NTU Journal of Engineering and Technology (2025) 4 (2): 24 - 30
DOI: https://doi.org/10.56286/ntujet.v4i2

IRAQI Academic Scientific Journals

P-ISSN: 2788-9971 E-ISSN: 2788-998X
NTU Journal of Engineering and Technology
Available online at: https://journals.ntu.edu.iq/index.php/NTU-JET/index

Numerical Simulation of the Effect of Rounded Edge Cylinders on Drag Coefficient

Elaf Ayad Hasan[1], Tahseen Ahmed Tahseen[2], Sami Ridha Aslan[1]
[1]Technical Engineering College Kirkuk, Northern Technical University, Mosul, Iraq
[2]College of Engineering. University of Kirkuk, Kirkuk, Iraq.
elaf.ayad@ntu.edu.iq, tahseen@uokirkuk.edu.iq aslan.sami@ntu.edu.iq

Article Informations

Received: 02-11- 2023,
Revised: 06-01-2024,
Accepted: 08-02-2024,
Published online: 23-06-2025

Corresponding author:
Name: Elaf A. Hasan
Affiliation : Technical Engineering College Kirkuk
Email: elaf.ayad@ntu.edu.iq

Key Words:
Ansys fluent,
drag coefficient,
Rounded edged cylinder,
Numerical simulation.

ABSTRACT

This work aims to investigate the influence of the presence of three cylinders with rounded edges on the drag coefficient. Three heated cylinders, whose shapes varied from triangle to circle, were placed in a 2-dimensional horizontal channel with laminar, steady flow. The effect of five different Reynolds numbers (Re = 100, 200, 400, 800, and 1200) and three vertical distances (S_T = 1.5D, 2D, and 3D) has been studied. A constant heat flux (q" = 5000 W/m^2) subjected to the cylinders' walls and the rounded corners radii of the triangular cylinders was (1 mm, 1.6667 mm). The finite volume method is used to solve the governing equations (ANSYS FLUENT 2022 R2). It was observed that the coefficient of drag decreases as the Re increases and decreases with decreasing longitudinal distance. The drag coefficient decreased from (37.01%) to (72.3%) when the Reynolds number increased from (100) to (1200) at S_T = 3D and from (33.1%) to (62.4%) at S_T = 1.5D.

1. Introduction

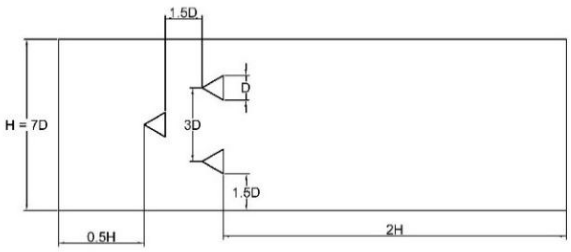
In the fluid mechanics field, one of the important problems from pragmatic and practical analysis is the flow of fluids over bluff bodies [1]. These flow models are used to have useful information about physical processes, for example, the flow field structure, vortex shedding, the wake phenomenon, etc. It also gives a prediction of the rates of mass and heat transfer and the hydrodynamic forces (drag and lift) acting on immersed objects [2]. The focus point of many experimental or numerical investigations is the influence of the cylinder number, position, and form; they also focused on the boundary of the computational zone and the Reynolds number [3]. However, one of the significant practical concerns is studying the drag of the flow over bluff bodies. It's very important to investigate the drag force, especially for submarines, and also to calculate the power required for airplanes, etc [4]. There are a lot of previous works that focused on the investigation of fluid flow characteristics and heat transfer using cylinders with various cross-sections. Alam et al. [5] inspected, using a numerical method, the heat transfer topology and 2D, laminar, incompressible flow of a variable cross-sectional cylinder from square to circular by varying the ratio of the edge-radius of the cylinder (r/R = 0–1), where R represents the width of half the side of the cylinder and r represents the radius of the corner. The angle of flow (α) is changed from 0° to 45°. It was observed that heat transfer improved with the angle facing the flow (α) increment, where the minimum improvement in heat transfer was when (r/R = 0) and (α = 0°), and its ultimate value was when (r/R = 0.5) and (α = 45°). Rosales et al. [6] analyzed numerically the heat transfer and laminar, unsteady flow characteristics of a pair of tandem nonidentical cylinders that are square in shape inside a channel, where the small cylinder is placed in front of the large one. They studied the drag and lift coefficient in addition to the coefficient of heat transfer of the heated rear cylinder due to the in-line and offset cylinders enhancing the vortexes. They observed that as the heated cylinder gets near the channel wall, the Nusselt number and the coefficient of drag of the cylinder decrease. Agarwal and Dhiman [7] studied the heat transfer and flow through two extended triangle-shaped rods, where the triangles were placed in a tandem array inside the channel. The Reynolds number range was (Re = 1–40), the ratio of the gap (S/B) was (1–4) with a 25% blockage ratio, and the Prandtl number was (Pr = 0.71–50). They observed that the average Nusselt number for the second cylinder is less than the first cylinder. Jue et al. [8] carried out a numerical study of forced and mixed convection heat transfer for laminar-transient flow between two parallel plates that contain three cylinders that are heated and placed in the form of a right-angled isosceles triangle. For mixed convection, the Grashoff number ranged between 80,000 and 200,000; for forced convection, the Reynolds number ranged between 100 and 300; and the ratio of the gap to the diameter ranged from 0.5 to 1.25. The results that were obtained showed the time and surface average Nusselt number for both cases had the highest value when the ratio of gap-to-diameter was 0.75. Eleiwi et al. [9] conducted a numerical study to improve the laminar flow characteristics and heat transfer around three adiabatic non-rotative circular cylinders placed inside a channel with a backward-facing step. The study included examining the effect of heat flux, Reynolds number, and horizontal distance between two successive cylinders on the characteristics of heat transfer. Five values of the Reynolds number were examined (Re = 50, 100, 150, 200, 250) and three values of the heat flux on the bottom surface of the channel (q" = 250, 500, 750 W⁄m2), and the horizontal distance was (2H, 3H, 4H), where (H = 10 mm) is the cylinder diameter. The results showed an improvement in heat transfer within the channel and a decrease in the reattachment distance in the presence of cylinders. Admi et al. [10] examined numerically the laminar flow control using a flat plate with three square cylinders that are heated and placed beside each other in a 2D channel. The height, length, and location effects of the flat plate on heat transfer and flow were studied using the double relaxation time multiple network method of Boltzmann (DMRT-LBM). The study was carried out by placing the plate horizontally once and vertically the second time around one cylinder. The results showed that when the plate is placed horizontally at the location (g = 3) behind the middle cylinder and when its length is (Lp = 4D), this leads to a reduction in the capacity of the Kármán vortex wave and a major and regular increase in heat transfer. When the plate is vertically positioned at the location (g = 3) beyond the middle cylinder and when its height is (h = 2D), it ameliorates the exchange of heat between the heated cylinders and the inlet fluid. Oztop et al. [11] numerically examined the heat transfer of steady forced convection of two-dimensional laminar flow around isothermal blocks placed inside a channel. Three squares in shape were mounted on the bottom wall of the channel, and a triangular bar was used as a control inside the channel, where both the triangular bar and top wall were isothermal. The rod was fixed at two various points in the direction of the y-axis. Three values of the Reynolds number were used, ranging from (400 ≤ Re ≤ 1300). They compared the results with the situation of the channel without the existence of the triangle rod. It was observed from the results for all values of the Reynolds number that the presence of the triangular rod improves heat transfer. After a serious consideration of the literature review, it is observed that the previous works did not study the effect of rounded edges of a triangular cylinder on drag coefficient or on heat transfer and fluid flow in general, so this paper aims to investigate the effect of changing the tip of a triangular cylinder from a pointed tip to a rounded one by using different radii for the rounded tip, as well as the effect of changing the vertical distance between the downstream cylinders on drag coefficient, and examine if this kind of change in shape of the cylinder and the vertical distances reduce the drag coefficient or not, and observe the percentage of reduction if it is worth using them in industrial fields in the future.

2. Problem formulation

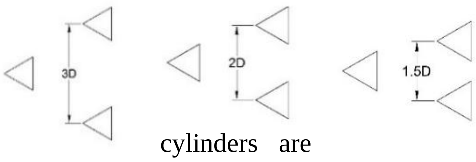
In this paper, the steady, laminar, incompressible fluid flow inside a two-dimensional channel that contains three heated cylinders with their shapes varies from triangle to circle, as illustrated in Figure 1. The side length of the triangle is (D), and other dimensions are given as a function of (D).

Fig. 1. Illustrative diagram of the channel and the vertical distances between the downstream cylinders.

At the inlet, a developing air stream flows into the channel with a constant temperature (T_∞ = 300 K) and five different inlet velocities (Re = 100, 200, 400, 800, 1200). A constant heat flux is applied to the walls of the cylinders (q" = 5000 W/m2). The



vertical distances between the downstream



cylinders are (S_T = 1.5D, 2D, 3D). Two different corner radiuses are used (r = 1, 1.6667 mm) for the triangular cylinders with rounded corners, as shown in Figure 2.

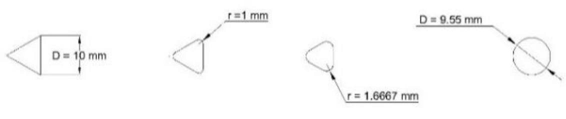


Fig. 2. The cylinders with its shape varying from triangle to circle

3. Governisng equations and Boundary conditions

The equations that are used to govern the forced convection heat transfer and steady, laminar, incompressible fluid flow are continuity, momentum, and energy equations, as illustrated below [12]:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \quad (2)$$

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = -\frac{\partial P}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \quad (3)$$

$$\left(u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y}\right) = \alpha\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \quad (4)$$

The cylinders are subjected to a constant heat flux (q'' = 5000 W/m^2) and the walls of the channel are presumed to be adiabatic with a no-slip condition. The working fluid is air (Pr = 0.71), and the Reynolds numbers are (Re = 100, 200, 400, 800, 1200).

The Reynolds number can be computed from the equation below:

$$Re_H = \frac{\rho v_{in} H}{\mu} \quad (5)$$

Where (H) represents the channel height.

The drag coefficient equation [12]:

$$C_D = \frac{F_D}{\frac{1}{2}\rho {v_{in}}^2 D} \quad (6)$$

The friction factor can be calculated from equation [12]:

$$f = 4C_f \quad (7)$$

C_f is friction drag coefficient (also called Fanning friction factor) is defined as the drag caused by the friction of a fluid against the surface of an object that is moving through it. It is the ratio of wall shear stress over the dynamic pressure of the flowing fluid.

$$C_f = \frac{\tau_s}{\frac{1}{2}\rho V^2} \quad (8)$$

The boundary conditions for the present study are expressed as:

1. At the inlet, the values of the velocity are (0.0222, 0.0445, 0.0889, 0.178, and 0.267) at ($T_\infty = 300\ K$).
2. A constant wall heat flux (q'' = 5000 W/m^2) is applied to the walls of the cylinders.
3. The channel walls are solid and adiabatic, with no-slip conditions.
4. The fluid flows out of the channel at gauge pressure (P = 0 Pa).

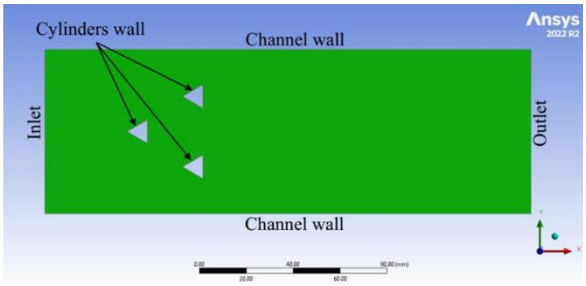


Figure 3 Illustrative diagram of computational domain and boundary.

4. Methodology of numerical analysis

The present investigation has been done using Ansys Fluent (2022 R2) to solve the governing equations using the finite volume method (FVM). A non-uniform grid is used around the cylinders, while the grid structure becomes uniform in other areas. To get a high gradient in the thermal and hydrodynamic boundary layer around the cylinders, the grid is chosen to be fine by creating six layers of inflation around them, and (29619) elements are used to divide the computational domain. The convective terms in the momentum and energy equations are discretized using a second-order upwind scheme. The coupled scheme has been used to solve the velocity-pressure decoupling.

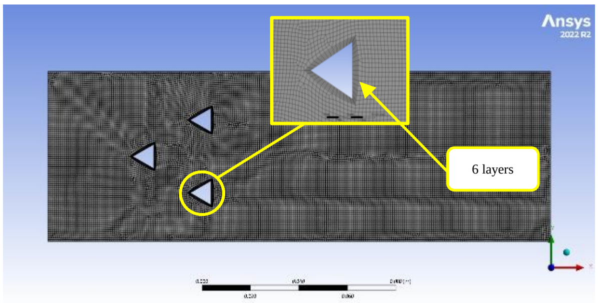


Fig. 4. The computational domain's grid

5. Grid independent test

The grid-independent test has been carried out in order to examine the effect of cell number on the results. The test is done for (Re = 100), for vertical distance (S_T = 3D), and six different grids are used. The results of the test of the grid independency are shown below in Table 1. The error percentage of the average heat transfer coefficient did not exceed 1%.

Table 1. The results of grid independent test at Re = 100 and S_T = 3D

Case	Element size (mm)	Number of elements	h_{ave} $W/m^2.K$	Error (%)
G_1	1.2	21034	7.32591	-
G_2	1.1	11815	7.32350	0.0329
G_3	1	14388	7.32755	0.0553
G_4	0.9	17037	7.32765	0.0014
G_5	0.8	22716	7.32759	0.000899
G_6	0.7	29619	7.32735	0.0033

After choosing the element size (0.7mm), to check the mesh quality of the grid, the minimum and maximum values of the orthogonal quality and ortho skew have been checked, as shown in Table 2.

Table 2. The ranges of the mesh quality.

Range	Orthogonal quality	Ortho skew
Minimum value	0.51286	1.3057e-01
Maximum value	1	0.66667

It can be observed from the table that the mesh quality is good depending on the range of values of the orthogonal quality and ortho skew that are given in [13].

6. Model validation

In this paper, the work done by Kumar and Dhiman [14] is used as a model to validate the used program. Kumar and Dhiman [14] numerically examined the flow in a 2D backward-facing step channel that contains an adiabatic circular cylinder in order to improve the forced convection flow characteristics. They studied the effect of changing the cross-stream position using three different values. The result obtained for the cylinder location with respect to channel height (yc = 1.5) showed great agreement with the maximum deviation (±8.99%). Figure 5 shows the results of validation with Kumar and Dhiman [14].

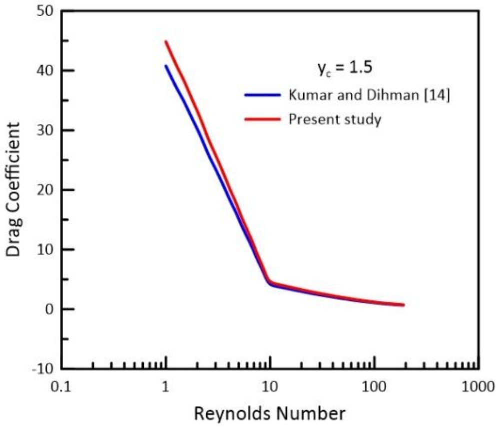
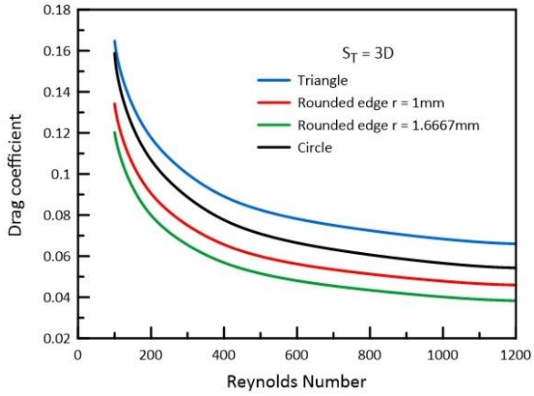


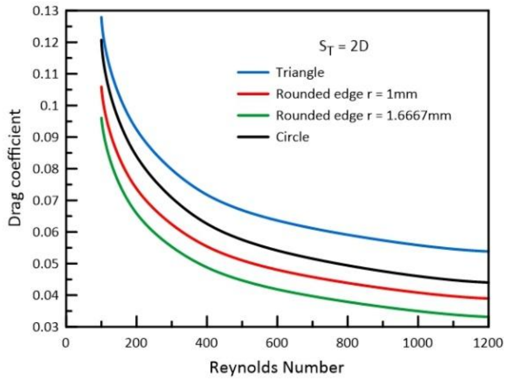
Fig. 5. Comparison of drag coefficient of the present study with the results of Kumar and Dhiman [14]

7. Results

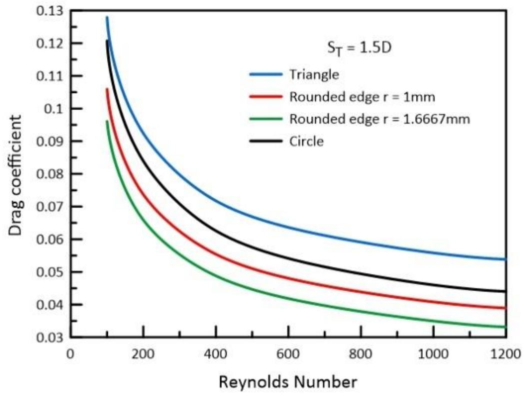
This work aims to study the influence of changing the cylinder shape and the vertical distance between the downstream cylinders on the drag coefficient. The Reynolds numbers are (Re = 100, 200, 400, 800, 1200), and the vertical distances are (S_T = 3D, 2D, 1.5D). The drag coefficient of various shapes and vertical distances has been calculated, and it is shown in Figures 5 and 6. It is known that the coefficient of drag decreases as the Reynolds number increases, and it is remarked that the triangular cylinder has the highest drag coefficient among the cylinders, while the rounded-edged cylinder with a corner radius (r = 1.6667mm) has the lowest value of drag coefficient for all vertical distances, as shown in figure 5. This is because when the tips get more rounded, it makes the shape more streamlined, which reduces the turbulence behind the cylinders by reducing the formation of the eddy currents. The reason for that is the rounded-edged cylinders have a drag coefficient even less than the circular ones because when the pointed tip is changed to a rounded tip the cylinders decrease in shape while the perimeter remains the same for all the used cylinders. For the vertical distances, it is noticed that the increase in vertical distance between the downstream cylinders leads to an increment in the drag coefficient. As shown in Figure 6, for all cylinder shapes, S_T = 3D has the highest drag coefficient and S_T = 1.5D has the lowest. This is due to the upstream cylinder, which acts as shielding when there is a small vertical distance between the cylinders. As the distance increases, the shielding effect of the upstream cylinder weakens, leading to an increase in the drag coefficient and the same effect can be noticed in [15].



(a)

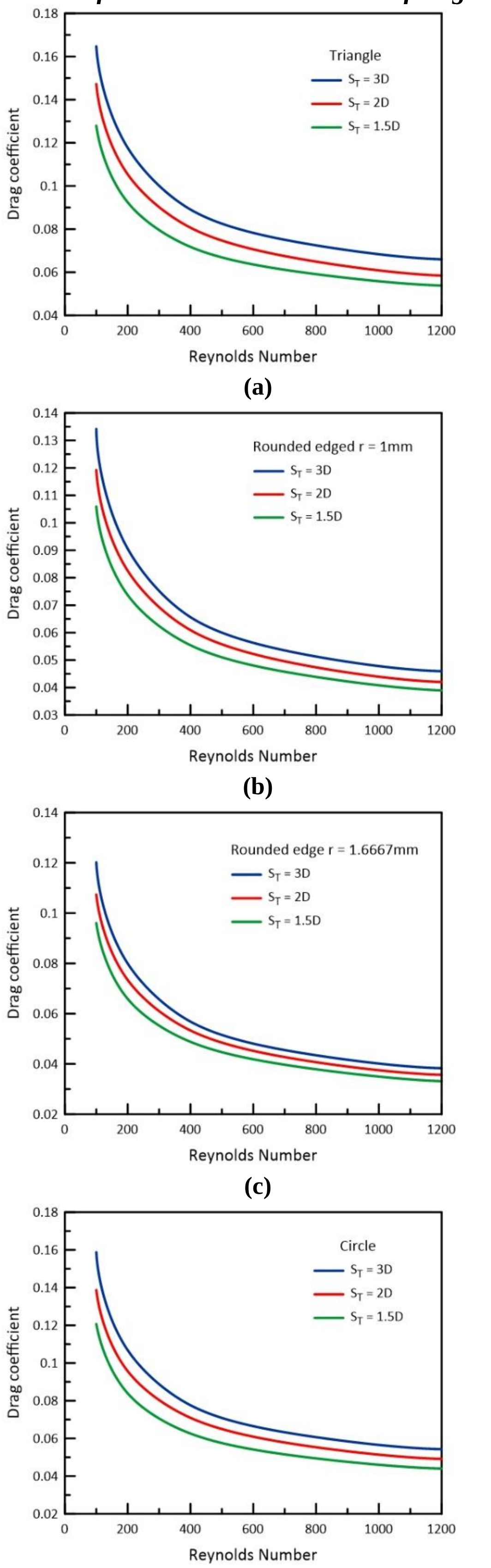


(b)



(c)

Figure 6. Drag coefficient for different vertical distances



(a)

(b)

(c)

(d)

Fig. 7. Drag coefficient for different cylinder shapes

8. Conclusion

A fluid flow over three heated cylinders with various shapes is numerically investigated in a 2D channel. The flow is considered a laminar regime with Reynolds number values (Re = 100, 200, 400, 800, 1200). A constant heat flux (q"= 5000 W⁄m2) is subjected to the cylinder wall. The effect of changing the shape of the cylinders and the vertical distance between the downstream cylinders on the drag coefficient has been studied. Three values of vertical distance have been examined (S_T = 3D, 2D, and 1.5D). Air (Pr = 0.71) is taken as the working fluid, and the flow is assumed to be steady with the temperature of the inlet stream (T_∞ = 300 K). The results show that the coefficient of drag decreases as the vertical distance between the cylinders decreases due to the presence of the upstream cylinder that acts as a shield because the recirculation behind it reduces the velocity of the fluid that attacks the downstream cylinders, which in turn makes the fluid around the downstream cylinder more stabilized, which reduces the turbulence behind them. It decreased from (37.01%) to (72.3%) as the Reynolds number increased from (100) to (1200) at (S_T = 3D) and from (33.1%) to 62.4%) at (S_T = 1.5D). It is also observed that the triangular cylinder with a tip radius of 1.6667mm has the lowest value of drag coefficient, while the normal triangular cylinder has the highest value. This is because the rounded edges give the cylinders a more streamlined effect, which reduces the drag of the cylinders.

Nomenclature

C_D	drag coefficient	S_T	vertical distance between downstream cylinders (mm)
C_f	fanning friction factor	T	Temperature (K)
D	Diameter (mm)	T_∞	Air inlet temperature (K)
f	Friction factor	u	Velocity in x direction (m/s)
F_D	Drag force (N)	v	Velocity in y direction (m/s)
h	heat transfer coefficient ($W/m^2.K$)	v_{in}	Inlet velocity (m/s)
H	channel height (mm)	x, y	Cartesian coordinates
P	Pressure (Pa)	Greek symbols	
Pr	Prandtl number	α	Thermal diffusivity
q''	heat flux (W/m^2)	μ	Viscosity (Pa. s)
r	cylinder tip radius (mm)	ρ	Density (kg/m^3)
Re	Reynolds number		

References

[1] A. Gode, A. K. Sahu, and R. Chhabra, "Two-dimensional steady flow over a semi-circular cylinder: drag coefficient and Nusselt number," Int. J. Adv. Eng. Sci. Appl. Math., vol. 3, pp. 44–59, 2011.

[2] A. Prhashanna, A. K. Sahu, and R. Chhabra, "Flow of power-law fluids past an equilateral triangular cylinder: Momentum and heat transfer characteristics," Int. J. Therm. Sci., vol. 50, no. 10, pp. 2027–2041, 2011.

[3] M. Moussaoui, Y. Admi, E. Lahmer, and A. Mezrhab, "Numerical investigation of convective heat transfer in fluid flow past a tandem of triangular and square cylinders in channel," in IOP Conf. Ser.: Mater. Sci. Eng., 2021.

[4] P. Sooraj, A. Agrawal, and A. Sharma, "Measurement of drag coefficient for an elliptical cylinder," J. Energy Environ. Sustain., vol. 5, pp. 1–7, 2018.

[5] M. M. Alam, T. Abdelhamid, and A. Sohankar, "Effect of cylinder corner radius and attack angle on heat transfer and flow topology," Int. J. Mech. Sci., vol. 175, p. 105566, 2020.

[6] J. Rosales, A. Ortega, and J. Humphrey, "A numerical simulation of the convective heat transfer in confined channel flow past square cylinders: comparison of inline and offset tandem pairs," Int. J. Heat Mass Transf., vol. 44, no. 3, pp. 587–603, 2001.

[7] R. Agarwal and A. Dhiman, "Flow and heat transfer phenomena across two confined tandem heated triangular bluff bodies," Numer. Heat Transf. Part A Appl., vol. 66, no. 9, pp. 1020–1047, 2014.

[8] H.-W. W. Jue, S.-Y. Huang, and T.-C. Tswen-Chyuan, "Heat transfer predictions around three heated cylinders between two parallel plates," Numer. Heat Transf. Part A Appl., vol. 40, no. 7, pp. 715–733, 2001.

[9] T. A. Tahseen, M. A. Eleiwi, and A. F. Hameed, "Numerical study of fluid flow and heat transfer in a backward facing step with three adiabatic circular cylinders," J. Adv. Res. Fluid Mech. Therm. Sci., vol. 72, no. 1, pp. 80–93, 2020.

[10] Y. Admi, M. A. Moussaoui, and A. Mezrhab, "Numerical investigation of convective heat transfer and fluid flow past a three-square cylinders controlled by a partition in channel," Int. J. Renew. Energy Dev., vol. 11, no. 3, pp. 766–781, 2022.

[11] H. F. Oztop, Y. Varol, and D. E. Alnak, "Control of heat transfer and fluid flow using a triangular bar in heated blocks located in a channel," Int. Commun. Heat Mass Transf., vol. 36, no. 8, pp. 878–885, 2009.

[12] Y. A. Çengel, Heat Transfer: A Practical Approach. New York, USA: McGraw-Hill, 2002.

[13] G. Munoz, "Lecture 7: Mesh Quality & Advanced Topics," ANSYS Inc., Canonsburg, PA, USA, 2015.

[14] A. Kumar and A. K. Dhiman, "Effect of a circular cylinder on separated forced convection at a backward-facing step," Int. J. Therm. Sci., vol. 52, pp. 176–185, 2012.

[15] R. Prastianto, K. Dwipayana, N. Syahroni, and B. Pumbarino, "Experimental study on interference effects of two tandem cylinders wrapped around by triple helical rods with gap on induced drag," in IOP Conf. Ser.: Earth Environ. Sci., 2018.