Modelling and Analysis of Convergent Divergent Nozzle with Sudden Expansion Duct using Finite Element Method

Open Access

Abdul Aabid[1], Zakir Ilahi Chaudhary[2], Sher Afghan Khan[1,*]

1 Department of Mechanical Engineering, Faculty of Engineering, International Islamic University Malaysia, 53100, Kuala Lumpur, Malaysia
2 Department of Automobile Engineering, M. H. S. S. College of Engineering, Byculla, Mumbai, Maharashtra, India

ARTICLE INFO	ABSTRACT
	In most of the engineering problems, the sudden expansion at the base is encountered in automobile industries, at the base of the fuselage of the aircraft, at the base of the artillery shells, at the base of the unguided rockets and missiles. At transonic Mach numbers, the contribution of the base drag is more than sixty percent. Hence it is mandatory on the part of the researcher to control the base pressure, depending upon the situation. For external aerodynamics, the base pressure should be closed to the ambient pressure to make base drag almost zero. However, for combustion chambers, the base pressure should be very low so that fuel-air mixing is so that the combustor is efficient. With this idea in mind, this study was undertaken. A finite element (FE) method has been used to investigate the effectiveness of the microjets on wall pressure distribution in a convergent-divergent (CD) nozzle with sudden expansion. For control of the base pressure, four microjets of 1 mm orifice diameter placed at 90-degree intervals along a pitch circle distance (PCD) of 1.3 in the base region were used as a control mechanism. The variables considered are the Mach number, area ratio, and length-to-diameter ratio (L/D) as 2.2, 2.56, and 8. The simulation was done for nozzle pressure ratio (NPR) 3, 5, 7, 9, and 11 with and without the presence of the microjets. The current results for all the cases of this study indicate that one can identify the Mach number, NPR, and the L/D, which will result in a maximum increase in the pressure. Moreover, the results also proved that the influence of microjet control does not affect the flow field in the duct adversely. The two-dimensional planar CD nozzle with sudden expansion duct has been modelled, and analyses using the K-ε turbulence model independently checked with the ANSYS tool.

1. Introduction

In subsonic and supersonic regimes, a sudden expansion of the flow at the base corner is the main problem for various engineering applications. At the blunt base, the pressure is very low, and the contribution of the base drag is more than two-thirds of the total drag of the aerodynamic vehicle. Hence, at transonic and supersonic flow, it is mandatory on the part of the researcher working in the

---

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

field of sudden expansion to control the base pressure and hence the base drag. The effectiveness of the microjets as an active controller plays a prominent role in the regulation of the base flows with sudden expansion. In the case of jet and rocket engine test cells, it has been observed that the system is used to simulate high altitude conditions; the discharging of jet results in enough discharge pressure that is sub-atmospheric. Many scholars are investigating the behavior of fluid in suddenly expanded duct because of its various applications in many engineering problems. Khan and Rathakrishnan [1] experimentally investigated to control the base pressure with active control in the form of the tiny jets. However, to the best of the author's information, no work is reported in the literature for active control through experiments as well as the numerical simulations. Thus, in the current study, an effort is made to study the base pressure with microjets as active control through the Computational fluid dynamics (CFD) method in ANSYS FLUENT.

Computational fluid dynamics (CFD) is a versatile technique that conveys relevant results about the flow characteristics of an object by modeling and simulation of flow fields. The Reynolds averaged Navier-Stokes (RANS) equations solution is transitory, which inflicts complexity in the computational studies for flow field through suddenly expanded convergent-divergent (CD) nozzles and the execution of suitable turbulence model for the closure of the RANS equations. The numerical simulations and the experimental measurements show discrepancies for the compressible flow regions of the CD nozzle, which are influenced by intense pressure gradients and complex secondary flows.

The objectives of this study are to compute the effect of the microjet control in CD nozzle, to calculate the flow field through the suddenly expanded CD nozzle using CFD method, and to validate the experimental result with the simulation results.

2. Literature Review

Studies have been conducted experimentally to investigate the influence of microjet control in a CD nozzle; few of them are discussed here. The designed Mach number at the entry of duct from 1 to 3 have been represented. In these cases, the NPR ranges from 3 to 11, and at these NPRs, the level of expansion is such that the jets at the exit of the nozzles are under, over, and correctly expanded. Moreover, the L/D ratio also a parameter that influences the results; therefore, the L/D was used as an essential variable in order to perform different simulations for L/D ratios in the range from 10 to 1. The microjets were located at the pitch circle diameter (PCD) of 1.3 in the suddenly expanded duct control to control the base pressure. For base pressure control at the base region, four microjets were placed 1 mm diameter as a control mechanism, and the control pressure in the control chamber was the same as the respective NPR. The results show the influence of microjet can regulate the flow in the base region. Also, the flow field in the duct is not adversely influenced by the microjets. Finally, in all cases, they proved that one could identify the Mach number, NPR, and the L/D ratio, which will result in a maximum increase in the base pressure values [1-7]. A similar study also were done by Baig *et al.,* [8] for area ratio of 2.56 and Ahmed and Baig [9] for area ratio of 4.84.

The numerical method was used to investigate the processes of controlling the thrust in a CD nozzle by secondary fluidic injections. Modulation of thrust by fluidic means can be of considerable advantage in some cases like the solid-fueled rockets. Two methods of Fluidic Thrust Modulation (FTM) was used; Shock Thrust Modulation (STM) and the Throat Shifting Thrust Modulation (TSTM) [10]. CFD method was used to investigate the supersonic flow through a De-Laval nozzle, for maximum thrust, flow separation due to the shock waves [11]. Numerical simulation was used to investigate the flow field through a CD nozzle. In nozzle geometry, initially, the flow passes through the converging part of the nozzle till the throat of the CD nozzle and then expands after the throat.

ANSYS FLUENT was utilized to simulate the results and for pre-processing and processing steps. The finite element (FE) model developed for two-dimensional (2D) and three-dimensional (3D) for approximate results. Also, the effectiveness of the turbulence model was studied by Shariatzadeh *et al.,* [12]. They investigated the detailed of flow-field inside the supersonic ejector using the CFD technique and enhanced the ejector's mixing chamber wall shape to obtain a maximum entrainment ratio (ER) in order to get the uppermost probable proficiency that can be achieved by the ejector. The ANSYS FLUENT 14.5 with ICEM 14.5 meshing tool was used to conduct the modelling and examine the ejector performance using two different turbulence models: k-ε realizable and k-ω SST [13]. CFD method was used to find the performance of the micro-nozzle and investigation primarily by its machined surface topology. A circular cross-section micro-Laval nozzle was modelled and studied by Cai *et al.,* [14].

Flow-through the CD nozzle investigation was carried out by using a finite volume readable code, ANSYS FLUENT 6.3. The nozzle geometry modelling and mesh generation was done using GAMBIT 2.4 Software. The numerical analysis shows good agreement with the experimental results obtained from Belega *et al.,* [15]. Patel *et al.,* [16] extracted the concepts of De-Laval nozzle with the principle of working of the nozzle was discussed. The discrepancy of flow parameters like pressure, temperature, velocity, and density is visualized using the CFD technique.

The CFD analysis has been used to analyses geometry the effectiveness and flow parameters on thrust force generated by the flow from CD nozzles to an enlarged duct. The nozzles are designed for different Mach numbers, which are varied from 1 to 3. The analysis was done with the various area ratios (2 to 12) and NPR's (3 to 11) for all Mach numbers [17-20]. Optimize the configuration of rocket air ejector using the numerical method with 3D models through AutoCAD, meshed, and analyzed using Ansys CFX [21]. The performance and flow structure in a 3D rectangular ejector were numerically investigated by using air as the working fluid. The results were obtained using ANSYS commercial software, for supersonic, compressible, turbulent flow ejectors. The main objective is to optimize the C-D nozzle position for each working condition and to bring out the effect of reflected shock waves and boundary layer in the constant area mixing chamber on the performance of the ejector [22].

Mathematical modelling and numerical simulations were carried out to study particle–gas flows through a CD nozzle. Equilibrium and homogeneous model that no-slip in temperature and velocity arises between the particle phase and the gas phase were measured to derive mass flow rate and sound speed of multiphase flows. Particle number density and particle velocity vectors were also attained to clarify particle motion through the nozzle [23]. The design and modelling of a pure jet noise reduction concept were studied. The idea is proposed for use on the type of supersonic exhaust nozzles classically hired on tactical aircraft, the design talks both turbulent mixing noise and broadband shock-associated noise [24].

A numerical method was used to carry out the flow simulation using ANSYS FLUENT for enlarged duct flow through CD nozzle. The L/D of 10 for the duct was used in the analysis. Also, an Area Ratio between the CD Nozzle exit area and the Duct Area was used to be 2.56. The effects of Nozzle Exit Mach number, and the NPR were evaluated by conducting the simulations for Mach numbers were 1.87, 2.25, and 2.58. For each Mach number, the NPR of 3, 5, 7, 9, and 11 were set respectively. The simulation was done using the k-ω turbulence model [25]. Next, this concept was also used for different area ratio, Mach number, NPR's and L/D using CFD method [26-32]. The CFD method was also used to investigate the flow field around the wedge [33-34]. A similar type of CD nozzle was used for experimental studies and observed the wall pressure performance with and without control using the microjet controller for different Mach number and area ratios of duct [35-44]. The experimental

study has been found to obtain the effectiveness of combined factors for nozzle operating, diameter and riser height on sprinkler irrigation uniformity [45].

3. Methodology

This section illustrates the methodology which is used to design and model the nozzle. A 2D FE model was used and generate fine mesh to obtain the accuracy in results. It also shows the analysis and boundary conditions [46].

3.1 Two-Dimensional CD Nozzle

The CD nozzle with an enlarged duct is designed based on the Mach number and the experimental work done by Khan and Rathakrishnan [1]. All the dimensions in this model are in mm, as shown in Figure 1. The purpose of this study is to examine the flow-field through the nozzle and estimate the flow parameters such as pressure and velocity by CFD simulation in 2D modelling with and without tiny jets.

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

The dimensions of the nozzle are; the inlet diameter (Di) and the convergent length (Lc) is 26.52 mm and 35 mm respectively, the throat diameter (Dt) is 6.45 mm, and exit diameter (De) and length of divergent (Ld) is 10 mm and 16.88 mm respectively. Next, the suddenly expanded diameter (D) of the tube is 16 mm, and the diameter of the microjet (Dm) control is 1 mm.

3.2 Geometry and Modelling

The experimental designed model was used to design the FE model [1]. In the present case, only internal flow-field was observed in order to see the effects of the flow variables based on the variation of NPR. The designed FE model has been shown in Figure 2.
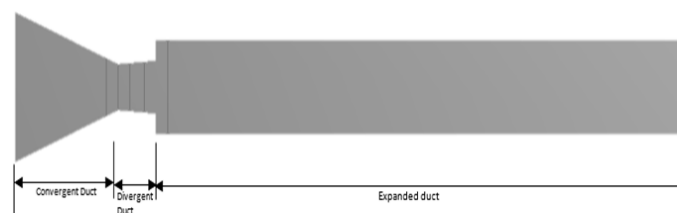

Fig. 2. 2D planar CD nozzle

3.3 Meshing and Boundary Conditions

Before the type mesh to be used, the designed model considered a structured grid element, which is shown in Figure 3. The element numbers are increased in the compressed surface, the mesh size increases, and it is useful to get the best results. Total binary nodes 39,969 are used during the simulation runs. Suitable boundary conditions (BC's) are applied through selecting the lines of the 2D planar model to run the solution with a perfect ideal flow.
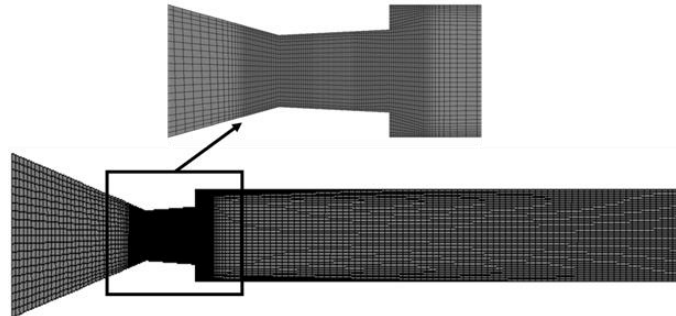

Fig. 3. Structural mesh for 2D planar CD nozzle

3.4 Calculation Procedure

For setting up the solution in FLUENT runs the different cases of simulation, a Reynolds-Averaged Navier-Stokes equation (RANS) with the k-ε standard turbulent model [47] was used. The setup which was used to run the simulation is as follows; selected the solver as a pressure-based absolute 2D planar. Following ideal gas, viscosity by Sutherland fluid and the BC's are inlet as pressure inlet, outlet as pressure outlet, and the wall is the stationary wall. To simulate the flow initialized the solution from the inlet of convergent line with the reference of the uninterrupted surface inlet. Finally, the solutions run until it is converged.

3.4.1 k-ε turbulent model

The selection of the turbulence model is significant for analyzing the fluid flows. Generally, the study of turbulence will give an idea of a fluid flow phenomenon characterized by unsteadiness, fluid motions, and contaminants such as smoke and heat. To simulate a flow such as turbulent flow, it has effort to attain the numerical solution for the unsteady Navier-Stokes equations. This type of method called direct numerical simulation of turbulence, and it is available for research tools at relatively low Reynolds numbers. But the problem with direct numerical simulation is in grid and time scale requirements to model all the significant motions in a turbulent flow.

A simple turbulence model comprises stating that μ_{eff} is constant during the flow. It means that viscosity is meaningful, and in the present study, the viscosity selected Sutherland. The constant eddy viscosity model will not allow accurate prediction of the smaller-scale features of the flows. The estimation of μ_{eff} is obtained by assuming that the l_m is the same in the flow field and in the duct. In the present study, the model contains a duct. The flow over a duct can easily estimate the magnitude, and Prandtl's model is

$$\mu_t = \rho V_t l_m \tag{1}$$

The Reynolds numbers

$$Re_{eff} = \frac{\rho U D}{\mu_t} \tag{2}$$

where,
μ_t = Turbulence Viscosity
V_t = turbulent velocity fluctuations
ρ = Density
U = Average value
D = Diameter of duct

Next, the k-ε model, which was proposed by Jones and Launder [48] with the consideration of standard turbulent model for engineering flows simulation. To relate μ_t to the Reynolds stresses, and assume that

$$V_t \propto \sqrt{k} \tag{3}$$

So that,

$$\mu_t = C\rho l_m \sqrt{k} \tag{4}$$

where C is a constant. By means of this model μ_t is non-zero all over the place of the flow that k is non-zero. Therefore, a new independent variable of the turbulence model is l_m and k.

The current model has a duct in addition to CD nozzle in order to study the sudden expansion of flows. Therefore, the K-ε turbulent model most likely suitable for pipe, duct, or cylindrical type of model to find the flow-field insider the fluid region.

4. Analysis

The present study based on the computational fluid dynamics (CFD) method which is applicable for the solution of different types of fluid flow problems. The fluid flows performances the internal flow of the fluid model and the external flow of the solid model. The fluid flow problem simulates with appropriate boundary conditions by using fundamental equations for the solutions. CFD method is applicable for different types of solutions such physics, science, mathematical and engineering problems. The fundamental equations of fluid flow analysis are the continuity equation, momentum equation, and energy equation. These equations can be expressed as Eq. (5)-(7), respectively.

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0 \tag{5}$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_i} = \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_i} \tau_{ij} \tag{6}$$

$$\frac{\partial}{\partial t}\left[\rho\left(e + \frac{V^2}{2}\right)\right] + \frac{\partial}{\partial x_i}\left[\rho u_j\left(e + \frac{V^2}{2}\right) + P + q_j - u_i \tau_{ij}\right] = 0 \tag{7}$$

where, u, ρ, and P indicate velocity, density, and pressure, respectively. The terms q_j and τ_{ij} represent the heat flux and the viscous stress tensor. Where V denotes the modulus velocity.

These mathematical expressions are bases on the three fundamental physical principles – mass is conserved, Newton's second law, which states that the momentum and energy are conserved.

5. Results

The primary purpose of this work is to optimize the flow-filed at different NPR's and the influence of the microjet on control. The pressure, Mach number contours, and plots have been obtained.

5.1 Validation of Present Finite Element Results

To validate the present finite element model, considered the experimental work from Khan and Ratnakrishnan [1] which is shown in Figure 1. The case was found from the experimental data at Mach number 2.58 for area ratio of 3.24, NPR of 11, and L/D ratio of 6. The present results show a good agreement with the experimental work. Figure 4 shows the comparison of results. The base pressure valves are non-dimensional (divided into atmospheric pressure). The relative error for base pressure with control is around 1.7 %. Whereas. without control, it is almost 7.4 % that has been found in current study.
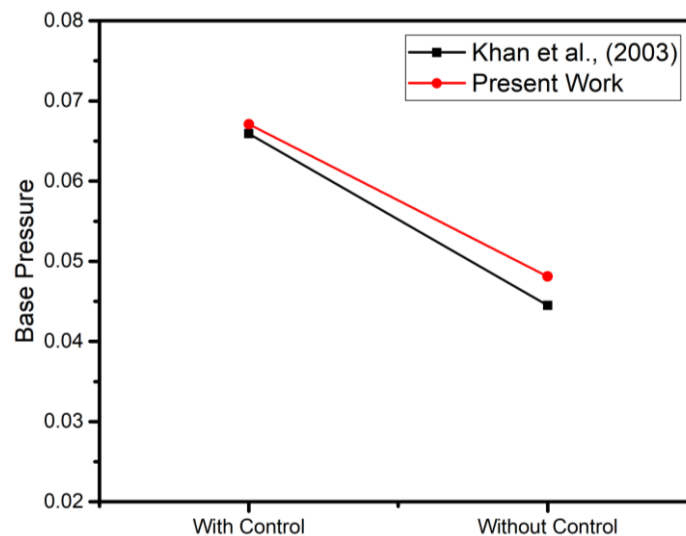

Fig. 4. Velocity streamline

5.2 Mesh Independence Test

The grid independence study was done with three relevance, which is for coarse, medium, and fine in order to see which test has the most excellent result. So, the results show the coarse have less pressure compare to medium and fine. For the case of fine mesh, results show superior performance, but the computational time is twice compared to the medium with less discrepancy in the results. Thus, the medium mesh has been selected for this study to evaluate the simulation results.

The processor which was used to simulate the number of cases intel Core i7 with 16 GB RAM and case was selected to find the mesh test is NPR 3 without microjet control of pressure results. The meshing independence test is shown in Table 1.

Table 1
The Mesh Independence Study

Mesh Type	No. of Element	CFD Runtime	Exit Pressure
Coarse	19,663	190 seconds	0.3551 Pa
Medium	39,969	379 seconds	0.8971 Pa
Fine	46,851	891 seconds	0.9262 Pa

5.3 Velocity Streamline

The flow formation from inlet to outlet of nozzle is essential in this study. Therefore Figure 4 shows the velocity streamlines without (Figure 5(a)) and with (Figure 5(b)) microjet control. From the figure, it is seen that the re-circulation formation in the suddenly expanded duct will reduce when microjet controls are employed. In the absence of the microjets, the re-circulation zone is significant. From this result, it is evident that the microjet controls the formation of wake at sudden expansion region; hence, the base pressure will reduce. Moreover, this will not influence wall pressure distribution. The case considered for velocity streamlines is NPR 11, L/D of 8, and further studies all NPR with and without control is continued from the next sub-sections.

(a)

(b)
Fig. 5. Velocity streamline (a) Without Control (b) With Control

5.4 Pressure Effect

The pressure distribution is the most crucial observation of this study, and pressure has a different distribution in each zone, such as convergent, throat, divergent, and enlarged duct in a CD nozzle. Therefore, in this section, the results considered from all zones of the CD nozzle.

According to the obtained results, which are shown in Figure 6 to 10 for the case without control of microjets. It has been illustrated that the distribution of wall pressure in the duct is consistently varying and reaching relatively higher value at the exit of the duct in Figure 6-10(b). Meanwhile, in the sudden expansion region, the pressure is also expanding with the formation of shock waves, which is formed at the recirculation zone due to which the base pressure increases. The same can be observed in contours from Figure 6-10(a). The blue color at the sudden expansion region is high, which it has a low-pressure effect and high drag formation.

(a)   (b)

Fig. 6. Pressure distribution for L/D = 8 without control of micro-jets for NPR 3 (a) contour (b) plot
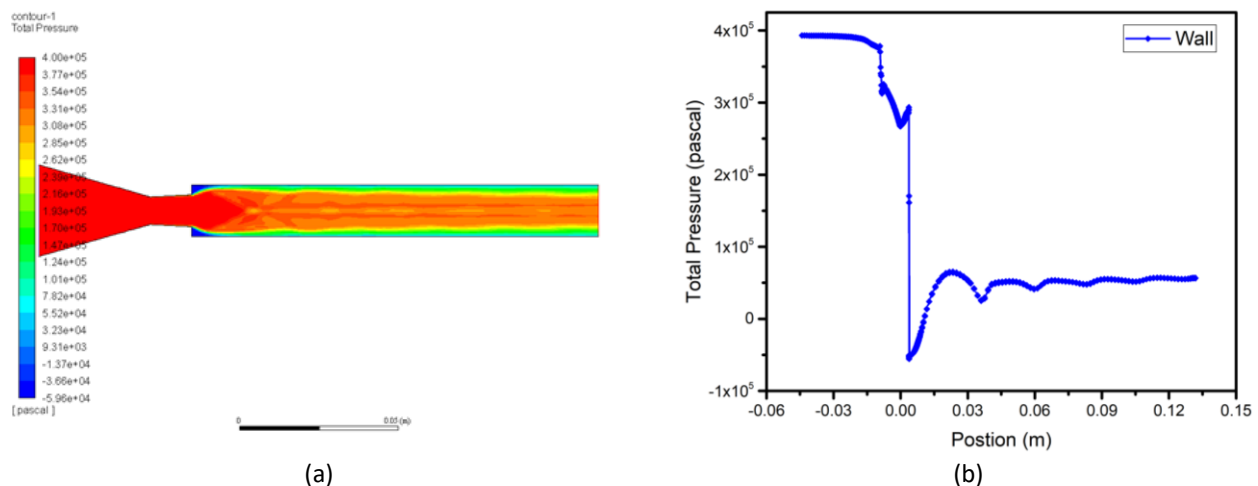

(a)   (b)

Fig. 7. Pressure distribution for L/D = 8 without control of micro-jets for NPR 5 3 (a) contour (b) plot

According to our previous discussion of pressure flow in CD nozzle without any influence of the microjets as control, it is found the wake formation is high and hence the high values of the drag. Now, in order to control the base drag in a sudden expansion region, the microjet was employed as control, and later the flow was simulated.
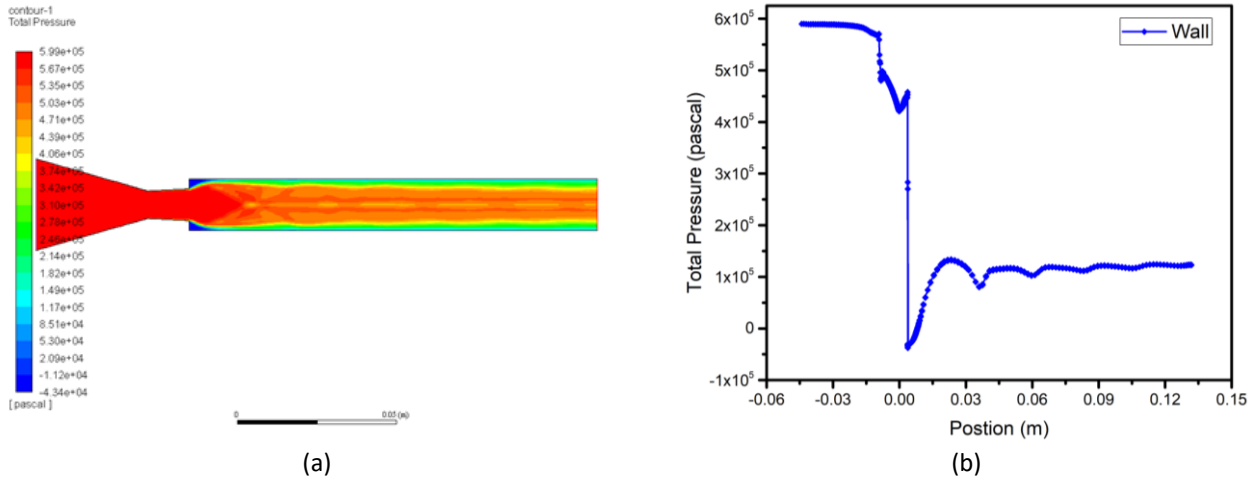
(a)   (b)

Fig. 8. Pressure distribution for L/D = 8 without control of micro-jets for NPR 7 3 (a) contour (b) plot
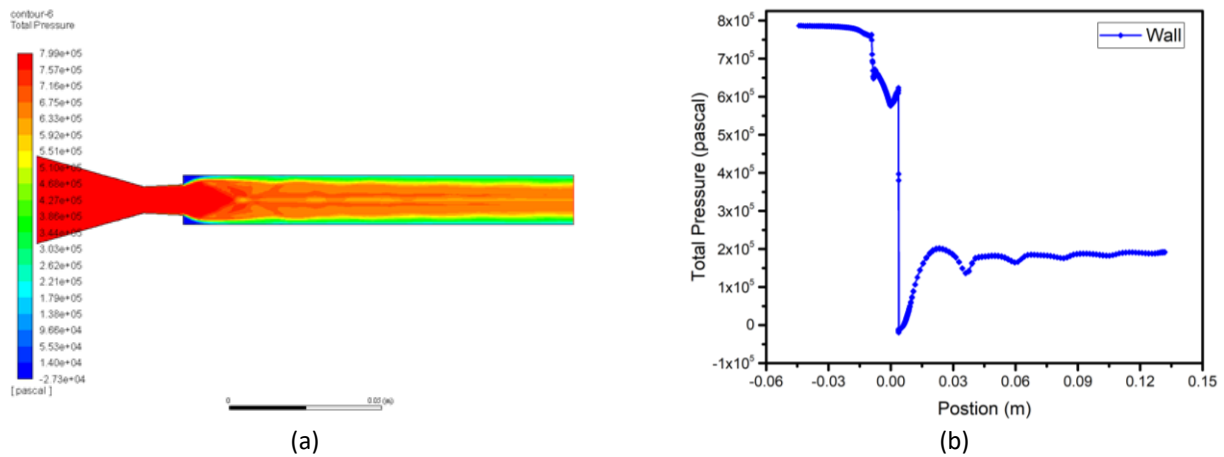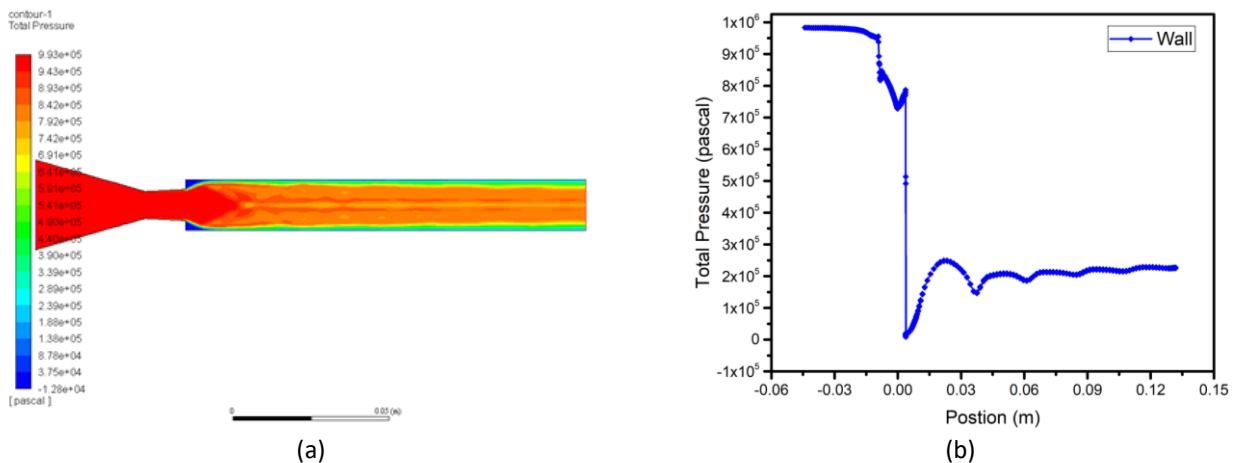

(a)   (b)

Fig. 9. Pressure distribution for L/D = 8 without control of micro-jets for NPR 9 3 (a) contour (b) plot

(a)   (b)

Fig. 10. Pressure distribution for L/D = 8 without control of micro-jets for NPR 11 3 (a) contour (b) plot

The results show that when control is employed at the base, as shown in Figure 11 to 15. Based on the results shown in Figure 11-15(b), the nature of the flow, as well as wall pressure distributions for different NPR's, is unaltered. When we look at the contours in Figure 11-15(a), it is straightforward to quantify the effectiveness of the microjets on base pressure in the recirculation zone as well as the influence of the microjets, and the blue color is limited in a small portion of sudden expansion

region. Therefore, in this case, the use of microjet will result in an increase of the base pressure, which results in decreasing the base drag at the separated region and increasing the thrust for supersonic aerospace vehicles.
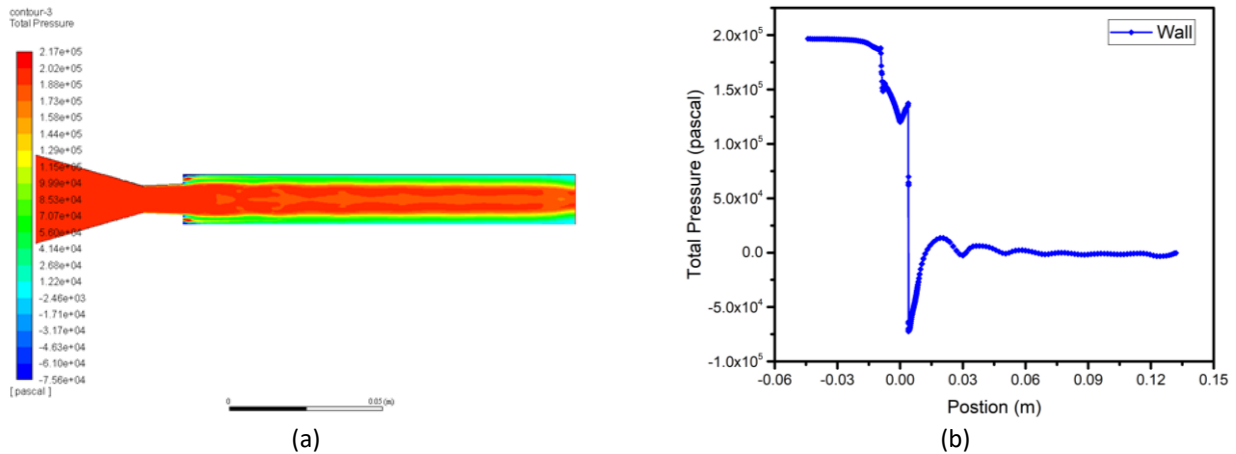

Fig. 11. Pressure distribution for L/D = 8 with control of micro-jets for NPR 3 3 (a) contour (b) plot
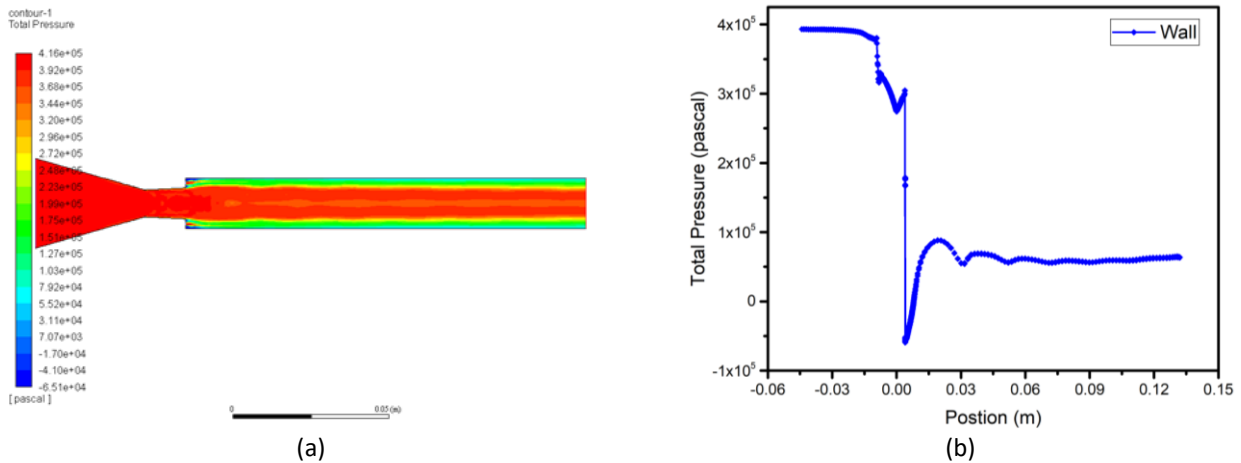

Fig. 12. Pressure distribution for L/D = 8 with control of micro-jets for NPR 5 3 (a) contour (b) plot
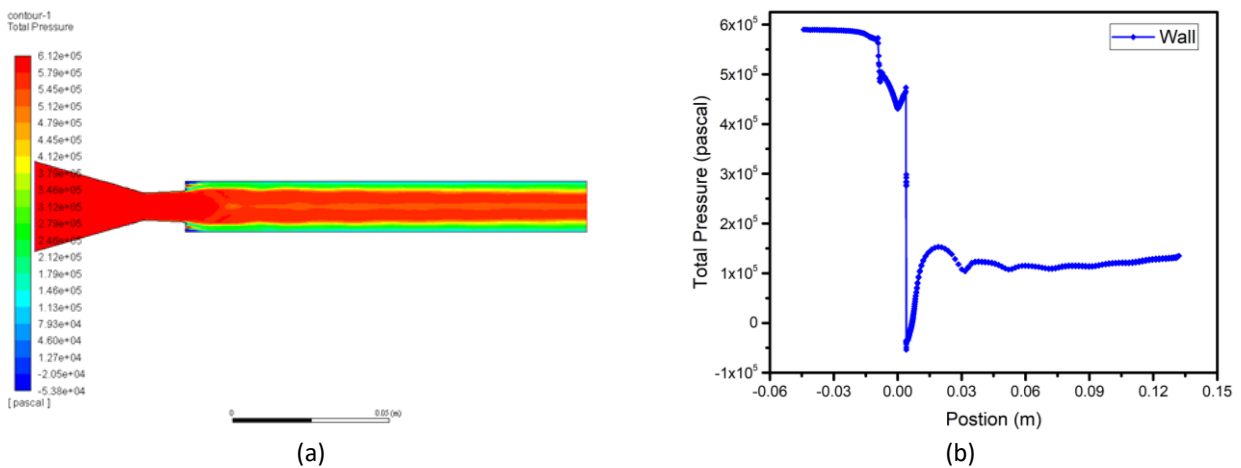

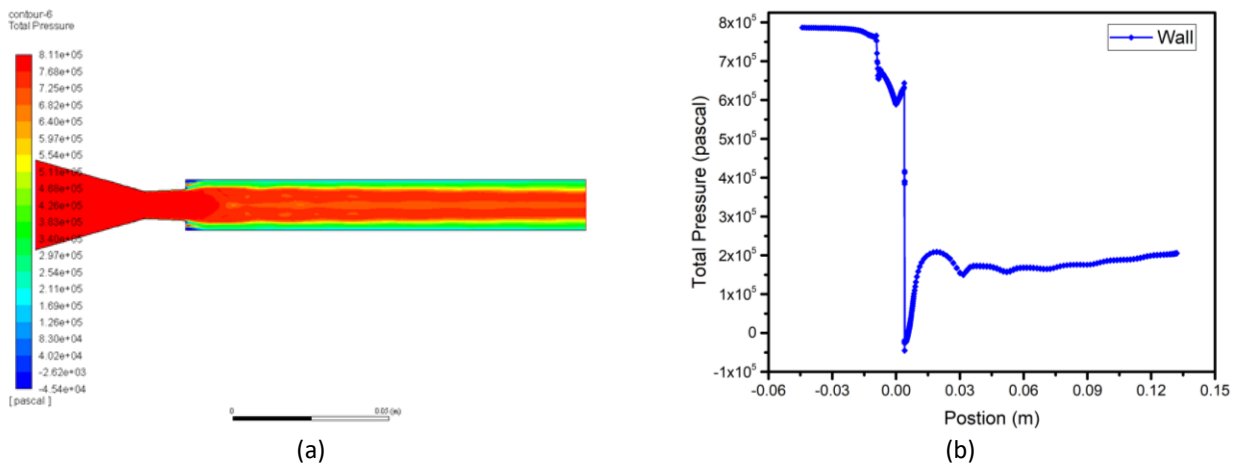**Fig. 13.** Pressure distribution for L/D = 8 with control of micro-jets for NPR 7 3 (a) contour (b) plot

(a)   (b)

Fig. 14. Pressure distribution for L/D = 8 with control of micro-jets for NPR 9 3 (a) contour (b) plot

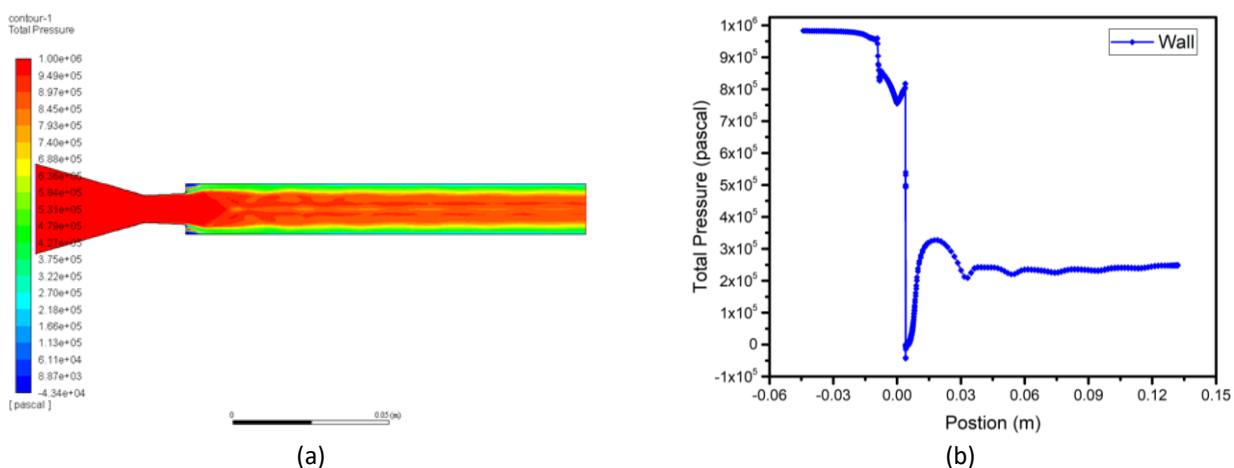(a)   (b)

Fig. 15. Pressure distribution for L/D = 8 with control of micro-jets for NPR 11 3 (a) contour (b) plot

5.5 Mach Number

The pressure and velocity are related to each other; when pressure increases, then the velocity will decrease from the energy equation. Therefore, keeping this fundamental phenomenon into consideration, the Mach number has been evaluated at different nozzle pressure ratios as a function of Mach number variation from inlet to the outlet of the CD nozzle.

With a similar analogy, all the performance for Mach number effects the contours plots are generated. The same can be observed in the contours of Figure 16-20(a). The trends are found to be opposite to the pressure contours. Here, from the figure, it is seen that the Mach number at sudden expansion region is suddenly increasing, after exiting from the nozzle and flowing through the wall then decreasing when it reaches to exit portion of the duct. The results are shown in Figure 16-20(b), the flow-field varying with the sinusoidal effects, and the same will decrease at the exit of the duct.

Next, the influence of microjet shows the drag reduction at the base area as studied for pressure effects as shown in Figure 21 to 25 without any adverse impact of the wall pressure flow field.
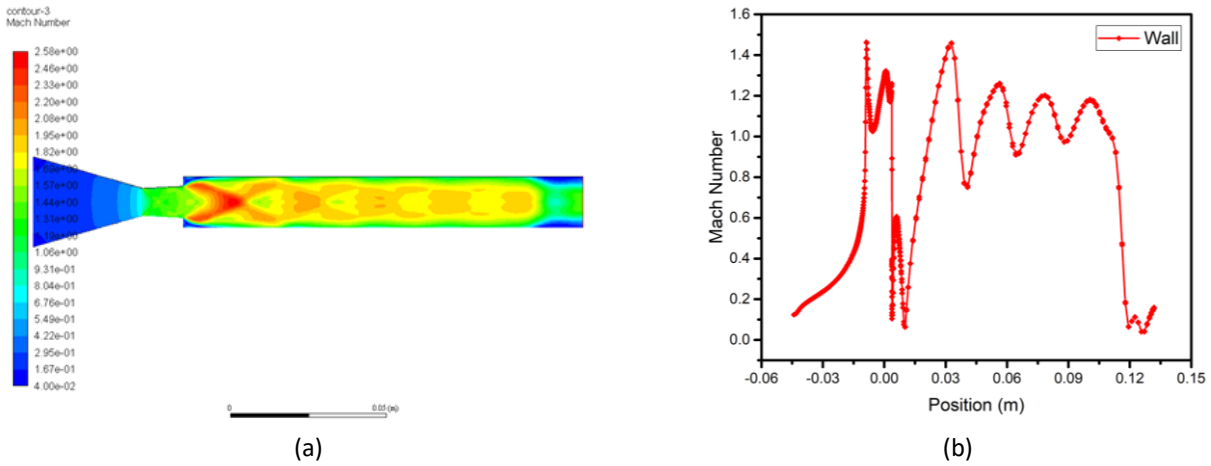
(a)   (b)

Fig. 16. Mach number variation for L/D = 8 without micro-jets Control for NPR 3 3 (a) contour (b) plot

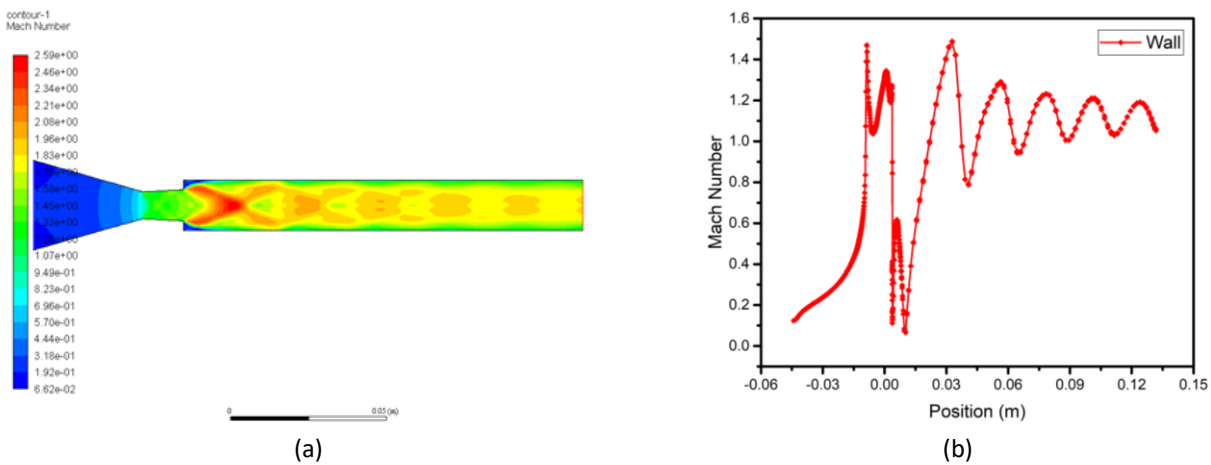(a)   (b)

Fig. 17. Mach number variation for L/D = 8 without micro-jets control for NPR 5 3 (a) contour (b) plot
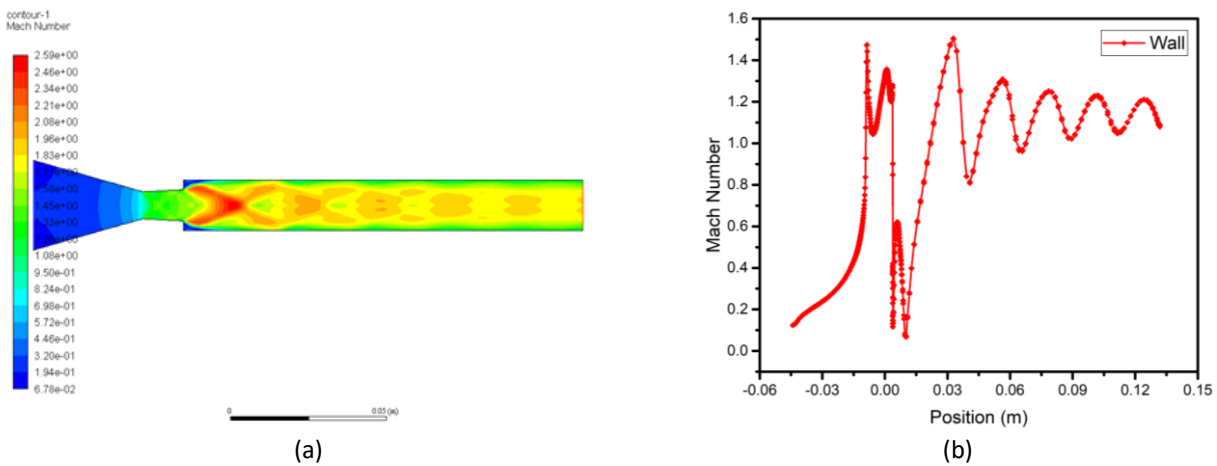

(a)   (b)

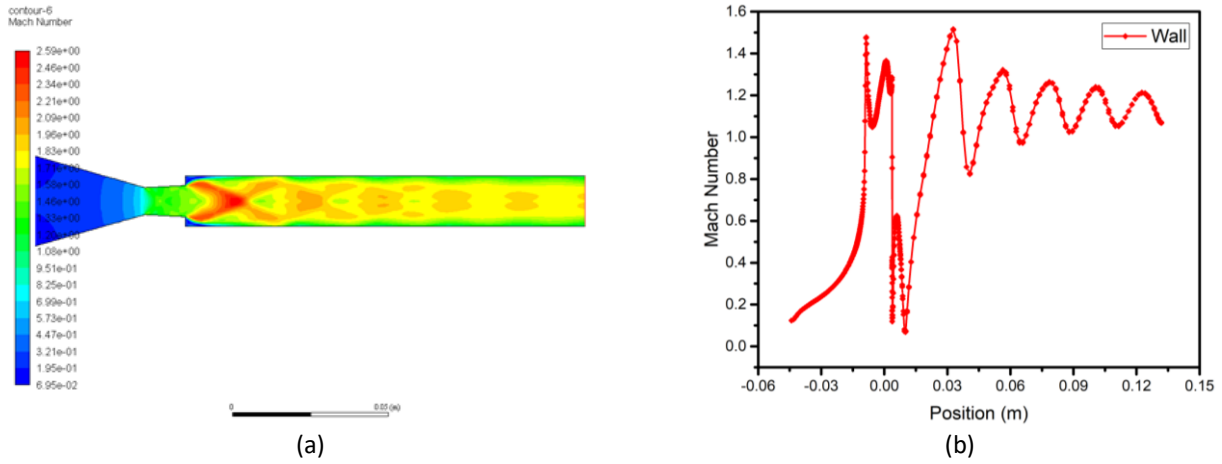**Fig. 18.** Mach number variation for L/D = 8 without micro-jets control for NPR 7 3 (a) contour (b) plot

(a)   (b)

Fig. 19. Mach number variation for L/D = 8 without micro-jets control for NPR 9 3 (a) contour (b) plot
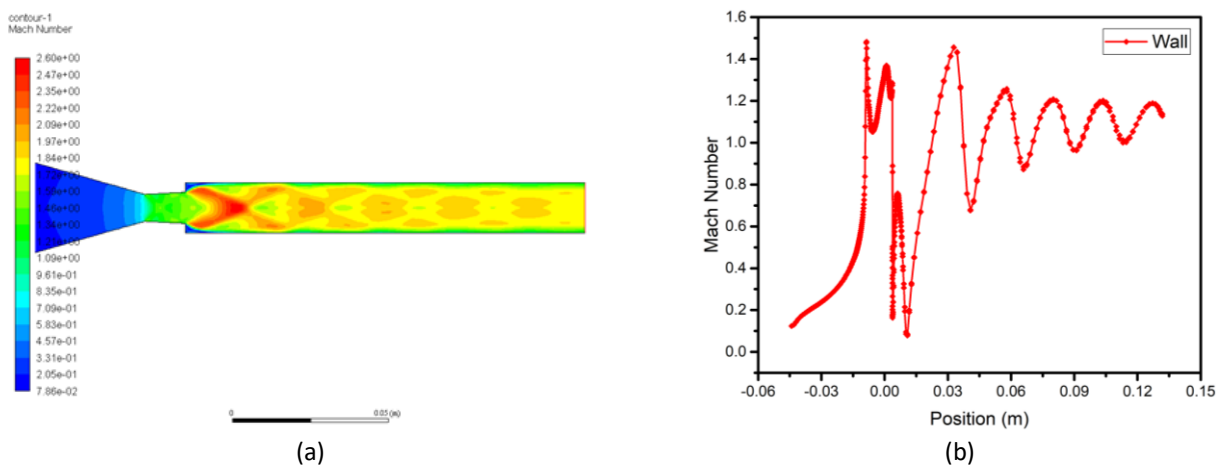

(a)   (b)

Fig. 20. Mach number variation for L/D = 8 without micro-jets control for NPR 11 3 (a) contour (b) plot
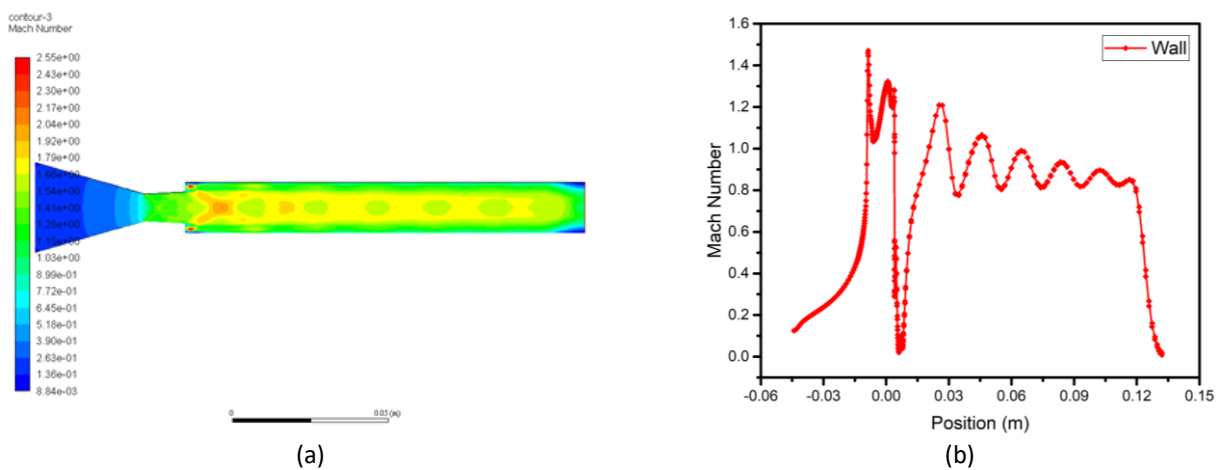

(a)   (b)

Fig. 21. Mach number variation for L/D = 8 without micro-jets control for NPR 3 3 (a) contour (b) plot

(a)

(b)

Fig. 22. Mach number variation for L/D = 8 without micro-jets control for NPR 5 3 (a) contour (b) plot
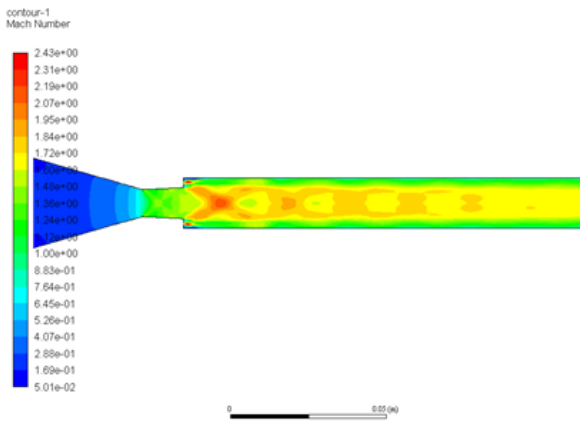

(a)

(b)

Fig. 23. Mach number variation for L/D = 8 without micro-jets control for NPR 7 3 (a) contour (b) plot

(a)

(b)

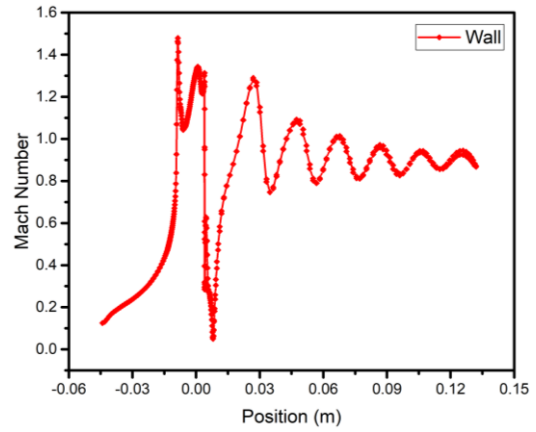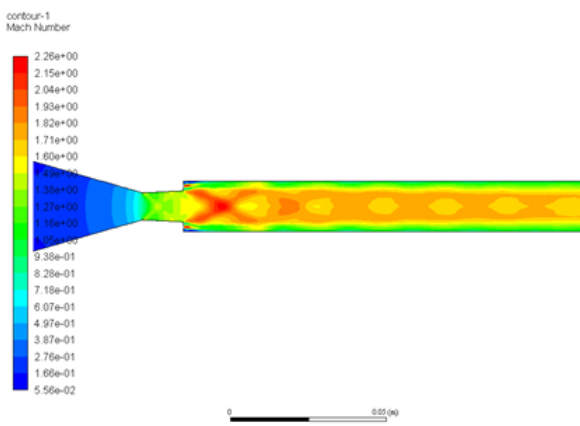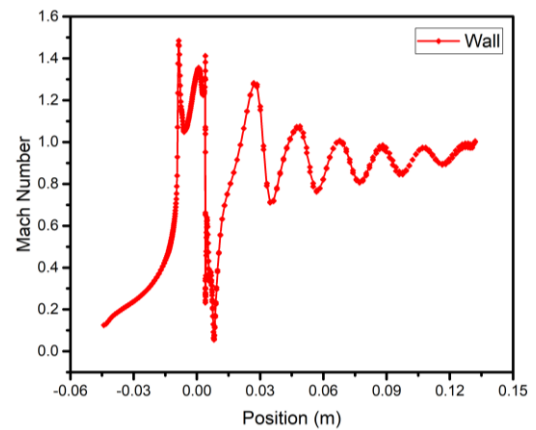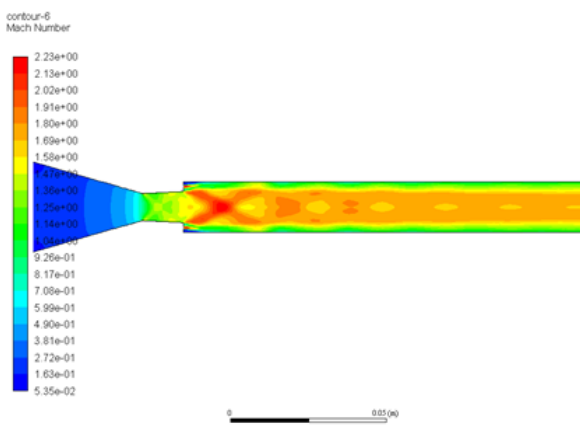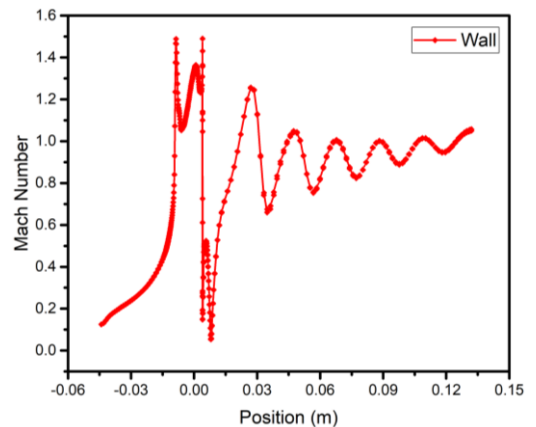**Fig. 24.** Mach number variation for L/D = 8 without micro-jets control for NPR 9 3 (a) contour (b) plot
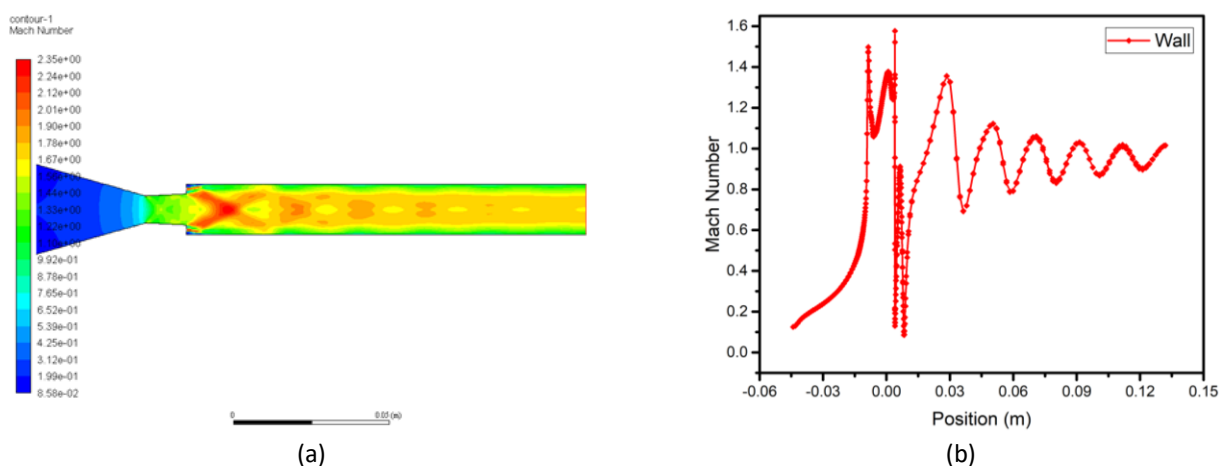
(a)   (b)

Fig. 25. Mach number variation for L/D = 8 without micro-jets control for NPR 11 3 (a) contour (b) plot

6. Conclusions

The numerical investigation of flow-field and wall pressure distribution through a CD nozzle has been studied using the finite element method. The effectiveness of base pressure control on wall pressure and Mach number has been demonstrated for with and without control through ANSYS simulation. The micro-jets can control the base pressure and are able to reduce the base suction without any adverse impact on the wall pressure field. The factors which are influencing the base pressure are the Mach number, the relief available to the flow, the level of expansion, and the L/D ratio. The minimum duct length needed for this Mach number, NPR and the area ratio is L/D = 2. The reattachment, which dictates the level of base pressure also, influenced by the backward-facing step height. The level of expansion plays an essential role in fixing the base pressure values. The results demonstrate that control is useful in the presence of a favorable pressure gradient. Based on the simulation results; it can be concluded that the total wall pressure variable from inlet to outlet and pressure value is high in the case of NPR 11.

References

[1]   Sher Afghan Khan, and Ethirajan Rathakrishnan. "Control of suddenly expanded flows with micro-jets." *International journal of Turbo and Jet engines* 20, no. 1 (2003): 63-82.
[2]   Sher Afghan Khan, and Ethirajan Rathakrishnan. "Active Control of Suddenly Expanded Flows from Underexpanded Nozzles." *International Journal of Turbo and Jet Engines* 21, no. 4 (2004): 233-254.
[3]   Sher Afghan Khan, and Ethirajan Rathakrishnan. "Control of suddenly expanded flows from correctly expanded nozzles." *International Journal of Turbo and Jet Engines* 21, no. 4 (2004): 255-278.
[4]   Sher Afghan Khan, and E. Rathakrishnan. "Active control of suddenly expanded flows from underexpanded nozzles-Part II." *International Journal of Turbo and Jet Engines* 22, no. 3 (2005): 163-184.
[5]   Sher Afghan Khan, and E. Rathakrishnan. "Control of suddenly expanded flow." *Aircraft Engineering and Aerospace Technology* 78, no. 4 (2006): 293-309.
[6]   Sher Afghan Khan, and Ethirajan Rathakrishnan. "Nozzle expansion level effect on suddenly expanded flow." *International Journal of Turbo and Jet Engines* 23, no. 4 (2006): 233-258.
[7]   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.
[8]   Baig, Maughal Ahmed Ali, Fahad Al-Mufadi, Sher Afghan Khan, and Ethirajan Rathakrishnan. "Control of base flows with micro jets." *International Journal of Turbo and Jet Engines* 28, no. 1 (2011): 59-69.
[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] Ali, Ashraf, Andrew Neely, John Young, Brendan Blake, and Jian Yang Lim. "Numerical simulation of fluidic modulation of nozzle thrust." In *17th Australian Fluid Mechanics Conference*, pp. 5-9. 2010.

[11]   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): 119-124.

[12]   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.

[13]   Su, Liju. "CFD Simulation and Shape Optimization of Supersonic Ejectors for Refrigeration and Desalination Applications." (2015).

[14]   Cai, Yukui, Zhanqiang Liu, Zhenyu Shi, Qinghua Song, and Yi Wan. "Residual surface topology modeling and simulation analysis for micro-machined nozzle." *International Journal of Precision Engineering and Manufacturing* 16, no. 1 (2015): 157-162.

[15]   Belega, Bogdan-Alexandru, and Trung Duc Nguyen. "Analysis of flow in convergent-divergent rocket engine nozzle using computational fluid dynamics." In *International Conference of Scientific Paper*. 2015.

[16]   Patel, Malay S., Sulochan D. Mane, and Manikant Raman. "Concepts and CFD Analysis of De-Laval Nozzle." *International Journal of Mechanical Engineering and Technology (IJMET)* 7, no. 75 (2016): 221–40.

[17]   Pathan, Khizar Ahmed, Sher Afghan 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.

[18]   Pathan, Khizar Ahmed, Sher Afghan 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.

[19]   Pathan, Khizar Ahmed, Sher Afghan 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.

[20]   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.

[21]   CH V K N S N Moorthy, V Srinivas, and V V S H Prasad & T Vanaja. "Computational Analysis of a Cd Nozzle With 'Sed' for a Rocket Air Ejector in Space Applications." *International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)* 7, no. 1 (2017): 53–60.

[22]   Arun, K. M., Shaligram Tiwari, and A. Mani. "Three-dimensional numerical investigations on rectangular cross-section ejector." *International Journal of Thermal Sciences* 122 (2017): 257-265.

[23]   Zhang, Guang, and Heuy 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 Flows* 10, no. 1 (2018): 19-32.

[24]   Pilon, Anthony R., Russell W. Powers, Dennis K. McLaughlin, and Philip J. Morris. "Design and analysis of a supersonic jet noise reduction concept." *Journal of Aircraft* 54, no. 5 (2017): 1705-1717.

[25]   Sher Afghan Khan, M. Mohiuddin, C. Ahmad Saleel, 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.

[26]   Khan, Ambareen, Abdul Aabid, and Sher Afghan 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.

[27]   Sher Afghan Khan, 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): 1147-1158.

[28]   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): 373–382.

[29]   Sher Afghan Khan, Abdul Aabid, and 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: 70-82.

[30]   Aabid, Abdul, Ambreen Khan, Nurul Musfirah Mazlan, Mohd Azmi Ismail, Mohammad Nishat Akhtar, and Sher Afghan Khan. "Numerical Simulation of Suddenly Expanded Flow at Mach 2.2." *International Journal of Engineering and Advanced Technology* 8, no. 3 (2019): 457-462.

[31]   Muhammad Fahmi Mohd Sajali, Abdul Aabid, Sher Afghan Khan, Fharukh Ahmed Ghasi Mehaboobali, and Erwin Sulaeman. "Numerical Investigation of Flow Field of a Non-Circular Cylinder." *CFD Letters* 11, no. 5 (2019): 37-49.

[32]   Sher Afghan Khan, Abdul Aabid, Fharukh Ahmed Mehaboobali Ghasi, Abdulrahman Abdullah Al-Robaian, and Ali Sulaiman Alsagri. "Analysis of Area Ratio In a CD Nozzle with Suddenly Expanded Duct Using CFD Method." *CFD*

Letters 11, no. 5 (2019): 61-71.

[33] Sher Afghan Khan, Abdul Aabid, and Ahmed Saleel C. "CFD Simulation with Analytical and Theoretical Validation of Different Flow Parameters for the Wedge at Supersonic Mach Number." *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS* 19, no. 1 (2019): 170–177.

[34] Sher Afghan Khan, Abdul Aabid, Imran Mokashi, Abdulrahman Abdullah Al-Robaian, and Ali Sulaiman Alsagri. "Optimization of Two-Dimensional Wedge Flow Field at Supersonic Mach." *CFD Letters* 11, no. 5 (2019): 80-97.

[35] Mohammed Faheem, Mohammed Kareemullah, Abdul Aabid, Imran Mokashi, and Sher Afghan Khan. "Experiment on of Nozzle Flow with Sudden Expansion at Mach 1.1." *International Journal of Recent Technology and Engineering* 8, no. 2S8 (2019): 1769–1775.

[36] Sher Afghan Khan, Abdul Aabid, Imran Mokashi, and Zaheer Ahmed. "Effect of Micro Jet Control on the Flow Filed of the Duct at Mach 1 . 5." *International Journal of Recent Technology and Engineering (IJRTE)* 8, no. 2S8 (2019): 1758–1762.

[37] Sher Afghan Khan, Zaheer Ahmed, Abdul Aabid, and Imran Mokashi. "Experimental Research on Flow Development and Control Effectiveness in the Duct at High Speed." *International Journal of Recent Technology and Engineering (IJRTE)* 8, no. 2S8 (2019): 1763–1768.

[38] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Khan. "Effects of Micro Jets on the Flow Field of the Duct with Sudden Expansion." *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* 8, no. 9S2 (2019): 636–640.

[39] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Khan. "Numerical Simulations of a CD Nozzle and the Influence of the Duct Length." *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* 8, no. 9S2 (2019): 622–630.

[40] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Kha. "Control of CD Nozzle Flow Using Microjets at Mach 2.1." *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* 8, no. 9S2 (2019): 631–635.

[41] Sher Afghan Khan, Imran Mokashi, Abdul Aabid, and Mohammed Faheem. "Experimental Research on Wall Pressure Distribution in C-D Nozzle at Mach Number 1.1 for Area Ratio 3.24." *International Journal of Recent Technology and Engineering* 8, no. 2S3 (2019): 971–975.

[42] Azami, Muhammed Hanafi, Mohammed Faheem, Abdul Aabid, Imran Mokashi, and Sher Afghan Khan. "Experimental Research of Wall Pressure Distribution and Effect of Micro Jet at Mach." *International Journal of Recent Technology and Engineering* 8, no. 2S3 (2019): 1000–1003.

[43] Azami, Muhammed Hanafi, Mohammed Faheem, Abdul Aabid, Imran Mokashi, and Sher Afghan Khan. "Inspection of Supersonic Flows in a CD Nozzle Using Experimental Method." *International Journal of Recent Technology and Engineering* 8, no. 2S3 (2019): 996–999.

[44] Sher Afghan Khan, Abdul Aabid, and Zakir Ilahi Chaudhary. "Influence of Control Mechanism on the Flow Field of Duct at Mach 1.2 for Area Ratio 2.56." *International Journal of Innovative Technology and Exploring Engineering* 8, no. 6S4 (2019): 1135–1138.

[45] Manal Osman, Suhaimi Hassan, and Khamaruzaman Wan Yusof. "Effect of Combination Factors of Operating Pressure, Nozzle Diameter and Riser Height on Sprinkler Irrigation Uniformity." *Journal of Advanced Research in Applied Mechanics* 2, no. 1 (2014): 1-5.

[46] Stefanou, George. "The stochastic finite element method: past, present and future." *Computer methods in applied mechanics and engineering* 198, no. 9-12 (2009): 1031-1051.

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

[48] Jones, W. P., and Brian Edward Launder. "The prediction of laminarization with a two-equation model of turbulence." *International journal of heat and mass transfer* 15, no. 2 (1972): 301-314.