CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363

Effect of Cavities in Suddenly Expanded Flow at Supersonic Mach Number

Atifatul Ismah Ismail[1], Sher Afghan Khan[1], Parvathy Rajendran[2, 3,*], Erwin Sulaeman[1]

[1] Department of Mechanical Engineering, Faculty of Engineering, IIUM, Gombak Campus, Kuala Lumpur 53100, Malaysia
[2] School of Aerospace Engineering, Universiti Sains Malaysia, 14300 Nibong Tebal, Penang, Malaysia
[3] Faculty of Engineering & Computing, First City University College, Bandar Utama, 47800 Petaling Jaya, Selangor, Malaysia

ARTICLE INFO

Article history:
Received 14 August 2021
Received in revised form 9 September 2021
Accepted 10 September 2021
Available online 17 September 2021

Keywords:
Nozzle Pressure Ratio (NPR); Sudden expansion; Computational Fluid Dynamics (CFD); Aspect cavity ratio

ABSTRACT

The contribution from the base drag due to the sub-atmospheric pressure is significant. It can be more than two-thirds of the net drag. There is a need to increase the base pressure and hence decrease the base drag. This research examines the effect of Mach Number on base pressure. To accomplish this objective, it controls the efficacy in an enlarged duct computed by the numerical approach using Computational Fluid Dynamics (CFD) Analysis. This experiment was carried out by considering the expansion level and the aspect cavity ratio. The computational fluid dynamics method is used to model supersonic motion with the sudden expansion, and a convergent-divergent nozzle is used. The Mach number is 1.74 for the present study, and the area ratio is 2.56. The L/D ratio varied from 2, 4, 6, 8, and 10, and the simulated nozzle pressure ratio ranged from 3 to 11. The two-dimensional planar design used commercial software from ANSYS. The airflow from a Mach 1.74 convergent-divergent axi-symmetric nozzle expanded suddenly into circular ducts of diameters 17 and 24.5 mm with and without annular rectangular cavities. The diameter of the duct is taken D=17mm and D=24.5mm. The C-D nozzle was developed and modeled in the present study: K-ε standard wall function turbulence model was used with the commercial computational fluid dynamics (CFD) and validated. The result indicates that the base pressure is impacted by the expansion level, the enlarged duct size, and the passage's area ratio.

1. Introduction

In today's scenario, the sudden expansion of flow finds applications in numerous engineering applications. These kinds of issues can be found in the automobile as well as the aerospace industry. The need for regulating such a stream has encouraged studies of this flow to accomplish engineering efficacy, technologic ease, financial prudence, observance to guidelines, etc. Another essential feature is bluff body flow, a powerful interaction between the viscous and inviscid regions. Moreover, it is also characterized by separated shear layers that skip from the trailing edges of the body. These shear layers are intrinsically unstable and relate to form powerful vortices frequently at a regular frequency. The vortices provoke minimal pressure at the base and hence increase the drag of the

* *Corresponding author.*
E-mail address: aeparvathy@usm.my (Parvathy Rajendran)

https://doi.org/10.37934/cfdl.13.9.5771

body. There have been many techniques applied to reduce the base drag. One of these techniques is active control, while the other is known as passive control. However, this report's primary focus is the passive control technique, an energy-free technique to reduce pressure drag. These are also known as the base cavity. This report studied cavities as a passive controller on base pressure and wall pressure at supersonic flow.

The objective of this analysis is to ascertain the need to control the flow fields of these flows. The ability to modify the flow of mixing characteristics was a part of the control role. However, various types of flow management methods are used in active and passive control. Dynamic control can only control the jet characteristics using the auxiliary power source, while passive control is used to control energy directly from the flow. Another purpose in investigating is to improve the execution and conserve momentum, and it is desired to lower the drag of a bluff body due to this sub-atmospheric pressure.

Moreover, another purpose in investigating is to improve the efficacy and conserve power, and it is desired to lower the base drag of a bluff body. Next, the establishment of a low-pressure near the base region is usually caused by the flow separation is of considerable concern to the researchers working in sudden expansion. In this region, the pressure is lesser than atmospheric pressure. This difference in pressure caused in this region is called base drag. However, as in Supersonic speeds, the base drag decreases around one-third of the total drag compare to transonic, where the base drag is up to 70% of the net drag for the body of revolution. While scanning the literature, it is found that passive control of base pressure is studied experimentally, and no study is done using CFD. Hence in this study, CFD is used to simulate the flow field numerically.

Computational fluid dynamics is a computer tool to simulate any system's efficiency involving fluid flow, heat transfer, and other physical processes. By solving the equations, it affects fluid flow over the region of interest with defined boundary conditions. A variety of technologies are applied by CFD, including mathematics, computer sciences, engineering, and physics. The equations considered are conservation of mass, momentum, and energy [1].

Compressible flow analysis through a CD nozzle was studied with ANSYS FLUENT using the turbulence model K-ε and Spalart-Allmaras. A comparative analysis between the model's pressure velocity, temperature contours, and vectors determines the efficient design conditions for CD nozzles. From the comparison of the turbulence models, the realizable k-ω model was proposed. The achievable k-ε model for flat and round jets gives quite good results, showing the dissipation rate distribution. It demonstrates a reliable estimation of the boundary layers for large and different pressure ratios and flows. The turbulent model SST k-ω (Shear Stress Transport) was introduced in the year 1993. Models of k-ε and k-ω are very similar. The k-ω model uses a formula to find the dissipation frequency of turbulent energy instead of the dissipation equation. K determines the turbulence energy; ω identifies the typical linear turbulence scale. The k-ω model is sensitive to the original conditions. It soon turned out that the SST k-ω model (Shear Stress Transport) was very commonly used. It was discovered that the SST k-ω shows excellent results at medium pressure gradients in mixing layers. However, the SST k-ω produces too high turbulence levels. It results in a massive change in quality in the calculated flow pattern. The SST model helps one to describe the turbulence more accurately. Calculating supersonic flows for modern jet engines reveals that the best performance can be achieved from k-ω and SST turbulence transitional models [2].

Cavities are one of the most used flow control mechanisms in unexpectedly expanded flows, increasing or decreasing the base pressure. Figure 1 shows the schematic sketch of the cavities in an enlarged duct. According to Khan *et al.,* [3], many parameters are varied, such as the area of the enlarged duct, size of the extended duct, the aspect ratio, the number of cavities, and the sudden expansion inlet Mach number. In addition, some parameters need to be varied, which are the entry

Mach number and the geometrical parameters. In contrast, the other parameter such as base and wall pressures along the enlarged duct's length and the static pressure at the cavities are usually measured. In studying the sudden expansion phenomenon and pressure expansion development at the base, the cause of the high pressure, which suddenly expands into the duct with an area or greater cross-section, must be considered.

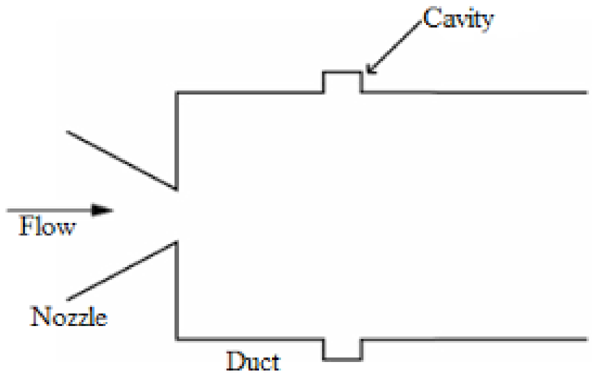


Fig. 1. Cavities in an enlarged duct [2]

As Wick [4] mentioned in his research, the recirculation region will be formed at the base of missiles, projectiles, or automobiles. That happened below the shear layer, leading to the breakdown of large-scale vortices into small vortices. The shear layer will also hit the duct wall, and the point where it hits the duct's surface is called the reattachment point. It will affect the base pressure to increase if the reattachment length increase. He also explained the theory of vortex generation and the mass ejection that become the main reason for the backflow. Wick [4] formulated that the boundary layer is the source of two fluids: the recirculation zone and the backflow from the wall's expanded section. Based on the research of Sethuraman [5], they concluded that the strength of the vortex controls the base pressure and the flow towards the trailing edge. That is prompted by the sudden increase in the area of the flow. Thus, both of the researchers believe in the need for controlling the base pressure.

The presence of cavities aid as a passive control to decrease the drag. Cavities enhance the base pressure to minimize drag forces. The base pressure changes depending on the shape, geometry, or dimension of the cavity employed. As de la Cruz *et al.,* [6] mentioned, a cavity with more depth will increase the base pressure. There was an experimental study of the impacts of the base cavity on compressible near wake flow. They have examined the base cavity's impact on a slender two-dimensional body's near wake flow field in the low-speed range. They considered basic trio configurations: a blunt base, a shallow rectangular cavity at the base, and a bottomless rectangular cavity base equal to the base height. Flow visualization showed that the vortex's fundamental gross structure was unmodified by the base cavity's presence. However, the feebler vortex generated more significant pressures for the cavity bases in the near wake and increases in the range of 10 percent to 14 percent in the base pressure coefficients. In addition, it increased 4% to 6% shedding frequencies similar to the blunt-based configuration.

At the same time, the same base pressure can be obtained with many cavities with less depth. According to Pandey and Rathakrishnan [7], the experiment was conducted to find cavities' influence on the suddenly expanded flow field. Their main objective in delivering cavities was to establish secondary vortices in the duct. The primary purpose is to ascertain; some parameters need to be varied, such as the stagnation pressure, size of the duct, area of the duct, and length to width ratio of the cavity.

The average flow growth in the duct is reasonably smooth, caused by the static pressure variation along the model's length. It was done by the function of the stagnation pressure of two area ratios. However, according to Anderson and Williams [8], flow development's oscillating nature is different from the above investigators' result. A similar study that both investigators have obtained is on average flow in the base's surrounding area, resulting in the low base pressure to increase the same as the static wall pressure near the blunt base with a rise in area ratio. Figure 2 shows that the pressure rises from the base pressure value to atmospheric pressure. The difference in result can be seen when the pressure boost from base pressure to ambient atmospheric levels persists beyond the area ratio even though the experiment had been conducted with similar models Pandey and Rathakrishnan [7]. So, it is considered that the cavity influences the stream in the duct.

Khan *et al.,* [9] investigated a test for multiple cavities and without numerous cavities. The experimental results show that multiple cavities have a perfect effect in reducing the base drag by decreasing the base suction and boosting base pressure. It is concluded that the inertia level significantly impacts the base pressure and influences the control for base pressure. Srikanth and Rathakrishnan [10] investigated the flow in a pipe with a sudden enlargement at sonic Mach number. Their experiments concluded that non-dimensionalized base pressure is firmly inspired by the overall pressure ratio, the expansion area, and the duct size.

Tanner [11] conducted studies on the base cavity's impact at angles of incidence for base pressure. It is known that the base cavity can raise the base pressure, thus lessen the base drag in axisymmetric flow. The angle of attack was varied from 0° to 25°. At 2° incidence, there happened to be the maximum drag decrease. Nusselt [12] studied a test with the high-velocity gas flow through sudden expansion. He concluded that if the entrance flow is supersonic, the base pressure P_b =< or > the entrance pressure.

According to Bonnavion *et al.,* [13] drag decline by nearly 9% was accomplished using a base cavity on a 3-D bluff body, using the square back Ahmed body. In the sense of aerospace engineering, the beneficial influence of a base cavity on the resistance to flow has been recognized for years in axisymmetric bluff bodies. The base cavity removes the oscillations in the flow; this reduces the drag associated with asymmetry and elongates the base of the model’s recirculating area to allow for pressure recovery in the duct.

Figure 2 exhibits the investigational setup used for the experiments. The stagnation chamber and the model's exit are depicted in Figure 2. An axi-symmetric convergent-divergent nozzle is followed by a concentric axi-symmetric circular duct with a larger duct area. Three models were selected with area ratios of 10,6.0, 2.89, and 2. 89.

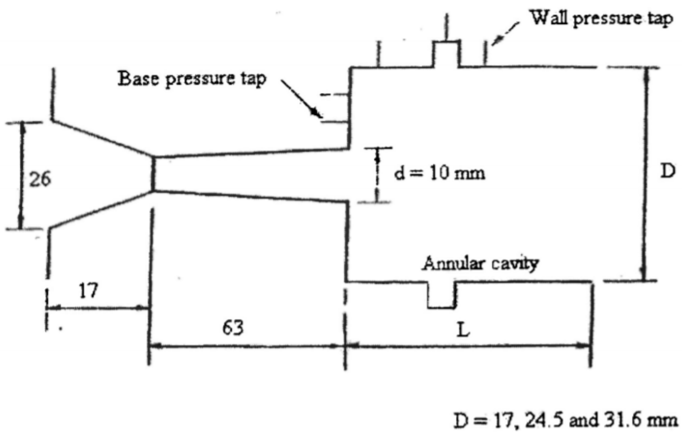


Fig. 2. Schematic of experimental work

The researchers carried out ten pieces of length to the diameter of ratio one so as the entire set can be for duct size varying from 1D to 10D. The researchers also concentrated on the pressure tapings where the pressure tapping was produced with 1.6 mm OD steel tubes. It is introduced in the models, and tubes were sealed with Araldite. After the straight models' investigation, the researchers will experiment with models without annular cavities. These cavities have depth and width and also have field intervals. The static pressure tapping is equal intervals, and the distances between tapping were kept at 5 and 8 mm. The tests were carried out appropriately in expanded cases and 10-1 L/D points. There were 2.89, 6.0, and 10 models of area ratios [7].

2. Methodology

This section illustrated 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 the boundary conditions.

2.1 Two-Dimensional CD Nozzle

The CD nozzle with an enlarged duct is modeled based on inertia level and the authors' investigational work; all the dimensions in this model are in mm, shown in Figure 3 [7]. The cavity has a width and depth of 3 mm and is located at 17 mm from the divergent diameter duct. The study examines the flow field through the nozzle and estimates theoretical flow parameters such as pressure and velocity by CFD simulation in 2D modeling with and without a cavity.

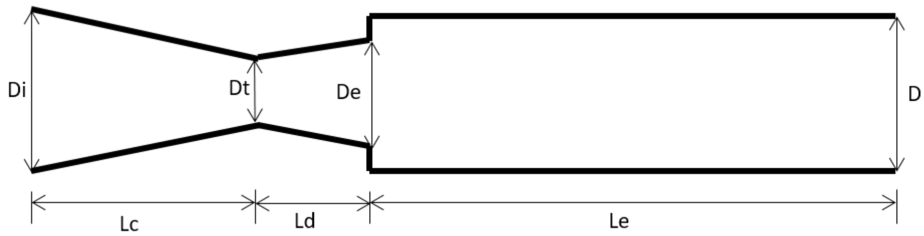


Fig. 3. FE design of CD nozzle with the suddenly expanded duct

A CFD-based model can obtain acceptable results, and to validate the experiment should follow several steps. The minor error can lead to a considerable amount of deviation from actual results. First, the model's geometry is created for the process to proceed, followed by mesh generation. Next, physics and fluid properties are selected, and boundary conditions are defined. Finally, the operation of initialization is compulsory for the calculation to be simulated. Moreover, post-processing is performed to confirm and check the results [14]. The dimensions of the nozzle are mentioned in Table 1.

Table 1
Dimension of CD nozzle

Mach Number	1.74
Inlet diameter (Di)	30mm
Throat diameter (Dt)	8.487mm
Exit diameter (De)	10 mm
Extended diameter (D)	16 mm
Convergent length (Lc)	40.143mm
Divergent length (Ld)	8.646mm
Extended Length (Le)	170mm (Variable)

The nozzle geometries are performed in ANSYS Workbench 18.0 by the 2D planar structure by taking up the dimensions in Table 1. The extended length varies depending on the L/D set to 2, 4, 6, 8, and 10.

2.2 Meshing and Boundary Conditions

Meshing is the essential component at ANSYS Fluent. The reliability and the precision of the simulations depend upon the meshing. When the geometry is designed, meshing is one of the crucial parts needing the care of. Meshing needs to be done with utmost care because it is one of the essential components to discuss. High-density meshing will lead to statistical reliability and higher precision of meshing in computational fluid dynamics. For excellent meshing, the mesh density should be divided carefully. For example, the mesh density between the throat region and the nozzle walls was set at a higher meshing division than the inlet and outlet. Figure 4 and Figure 5 show the mesh details of the 2D planar C-D nozzle. The structured mesh is used for the analysis. The grid independence test has been carried out to find the optimum size of the grid to save time. Table 2 shows the results of the grid independence test.

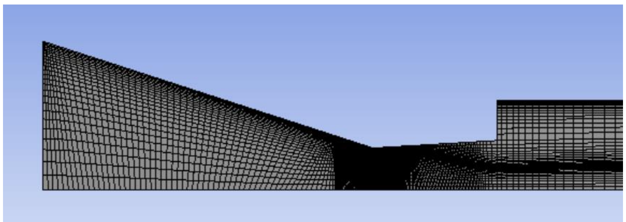


Fig. 4. Closer view Structured Mesh of 2D planar C-D Nozzle

Fig. 5. Structured Mesh of 2D planar C-D Nozzle

Table 2
Grid independence test for various mesh sizes

Mesh Max Size (mm)	Mesh Elements	Static Pressure at Base Edge	Total Pressure at Base Edge
5	56	0.982321125	0.983963473
4	92	0.924344944	0.92464879
3	162	0.860645547	0.862173393
2	352	0.878006385	0.881550664
1	1310	0.885458179	0.886654320
0.8	2030	0.883369159	0.884080365
0.6	3609	0.885199250	0.886767540

Based on the result, it is found that the result of base pressure is stable at a grid size of 1 mm. Therefore for further analysis, the grid size of 1 mm is considered.

2.3 Calculation Procedure

The numerical modeling involves selecting proper mathematical models, i.e., governing equations, boundary conditions, the mesh quality, and the numerical scheme to solve the governing equations simultaneously. Although the computational method does not precisely represent physical phenomena, it provides sufficient insight into the flow behavior and is trusted over decades. Therefore, this requires a proper selection of aspects that can closely mimic the flow behavior. In this work, the assumptions are identified to compromise the exact physical situation. The beliefs and features discussed in this study are as follows:

(i) The flow is supposed to be a steady-state two-dimensional, 2D flow due to the flow being symmetric along the flow direction.
(ii) As for the flow velocity, the turbulent viscous dissipation effects are considerable, so the turbulent flow is considered.
(iii) The fluid is compressible, and its viscosity is a function of temperature.
(iv) The flows exit from the duct at ambient atmospheric pressure.

The nozzle flow is considered turbulent; hence, the K-ε standard model is applied for the compressible flow field. The following equations represent the turbulent flow field most appropriately. Continuity equation for the compressible flow (density ρ, based) with the steady-state condition in the 2-dimensional cylindrical coordinate system:

$$\frac{1}{r}\frac{\partial(\rho r u)}{\partial r}+\frac{\partial(\rho v)}{\partial z}=0 \tag{1}$$

Where r and u are the radius and the velocity in the z-direction. ρ is the density of the flow. V is the velocity in the y-direction. The time-averaged axial z-momentum equation representing u velocity of flow is given by

$$\frac{1}{r}\frac{\partial(\rho r u u)}{\partial z}+\frac{1}{r}\frac{\partial(\rho v u)}{\partial r}=-\frac{\partial p}{\partial r}+(\mu+\mu_t)\frac{\partial}{\partial z}\left[2\frac{\partial u}{\partial z}-\frac{2}{3}(\nabla.\vec{v})\right]+(\mu+\mu_t)\frac{\partial}{\partial r}\left[\frac{\partial u}{\partial r}+\frac{\partial v}{\partial z}\right] \tag{2}$$

Where μ and μ_t are the dynamic viscosity and turbulent viscosity, here, $\nabla.\vec{v}$ is the divergence of the velocity vector. In this analysis of compressible flow, the density of a gas is selected as an ideal gas constant, and the viscosity is applied according to the law of Sutherland as the dynamic viscosity is a function of temperature, T. Sutherland's viscosity model based on three coefficients is given by the form:

$$\mu=\mu_o\left(\frac{T}{T_o}\right)^{3/2}\frac{T_o+S}{T+S} \tag{3}$$

The radial r-momentum equation for velocity, v is given by:

$$\frac{1}{r}\frac{\partial(\rho r u v)}{\partial z}+\frac{1}{r}\frac{\partial(\rho v v)}{\partial r}=-\frac{\partial p}{\partial r}+(\mu+\mu_t)\frac{\partial}{\partial r}\left[\left(2\frac{\partial u}{\partial z}-\frac{2}{3}(\nabla.\vec{v})\right)\right]+(\mu+\mu_t)\frac{\partial}{\partial z}\left[\left(\frac{\partial u}{\partial r}+\frac{\partial v}{\partial z}\right)\right]-2\frac{(\mu+\mu_t)v}{r^2}+\frac{2}{3}\frac{1}{r}(\mu+\mu_t)(\nabla.\vec{v}) \tag{4}$$

The term $\vec{v}$ in Eq. (2) and Eq. (3) is given by:

$$\nabla.\vec{v} = \frac{\partial u}{\partial z} + \frac{\partial v}{\partial r} + \frac{v}{r} \tag{5}$$

Here, μ is the viscosity, and μ_o is the reference viscosity (kg/m-s), T is the static temperature, and T_o is the reference temperature (K). S is Sutherland constant, depending upon sufficient temperature. The quantity $\left(\frac{k}{C_p} + \frac{\mu_t}{Pr_t}\right)$ represents the applicable thermophysical property of the fluid where k is the thermal conductivity (W/m2k), C_p is specific heat capacity (kJ/kg-K), μ_t is turbulent viscosity in (kg/m-s), and Pr_t is a turbulent Prandtl number.

The K-ε turbulence model is among the most used models that provide economy, sturdiness, and adequate correctness for several kinds of flow conditions. The K-ε turbulence model utilized in this study is by the Ansys Fluent software. The turbulent kinetic energy, i.e., K-equation, is given by:

$$\frac{\partial(\rho u \mathrm{K})}{\partial z} + \frac{1}{r}\frac{\partial(\rho v \mathrm{K})}{\partial r} = \frac{\partial}{\partial z}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \mathrm{K}}{\partial z}\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial \mathrm{K}}{\partial r}\right] - \rho\varepsilon + G \tag{6}$$

The term σ_k is the turbulent Prandtl number for K, and ε is turbulent kinetic energy dissipation rate, and G is turbulence generation term given by:

$$G = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j} - \frac{2}{3}k\delta_{ij}\frac{\partial u_i}{\partial x_j} \tag{7}$$

The kinetic energy of turbulence dissipation, i.e., ε-the equation is given by:

$$\frac{\partial(\rho u \varepsilon)}{\partial z} + \frac{1}{r}\frac{\partial(\rho v \varepsilon)}{\partial r} = \frac{\partial}{\partial z}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial z}\right] + \frac{1}{r}\frac{\partial}{\partial r}\left[r\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial r}\right] - C_1 f_1\left(\frac{\varepsilon}{\mathrm{K}}\right)G - C_2 f_2\left(\frac{\varepsilon^2}{\mathrm{K}}\right) \tag{8}$$

The term $\mu_t = \rho f_\mu C_\mu k^2/\varepsilon$ represents turbulent viscosity, and C_μ, C_1, C_2, f_μ, σ_k, and σ_ε are arbitrary constants for the kinetic energy of turbulence dissipation model in the above equations.

3. Results and Discussion

The experiment with annular cavities and without cavities was conducted for different combinations of process parameters. The experimental data was extracted from the paper [7]. The convergent-divergent nozzle with and without a cavity is used to validate the result. The variation of L/D 2,4 and 6 in Figure 6 as per Pandey and Rathakrishnan [7] is considered. By comparing the results obtained by Pandey and Rathakrishnan [7] and the present work, there are in good agreement, as shown in Figure 6.

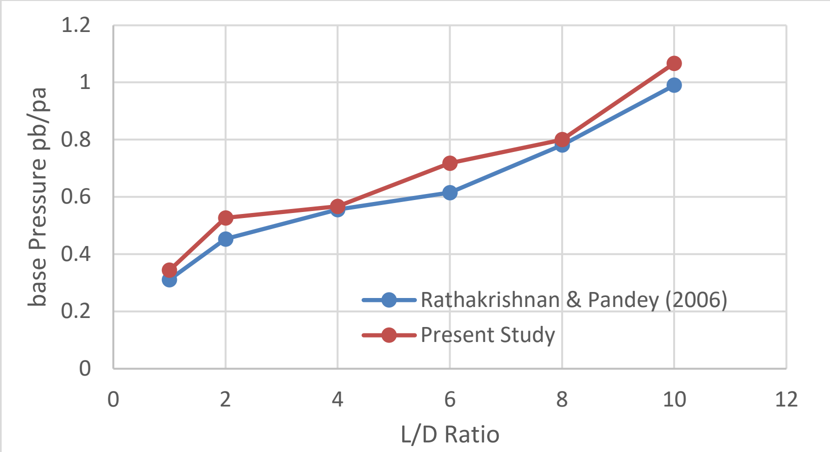


Fig. 6. Verification of present study

The structure considered is selected to verify the effects of base pressure variation with the L/D ratio, as seen in Figure 6, which is chosen to validate the base pressure results [7]. For NPR 3 to 11, Nusselt [12] performed Mach Number 1.74 and area ratios of 2.10, 2.65, and 3.48 studies with and without cavities. The agreement for the plots of base pressure variations with the NPR for various duct L/D is shown in Figure 6 by comparing the experimental findings obtained by Pandey and Rathakrishnan [7] as per the latest results.

3.1 Effect of NPR on Pressure

This study's most important finding is that pressure is distributed differently in each region, such as the throat, convergent, divergent, and expanded duct in a CD nozzle. As a result, pressure from all zones of the CD Nozzle is considered in this segment. Results obtained by findings for the case without a cavity, the distribution of wall pressure in the duct varies consistently. It reaches a relatively higher value at the duct's exit. Meanwhile, the sudden expansion area's pressure increases due to the development of shock waves in the recirculation zone due to increased base pressure. We can validate this result by observing the contour color where the blue color shows the sudden expansion region is high, which means it has a low-pressure effect and at the same time shows high drag formation. From the previous study by Aabid *et al.,* [15], the microjets were used to control the base drag in the sudden expansion region. More studies in the literature are available on suddenly expanded flows to find the parameters affecting base pressure [16-30].

A passive control, also known as the cavity, controls the base drag in the sudden expansion region for the present study. At the same time, for the present study, a passive control, also known as the cavity, is used to control the base drag. Figure 7 displays that when control is used at the base, the nature of the flow and the distribution of wall pressure for different NPRs remain unchanged.

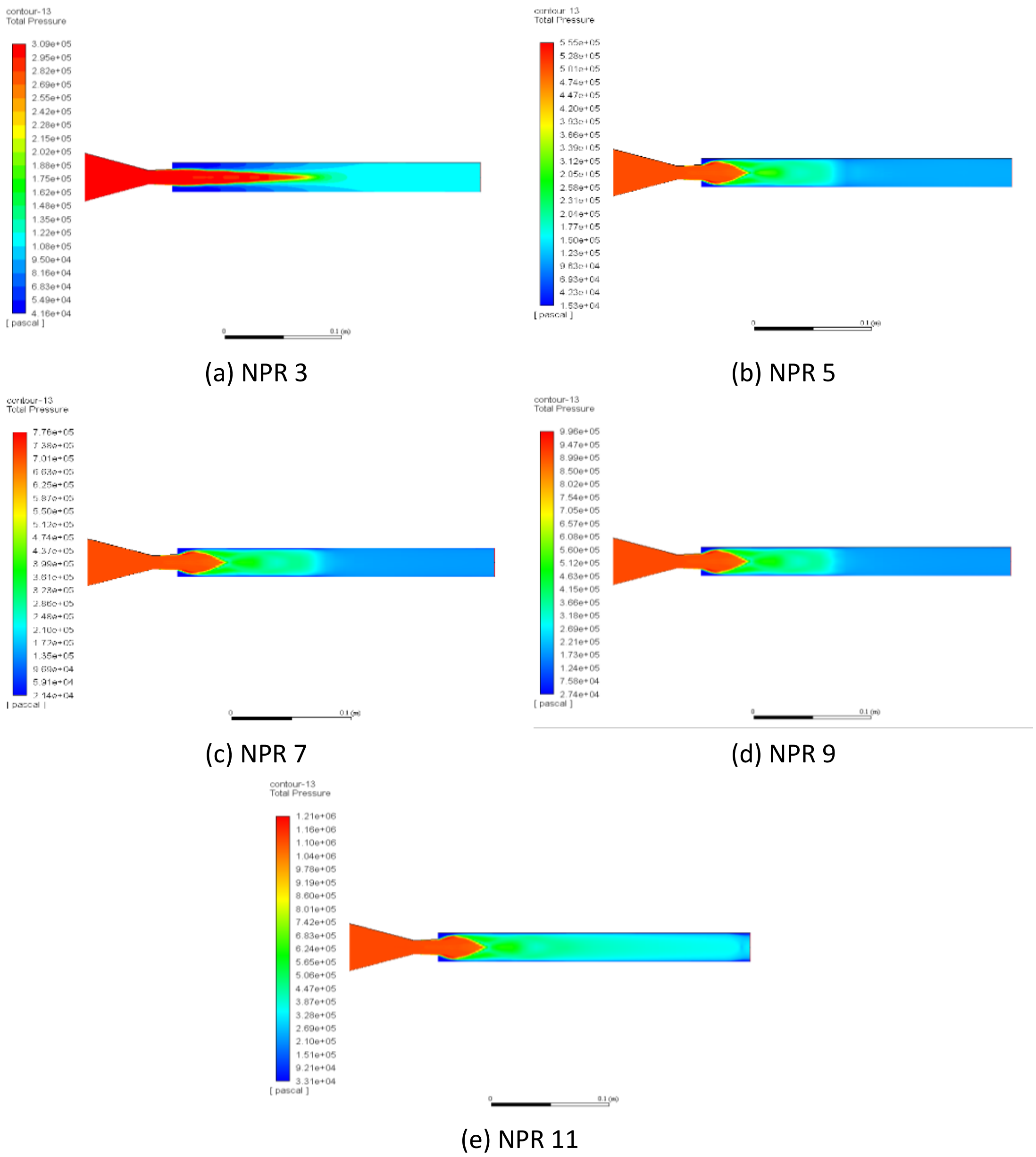
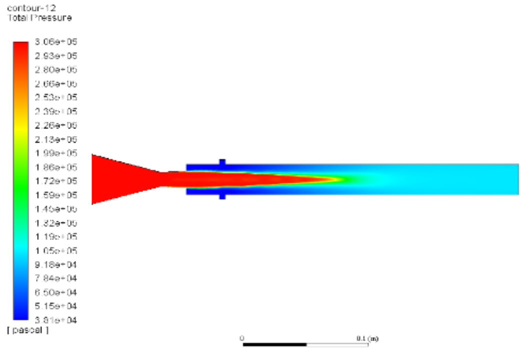
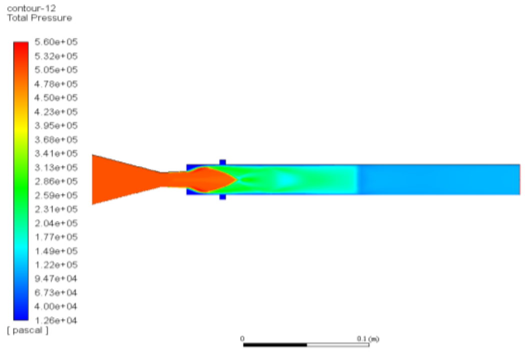


Fig. 7. Effect of NPR on total pressure for suddenly Expanded Flows of L = 10D Extended Length = 17 mm

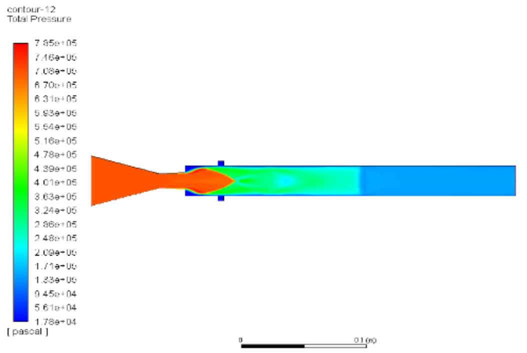
It is easy to measure the cavity's effectiveness when looking at the contours since blue is restricted to a small portion of the sudden expansion area. It can be concluded that the use of cavity did affect an increase in base pressure, which also decreased the base drag at few regions and increased the thrust for supersonic aerospace vehicles. Figure 8 shows the effect of NPR on total pressure of L/D 10 with Aspect Ratio 3:3 using 1 Cavity for Suddenly Expanded Flows Extended Length = 17mm.



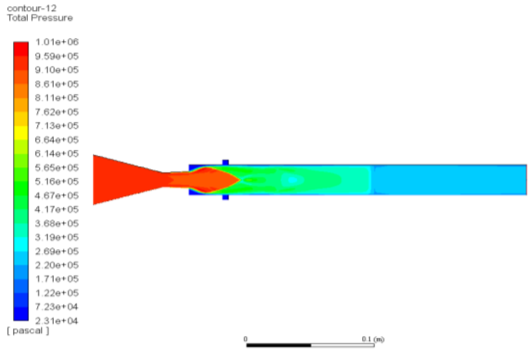
(a) NPR 3



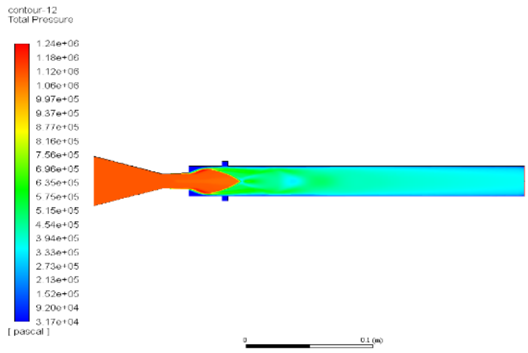
(b) NPR 5



(c) NPR 7



(d) NPR 9



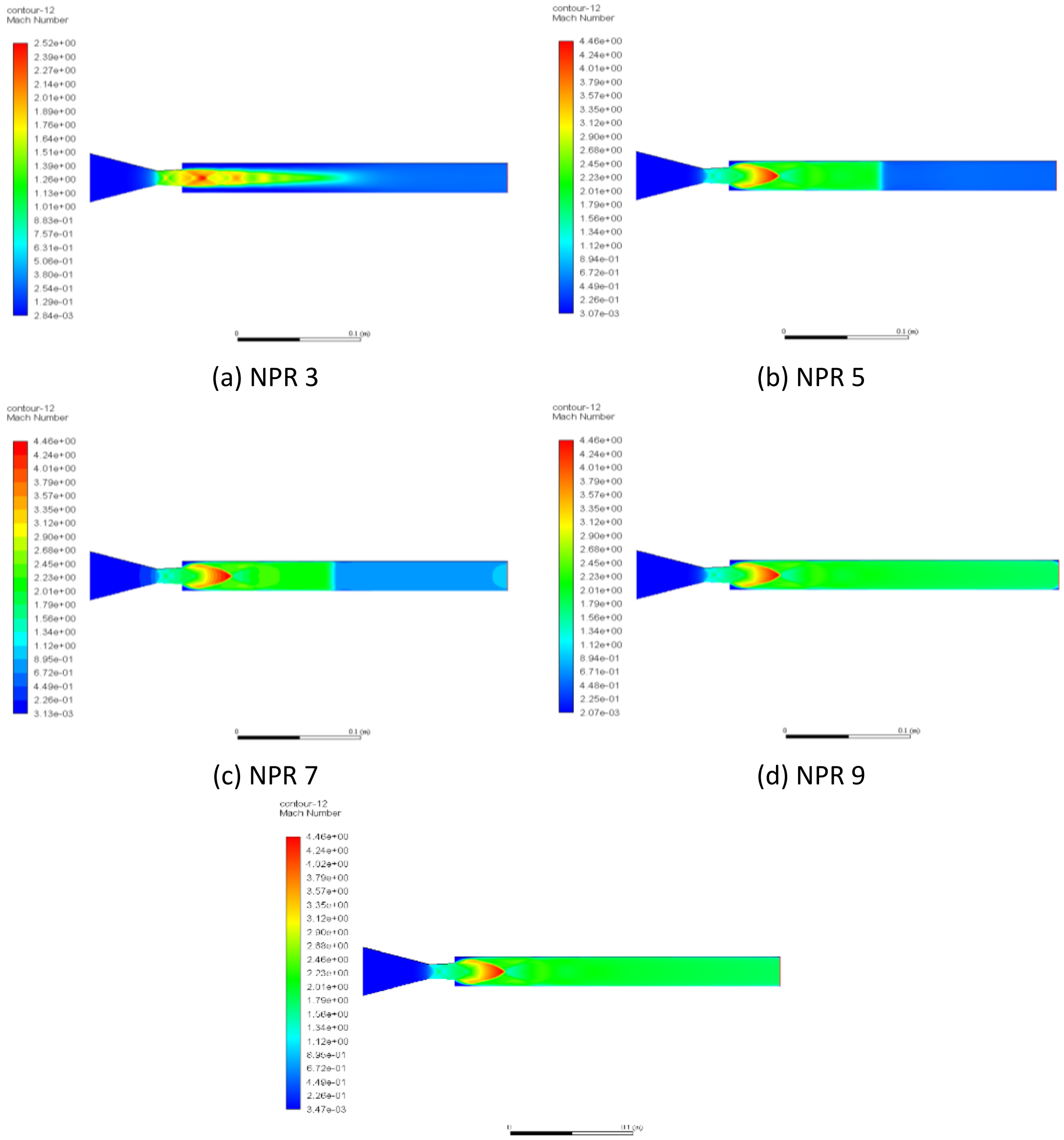
(e) NPR 11

Fig. 8. Effect of NPR on total pressure of L/D 10 with Aspect Ratio 3:3 using 1 Cavity for Suddenly Expanded Flows Extended Length = 17mm

3.2 Effect of NPR on Velocity (Mach No)

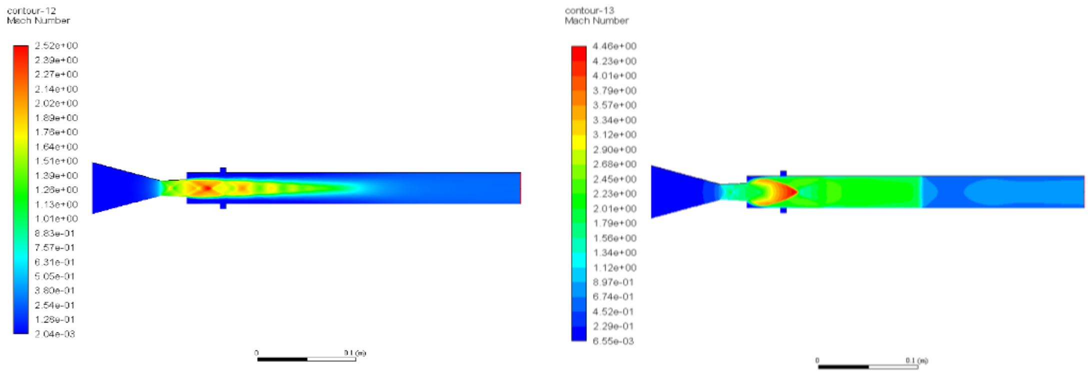
According to the energy equation, the pressure and the velocity are related. And as the pressure increase, the velocity decrease. The Mach number has been evaluated as a function of inertia level from the inlet to the CD nozzle outlet at various NPRs. The contour plots for Mach number effects are made using the same concept, as seen in the contours. The pressure contours are found to be in contrast to the patterns.

The Mach number at the sudden expansion region suddenly increases, then decreases after exiting the nozzle and flowing through the wall. It then falls again when it starts to reach the exit portion of the duct, as shown in Figure 9. The cavity's influence shows that the drag reduction at the base area is studied for pressure effects, and there is no adverse impact on the wall pressure flow field. Figure 10 shows the relevant effect on NPR using the control condition.

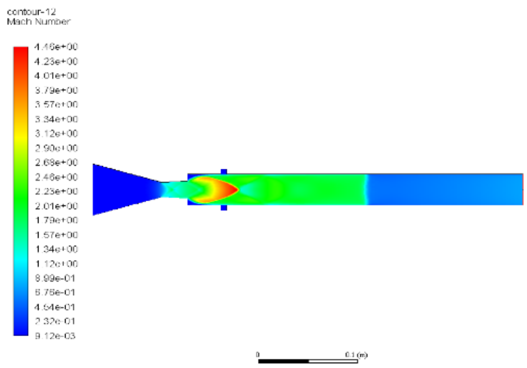


(a) NPR 3
(b) NPR 5
(c) NPR 7
(d) NPR 9
(e) NPR 11

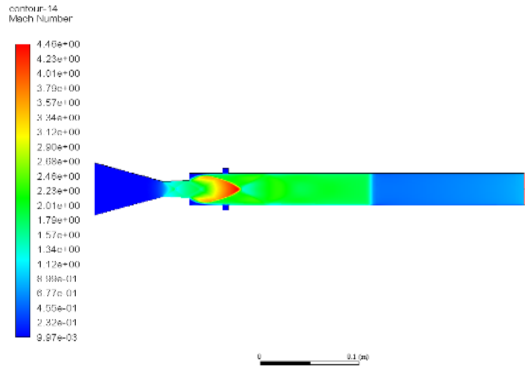
Fig. 9. Effect of NPR on Velocity (Mach Number) of L = 10D for Suddenly Expanded Flows Extended Length = 17mm



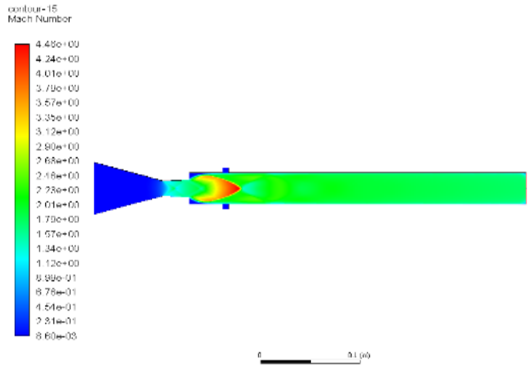
(a) NPR 3
(b) NPR 5



(c) NPR 7



(d) NPR 9



(e) NPR 11

Fig. 10. Effect of NPR on Velocity (Mach Number) of L = 10D with Aspect Ratio 3:3 using 1 Cavity for Suddenly Expanded Flows Extended Length = 17mm

4. Conclusion

In conclusion, cavities' presence affects the base pressure and can be considered one of the most valuable methods to decrease net drag. The suitable way that is going to be used for future investigation is successfully demonstrated. Further analysis can be performed to find the cavity's optimum geometry to achieve the maximum base pressure. Also, one can try various shapes of a cavity's geometry, which will result in the highest rise and decline in the base pressure. The cavity does not introduce any change inside the duct flow pattern. Instead, it is seen that the control smoothens and suppresses the flow's oscillatory nature in the enlarged duct.

Acknowledgment
This research was funded by Kementerian Pendidikan Malaysia, Grant No. 203/PAERO/6071437, and the APC was funded by Universiti Sains Malaysia and Kementerian Pendidikan Malaysia.

References
[1] Kruiswyk, R. W., and J. Craig Dutton. "Effects of a base cavity on subsonic near-wake flow." *AIAA Journal* 28, no. 11 (1990): 1885-1893. https://doi.org/10.2514/3.10495

[2] Rathakrishnan, E. "Effect of ribs on suddenly expanded flows." *AIAA Journal* 39, no. 7 (2001): 1402-1404. https://doi.org/10.2514/2.1461

[3] Khan, Sher Afghan, Mohammed Asadullah, Fharukh Ahmed G. M., Ahmed Jalaluddeen, and Maughal Ahmed Ali Baig. "Passive control of base drag in compressible subsonic flow using multiple cavity." *International Journal of Mechanical and Production Engineering Research and Development* 8, no. 4 (2018): 39-44. https://doi.org/10.24247/ijmperdaug20185

[4] Wick, Robert S. "The effect of boundary layer on sonic flow through an abrupt cross-sectional area change." *Journal of the Aeronautical Sciences* 20, no. 10 (1953): 675-682. https://doi.org/10.2514/8.2794

[5] Sethuraman, Vigneshvaran. "Investigation of base and wall pressure in suddenly expanded flow through ducts using ribs as passive flow control." *PhD diss., Universiti Sains Malaysia*, 2019.

[6] de la Cruz, Juan Marcos García, Anthony R. Oxlade, and Jonathan F. Morrison. "Passive control of base pressure on an axisymmetric blunt body using a perimetric slit." *Physical Review Fluids* 2, no. 4 (2017): 043905. https://doi.org/10.1103/PhysRevFluids.2.043905

[7] Pandey, K. M., and E. Rathakrishnan. "Annular cavities for Base flow control." *International Journal of Turbo and Jet Engines* 23, no. 2 (2006): 113-128. https://doi.org/10.1515/TJJ.2006.23.2.113

[8] Anderson, J. S., and T. J. Williams. "Base pressure and noise produced by the abrupt expansion of air in a cylindrical duct." *Journal of Mechanical Engineering Science* 10, no. 3 (1968): 262-268. https://doi.org/10.1243/JMES_JOUR_1968_010_038_02

[9] Khan, Ambareen, Nurul Musfirah Mazlan, and Mohd Azmi Ismail. "Analysis of flow through a convergent nozzle at Sonic Mach Number for Area Ratio 4." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 62, no. 1 (2019): 66-79.

[10] Srikanth, R., and E. Rathakrishnan. "Flow through pipes with sudden enlargement." *Mechanics Research Communications* 18, no. 4 (1991): 199-206. https://doi.org/10.1016/0093-6413(91)90067-7

[11] Tanner, Mauri. "Base cavity at angles of incidence." *AIAA Journal* 26, no. 3 (1988): 376-377. https://doi.org/10.2514/3.9903

[12] Nusselt, W. "Heat Dissipation From Horizontal Tubes and Wires to Gases and Liquids." *Ver DeutIng* 73 (1929): 1475-1478.

[13] Bonnavion, Guillaume, Olivier Cadot, Vincent Herbert, Sylvain Parpais, Rémi Vigneron, and Jean Délery. "Effect of a base cavity on the wake of the squareback Ahmed body at various ground clearances and application to drag reduction." In *Congrès français de mécanique*. AFM, Association Française de Mécanique, 2017.

[14] Magoulès, Frédéric, ed. *Computational fluid dynamics*. CRC Press, 2011. https://doi.org/10.1201/b11033

[15] Aabid, Abdul, Zakir Ilahi Chaudhary, and Sher Afghan Khan. "Modelling and analysis of convergent divergent nozzle with sudden expansion duct using finite element method." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 63, no. 1 (2019): 34-51.

[16] 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. https://doi.org/10.1016/j.csite.2018.09.006

[17] Pathan, Khizar Ahmed, Prakash S. Dabeer, and Sher Afghan Khan. "An investigation to control base pressure in suddenly expanded flows." *International Review of Aerospace Engineering (I. RE. AS. E)* 11, no. 4 (2018): 162-169. https://doi.org/10.15866/irease.v11i4.14675

[18] Pathan, Khizar Ahmed, Syed Ashfaq, Prakash S. Dabeer, and Sher Afgan Khan. "Analysis of Parameters Affecting Thrust and Base Pressure in Suddenly Expanded Flow from Nozzle." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 64, no. 1 (2019): 1-18.

[19] Pathan, Khizar A., Prakash S. Dabeer, and Sher A. Khan. "Enlarge duct length optimization for suddenly expanded flows." *Advances in Aircraft and Spacecraft Science* 7, no. 3 (2020): 203-214.

[20] Pathan, Khizar Ahmed, Prakash S. Dabeer, and Sher Afghan Khan. "Influence of Expansion Level on Base Pressure and Reattachment Length." *CFD Letters* 11, no. 5 (2019): 22-36.

[21] Pathan, Khizar Ahmed, Prakash S. Dabeer, and Sher Afghan Khan. "An investigation of effect of control jets location and blowing pressure ratio to control base pressure in suddenly expanded flows." *Journal of Thermal Engineering* 6, no. 2 (2019): 15-23. https://doi.org/10.18186/thermal.726106

[22] Pathan, Khizar Ahmed, Prakash S. Dabeer, and Sher Afghan Khan. "Effect of nozzle pressure ratio and control jets location to control base pressure in suddenly expanded flows." *Journal of Applied Fluid Mechanics* 12, no. 4 (2019): 1127-1135. https://doi.org/10.29252/jafm.12.04.29495

[23] Pathan, Khizar Ahmed, Prakash S. Dabeer, and Sher Afghan Khan. "Investigation of base pressure variations in internal and external suddenly expanded flows using CFD analysis." *CFD Letters* 11, no. 4 (2019): 32-40.

[24] Sajali, Muhammad Fahmi Mohd, 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.

[25] Khan, Sher Afghan, 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, no. 01 (2019): 70-82.

[26] Khan, S. A., Mohammed Asadullah, and Jafar Sadhiq. "Passive Control of Base Drag Employing Dimple in Subsonic Suddenly Expanded Flow." *International Journal of Mechanical & Mechatronics Engineering* 18, no. 3 (2018): 69-74.

[27] 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 (2011): 59-69. https://doi.org/10.1515/tjj.2011.009

[28] Fharukh Ahmed, G. M., Mohammad Asad Ullah, and Khan S. A. "Experimental study of suddenly expanded flow from correctly expanded nozzles." *ARPN Journal of Engineering and Applied Sciences* 11 no. 16 (2016): 10041-10047.

[29] Asadullah, Mohammed, Sher Afghan Khan, Waqar Asrar, and E. Sulaeman. "Passive control of base pressure with static cylinder at supersonic flow." In *IOP Conference Series: Materials Science and Engineering*, vol. 370, no. 1, p. 012050. IOP Publishing, 2018. https://doi.org/10.1088/1757-899X/370/1/012050

[30] Khan, Sher Afghan, Abdul Aabid, Imran Mokashi, Abdulrahman Abdullah Al-Robaian, and Ali Sulaiman Alsagri. "Optimization of two-dimensional wedge flow field at supersonic Mach number." *CFD Letters* 11, no. 5 (2019): 80-97.