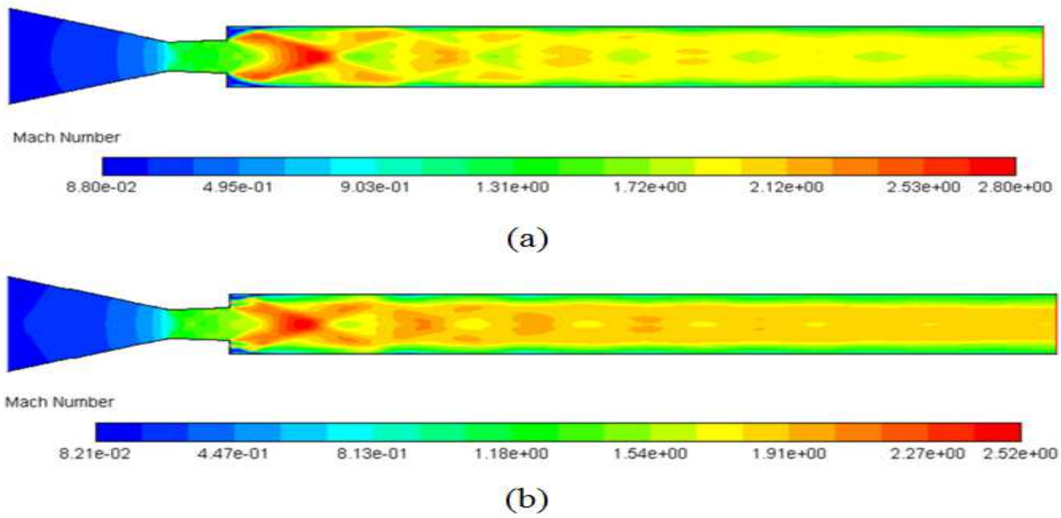


Figure 6: Close View of CD Nozzle with Mach number for L/D = 10 and NPR=11 (Mach number) (a) With Control (b) Without Control

Figure 7 illustrate the Mach number from inlet to the outlet with and without control of micro jets for L/D = 10 and NPR = 11. The singularity which shows in this figure 7, the increases when it reaches at the end of the enlarged duct it means that the Mach number is high at the exit region. When micro jets were employed flow-process slightly will change and control the Mach number at the suddenly expanded duct. Moreover, because of constant wall position at the suddenly enlarged duct position of the divergent region Mach number found vertical line.

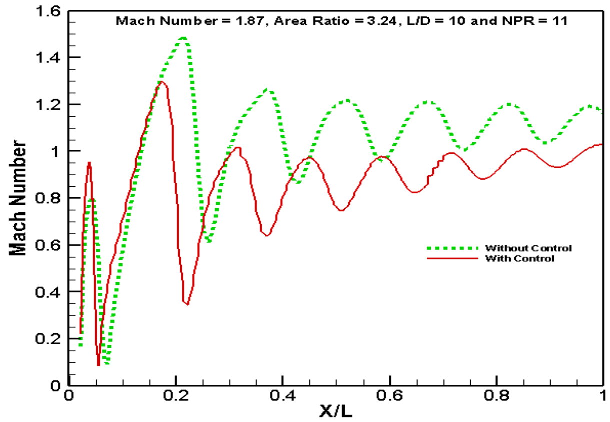
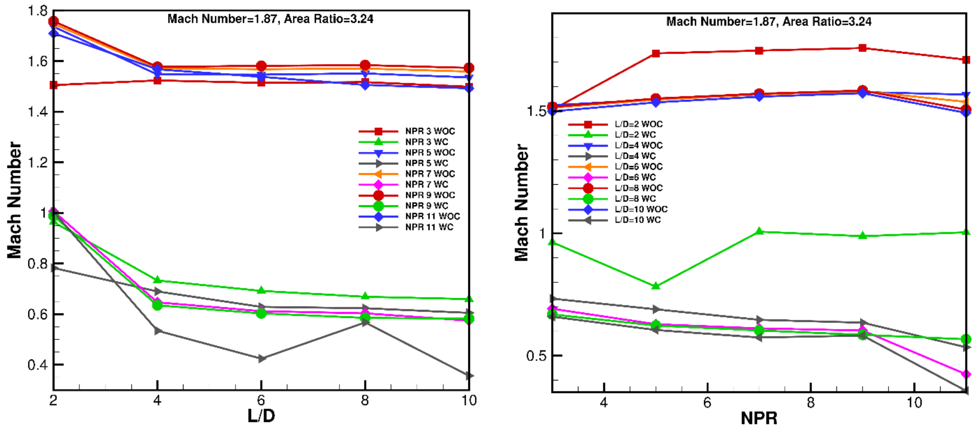


Figure 7: Mach number for L/D=10 and NPR=11

Subsequently, Figure 8 shows the Mach number variation for different NPR and L/D without and with micro jet control. From the figure 8 it has been identified that the variation of L/D and NPR is very less. As specified at literature Khan et al., (2018) obtained the results with respect to complete wall and complete base location. Therefore, this paper represents only for one selected location of the wall and base to perform all cases.



(a) **(b)**

Figure 8: Close View of CD Nozzle with Mach number (a) With Control (b) Without Control

CONCLUSIONS

The pressure and Mach number flow-field has been demonstrated using FE method by ANSYS software. Designed Mach number 1.87 considered for different NPR and L/D ratio of an area ratio 3.24. The effect of micro-jets was found effective in performing the expansion level and fixed level of inertia. In active control the pressure in the duct is not negatively affected. From the above discussion, conclude that, total pressure dramatically varying from inlet to the outlet and value of pressure is high for NPR 11. As pressure decreases velocity will increase and our results proved that the pressure is low by considering the total wall pressure flow (shown in Figure 4) and velocity is high detected by seeing Mach number (shown in Figure 7) at the exit.

REFERENCES

1. *A. Khan, A. Aabid, and S. A. Khan, "CFD analysis of convergent-divergent nozzle flow and base pressure control using micro-JETS," Int. J. Eng. Technol., vol. 7, no. 3.29, pp. 232–235, 2018.*

2. *S. A. Khan and A. Aabid, "CFD Analysis of CD Nozzle and Effect of Nozzle Pressure Ratio on Pressure and Velocity For Suddenly Expanded Flows," Int. J. Mech. Prod. Eng. Res. Dev., vol. 8, no. June, pp. 1147–1158, 2018.*

3. *J. S. Anderson and T. J. Williams, "Base pressure and noise produced by the abrupt expansion of air in a cylindrical duct," J. Mech. Eng. Sci., vol. 10, no. 3, pp. 262–268, 1968.*

4. *Amer, Y. A., & El-Shourbagy, S. M. Active Control Of A Cantilever Beam Subject To Parametric Excitation Via Negative Feedback Velocity.*

5. *S. A. Khan and E. Rathakrishnan, "Active Control of Suddenly Expanded Flows from Overexpanded Nozzles," Int. J. Turbo Jet Engines, vol. 19, pp. 119–126, 2002.*

6. *S. A. Khan and E. Rathakrishnan, "Control of Suddenly Expanded Flows with Micro-Jets," Int. J. Turbo Jet Engines, vol. 20, pp. 63–82, 2003.*

7. *S. A. Khan and E. Rathakrishnan, "Control of Suddenly Expanded Flows from Correctly Expanded Nozzles," Int. J. Turbo Jet Engines, vol. 21, pp. 255–278, 2004.*

8. *S. A. Khan and E. Rathakrishnan, "Active Control of Suddenly Expanded Flows from Underexpanded Nozzles," Int. J. Turbo Jet Engines, vol. 21, pp. 233–254, 2004.*

9. S. A. Khan and E. Rathakrishnan, "Active Control of Suddenly Expanded Flows from Underexpanded Nozzles - Part II," Int. J. Turbo Jet Engines, vol. 22, pp. 163–183, 2005.

10. S. A. Khan and E. Rathakrishnan, "Nozzle Expansion Level Effect on Suddenly Expanded Flow Sher," Int. J. Turbo Jet Engines, vol. 23, pp. 233–257, 2006.

11. S. A. Khan and E. Rathakrishnan, "Control of suddenly expanded flow," Aircr. Eng. Aerosp. Technol. An Int. J., vol. 78, no. 4, pp. 293–309, 2006.

12. S. Rehman and S. A. Khan, "Control of base pressure with micro-jets : part I," Aircr. Eng. Aerosp. Technol., vol. 80, no. 2, pp. 158–164, 2008.

13. M. Ahmed and A. L. I. Baig, "Active control of base pressure in suddenly expanded flow for area ratio 4. 84," Int. J. Eng. Sci. Technol., vol. 4, no. 05, pp. 1892–1902, 2012.

14. G. M. Kumar, D. X. Fernando, and R. M. Kumar, "Design and Optimization of De Lavel Nozzle to Prevent Shock Induced Flow Separation," vol. 3, no. 2, pp. 119–124, 2013.

15. Karna S. Patel, "Flow analysis and optimization of supersonic rocket engine nozzle at various divergent angle using Computational Fluid Dynamics (CFD)," IOSR J. Mech. Civ. Eng., vol. 11, no. 6, pp. 1–10, 2014.

16. N. D. Deshpande, S. S. Vidwans, P. R. Mahale, R. S. Joshi, and K. R. Jagtap, "theoretical and CFD Analysis of De-Laval Nozzle," Int. J. Mech. Prod. Eng., vol. 2, no. 4, pp. 2320–2092, 2014.

17. O. J. Shariatzadeh, A. Abrishamkar, and A. J. Jafari, "Computational Modeling of a Typical Supersonic Converging-Diverging Nozzle and Validation by Real Measured Data," J. Clean Energy Technol., vol. 3, no. 3, pp. 220–225, 2015.

18. O. Kostic, Z. Stefanovic, and I. Kostic, "CFD modeling of supersonic airflow generated by 2D nozzle with and without an obstacle at the exit section," FME Trans., vol. 43, no. 2, pp. 107–113, 2015.

19. T. J. Ajoko and T. J. Tuaweri, "Design Optimisation of Convergent-Divergent Aircraft Nozzle," vol. 8, no. 1, pp. 9–16, 2017.

20. K. A. Pathan, S. A. Khan, and P. S. Dabeer, "CFD Analysis of Effect of Area Ratio on Suddenly Expanded Flows," in 2nd International Conference for Convergence in Technology (I2CT) CFD, 2017, pp. 1192–1198.

21. K. A. Pathan, S. A. Khan, and P. S. Dabeer, "CFD Analysis of Effect of Mach number, Area Ratio and Nozzle Pressure Ratio on Velocity for Suddenly Expanded Flows," in 2nd International Conference for Convergence in Technology (I2CT) CFD, 2017, pp. 1104–1110.

22. K. A. Pathan, S. A. Khan, and P. S. Dabeer, "CFD Analysis of Effect of Flow and Geometry Parameters on Thrust Force Created by Flow from Nozzle," in 2nd International Conference for Convergence in Technology (I2CT) CFD, 2017, pp. 1121–1125.

23. D. S. Sahana and A. Aabid, "CFD Analysis of Box Wing Configuration," Int. J. Sci. Res., vol. 5, no. 4, pp. 706–709, 2016.

24. K. A. Pathan, P. S. Dabeer, and S. A. Khan, "Optimization of Area Ratio and Thrust in Suddenly Expanded Flow at Supersonic Mach Numbers," Case Stud. Therm. Eng., 2018.

25. ANSYS Inc, "ANSYS FLUENT 15.0: Theory Guidance," Canonsburg PA, 2016.

26. ANSYS Inc, "ANSYS FLUENT 18.0: Theory Guidance," Canonsburg PA, 2017.