Numerical Simulation of Suddenly Expanded Flow at Mach 2.2

Abdul Aabid, Ambareen khan, Nurul Musfirah. Mazlan, Mohd Azmi. Ismail, Mohammad Nishat Akhtar, S. A. Khan

ABSTRACT- A numerical simulation has been performed to investigate the control of base pressure with microjets in a suddenly expanded duct. Microjets placed at the pitch circle diameter (PCD) of 13 mm, two micro jets of 1 mm orifice diameter located at 900 for active control. The flow Mach number of the investigation was M = 2.2, the L/D ratio of the enlarged duct considered is 6, and the area ratio is 3.24. The convergent-divergent (CD) nozzle geometry has been modeled and simulated employing K-ε turbulence model for standard wall function. From the code independently was checked with the commercial computational fluid dynamics software. The numerical simulations carried for nozzle pressure ratio's (NPR) 3, 5, 7, 9 and 11. From the present numerical investigation, it is observed that the NPR, Mach number, and area ratio plays a vital role in fixing the base pressure values. NPR's of the present study is such that the flow mostly remained over expanded. Despite jets being over-expanded the control is effective in decreasing the base suction and hence the base drag.

Keyboards: M = 2.2, 3.24. 3, 5, 7, 9 and 11. NPR, Mach number

I. INTRODUCTION

In the field of high-speed flows by considering subsonic and supersonic regimes an unexpected expansion of flow is a foremost problem in many applications. In case of sudden expansion, the effectiveness of microjet plays an essential role in the various high-speed application. For rocket and jet engine test cells, it has been observed that the systems are used to simulate high-altitude and high-speed conditions; a jet produces sufficient discharge pressure which is a sub-atmospheric pressure. On the other hand, the base pressure at the blunt base too is sub-atmospheric which occurs at sub-sonic as well as at supersonic flow Mach numbers. At transonic Mach number, the contribution from this low base at the blunt base is significant, and it can be around seventy (%) percent of the net drag. Hence, it is critical to control the base pressure so that we can reduce the base drag at sonic Mach numbers and ultimately save the energy. Due to its wide-range application number of researchers have studied the behavior of fluid in the suddenly expanded duct. Khan et al. attempted to control the base pressure with active control using the finite element method [1], [2]. A similar study has also been carried out for Mach number 1.87 to control base pressure at supersonic Mach number [3]. Therefore, in the present study, an attempt is made to investigate the regulation of pressure in the base corner when control is present or not as the tiny jets using Computational fluid dynamics (CFD) method using ANSYS FLUENT.

The objective of this article is initially to design a supersonic CD Nozzle with the assumption of the isentropic flow of the perfect gas. The effect of NPR considered for the variation of pressure in the base of the recirculation zone as a function of inertia value of M = 2.2. However, the pressure and the velocity in the suddenly expanded duct have been observed by evaluating contour figures, base pressure, and wall pressure plots. Moreover, a design procedure which can determine the configuration of a suddenly expanded flow and the results of (Sher Afghan Khan & Rathakrishnan, 2003) are reproduced using ANSYS commercial software.

II. LITERATURE REVIEW

From the last few decades, in the literature, it is found that various studies conducted with sudden expansion. However, all cases are for flow and geometrical parameters. Numerical simulation and experiments investigated done for the effect of the entrance geometry of the gun nozzle on the high-velocity oxygen-fuel (HVOF) process. Using a numerical simulation method has been identified the process variation inside the nozzle associated with the coating properties [4]. The experimental method used to study the microjets usefulness to control pressure in the recirculation area of the flow from convergent-divergent (CD) nozzle with sudden expansion for different NPR, the diameter ratio and length to diameter (L/D) ratio of sudden expansion for different Mach number [5]–[12]. A numerical simulation for two different combustion noise source mechanisms was presented using a CFD approach in combination with appropriate acoustic boundary conditions. The Entropy Wave Generator (EWG) experiment is taken for validation of the proposed approach and for evaluating the acoustic sources of entropy noise.

Manuscript published on 28 February 2019.
* Correspondence Author (s)
Abdul Aabid Department of Mechanical Engineering, Faculty of Engineering, International Islamic University Malaysia, Kuala Lumpur, Malaysia.
Ambareen khan School of Aerospace Engineering, University Sains Malaysia, Nibong Tebal, Penang, Malaysia.
Nurul Musfirah. Mazlan School of Aerospace Engineering, University Sains Malaysia, Nibong Tebal, Penang, Malaysia.
Mohd Azmi. Ismail School of Mechanical Engineering, Engineering Campus, Universiti Sains Malaysia, Nibong Tebal, Penang, 14300, Malaysia
Mohammad Nishat Akhtar School of Aerospace Engineering, University Sains Malaysia, Nibong Tebal, Penang, Malaysia.
S. A. Khan Department of Mechanical Engineering, Faculty of Engineering, International Islamic University Malaysia, Kuala Lumpur, Malaysia.

The numerical approach RPM-CN approach was developed to predict broadband combustion noise. The accuracy of the RPM-CN approach demonstrated by a good agreement of the simulation results with acoustic measurements of the DLR-A flame. Ali et al. used secondary fluidic injection in a CD nozzle of modulating exhaust thrust using a numerical method. Fluidic Thrust Modulation (FTM) can be of significant benefit in some cases like the solid-fuelled rockets which are challenging to throttle through conventional means. It is divided into two methods; Shock Thrust Modulation (STM) and the Throat Shifting Thrust Modulation (TSTM) [13]. Experimentally investigated the flow from CD nozzle in suddenly expanded axisymmetric duct, concentrating our attention on the pressure in the base region, wall pressure, and the flow development in the sudden expansion duct. The study was done for different NPR, the area ratio, the L/D ratio and the Mach number [14]. Kumar et al., designed a de-Laval nozzle to accomplish supersonic flow and optimized it to achieve maximum thrust without any flow separation due to Shock waves. The flow direction must be in parallel to the nozzle exit for maximum thrust and efficiency. The pressure and temperature flow conditions selected and gases that are available at the exit of the combustion chamber. The Shock-induced flow separation due to overexpansion and optimum expansion conditions at the exit of the nozzle has also been studied [15]. Cai et al. used machined surface topology for the performance of the micronozzle were primarily determined. A circular cross-section micro-Laval nozzle is modeled and analyzed by using numerical simulation [16]. A numerical analysis of CD nozzle with Mach number M = 2.6 at nozzle exit of the flow has been studied and achieved a good agreement with available experimental data, obtained during supersonic wind tunnel tests at VTI Žarkovo institute. Structured mesh for both cases generated, and for the numerical simulation, the RANS equations with k-ω SST turbulent model have been applied [17]. A numerical simulation of a flow in a typical supersonic CD nozzle has been studied for the case of 2D or 3D domains to provide a better comparative platform. The effect of the turbulence model, differentiation and computational grid to the solution was also studied. Furthermore, the numerical comparison between CFD modeling results and corresponding available measured data were presented [18]. The flow through the CD nozzle using a finite volume rewarding code, FLUENT 6.3 were numerically simulated. The modeling and mesh have been done using GAMBIT 2.4 Software. The results obtained using numerical simulation found a good agreement with available experimental results from the literature [19]. They have determined the optimal Nozzle parameters such as Convergent angle, Divergent angle, Throat radius for a rocket nozzle. CFD simulation has been used from ANSYS FLUENT 14.5 with suitable Boundary condition. In CFD simulation suitable values have been determined by Flow Parameters. Taguchi design is implemented for Optimization purpose [20]. Theoretical method and numerical simulations have been carried out to optimize the particle–gas flows through a CD nozzle. Homogeneous and equilibrium model that no slip in velocity and temperature occurs between particle and gas phase has been considered to derive mass flow rate and sound speed of multiphase flows. Using CFD simulation a discrete phase model that consists of Lagrangian-Eulerian tracking method was used to calculate particle–gas flows. The mass flow rate and speed of sound for two-phase flows were investigated theoretically and compared with CFD results. [21]. Investigated in the divergent section of the engine exhaust nozzle was enhanced the active control device that deflects a fraction of the adjustable seals. The deflected seals are returned to their undeflected position after take-off, and favorable jet noise reductions were demonstrated at multiple observer angles for minor take-off conditions [22]. Numerical simulation has been used to investigate the effects of H_2S feed concentration, operating parameters and nozzle geometry on the condensation process [23]. Using CFD analysis, investigated the efficacy of the flow at high speed when the flow is exhausted into a circular duct with sudden expansion and that leading to generate a thrust force and also investigate the development of the flow in a pipe having a circular cross-section, measurement of its magnitude [24].

III. PROBLEM DEFINITION

The CD nozzle with sudden expansion duct has been modeled based on the designed Mach number and the experimental work done by the Khan et al. the dimension in this problem is considered in mm which is shown in figure 1. The active control microjets are of 1 mm of diameter and located at the PCD of 13 mm from the divergent diameter duct. Since the flow through the microjets, can be at most at Mach number unity for NPR around 2, with further increase in the NPR values will result in microjets becoming under-expanded. The purpose of the investigation is to analyses the flow-field through the CD nozzle and prediction of theoretical flow parameters such as pressure and velocity with the effect of different parameters proving by CFD simulation in 2D modeling with and without micro-jets.

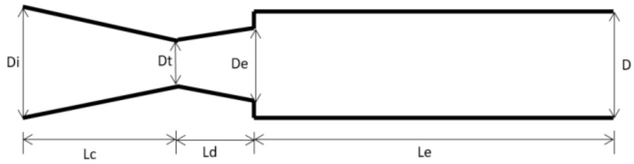


Figure 1: FE design of CD nozzle with the suddenly expanded duct.

The dimensions of the nozzle are mentioned in table 1.

Table 1: Dimension of CD nozzle

Mach Number	1.87
Inlet diameter (Di)	27 mm
Throat diameter (Dt)	6.5 mm
Exit diameter (De)	10 mm
Extended diameter (D)	18 mm
Convergent length (Lc)	35 mm
Divergent length (Ld)	17 mm
Extended length (Le)	108 mm
Micro-jets diameter (Dm)	1 mm

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

IV. CFD ANALYSIS

ANSYS commercially available tool has been employed to perform a CFD simulation using finite element (FE) method.

5.1. geometry and modeling

To design the CD nozzle used ANSYS workbench and the modeling has been done based on the available experimental data. For the present case, the only fluid area of nozzle considered to optimize the CFD simulation through the CD nozzle. Therefore, A two-dimensional planar model of CD nozzle was designed and modeled which is being shown in figure 1.

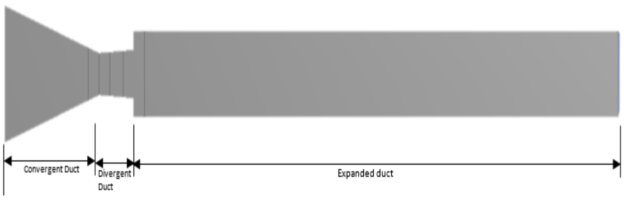


Figure 2: 2D planar CD nozzle

5.1. Meshing and boundary conditions

To mesh the design CD nozzle considered a finite element mesh by structural meshing showed in figure 3. In a high-stress area, the number of elements applied maximum due to that the mesh is higher which is more suitable for good results. Total, 38,368 binary nodes, were generated for the 2D planar model in the structural mesh. Suitable boundary conditions are defined by considering edges of the plane to initiate the solution with the perfect ideal flow.

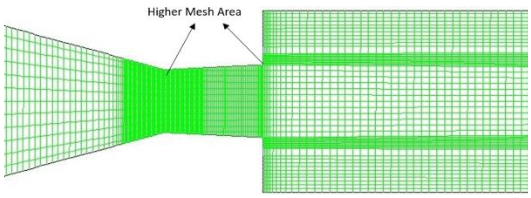


Figure 3: Structural mesh for 2D planar CD nozzle

5.1. Calculation Procedure

For CFD simulation, to analyses, the flow field inside the volume was done using RANS (Reynolds-Averaged Navier-Stokes) equations with the k-ε standard turbulent model [25]. Table 2 shows the most critical setting which has been used to simulate the results in the present study.

Table 2: Solution Set up

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

5.1. Validation of CFD results

In order to validate the present FE results, the CD nozzle with control of micro-jets at the base with suddenly expanded duct shown in figure 1 of Khan et al., [6] is considered. By comparing the results obtained from Khan et al., (2003) and the present FE results, the results found a good agreement with less than 5.0% discrepancy as illustrated in table 3.

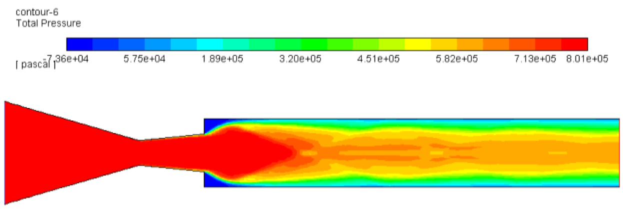
Table 3: Verification of Present FE Results

Pressure (Case: L/D = 6)	Khan et al., (2003) (Experimental)	Present Results (Numerical)	Relative Error
Without control at base (Pb/Pa) (NPR 3)	0.67	0.7	4.29%

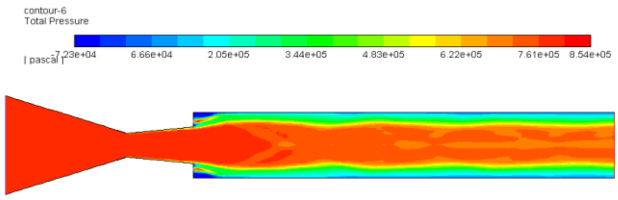
V. RESULTS AND DISCUSSION

5.1. Effect of Pressure

The pressure for a CD nozzle plays a crucial role in high-speed jets/flows. Figure 4 describes pressure flow for L/D = 6 and NPR = 9, at the base the oblique shock waves are created due to the sudden expansion of a CD nozzle as the nozzle is over-expanded. At the base corner having a recirculation flow which will create low pressure and hence the base drag. To minimize the drag and increase the forward flow to the outlet portion of the nozzle which may give high thrust. The performance of the microjets at the base area results in recirculation becoming low as shown in figure 4(b).



(a)

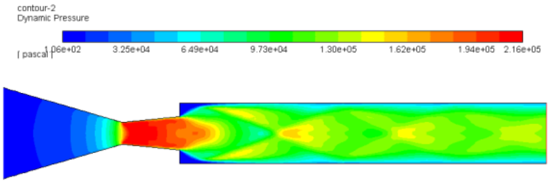


(b)

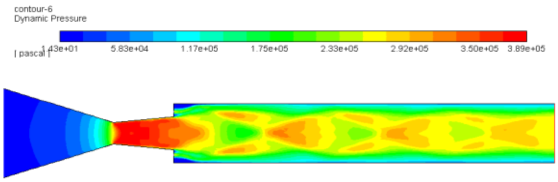
Figure 4: Total Pressure Distribution for NPR 9 and L/D = 6, (a) without Microjet Control, (b) with Microjet Control.

In order to study the pressure flow field, when we observe the dynamic pressure and it is found to be low in the base corner as shown in figure 5. The results show that pressure is high at divergent duct when it contacts with the base region it expands and generates shock waves. Moreover, the phenomenon is different when the microjets are presents shown in figure 5(b).

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)



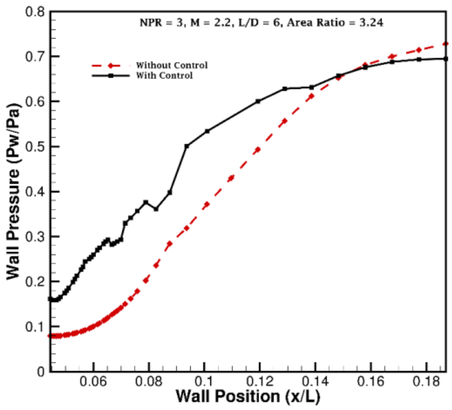
(a)



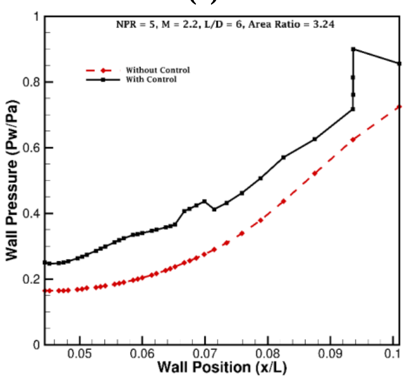
(b)

Figure 5: Dynamic Pressure Distribution for NPR 9 and L/D = 6, (a) without Microjet Control, (b) with Microjet Control.

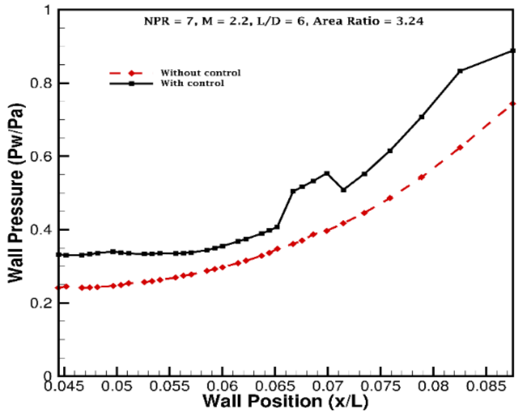
Base pressure/wall pressure values for L/D ratio = 6 are depicted in Figure 6 at Mach 2.2 at various NPR in the range from 3 to 11. Fig. 6(a) shows results for NPR = 3, when we compare the results with and without control, it is seen that there is a substantial increase in the base/wall pressure, and control reversal takes place at X/L = 0.15, where control results in a decrease of the base/wall pressure. Since we are analyzing the results within the reattachment length, hence the pressure within this reattachment length will be the base pressure. Similar results are seen in Fig. 6(b) where the control amounts in an enhancement of the pressure at the base corner by fifty percent all along the length of the duct and the control reversal do not take place in the downstream of the duct length. The reasons for this trend may be due to the decrease in the level of over-expansion the strength of the shock wave will decrease and also due to the small area ratio, the tiny jets are adequate even when the jets are overexpanded.



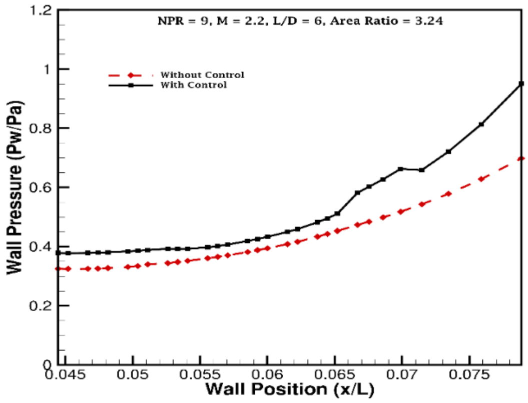
(a)



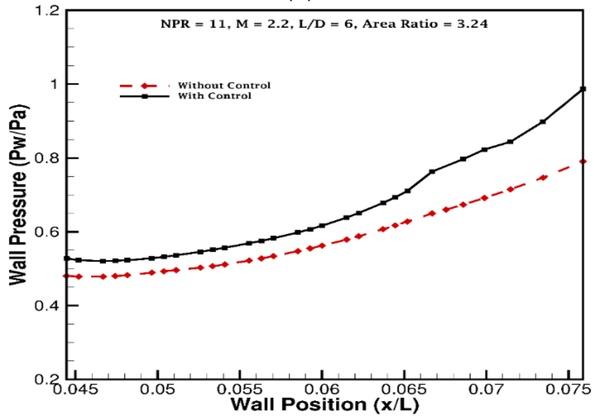
(b)



(c)



(d)



(e)

Figure 6: Duct Flow field in the presence and absence of Control

The base pressure results for the remaining NPR's are shown in Figs. 6((c) to (e)). They are the representative of the similar results as was seen for NPR 5 with the exception that due to the continuous increase of NPR that results in a continuous decrease over expansion resulting larger numerical value of the pressure at base even for the without control case. Under these circumstances when the controls have activated the control can decrease the suction given the presence of robust vortex as the active control in the form of microjets can break the vortices increasing base pressure appreciably. Further, the results presented in Figs. 6((a) to (e)) indicate that the deployment of the control as microjets does not aggravate the flow field in the duct which is the major advantage.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

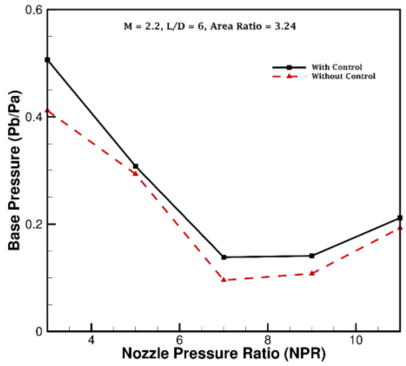


Figure 7: Base Pressure Variation Vs. NPR

The pressure at the base recirculation zone is shown in Fig. 7. The results demonstrate that the base pressure either in the presence or absence of the control mechanism behaves identically. Fig. 7 indicates that a continuous decrement in the base pressure values with increase in the NPR, resulting in a decrease in the level of overexpansion. It is seen that for NPR more magnificent than seven a sudden change in the base pressure trend is noticed, and the base pressure which was continuously decreasing with the NPR has started increasing for both with and without control. Under these circumstances when the controls are used the control amounts to an increment of the pressure in the base corner for the entire range of the NPR.

5.2. Mach Number Variation

The value of the Mach number is increased at the outlet of the CD nozzle. Therefore, to obtain all results, they selected the one location of the base and wall to identify all L/D and NPR values. Figure 7 illustrates the contours for the case of L/D = 6 and NPR 9 which shows from the different color variation lowest to the highest values at the CD nozzle. From the figure 8(b) it is clearly showing that at the recirculation zone Mach number becomes very low this means to reduce the drag, and the forwarded the flow to the exit of the nozzle which may result in increases the Mach Number.

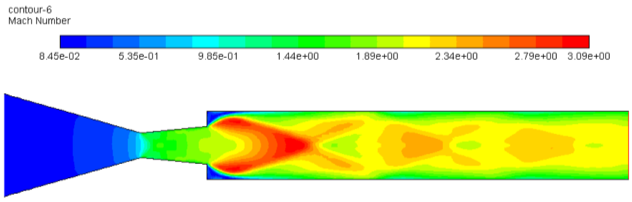


Figure 6: Mach Number Variation for NPR 9 without Microjet Control

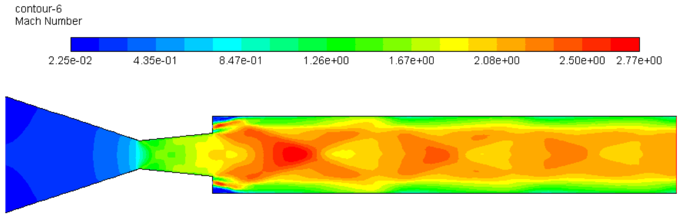


Figure 7: Mach Number Variation for NPR 9 with Microjet Control

Table 4: Effect of NPR on Mach Number for Suddenly Expanded Flow

S. No.	NPR	Without Control			With Control		
		Inlet	Outlet	Net	Inlet	Outlet	Net
1	3	0.13488306	0559352	0.30467063	0.23155495	0.73985856	0.42622441
2	5	0.140328275	1.9636366	0.86968185	0.25067901	1.3539075	0.67319205
3	7	0.14038914	1.9708018	0.87255416	0.26488771	1.8634172	0.87709046
4	9	0.14039682	1.9708711	0.87258649	0.2748451	1.8985907	0.8967051
5	11	0.14039269	2.019718	0.89190432	0.28075138	1.947251	0.91898525

VI. CONCLUSION

The usefulness of base pressure control in the form of tiny jets to regulate base pressure level is adequate. These jets serve as an adequate controller, raising the base pressure magnitude adequate for the above combinations of variables. There is no negative impact of the control on the flow field in the downstream beyond the nozzle, as the flow field is identical in the presence and absence of the flow regulation mechanism. The NPR has a vital role in fixing the numerical value of the pressure at the base corner either in the presence or absence of the control mechanism at large Mach number. The pressure and Mach number flow-field have been demonstrated using the FE method by ANSYS software. Designed Mach number 2.2 considered for different NPR and L/D = 6 of area ratios 3.24. The effect of microjets was found to be useful in decreasing base suction for various expansion level and a fixed level of inertia. In the case of active control, the pressure in the duct is not affected adversely. From the above discussion, it is concluded that the active control by micro jets seems to be effective at all the NPR of the present investigation.

REFERENCES

1. 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," *International Journal of Mechanical and Production Engineering Research and Development*, vol. 8, no. June, pp. 1147–1158, 2018.
2. A. Khan, A. Aabid, and S. A. Khan, "CFD analysis of convergent-divergent nozzle flow and base pressure control using micro-JETS," *International Journal of Engineering and Technology*, vol. 7, no. 3.29, pp. 232–235, 2018.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

3. A. G. M. Fharukh, A. A. Alrobaian, A. Aabid, and S. A. Khan, “Numerical Analysis of Convergent-Divergent Nozzle Using Finite Element Method,” *International Journal of Mechanical and Production Engineering Research and Development*, vol. 8, no. 6, pp. 373–382, 2018.
4. K. Sakaki and Y. Shimizu, “Effect of the Increase in the Entrance Convergent Section Length of the Gun Nozzle on the High-Velocity Oxygen Fuel and Cold Spray Process,” *Journal of Thermal Spray Technology*, vol. 10, no. 3, pp. 487–496, 2001.
5. S. A. Khan and E. Rathakrishnan, “Active Control of Suddenly Expanded Flows from Overexpanded Nozzles,” *International Journal of Turbo and Jet Engines*, vol. 19, pp. 119–126, 2002.
6. S. A. Khan and E. Rathakrishnan, “Control of Suddenly Expanded Flow with Micro-Jets,” *International Journal of Turbo and Jet Engines*, vol. 20, pp. 63–82, 2003.
7. S. A. Khan and E. Rathakrishnan, “Control of Suddenly Expanded Flows from Correctly Expanded Nozzles,” *International Journal of Turbo and Jet Engines*, vol. 21, pp. 255–278, 2004.
8. S. A. Khan and E. Rathakrishnan, “Active Control of Suddenly Expanded Flows from Underexpanded Nozzles,” *International Journal of Turbo and 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,” *International Journal of Turbo and Jet Engines*, vol. 22, pp. 163–183, 2005.
10. S. A. Khan and E. Rathakrishnan, “Control of suddenly expanded flow,” *Aircraft Engineering and Aerospace Technology: An International Journal*, vol. 78, no. 4, pp. 293–309, 2006.
11. S. A. Khan and E. Rathakrishnan, “Nozzle Expansion Level Effect on Suddenly Expanded Flow Sher,” *International Journal of Turbo and Jet Engines*, vol. 23, pp. 233–257, 2006.
12. S. Rehman and S. A. Khan, “Control of base pressure with micro-jets : part I,” *Aircraft Engineering and Aerospace Technology*, vol. 80, no. 2, pp. 158–164, 2008.
13. A. Ali, A. Neely, J. Young, B. Blake, and J. Y. Lim, “Numerical Simulation of Fluidic Modulation of Nozzle Thrust,” in *17th Australasian Fluid Mechanics Conference*, 2010, no. December, pp. 5–8.
14. M. A. A. Baig, S. A. Khan, C. Ahmed Saleel, and E. Rathakrishnan, “Control of base flows with microjet for an area ratio of 6.25,” *ARPN Journal of Engineering and Applied Sciences*, vol. 7, no. 8, pp. 992–1002, 2012.
15. G. M. Kumar, D. X. Fernando, and R. M. Kumar, “Design and Optimization of De Laval Nozzle to Prevent Shock-Induced Flow Separation,” vol. 3, no. 2, pp. 119–124, 2013.
16. Y. Cai, Z. Liu, Z. Shi, Q. Song, and Y. Wan, “Residual surface topology modeling and simulation analysis for the micro-machined nozzle,” *International Journal of Precision Engineering and Manufacturing*, vol. 16, no. 1, pp. 157–162, 2015.
17. O. Kostic, Z. Stefanovic, and I. Kostic, “CFD modeling of supersonic airflow generated by a 2D nozzle with and without an obstacle at the exit section,” *FME Transaction*, vol. 43, no. 2, pp. 107–113, 2015.
18. O. J. Shariatzadeh, A. Abrishamkar, and A. J. Jafari, “Computational Modeling of a Typical Supersonic Converging-Diverging Nozzle and Validation by Real Measured Data,” *Journal of Clean Energy Technologies*, vol. 3, no. 3, pp. 220–225, 2015.
19. B.-A. Belega and T. D. Nguyen, “ANALYSIS OF FLOW IN CONVERGENT-DIVERGENT ROCKET ENGINE NOZZLE USING COMPUTATIONAL FLUID DYNAMICS,” 2015.
20. D. D. Majil, A. Poojitha, and G. Devi, “Computational Study on Optimization of Rocket Nozzle,” 2016, pp. 8–13.
21. G. Zhang and H. D. Kim, “Theoretical and numerical analysis on choked multiphase flows of gas and solid particle through a convergent-divergent nozzle,” *The journal of computational multiphase flow*, vol. 0, no. 0, pp. 1–14, 2017.
22. A. R. Pilon, R. W. Powers, D. K. McLaughlin, and P. J. Morris, “Design and Analysis of a Supersonic Jet Noise Reduction Concept,” *Journal of Aircraft*, vol. 54, no. 5, pp. 1705–1717, 2017.
23. W. Sun, X. Cao, W. Yang, and X. Zhao, “CFD modeling on non-equilibrium condensation process of H2S in CH4-H2S mixture expansion through supersonic nozzles,” *Fuel Processing Technology*, vol. 170, no. November 2017, pp. 53–63, 2018.
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 Studies in Thermal Engineering*, 2018.
25. ANSYS Inc, “ANSYS FLUENT 18.0: Theory Guidance,” Canonsburg PA, 2017.

Published By:
Blue Eyes Intelligence Engineering and Sciences Publication (BEIESP)