Penerbit Akademia Baru

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

Analysis of Area Ratio In a CD Nozzle with Suddenly Expanded Duct using CFD Method

Open Access

Sher Afghan Khan[1*], Abdul Aabid[1], Fharukh Ahmed Mehaboobali Ghasi[2,3], Abdulrahman Abdullah Al-Robaian[4] , Ali Sulaiman Alsagri[4]

[1] Department of Mechanical Engineering, Faculty of Engineering, International Islamic University Malaysia, 53100, Kuala Lumpur, Malaysia
[2] Research Scholar, Department of Mechanical Engineering, Bearys Institute of Technology, Mangalore, Karnataka, India
[3] Assistant Professor, Department of Mechanical Engineering, Government Engineering College, Huvinahadagali, Karnataka, India
[4] Department of Mechanical Engineering, Qassim University, Saudi Arabia

ARTICLE INFO

Article history:
Received 31 March 2019
Received in revised form 3 May 2019
Accepted 11 May 2019
Available online 20 May 2019

Keywords:
CFD, NPR, L/D, Area Ratio, CD Nozzle, and Mach number

ABSTRACT

Whereas the demand for missiles and rockets has exponentially augmented, but the difficulties related to the gas dynamics of these vehicles remains to be a threat. Whenever there is a sudden expansion, the pressure in the downstream is sub-atmospheric. This low pressure in the recirculation zone leads to a considerable amount of drag, which is nearly two-thirds of the net drag of the aerospace vehicles. Hence, in view of the above problem, many researchers tried to control the base pressure depending upon the mission requirements. For example, in case of combustion chamber, we would like to decrease the base pressure as low as possible to have better mixing and efficient combustion, whereas, in case of the external ballistics application we would like to increase the base pressure as high as possible to reduce the drag of the projectiles to enhance the range. In this paper, investigated the effects of an area ratio in suddenly expanded duct and base pressure control with microjets using the computational fluid dynamics (CFD) method. A 1 mm orifice diameter of microjets placed at the pitch circle diameter (PCD) of 13 mm, located at 90^0 for active control. The Mach number is M = 2.2, the L/D ratio is 8, the nozzle pressure ratio's (NPR) is 9 and the area ratios are 2.56, 3.24, 4.86 and 6.25 considered in the present study. The design and modelling of convergent-divergent (CD) nozzle simulated using K-ε turbulence model for standard wall function. From the present results, it has been observed that the area ratio plays a crucial role in fixing the base pressure values.

1. Introduction

In the field of high-speed flows such as subsonic and supersonic flow regimes, an un-expected flow expansion is a leading problem in many of the fluid-flow applications. In case of sudden expansion flows, the microjets effect plays a vital role in a high-speed application. For jet and rocket engine's test cells, it is noticed that the systems are used to simulate high-speed and high-altitude conditions; a jet produces a forceful ejection of pressure which is called sub-atmospheric pressure.

* *Corresponding author.*
E-mail address: sakhan@iium.edu.my (Sher Afghan Khan)

On the other hand, a base pressure at the round base is sub-atmospheric which occurs at subsonic and supersonic flow Mach numbers. At transonic Mach number, the contribution from this low base at the round base is significant, and it can be as high as seventy percent of the total drag. Hence, it is essential to control the base pressure so that it can reduce the base drag at sonic Mach numbers and ultimately save the energy. Due to its wide-range application number of researchers were studying the behavior of fluid in the suddenly expanded duct. Khan *et al.,* [1–6] experimentally attempted to control the base pressure with active control.

The effect of microjets control for different NPR and L/D ratio has been studied very well and concluded that the base pressure drag reduces with uses of microjets without any loss of wall pressure [7–9]. Experimentally investigated the active controller to control the base pressure in suddenly expanded CD nozzle with the effect of the rotating cylinder for the different level of expansion for different Mach number and area ratio [10–13] and a passive controller to control the base pressure [14]. Quadros *et al.,* [15, 16] have reported the study of the efficacy of the flow variables on the pressure in a base region with suddenly expanded duct at significant inertia level using simulation and statistical methods.

The de-Lavel nozzle is designed to attempt supersonic flow and optimized to achieve maximum thrust without flow separation due to shock waves. A CD nozzle is designed based on the characteristic's method and the CFD method for various performance parameters analysis [17]. Numerical simulation has been used to optimize flow field in a typical CD nozzle for a supersonic flow regime by ANSYS Fluent commercial code. The CD nozzle designed and modeled for the case of two-dimensional (2D) and three-dimensional (3D) domain to get a better result [18]. The numerical study has been used to evaluate the pressure loss effect in CD nozzle with suddenly expanded duct with and without microjet controller at the base region to control the base pressure. To optimize the supersonic flow regime different parametric studies have been performed such as NPR, area ratio, L/D ratio for different Mach number using CFD method [19–23] and the CFD method used to simulate flow over a delta airfoil for the supersonic flow [24]. Pathan *et al.,* studied the flow distribution through an axisymmetric CD nozzle using CFD method. The important parameters considered by Pathan *et al.,* is the effect of area ratio, L/D ratio, NPR for different Mach number with and without microjet control [25–28]. Numerical simulation was carried out using ANSYS software and designed and modeled the CD nozzle with a suddenly expanded duct for Mach number 1.87, 2.22 and 2.58 to optimize pressure effect [29] and CFD simulation used for some other case of the study by the others [30-33].

From the previous work, it has been found that the number of studies has been taken in to account based on the Mach number, NPR, L/D ratio and area ratio separately and combined. Most of the studies have been found experimentally only few case studies using CFD method. Therefore, the present objective of this paper is to investigate the effect of an area ratio in a suddenly expanded duct of a CD nozzle for Mach number 2.2 using CFD method and the nozzle designed and modeled in ANSYS 18.0 commercial software. The parameter has been considered is NPR = 9 and L/D = 8. To optimize the results considered velocity streamlines and pressure plots for pressure and velocity effects from the post-processing method.

2. Methodology

2.1 The Geometry of a CD Nozzle with Suddenly Expanded Duct

The CD nozzle with sudden expansion duct has been designed and modeled based on the designed Mach number and the experimental work done by the Khan *et al.,* all the dimensions in this study is in "mm" which is shown in Figure 1. The micro-jets controller to actively control base pressure located at the PCD of 13 mm. The primary purpose of this study is to analyze the effect of area ratio and flow-field through a CD nozzle and the prediction of 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.

To design a CD nozzle with suddenly expanded duct illustrated in the Figure 1 and all the dimensions are; 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, and Micro-jets diameter (Dm) = 1 mm.

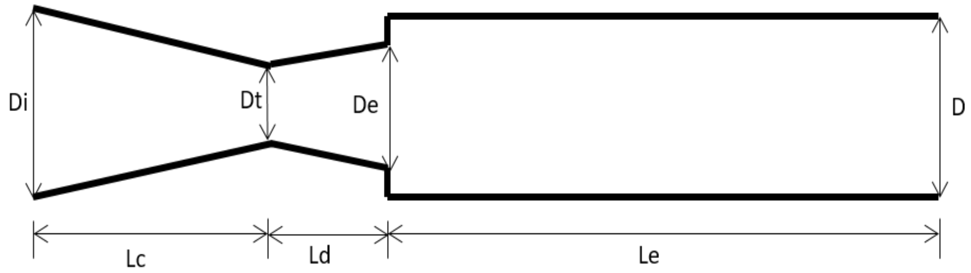


Fig. 1. FE Design of CD Nozzle with Suddenly Expanded Duct

2.2 Finite Element Modelling and Analysis

ANSYS Workbench 18.0 [34] available in our university for the academic purpose was used for numerical simulations and generation of the geometry. ANSYS geometry was utilized for pre-processing. The dimensions of the model are considered based on the designed Mach number data which is available in the experimental work of Khan *et al.,* [1]. The 2D model has been created with the surface of the CD nozzle in an X-Y plane which is shown in Figure 2. The boundary conditions were applied with the selection of edges (lines) by defined inlet is pressure inlet, the outlet is pressure outlet and the nozzle wall, the base wall suddenly expanded duct wall is considered as a wall.

Fluent was utilized for simulations and post-simulation data interpretations. To achieve very authentic results by the simulation, the mesh type of mesh considered was a structured one which has identical elements and nodes. Also, for the excellent results, the mesh must be refined manually using the size of the element by applying some element division. To develop a complete grid structured the converging length, throat location, diverging part, and the enlarged duct sections of the nozzle are divided in a different part, and every part meshes separately. Figure 3 shows the thoroughly structured meshed model.

For CFD simulation, the selection of a model is significant in the ANSYS software. The solutions of the governing equations were arrived by using (RANS) with the best and effective turbulent models to simulate the flow numerically. Well, the known k-ε standard model for modelling of turbulent flow is employed for the simulation of the viscous effects. The governing equations used for the simulations of the flow were a pressure-based and steady state for the analysis. Iterations were kept on changing till the flow is converged. The results were computed from the above method using an Intel i7 Core Xeon having a processor with a clock speed of 3.5 GHz, with random access memory (RAM) of 16 GB.

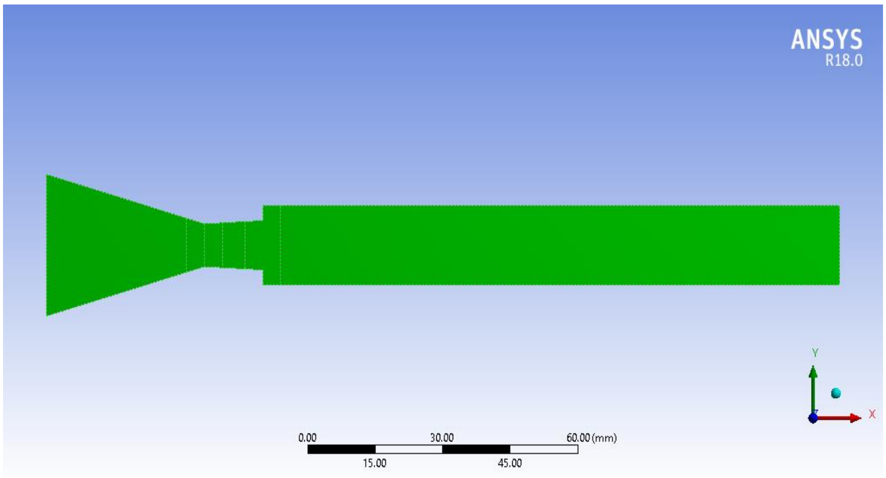


Fig. 2. Finite Element Model 2D planar CD nozzle for Area Ratio 2.56

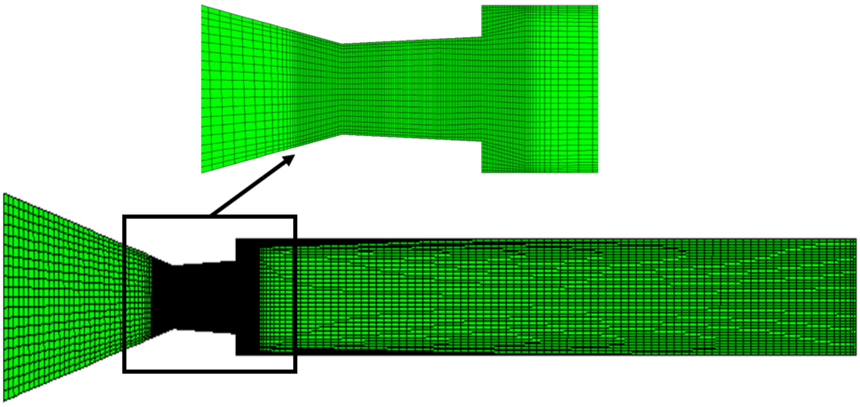


Fig. 3. Structural mesh for 2D planar CD nozzle for Area Ratio 2.56

2.3 Validation of CFD Results

In order to validate the present finite element model, considered a Khan *et al.,* [2] experimental work which is shown in Figure 1. The case was considered from the experimental data at Mach number 2.2 for area ratio 3.24, NPR 9 and L/D ratio 8. The present results show a good agreement with the experimental work. Table 1 illustrates the comparison of results.

Table 1
Comparison of present results

Base Pressure	Khan *et al.,* [2]	Present Work	Percentage Error
With control	0.12310	0.11609	5.6945
Without Control	0.10725	0.10888	1.4246

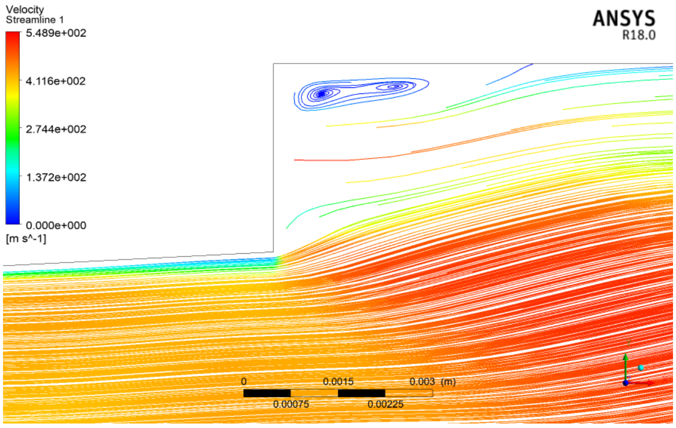
3. Results

In this finite element analysis with some cases has been considered and obtained the results using post-processing (results) tool in ANSYS workbench software. The velocity streamline and the plots for static pressure were developed from the simulation results. For area ratio 2.56, 3.24, 4.86 and 6.25, NPR 9 and Mach number 2.2 the velocity streamlines and pressure plot are shown in Figure 4 to 12 respectively.

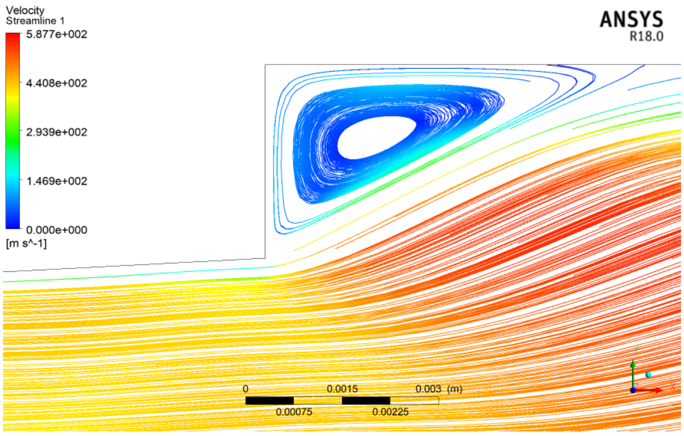
For the Figure 4, 6, 8, and 10 it can be seen that the close view of flow velocity streamlines immediate next to the diverging section of the nozzle is very high, and in the base region it is deficient which can be seen by the blue color. Obtain high velocity the adequate relief for expansion for the air should be available at the nozzle exit. The pressure at the base corner of the suddenly expanded pipe is minimum which is unwarranted as it gives rise to a large amount of base drag. From the previous studies, it is observed that with an increment in the area at the exit of the flow accelerating device, the effect of form drag is increased in the pipe and the pressure at the round base is decreased which is to be avoided [25–28].

For area ratio in the range 2.56 and 3.24 at NPR 9, the flow rate will be decreased in view of unattainability of the area for air expansion of the air from the stagnation chamber via CD nozzle, the net flow rate reduces, therefore, for NPR 9, the lesser relief to the flow is not appropriate. For higher relief and NPR 9 that gives the most significant mass flow rate of air at Mach number 2.2 due to the availability of the space.

Figure 5, 7, 9, 11, and 12 show the wall pressure distribution for L/D = 8 for area ratios 2.56, 3.24, 4.84, and 6.25. Figure 5 shows the wall pressure results for NPR = 9, when we compare the results with and without control, it is seen that there is a substantial increase in the wall pressure, and control reversal takes place at X/L = 0.0625, where control results in a decrease of the wall pressure. Since we are analyzing the results within the reattachment length, hence the pressure within this reattachment length will be the base pressure.



(a)



(b)

Fig. 4. Velocity Streamline for Area Ratio 2.56 (a) With Control (b) Without Control

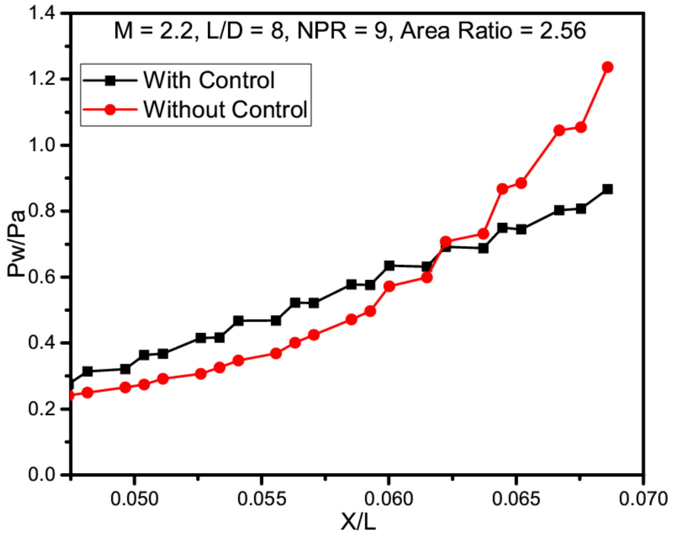
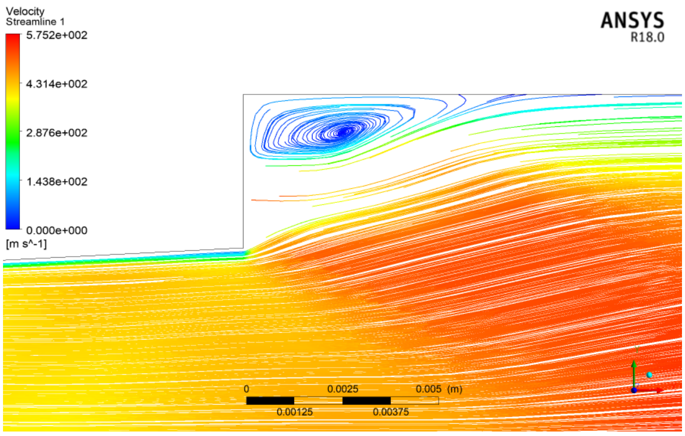
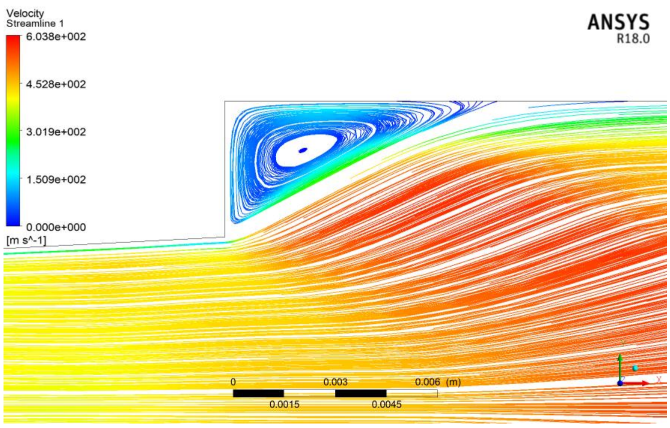


Fig. 5. Wall Pressure Distribution for Area Ratio 2.56



(a)



(b)

Fig. 6. Velocity Streamline for Area Ratio 3.24 (a) With Control (b) Without Control

For area ratio 3.24 the wall pressure is shown in Figure 7. Results indicate that there is continues increase in the wall pressure unlike for area ratio 2.56 where reattachment length is smaller than for this area ratio. The reason for this trend may be due to the additional relief available to the flow at the nozzle exit.

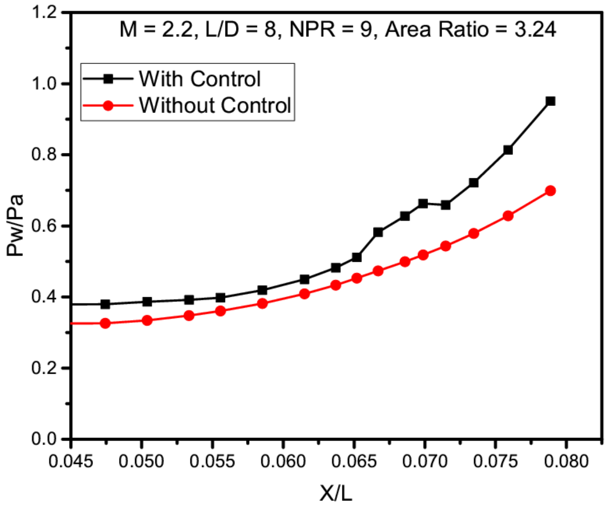
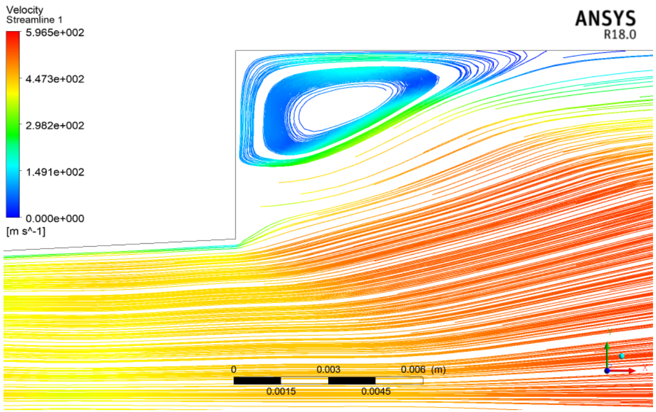
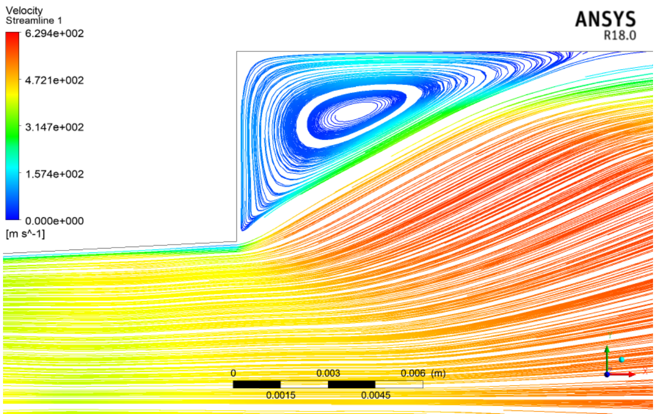


Fig. 7. Wall Pressure Distribution for Area Ratio 3.24



(a)



(b)

Fig. 8. Velocity Streamline for Area Ratio 4.86 (a) With Control (b) Without Control

Wall pressure results for area ratio 4.84 are shown in Figure 9. Since due to the increase in area ratio the base pressure will assume higher values, under these circumstances when the microjets are activated there is a marginal increase in the wall pressure, however, in the downstream of the duct, the wall pressure is increased substantially for X/L = 0.065 and above. This increase may be attributed due to the interaction of the shock wave with the duct wall.

CFD Letters
Volume 11, Issue 5 (2019) 61-71

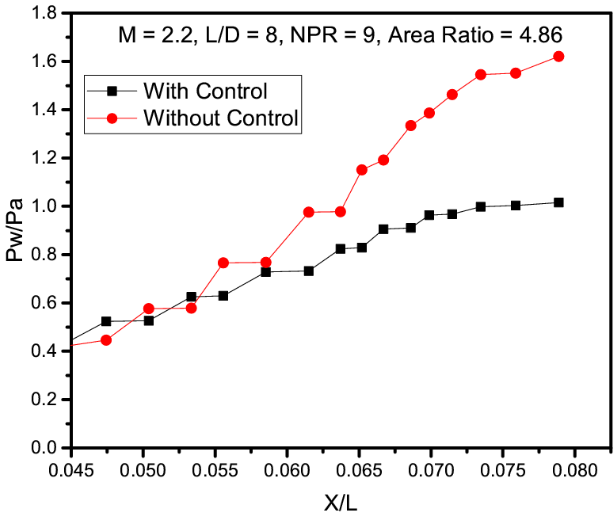
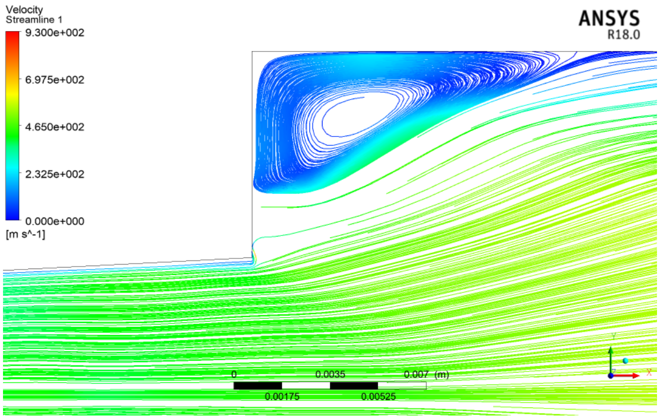
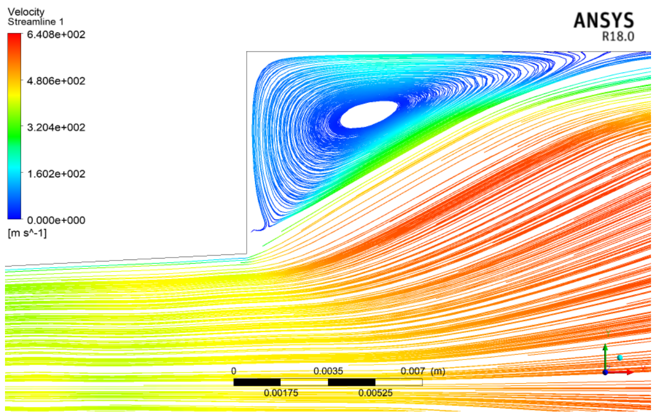


Fig. 9. Wall Pressure Distribution for Area Ratio 4.86



(a)



(b)

Fig. 10. Velocity Streamline for Area Ratio 6.25 (a) With Control (b) Without Control

Similar results are seen in Figure 11 where the area ratio is the maximum of the present study. When the control is employed results in an enhancement of the wall pressure all along the length of the duct. This increase in area ratio will result in a progressive increase in the reattachment length, under these circumstances when the flow is coming out the base vortex is unable to influence the base pressure and the flow moves in the downstream resulting in the much higher values when compared with the results at lower area ratios.

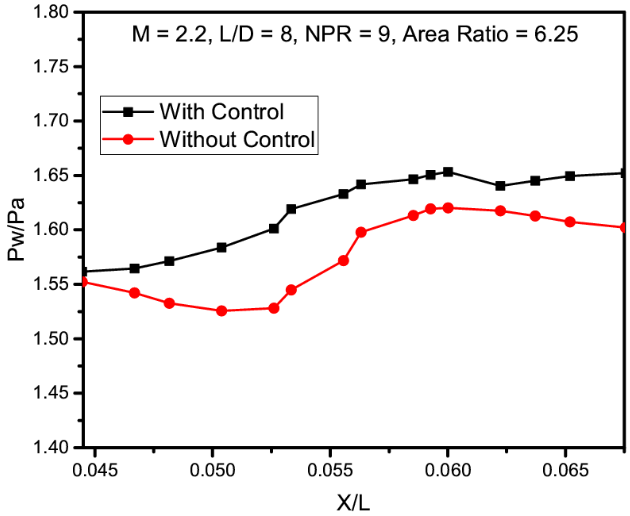


Fig. 11. Wall Pressure Distribution for Area Ratio 6.25

Base pressure results for all the four-area ratio for Mach 2.2 and NPR = 9 are shown in Figure 12. It is seen that for Mach numbers from 2.2, area ratio 6.25 results in maximum base pressure, but at area ratio 3.24, the base pressure is the minimum. Whenever a control is used for base flows region, one of the factors to be considered is the effect of these controls on the wall pressure field in the enlarged duct. In other words, it is essential to make sure that the control does not adversely influence the wall pressure field. To make sure that the nature of the flow in the duct is not made oscillatory in the present investigation also, the enlargement wall pressure distribution was recorded for all the tests. It was found that the wall pressure field is not adversely affected.

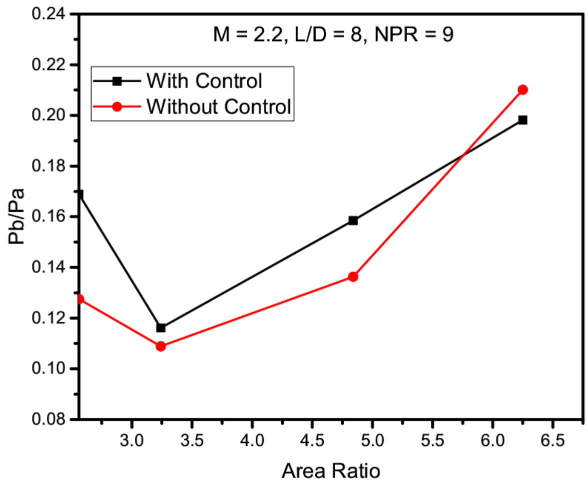


Fig. 12. Base Pressure Distribution

4. Conclusions

Based on the discussion in the previous section we may state that the relief to the flow plays a vital role in fixing the base pressure values. Due to the increase in the area ratio, there will be a progressive increase in the base pressure even though all other flow parameters remained the same. The results show that the initial values of the base pressure continue to decrease with the increase in area ratio. Due to the increase in area ratio when the controls are activated the efficacy of the control mechanism is marginal as compared to the lower area ratio. Control is most useful for lower area ratio namely 2.56 and 3.24. For present NPR 9, the nozzle exit pressure is less than the ambient pressure, and therefore the lowest area ratio will give the most significant value of the pressure at the base. The microjets are found to be useful for area ratios 2.56 and 3.24 for Mach number M = 2.2. The duct static wall pressure and the nature of the flow are not adversely influenced by the control mechanism.

References

[1] Khan, Sher Afghan, and E. Rathakrishnan. "Active control of suddenly expanded flows from overexpanded nozzles." *International Journal of Turbo and Jet Engines* 19, no. 1-2 (2002): 119-126.

[2] Khan, Sher Afghan, and E. Rathakrishnan. "Control of suddenly expanded flows with micro-jets." *International journal of Turbo and Jet engines* 20, no. 1 (2003): 63-82.

[3] Khan, Sher Afghan, and E. Rathakrishnan. "Control of suddenly expanded flows from correctly expanded nozzles." *International Journal of Turbo and Jet Engines* 21, no. 4 (2004): 255-278.

[4] Khan, Sher Afghan, and E Rathakrishnan. "Active Control of Suddenly Expanded Flows from Underexpanded Nozzles." *International Journal of Turbo and Jet Engines* 21 (2004): 233–54.

[5] Khan, Sher Afghan, and E. Rathakrishnan. "Active Control of Suddenly Expanded Flows from Underexpanded Nozzles - Part II." *International Journal of Turbo and Jet Engines* 22 (2005): 163–83.

[6] Khan, Sher Afghan, and E. Rathakrishnan. "Control of suddenly expanded flow." *Aircraft Engineering and Aerospace Technology* 78, no. 4 (2006): 293-309.

[7] Mishra, Nirmith Kumar, S Srinivas Prasad, and Ayub Padania. "Modeling & Simulation of Rocket Nozzle." *International Journal of Advanced Engineering and Global Technology* 2, no. 9 (2014): 988–95.

[8] Rehman, Shafiqur, and Sher Afghan Khan. "Control of base pressure with micro-jets: part I." *Aircraft Engineering and Aerospace Technology* 80, no. 2 (2008): 158-164.

[9] Baig, M. Ahmed Ali, Sher Afghan Khan, and E. Rathakrishnan. "Active Control Of Base Pressure In Suddenly Expanded Flow For Area Ratio 4.84." *International Journal of Engineering Science and Technology* 4, no. 5 (2012): 1892-1902.

[10] Asadullah, Mohammed, Sher Afghan Khan, Waqar Asrar, and E. Sulaeman. "Active control of base pressure with counter clockwise rotating cylinder at Mach 2." In *2017 4th IEEE International Conference on Engineering Technologies and Applied Sciences (ICETAS),* pp. 1-6. IEEE, 2017.

[11] Asadullah, Mohammed, and Sher Afghan Khan. "Control of Base Pressure with Variable Location of Clockwise Rotating Cylinder." In *International Conference on Advances in Thermal Systems, Materials and Design Engineering* 7, 2018.

[12] Asadullah, Mohammed, Sher Afghan Khan, Manzoore Elahi M. Soudagar, and T. R. Vaishak. "A Comparison of the Effect of Single and Multiple Cavities on Base Flows." In *2018 IEEE 5th International Conference on Engineering Technologies and Applied Sciences (ICETAS),* pp. 1-6. IEEE, 2018.

[13] Poh, S, A Baz, and B Balachandran. "Counter Clockwise Rotation of Cylinder with Variable Position to Control Base Flows Counter Clockwise Rotation of Cylinder with Variable Position to Control Base Flows Mach Number." In *IOP Conference Series: Materials Science and Engineering*, 2018.

[14] Khan, Sher Afghan, Abdulrahman Abdulla Al Robaian, Mohammed Asadullah, and Abdul Mohsin. "Fluid Grooved Cavity as a Passive Controller behind Backward Facing Step." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 53, no. 2 (2019): 185–93.

[15] Khan, Sher Afghan, and Jaimon Quadros. "Modelling of Suddenly Expanded Flow Process in Supersonic Mach Regime Using Design of Experiments and Response Surface Methodology." *Journal of Applied Fluid Mechanics* 11, no. 2 (2018): 483–96.

[16] Quadros, J. D., S. A. Khan, and A. J. Antony. "Study of effect of flow parameters on base pressure in a suddenly expanded duct at supersonic mach number regimes using CFD and design of experiments." *Journal of Applied Fluid Mechanics* 11, no. 2 (2018): 483-496.

[17] Mohan Kumar, G., Dominic Xavier Fernando, and R. Muthu Kumar. "Design and Optimization of De Lavel Nozzle to Prevent Shock Induced Flow Separation." *Advances in Aerospace science and Applications* 3, no. 2 (2013): 119–24.

[18] Shariatzadeh, Omid Joneydi, Afshin Abrishamkar, and Aliakbar Joneidi Jafari. "Computational Modeling of a Typical Supersonic Converging-Diverging Nozzle and Validation by Real Measured Data." *Journal of Clean Energy Technologies* 3, no. 3 (2015): 220-225.

[19] Fharukh, Ahmed G M, Abdulrehman A. Alrobaian, Abdul 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* 8, no. 6 (2018): 373–82.

[20] Khan, Sher Afghan, Abdul Aabid, and Maughal Ahmed Ali Baig. "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* 8 (2018): 1147–58.

[21] Khan, Ambareen, Abdul Aabid, and S. A. Khan. "CFD analysis of convergentdivergent nozzle flow and base pressure control using micro-JETS." *International Journal of Engineering and Technology* 7, no. 3.29 (2018): 232-235.

[22] Aabid, Abdul, Ambreen Khan, Nurul Musfirah Mazlan, Mohd Azmi Ismail, Mohammad Nishat Akhtar, and S A Khan. "Numerical Simulation of Suddenly Expanded Flow at Mach 2.2*." International Journal of Engineering and Advanced Technology* 8, no. 3 (2019): 457–62.

[23] Khan, Sher Afghan, Abdul Aabid, C. Ahamed Saleel. "Influence of Micro Jets on the Flow Development in the Enlarged Duct at Supersonic Mach Number." *International Journal of Mechanical and Mechatronics Engineering* 19, no. 1 (2019): 70–82.

[24] Khan, Sher Afghan, Abdul Aabid, C. Ahamed Saleel. "CFD Simulation with Analytical and Theoretical Validation of Different Flow Parameters for the Wedge at Supersonic Mach Number." *International Journal of Mechanical and Mechatronics Engineering,* no. 1 (2019).

[25] Pathan, Khizar Ahmed, 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 *2017 2nd International Conference for Convergence in Technology (I2CT),* pp. 1121-1125. IEEE, 2017.

[26] Pathan, Khizar Ahmed, S. A. Khan, and P. S. Dabeer. "CFD analysis of effect of area ratio on suddenly expanded flows." In *2017 2nd International Conference for Convergence in Technology (I2CT),* pp. 1192-1198. IEEE, 2017.

[27] Pathan, Khizar Ahmed, 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 *2017 2nd International Conference for Convergence in Technology (I2CT),* pp. 1104-1110. IEEE, 2017.

[28] Pathan, Khizar Ahmed, Prakash S. Dabeer, and Sher Afghan Khan. "Optimization of area ratio and thrust in suddenly expanded flow at supersonic Mach numbers." *Case studies in thermal engineering* 12 (2018): 696-700.

[29] Khan, Sher Afghan, M Mohiuddin, Ahmad Saleel C, and G M Fharukh. "Investigation of the Effects of Nozzle Exit Mach Number and Nozzle Pressure Ratio on Axisymmetric Flow through Suddenly Expanded Nozzles." *International Journal of Engineering and Advanced Technology (IJEAT) ISSN* 8 (2019): 570–78.

[30] Umair, Siddique Mohd, Sher Afghan Khan, Abdulrahman Alrobaian, and Emaad Ansari. "Numerical Study of Heat Transfer Augmentation Using Pulse Jet Impinging on Pin Fin Heat Sink." *CFD Letters* 11, no. 3 (2019): 84–91.

[31] Hamizi, Ilya Bashiera, and Sher Afghan Khan. "Aerodynamics Investigation of Delta Wing at Low Reynold ' s Number." *CFD Letters* 2, no. 2 (2019): 32–41.

[32] Khan, Sher Afghan, Abdulrahman Abdulla Al Robaian, Mohammed Asadullah, and Abdul Mohsin Khan. "Grooved Cavity as a Passive Controller behind Backward Facing Step." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 53, no. 2 (2019): 185–193.

[33] Khan, Sher Afghan, Abdulrahman A Alrobaian, and Mohammed Asadullah. "Threaded Spikes for Bluff Body Base Flow Control." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 53, no. 2 (2019): 194–203.

[34] ANSYS Inc, "ANSYS FLUENT 18.0: Theory Guidance," Canonsburg PA, 2017.