Journal of Advanced Sciences and Engineering Technologies (2024) 7 (1): 1-19
https:// https://doi.org/10.32441/jaset.07.01.01

JASET JOURNAL

Journal of Advanced Sciences and Engineering Technologies
https://isnra.net/ojs/index.php/jaset/index/

Air Flow Across Three Triangle Cylinders with Rounded Corners Edge in a Channel under Forced Convection

Elaf Ayad Hasan[1], Tahseen Ahmad Tahseen[2,*], Sami Ridha Aslan[1], Yousif Hashim Hussein[2]

1 Mechanical Power Department, Technical College/Kirkuk, Northern Technical University, Kirkuk, Iraq

2 Mechanical Engineering Department, College of Engineering, University of Kirkuk, Kirkuk, Iraq

Keywords:

Forced convection, triangle cylinder, rounded corner edge, CFD simulation, Enhancement of heat transfer

A R T I C L E I N F O

Article history:
Received : 01 December, 2023
Accepted : 30 December 2023
Available online 03 January 2024

Citation :
Hasan, E. A., Tahseen, T. A., Aslan, S. R., & Hussein, Y. H. (2024). Air Flow Across Three Triangle Cylinders with Rounded Corners Edge in a Channel under Forced Convection. Journal of Advanced Sciences and Engineering Technologies, 7(1), 1–19. https://doi.org/10.32441/jaset.07.01.01

A B S T R A C T

The analysis of heat transfer by forced convection steady, laminar flow around three triangle cylinders. A numerical investigation is conducted to analyze the variation in cross-section from a triangle to a circle by altering the edges of a triangular cylinder within a two-dimensional channel in the air. The effect of Reynolds number and transverse pitch (S_T) on the heat transfer characteristics are studied. Five Reynolds numbers (based on inlet height channel) are studied (Re = 100, 200, 400, 800, 1200) with a constant heat flux on cylinder surface (q''= 5000 W/m^2), and S_T = 1.5D, 2D, 3D, the D is triangle side length. The governing partial differential equations, which include the continuity, momentum, and energy equations, are resolved by computational methods using workbench 2022R2 program. The results reveal that heat transfer rises as the transverse distance and the diameter of the rounded edge change.

* Corresponding author: (Tahseen Ahmad Tahseen; Mechanical Engineering Department, College of Engineering, University of Kirkuk, Kirkuk, Iraq) tahseen444@gamil.com; tahseen@uokirkuk.edu.iq

1. Introduction

The development of life in its various aspects has led to an increase in the need to keep pace with this development, especially in terms of engineering and industry, as there has become an urgent need to develop systems and devices that contribute to providing the necessities of daily life. These include systems and devices that use and convert energy in the process of heat exchange [1]. Many thermal and chemical engineering processes employ cross-flow tube banks heat exchangers [2]. The process of heating and cooling the fluids passing the surfaces, including pipes (circular, oval, flat, triangular, etc.) is one of the important topics in heat transfer issues, so it has been observed that numerous research has been conducted to investigate the phenomenon of flow and heat transfer in various forms, with the aim of optimizing the performance of heat exchangers. [3-5]. Flow around or over steep bodies has frequently been the topic of extensive fluid mechanics research since it is relevant to numerous applications such as heat exchanger systems, electronic refrigeration, suspended bridges, and so on. Due to a range of problems related to fluid mechanics, it embodies a great deal of research interest. In recent years, this topic has received great interest in terms of experimental and numerical studies because of the ever-increasing development in computational techniques, especially numerical ones, and the emergence of new experimental techniques [6].

1.1. Real cylinder shape

Triangular baffles are one of the basic engineering shapes that can be used as a steep body due to their various engineering applications, flow metering devices, flame holders, and heat exchange systems such as tubular heat exchangers and compact heat exchangers are examples of such devices [7]. Many earlier studies investigated the laminar flow around different types of obstacles (square, triangle, circle) numerically but there are few studies that investigated this type of flow around rounded edged obstacles, especially rounded-edged triangular cylinders. An experiment was carried out by Eleiwi et al. [8] to study the influence of the intake air attack angle on the crossflow heat transfer coefficient over four flat tubes. Angles of 30, 45, and 90 degrees were employed for inlet airflow. The results showed that when the air attack angle increased, the Nu number decreased. Vyas et al. [9] conducted an experimental study on the unsteady laminar flow around a square cylinder positioned within a rectangle channel. The heat transfer rate of the channel was evaluated at five distinct Reynolds numbers, considering the presence and absence of the cylinder. The existence of a cylinder considerably enhanced the heat transfer rates of the wall, as compared to the situation where the channel had no additional components. Ganie et al. [10] conducted a numerical analysis of the flow of a non-isothermal fluid (air) via a perforated plate positioned within a rectangular channel. A heating element was positioned at the bottom edge of the channel, and the plate's angle of inclination was adjusted in a clockwise direction. The observation revealed a direct linear relationship between the Nusselt number and the angles. Kumar De and Dalal [6] performed a numerical analysis of thermal conduction in a flat channel with a triangular obstacle, examining the flow characteristics in two dimensions. The study focused on Reynolds numbers ranging from 80 to 200 and various blockage ratios. The findings obtained demonstrate a considerable increase in both the root mean square of the lift coefficient and the Strouhal number as the

Reynolds number and blockage ratio increase. Mustafa et al. [11] quantitatively analyzed the heat transfer through forced convection in a laminar flow inside a channel containing two heat sources on the upper and the lower walls with two semi-circular cylinders in front of each one on the opposite wall. The first heat source and its semi-cylinder were kept fixed while changing the location of the second heat source with its semi-cylinder. The study found that the average Nusselt number for the initial heat source rises as the distance between the semi-cylinders grows, regardless of the Reynolds numbers. Tahseen et al. [12] predicted the decrease in pressure and heat transfer in a straight tube arrangement subjected to a cross-flow. The Reynolds number ranges from 10 to 320. The findings of heat transfer and pressure drop were presented for a tube shape that has both transverse and longitudinal pitch. The calculated mean Nusselt number and dimensionless pressure align well with the findings of other studies. The ANFIS model yields an average relative error of 2.97%. Based on this model, the pressure experiences a 2.97% reduction or less than 1.9% for the average Nusselt number. Moussaoui et al. [13] numerically studied the flow characteristics and heat transfer properties around two heated cylinders triangular on the upstream and square on the downstream, placed in an (in-line) arrangement inside a channel. By maintaining a constant temperature for both cylinders, the experiment involved altering the horizontal distance between them. The findings indicated that the magnitude of the horizontal distance had an impact on the heat transfer. Aydin et al. [14] the heat transfer and flow properties of two circular cylinders arranged tandemly inside the channel were empirically investigated. Different blockage ratios and gaps between the cylinders were chosen. The results showed that increasing the size of the gap and the blockage ratio enhances heat transmission for the upstream cylinder. Huang et al. [15] numerically simulated the unsteady flow of water around an array of cylinders of unequal sizes. Reynolds number was chosen less than (150) depending on the cylinders' diameter of the first row and the velocity of the free stream. It was noted that through two rows of in-line cylinders, there is an incomplete vortex shedding at a separation distance of less than (2D) behind the first row of cylinders. Eleiwi et al. [16] utilized the heat transfer coefficient for four-flat tubes in cross-flow air using an artificial neural network (ANN). The experimental equipment has slope air entering flow orientation, with varying speeds of incoming air 0.2, 0.5, 0.6, 0.8, and 1.2 m/s, the coefficient of heat transfer was examined separately for three air intake angles (90^o, 45^o, and 30^o). The three heat flux values input are 13.2, 38.5, and 99.8 W/m^2. The experimental results and the predicted coefficient of heat transfer agree very well.

1.2. Non- circular Cylinder with rounded corner

Abdelhamid et al. [17] conducted a numerical analysis to validate the flow characteristics around a square cylinder, the heat transfer from it, and the aerodynamic forces in a laminar incompressible flow. The square cylinder's corner radius ratio has been modified from $(r/R = 0 - 1)$, where r represents half the angle of the cylinder and R denotes half the width of the side of the cylinder. The results concluded that the Strouhal number and the Nusselt number change equally with each of (r/R) and the Reynolds number as they increase with the increase of (r/R) and the Reynolds number.

The forced convection of air flow over two tandem cylinders with rounded corners in a channel at *Re* = 100 is investigated numerically by Zhang et al. [18]. With a radius of curvature of R, which may be non-dimensionalized as $R^* = R/D$ where D is the cylinder diameter, both cylinders have the geometry of a square that is rounded at both corners. As a result, the cylinder geometry can be square ($R^* = 0$), partially rounded (R^* = 0.1–0.4), or circular (R^* = 0.5). The results are displayed using both fluctuating and time-averaged amounts to show both mean and pulsing behaviors. The result shows that the near-wake flow unsteadiness is determined by the cylinder geometry following the downstream cylinder; for square-like cylinders with a small corner radius, the flow is always unsteady, but for circular-like cylinders with a larger corner radius, the flow can be stabilized to the extent that the flow fluctuation is significantly reduced or even completely suppressed at small parameter of gap ratio. The flow over a cylinder with rounded corners at a high Reynolds number with a range 10^4–10^6 was simulated by Tamura et al. [19]. For the rounded geometry, the Strouhal number is bigger and the base pressure coefficient at the cylinder rear is smaller than for the square cylinder. By studying the turbulent flow across the cylinder experimentally at Re = 30,000, Tamura and Miyagi [20] noticed that the rounded corners assist the separated shear layer on the lateral surface of the cylinder to attach, which reduces drag and increases lift at all inclinations between 5° and 30°. The influence of corner radius on the near-wake flow of an isolated cylinder at Re = 2600 and 6000 was investigated experimentally by Hu et al. [21]. The wake vortices of the rounded corner cylinder are much weaker, with the first roll-up happening significantly more distant from the cylinder; the Strouhal number increases, as seen by the decreased streamwise distances between consecutive vortices. The force fluctuations and average drag are greatly reduced by the rounded corners. The statistical quantities show sharp differences between the square cylinder and partially rounded cylinder configurations, as well as between the latter and the circular cylinder, rather than a monotonic variation with the corner radius. To investigate the physical process behind the stabilization of wake flow by rounding the corners, we recently conducted sensitivity analysis and two-dimensional global linear stability analysis on low-Re flow across a rounded corner cylinder at $Re \leq 110$ [22].

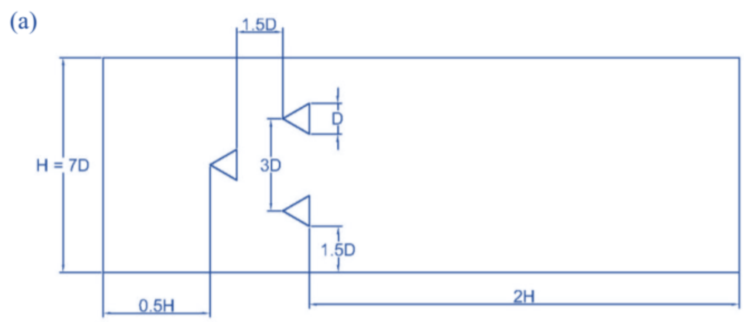
According to the context of the literature review that has been presented, it can be reported that although some efficient geometric configurations have been proposed by previous investigators to modify the fluid behavior to achieve the desired characteristics of reduced aerodynamic forces, a little researcher has discussed heat transfer characteristics, which are important for the best designs of cross flow heat exchangers. Thus, a numerical analysis is carried out to investigate the fluid flow and heat transfer properties of three triangular cylinders with rounded corners. Figure 1(b) depicts the corner configurations taken into consideration for this investigation, including corner rounding, that have not previously been included in studies. The effect of corners rounded, transverse pitch, and front air velocity on heat transfer and fluid flow characteristics were investigated. around three cylinders with shape varies from triangular to circular and using two different radiuses for the rounded-edged triangular cylinder. While Nusselt number and heat transfer coefficient have been studied to elucidate on heat transfer characteristics.

2. Problem Description and Mathematical Model

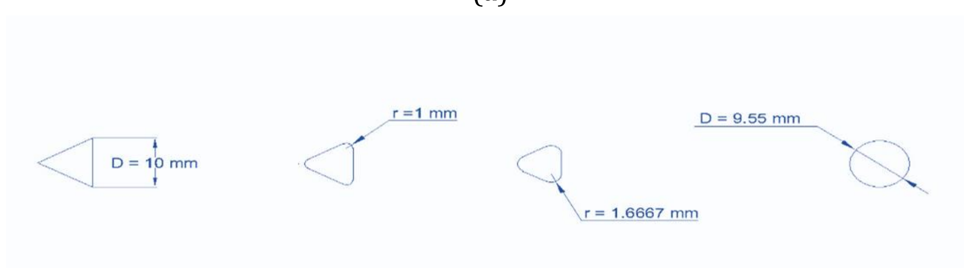
The geometry under consideration in this physical problem is shown in Figure 1. The fluid (air) flows around three heated cylinders with three different shapes (triangular, rounded edge triangular (with two different diameters of the rounded edge), and circular) set within a channel with adiabatic walls. The side length of the triangular cylinder (D = 10 mm) and the radius of the rounded edges (r = 1 mm, 1.6667 mm). Three different vertical distances between the downstream cylinders (S_T = 1.5D, 2D, 3D). The constant heat flux on the cylinder surface (q''= 5000 W/m^2) and five values of Reynolds number based on the inlet channel height used (Re = 100, 200, 400, 800, 1200).

2.1. Governing equations

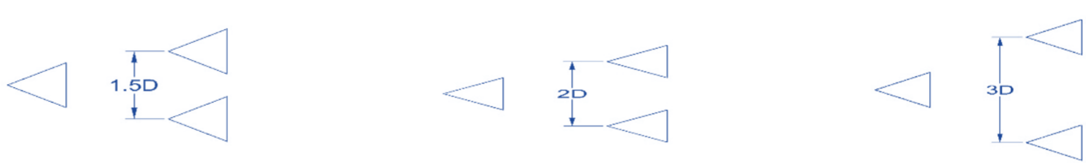
The equations governing the flow and heat transfer in a laminar, stable, two-dimensional, incompressible fluid with constant thermo-physical properties can be represented as follows [23]:



(a)



(b)



(c)**Figure 1.** Schematic diagram of the test rig (a) physical domain, (b) rounded corner, and (c) study cases.

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

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

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

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

The boundary conditions used for the present study are shown in Figure 2 and Table 1:

Table 1. The boundary conditions.

Selection	Boundary condition	Unit	Value
Inlet	Velocity at (T = 300 K)	m/s	0.0222, 0.0445, 0.0889, 0.178, 0.267
Outlet	Gauge pressure	Pa	0
Cylinder surface	Heat flux	W/m^2	5000
Channel wall	Heat flux	W/m^2	0

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

We define the Reynolds number as

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

where H is the channel height.
The average Nusselt number can be expressed as:

$$Nu = \frac{hD_h}{k} \quad (6)$$

The pressure drop can be expressed as follows [14]:

$$\Delta P = f\,\frac{L}{D}\,\frac{\rho V^2}{2} \quad (7)$$

where f is the friction factor ($f = 4C_f$) [14].

3. Numerical Model

3.1. Grid Generation

The computational grid structure employed for problem-solving is illustrated in Figure 3. The grid generated using the ANSYS FLUENT 2022R2 software is uniformly distributed throughout the whole domain, except for the region surrounding the cylinders where it becomes non-uniform. The number of elements that the computational domain is divided into is (29619) by using (0.7 mm) for element size and the mesh is finer near the cylinders by using 6 layers of inflation to achieve a significant gradient in the hydrodynamic and thermal boundary layer.

3.2. Grid Independent Test

A grid-independent test was conducted at Reynolds number (Re = 100) and (S_T = 3D) to examine the impact of the cells on the results. For this purpose, six various computational grids were utilized, with the element number varying for each grid G_1 = 21034, G_2 = 11815, G_3 = 14388, G_4 = 17037, G_5 = 22716, G_6 = 29619, respectively. The result of the test is shown below in Table 2 and the percentage difference in the average heat transfer coefficient did not surpass (1%).

3.3. Model validation

The model used in the present work is validated with the work obtained by Aydin et al. [14]. They conducted a numerical study that focused on heat transfer and flow characteristics in a channel using two tandem circular cylinders. The regime is assumed to be steady, 2D, laminar, and incompressible. The results of the average Nusselt number are obtained for each cylinder for blockage ratio (β = 0.6). Figure 4 shows the results of the validation which shows a good agreement with the maximum deviation (±4.5%, ±5.3%) for upstream and downstream cylinders, respectively.

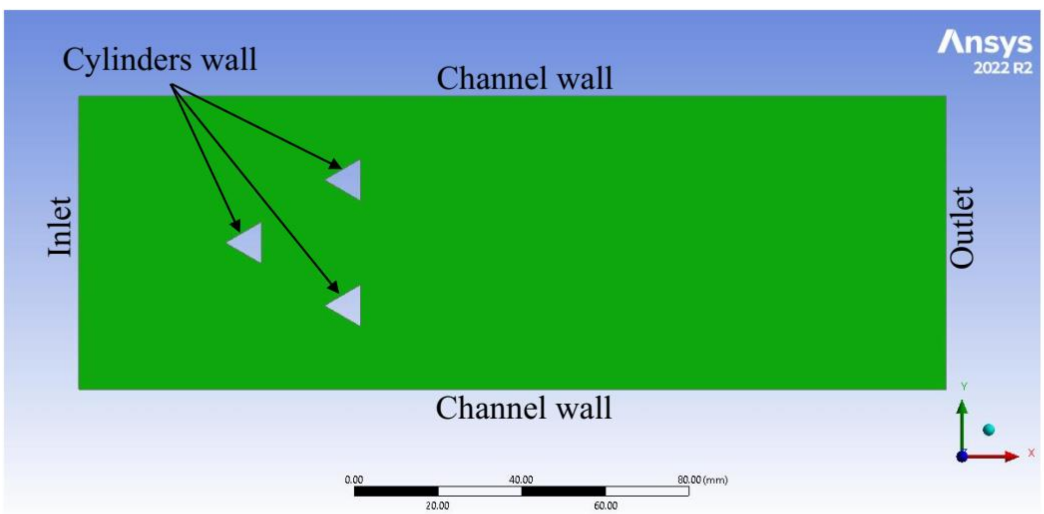


Figure 2. Schematic diagram of computational domain and boundary.

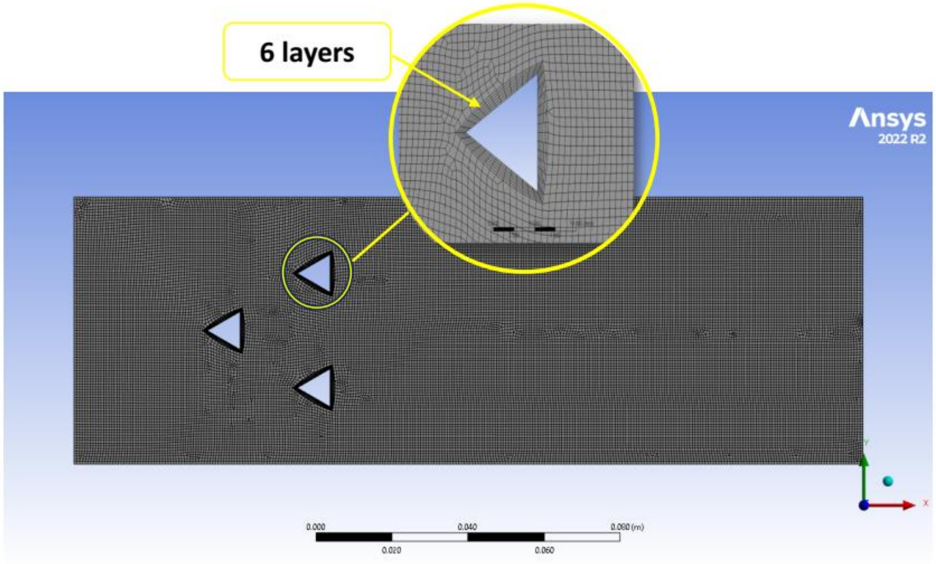


Figure 3. The computation grids

Table 2. The results of grid-independent test at Re = 100, and S_T = 3D

Case	**Element size (mm)**	**Number of elements**	h_{ave} $\mathrm{W}/(\mathrm{m}^2\mathrm{K})$	**Error (%)** $\frac{h_{i+1}-h_i}{h_{i+1}} \times 100$
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

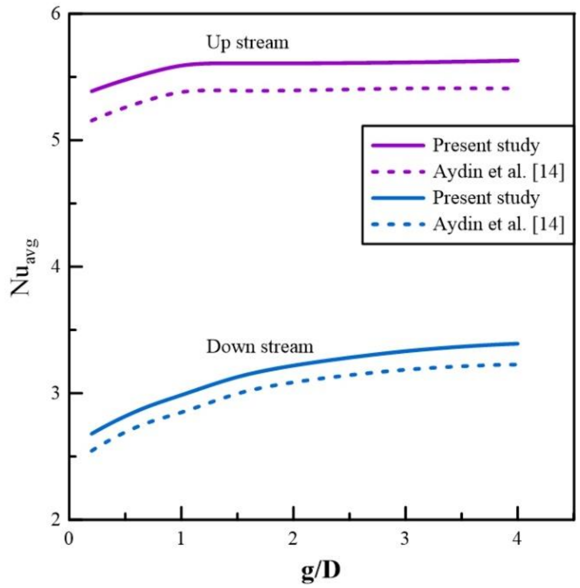


Figure 4. Comparison of Nu_{ave} with gap-to-diameter ratio between the present study and results of Aydin et al. [14]

4. Results and Discussion

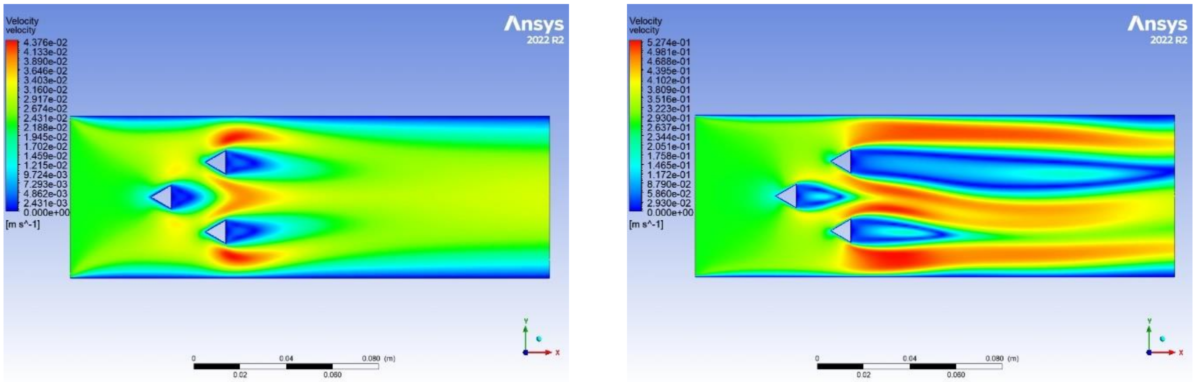
The purpose of this study is to examine the impact of a triangular cylinder with rounded edges on fluid flow and heat transfer. The investigation was carried out for Reynolds numbers (100, 200, 400, 800, 1200) and air (Pr = 0.71) as the working fluid with constant heat flux on cylinders wall (q'' = 5000 W/m^2), vertical cylinder spacing (transverse pitch) (S_T = 1.5D, 2D, 3D). Figures (5)-(8) Illustrate the patterns of distribution of velocity, pressure, and temperature, respectively. The presence of the cylinders led to a reduction in the flow separation's reattachment length, as seen below. The recirculation zone behind the cylinders contracts as the edges get more rounded. The thermal boundary layer decreases at a higher Reynolds number as a result of increasing the region of flow recirculation located behind the cylinders. The recirculation zone behind the downstream cylinders is affected by the transverse pitch between them which decreases with increasing the transverse pitch. Changing the transverse pitch has an impact on the pressure, the pressure decreases as the transverse pitch increases. The transverse pitch has an impact on temperature especially at lower Reynolds numbers. As the transverse pitch increased the thermal boundary layer thickness decreased. Figures 9 and 10 show the effect of the transverse pitch on the velocity, pressure, and temperature for rounded-edge triangular cylinders with r = 1 mm and r = 1.6667 mm, respectively. The thickness of the temperature gradient layer decreases throughout with an increase in velocity increases and becomes thinner at the cylinder surface.

Figures 11-14 show the velocity profile for the transverse pitch ($S_T = 3D$) for various Reynolds numbers for the locations (0.05116 mm, 0.07732 mm) which are between the cylinders and behind them. The y-axis zero value corresponds to the bottom wall at x = 0 mm. The presence of the cylinders has an impact on the flow zone both above and below them. A slight shift in the peak of the velocity magnitude that points to the positive y direction appears as the Reynolds number increases and the peak gets sharper as a result of free hydrodynamic flow near the walls and air between the cylinders in addition to the discordant direction of the fluid flow. In the recirculating zone, the velocity remains low after each cylinder meanwhile in the flow zone it is converted to jets after the cylinders.

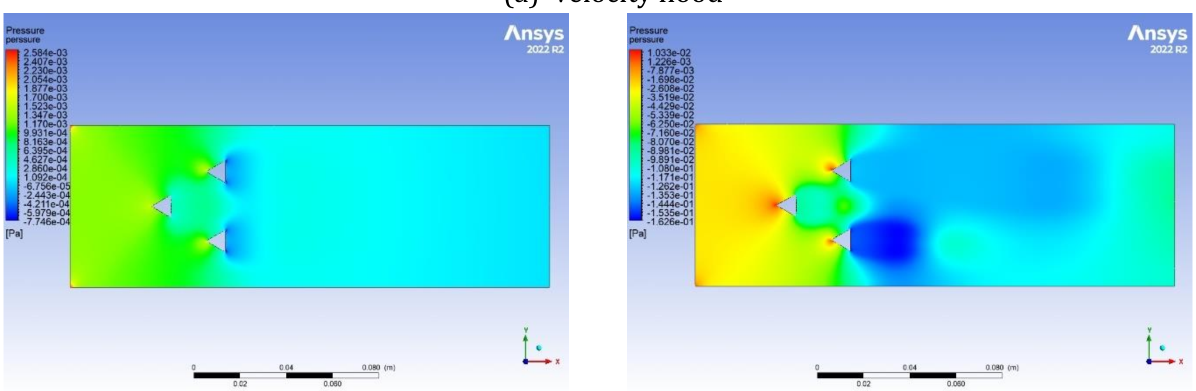
For all cylinder shapes it is observed that the Nusselt number increases with increasing in Reynolds number. The higher rate of heat transfer among in the rounded edge shape with (r = 1.6667 mm) edge diameter and the circular cylinder has the lowest heat transfer rate for the same Re and transverse pitch (S_T). Figures 15 and 16 show the relationship between Nusselt numbers, Nu and Re for different shapes of cylinders and each transverse pitch.

For the transverse pitch, it is noticed that the Nu increases with increasing in transverse pitch. Among the shapes of the cylinders (S_T = 3D) has the highest Nu number and (S_T = 1.5D) has the lowest for the same Re number.

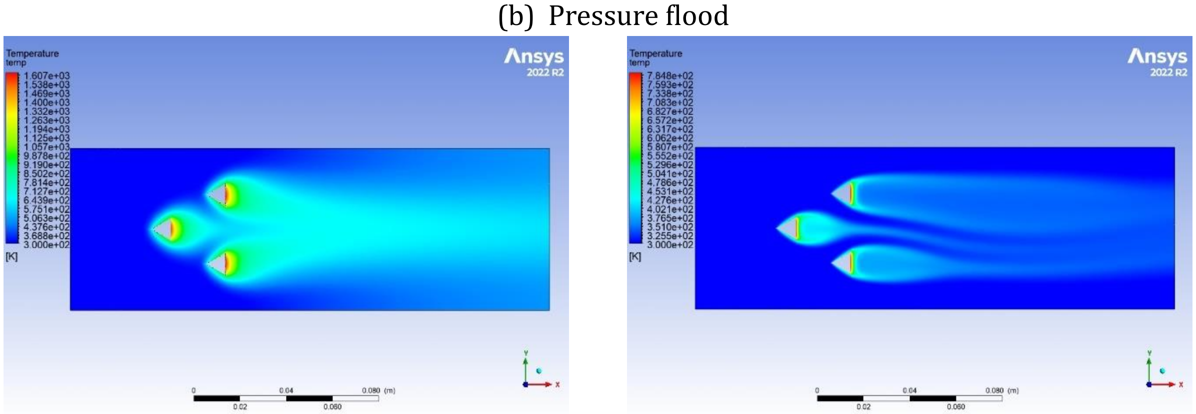
Re = 100 Re = 1200



(a) Velocity flood



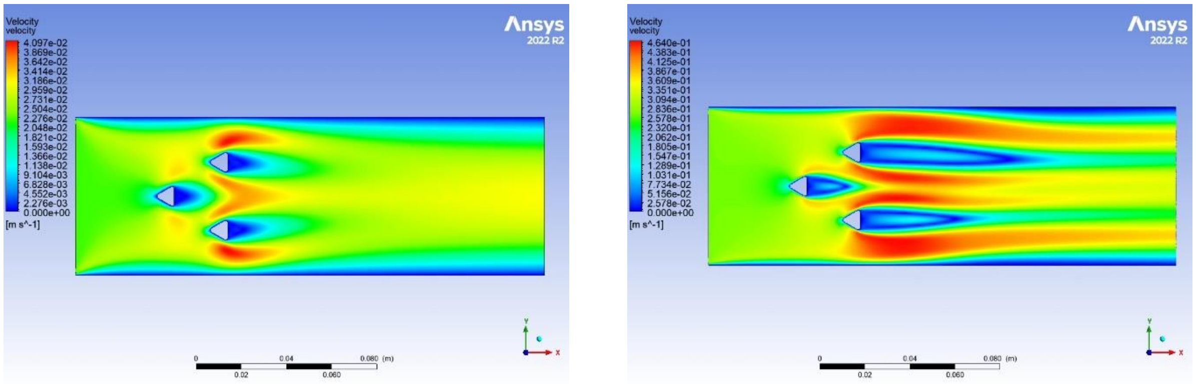
(b) Pressure flood



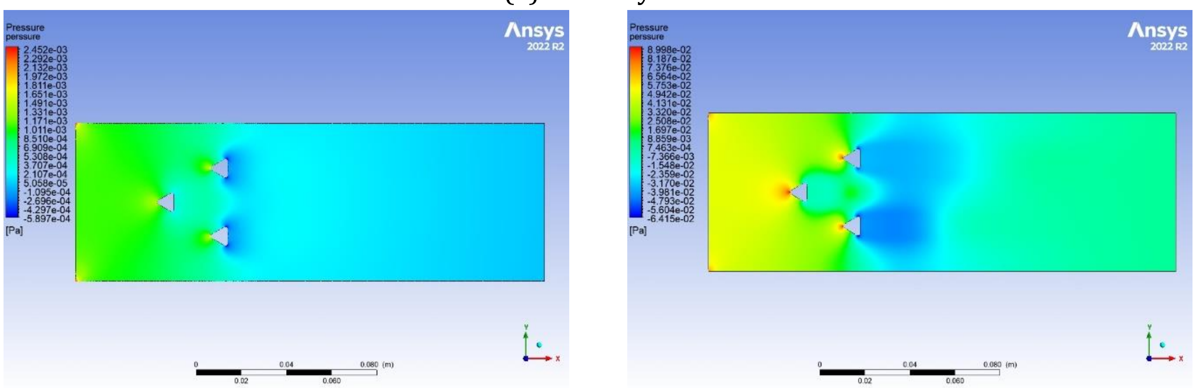
(c) Temperature flood

Figure 5. Velocity, pressure, and temperature contours for triangular cylinder for S_T = 3D.

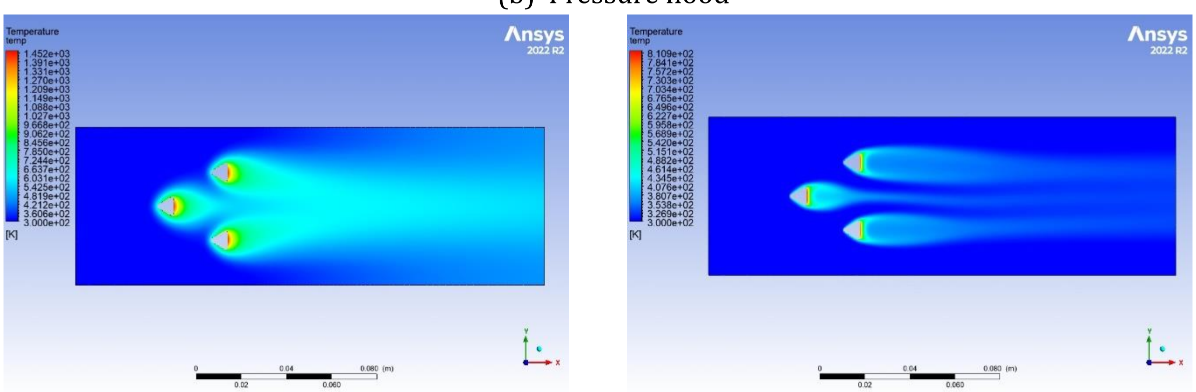
Re = 100 Re = 1200



(a) Velocity flood



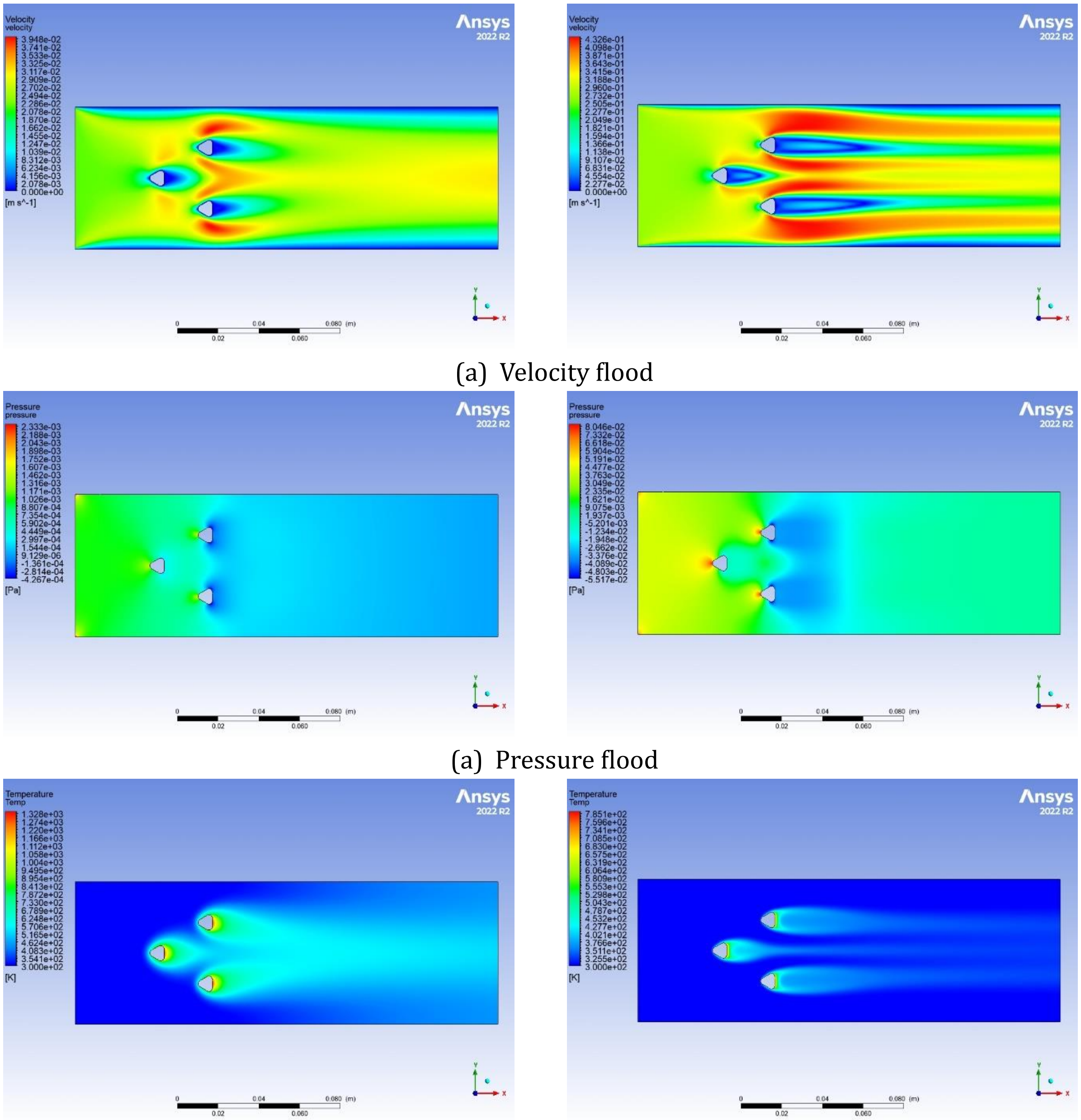
(b) Pressure flood



(c) Temperature flood

Figure 6. Velocity, pressure, and temperature contours for rounded edge triangular cylinder (r = 1 mm) for S_T = 3D

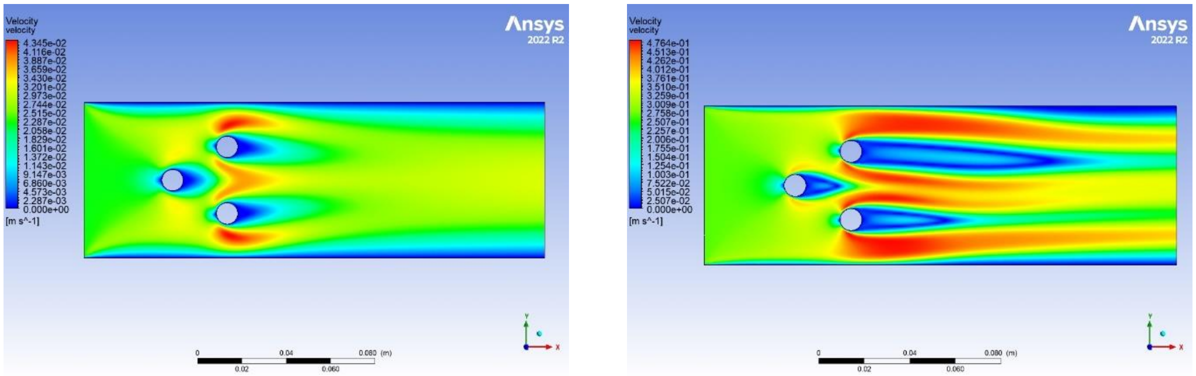
Re = 100 Re = 1200



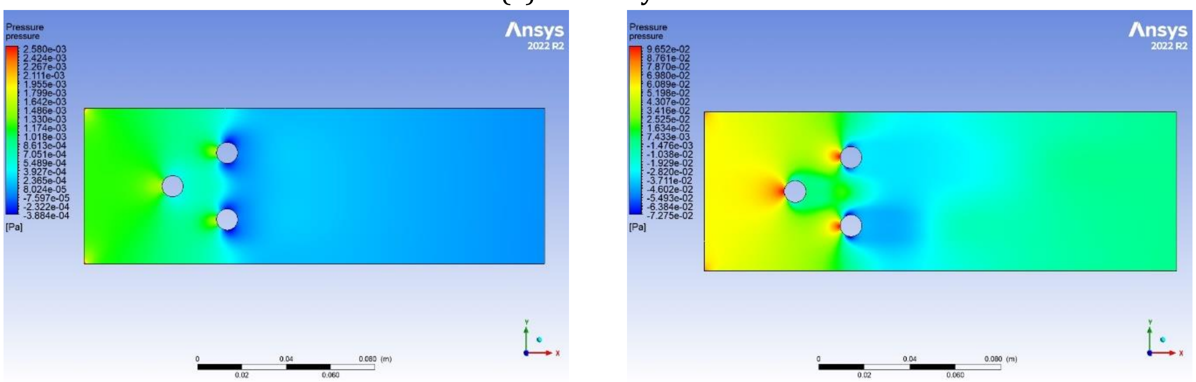
(c) Temperature flood

Figure 7. Velocity, pressure, and temperature contours for rounded edge triangular cylinder (r = 1.6667 mm) for S_T = 3D

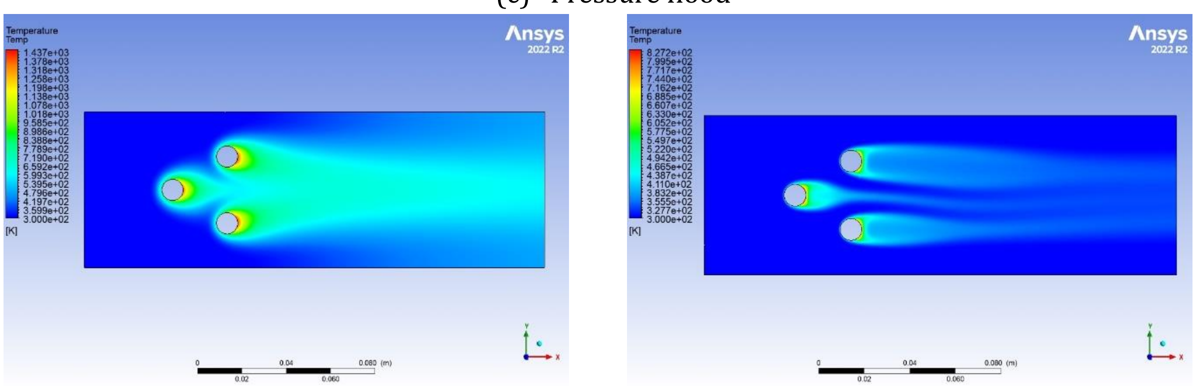
Re = 100 Re = 1200



(a) Velocity flood



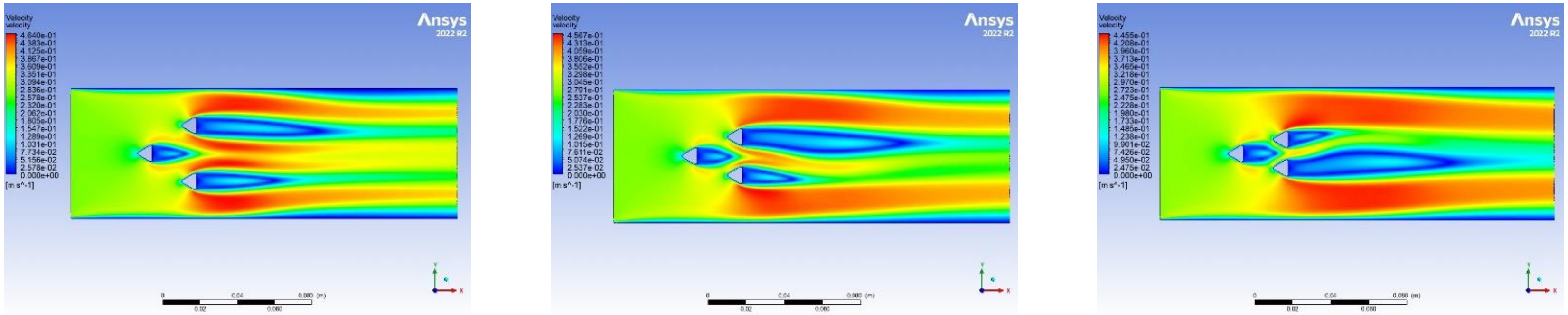
(c) Pressure flood



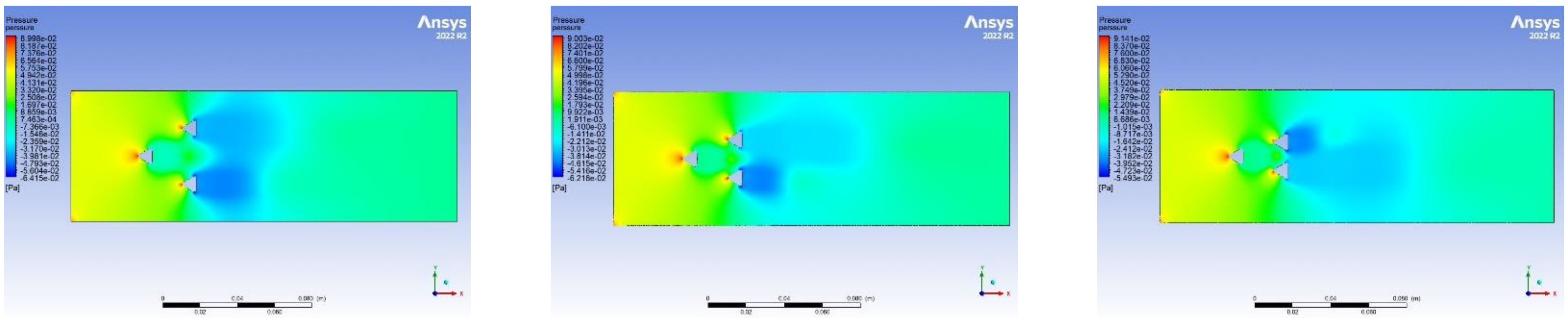
(c) Temperature flood

Figure 8. Velocity, pressure, and temperature contours for circular cylinder for S_T = 3D.

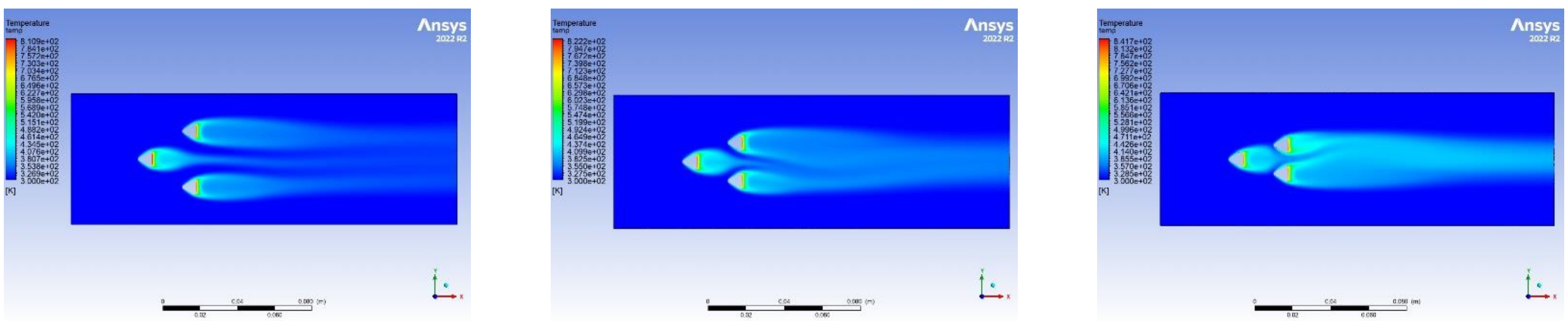
S_T = 3D S_T = 2D S_T = 1.5D



(a) Velocity flood



(b) Pressure flood



(c) Temperature flood

Figure 9. Velocity, pressure, and temperature contours for rounded edge triangular cylinder (r = 1 mm) for different S_T at Re = 1200

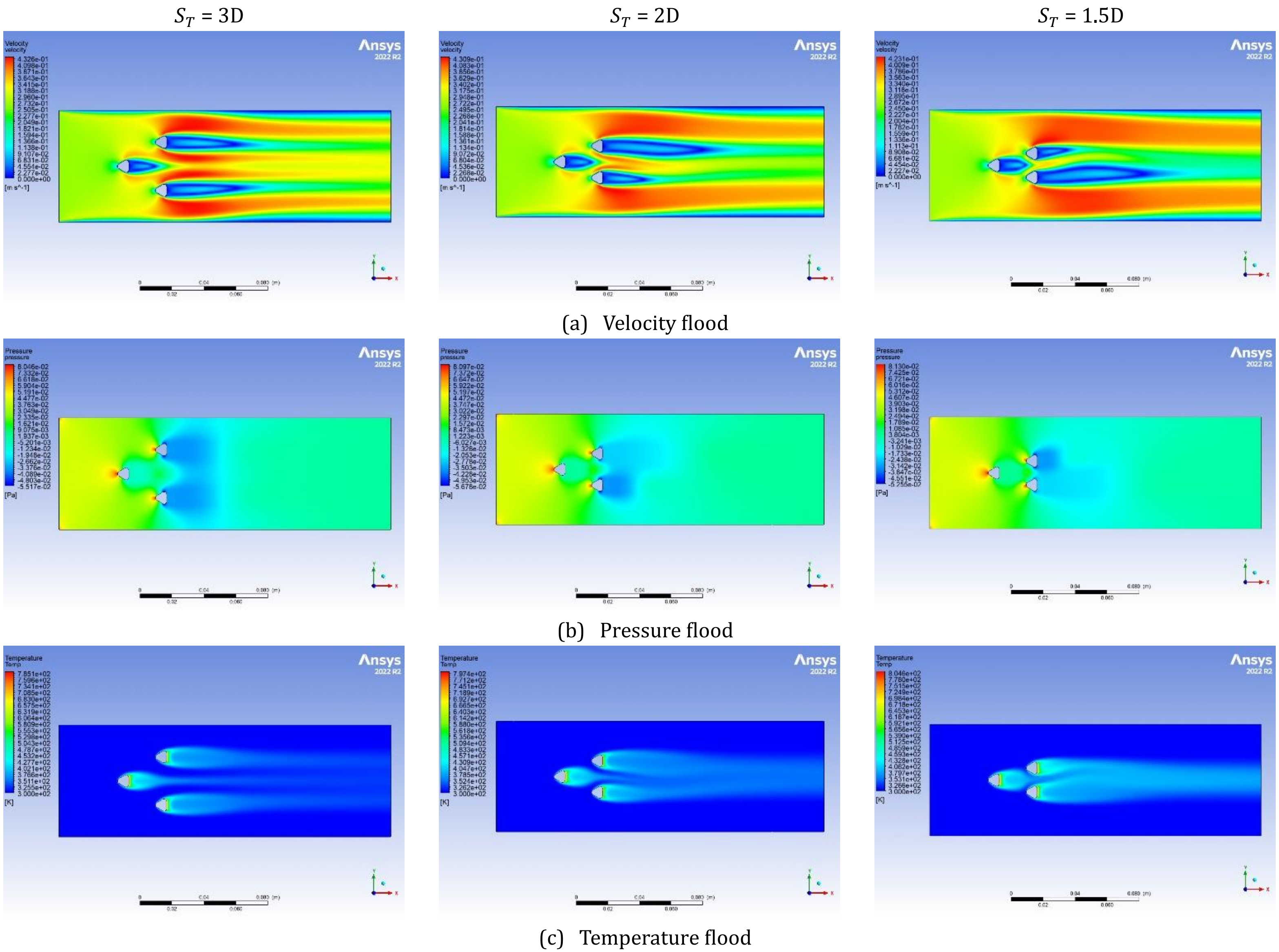
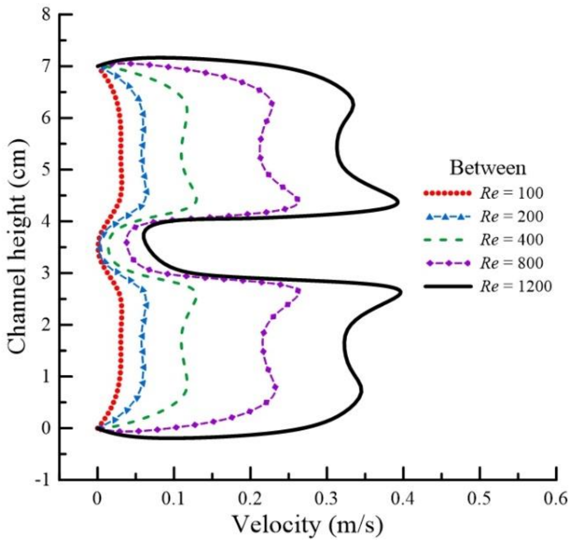
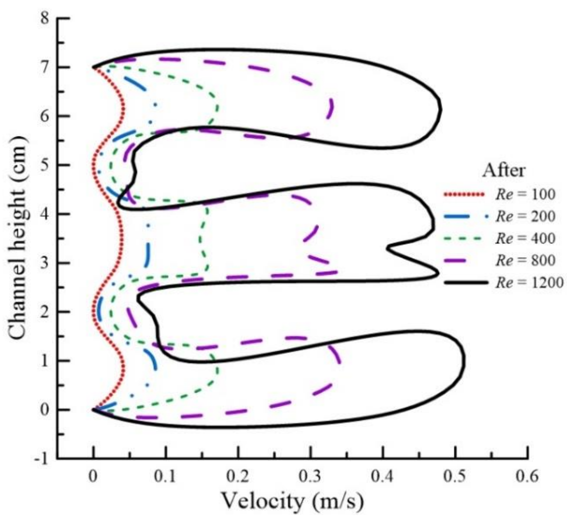


Figure 10. Velocity, pressure, and temperature contours for rounded edge triangular cylinder (r = 1.6667 mm) for different S_T at Re = 1200

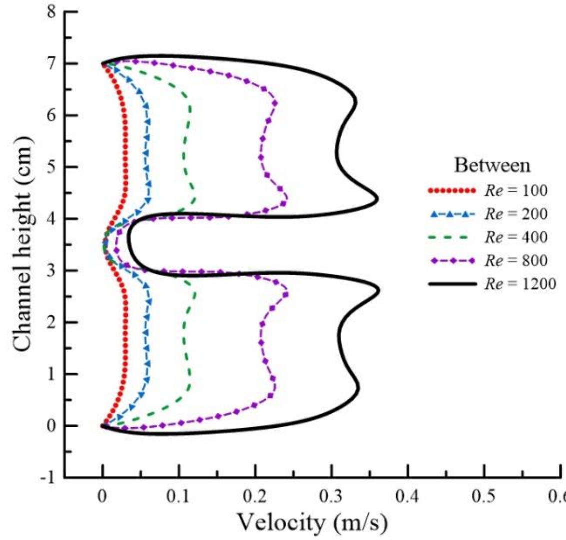


(a) Between

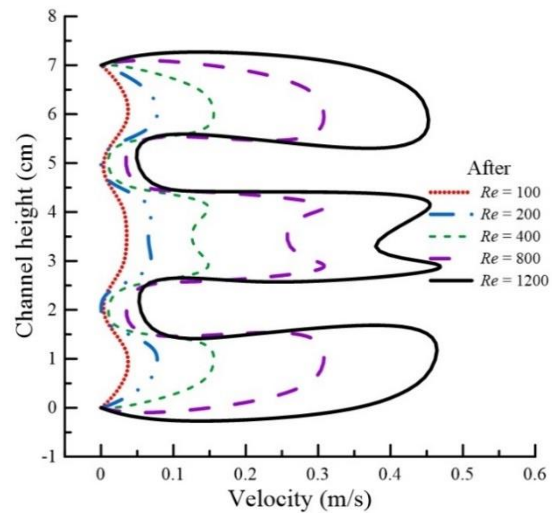


(b) After

Figure 11. Velocity is the file for the triangular cylinder at S_T = 3D.

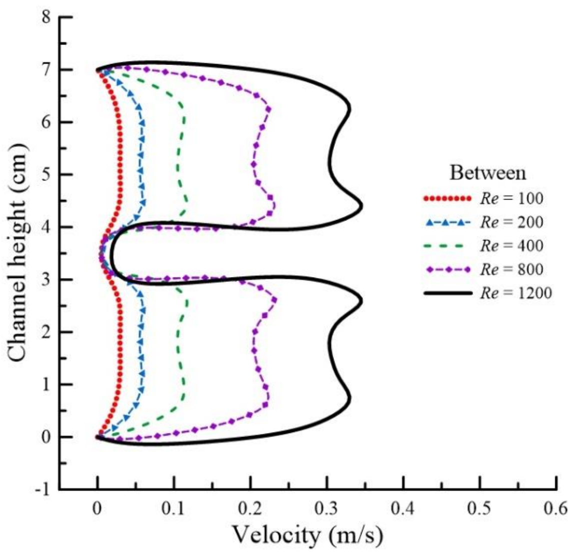


(a) Between

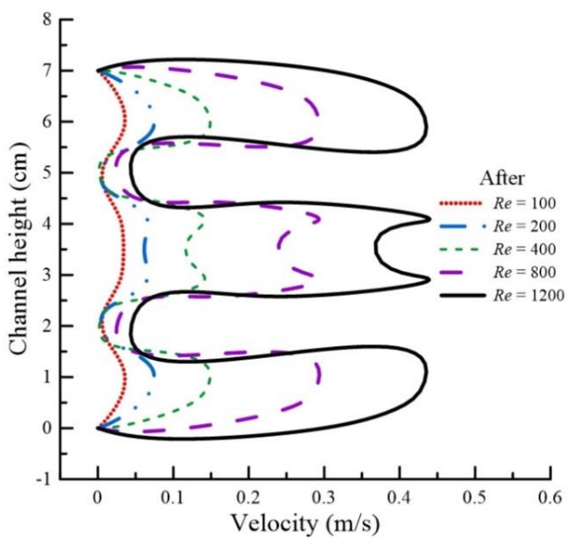


(b) After

Figure 12. Velocity profile for rounded edge triangular cylinder (r = 1 mm) at S_T = 3D.

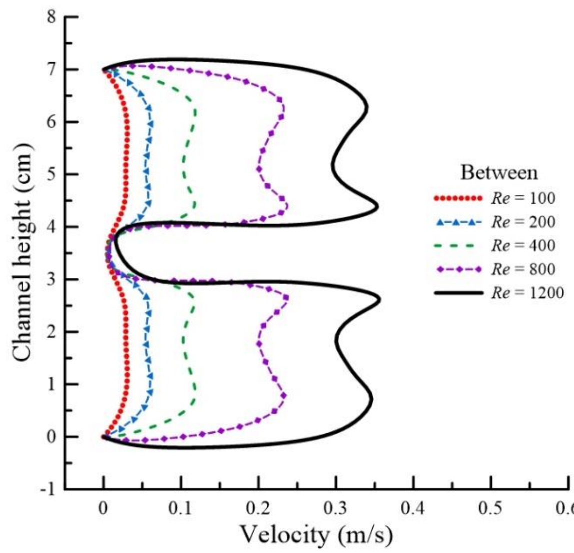


(a) Between

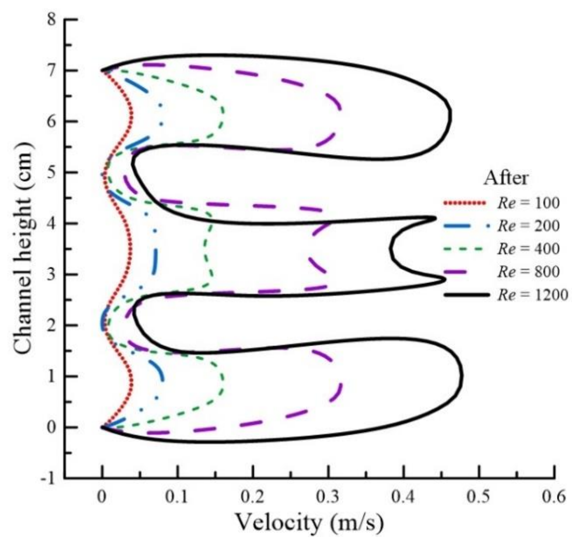


(a) After

Figure 13. Velocity profile for rounded edge triangular cylinder (r = 1.6667 mm) at S_T = 3D.

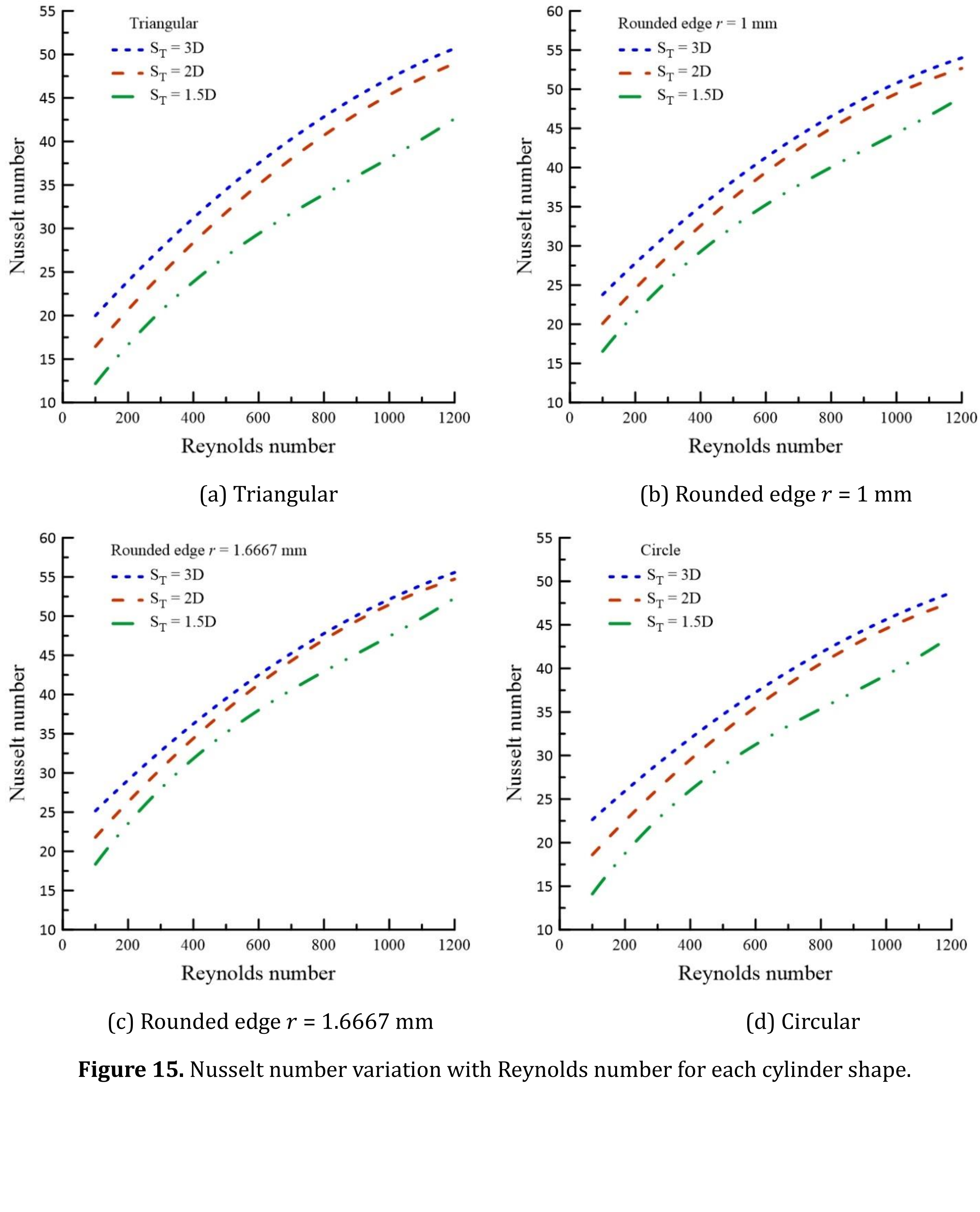


(a) Between



(a) After

Figure 14. Velocity profile for the circular cylinder at S_T = 3D.



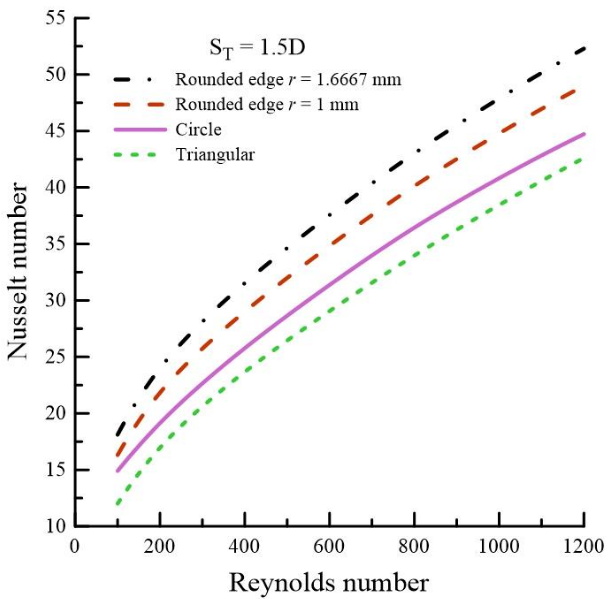
(a) Triangular

(b) Rounded edge r = 1 mm

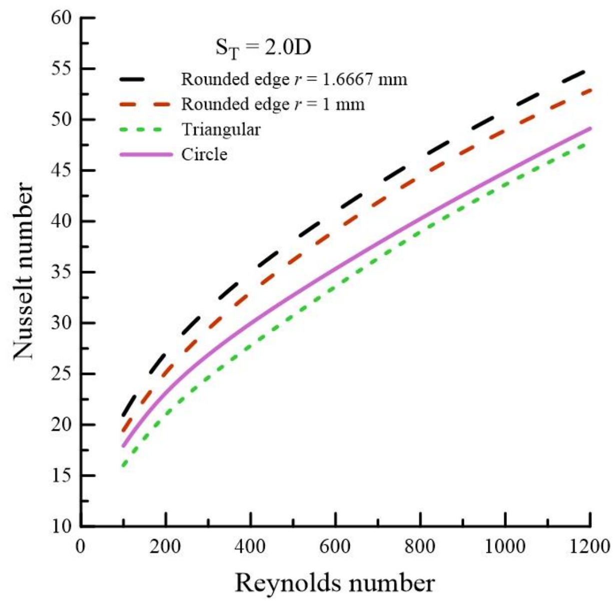
(c) Rounded edge r = 1.6667 mm

(d) Circular

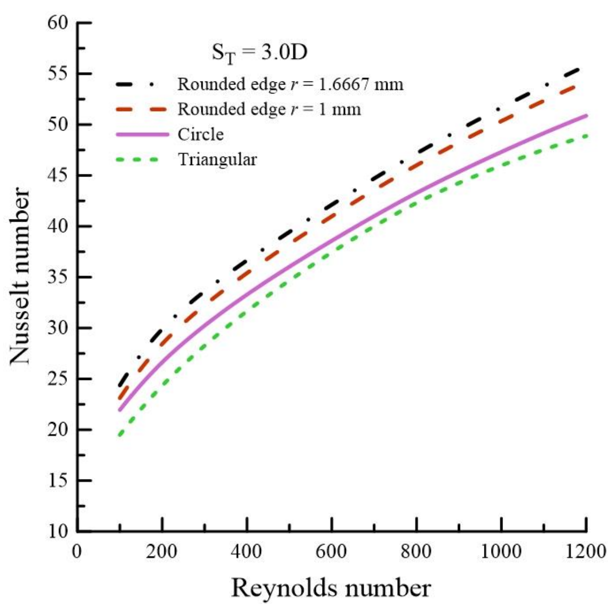
Figure 15. Nusselt number variation with Reynolds number for each cylinder shape.



(a)



(b)



(c)

Figure 16. Variation of the Nusselt number with the Reynolds number for transverse pitch (a) $S_T = 1.5D$, (b) $S_T = 2.0D$, and (c) $S_T = 3.0D$.

5. Conclusion

A numerical investigation of heat transfer enhancement and steady, laminar airflow characteristics for three heated cylinders with different shapes is performed within this manuscript. The influence of Reynolds number (Re = 100, 200, 400, 800, 1200), transverse pitch (S_T = 1.5D, 2D, 3D), and cylinder shape (triangle, rounded edge triangular r = 1 mm and 1.6667 mm, circle) is carried out. From the results, it can be the conclusion as follows:

1. The addition of the cylinders decreased the length at where the flow separation was reattached.
2. The cylinders have a strong effect on the velocity and the flow took a jet profile after the cylinders.
3. The thermal boundary layer is intense near the cylinders at a low *Re* number and weakens as the *Re* number increases.
4. The rounded edge triangular cylinder with edge radius (r = 1.6667 mm) has the maximum Nusselt number among the cylinders.
5. The highest Nusselt number for the transverse pitch was at (S_T = 3D).

References

[1]. Bejan, A. and Kraus, A. D. (2003). Heat Transfer Handbook (Vol. Ch.11). New Jersey, USA: John Wiley & Sons, Inc.

[2]. Tahseen, T. A., Ishak, M. and Rahman, M.M. Laminar Forced Convection heat transfer over Staggered Circular Tube Banks: A CFD Approach. Journal of Mechanical Engineering Science 2013, no. 4, 418-430. http://dx.doi.org/10.15282/jmes.4.2013.6.0039

[3]. Webb, R. L. and Kim, N.-H. (2005). Principle of Enhanced Heat Transfer (2nd ed.). New York: Taylor & Francis.

[4]. Tahseen, T. A., Ishak, M. and Rahman, M. An Overview on Thermal and Fluid Flow Characteristics in a Plain Plate Finned and Un-Finned Tube Banks Heat Exchanger. Renewable and Sustainable Energy Reviews *2015* no. 43, 363-380. https://doi.org/10.1016/j.rser.2014.10.070

[5]. Xu, Z., Han, Z., Wang, J., and Liu, Z. The Characteristics of Heat Transfer and Flow Resistance in a Rectangular Channel with Vortex Generators. International Journal of Heat and Mass Transfer 2018, no. 116, 61-72. https://doi.org/10.1016/j.ijheatmasstransfer.2017.08.083

[6]. Kumar, D. A., and Dalal, A. Numerical Study of Laminar Forced Convection Fluid Flow and Heat Transfer from a Triangular Cylinder Placed in a Channel. Journal of Heat Transfer 2007, vol. 129, no. 5, 646-656. https://doi.org/10.1115/1.2712848

[7]. Agarwal, R. and Dhiman, A. Flow and Heat Transfer Phenomena Across Two Confined Tandem Heated Triangular Bluff Bodies. Numerical Heat Transfer, Part A: Applications 2014, vol. 66, no. 9, 1020-1047. https://doi.org/10.1080/10407782.2014.894395

[8]. Eleiwi, M.A., Zainal, O.A., Tahseen, T.A. and Mustafa, A.W. Effect of Front Air Attack Angles on Heat Transfer Coefficient of the Crossflow of Four Flat Tube. Heat Transfer 2020, vol. 50, 638–654. https://doi.org/10.1002/htj.21897

[9]. Vyas, A., Mishra, B. and Srivastava, A. Experiments on Flow and Heat Transfer Characteristics of a Rectangular Channel with a Built-in Adiabatic Square Cylinder. International Journal of Heat and Mass Transfer 2020, vol. 147, 118908. https://doi.org/10.1016/j.ijheatmasstransfer.2019.118908

[10]. Ganie, A. H., Memon, A. A., Memon, M. A., Al-Bugami, A., Bhatti, K. and Khan, I. Numerical Analysis of Laminar Flow and Heat Transfer Through a Rectangular Channel Containing Perforated Plate at Different Angles. Energy Report 2022, vol. 8, 539-550. https://doi.org/10.1016/j.egyr.2021.11.232

[11]. Mustafa, A. W., Saied, A. A. M. and Sadiq, H. A. Effect of the Presence of Semi-Circular Cylinders on Heat Transfer from Heat Sources Placed in Two-Dimensional Channel. Tikrit Journal of Engineering Sciences 2012, vol. 19, no. 4, 24-47. https://doi.org/10.25130/tjes.19.4.09

[12]. Tahseen, T. A., Ishak, M. and Rahman, M.M. Performance Predictions of

Laminar Heat Transfer and Pressure Drop in an In-Line Flat Tube Bundle Using an Adaptive Neuro-Fuzzy Inference System (ANFIS) Model. International Communications in Heat and Mass Transfer 2014, vol. 50, 85–97. https://doi.org/10.1016/j.icheatmasstransfer.2013.11.007

[13]. Moussaoui, M., Admi, Y., Lahmer, E. and Mezrhab, A. (2021). Numerical Investigation of Convective Heat Transfer in Fluid Flow Past a Tandem of Triangular and Square Cylinders in Channel. *Proceedings of the 3rd International Conference on Inventive Research in Material Science and Technology (ICIRMCT 2021)*, 22-23 January 2021, Coimbatore, India, 1-9. https://doi.org/10.1088/1757-899X/1091/1/012058

[14]. Aydin, N., Ozalp, A. and Karagoz, I. Numerical Investigation of Heat and Flow Characteristics in a Laminar Flow Past Two Tandem Cylinders. Thermal Science 2021, vol. 25, no. 4 Part A, 2807-2818. https://doi.org/10.2298/TSCI201119175A

[15]. Huang, Z., Olson, J., Kerekes, R. and Green, S. Numerical Simulation of the Flow Around Rows of Cylinders. Computers & Fluids 2006, vol. 35, no. 5, 485-491. https://doi.org/10.1016/j.compfluid.2005.03.001

[16]. Eleiwi, M. A., Tahseen, T. A. and Ghareeb, A. H. Intelligent Control Based Estimation of Heat Transfer Coefficient from Four Flat Tubes with Different Attack Air Angles. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 2020, vol. 72, no. 2, 65-78. https://doi.org/10.37934/arfmts.72.2.6578

[17]. Abdelhamid, T., Alam, M. M. and Islam, M. Heat Transfer, and Flow Around Cylinder: Effect of Corner Radius and Reynolds Number. International Journal of Heat and Mass Transfer 2021, vol. 171, 121105. https://doi.org/10.1016/j.ijheatmasstransfer.2021.121105

[18]. Zhang, W., Chen, X., Yang, H., Liang, H., and Wei, Y. Forced Convection for Flow Across Two Tandem Cylinders with Rounded Corners in a Channel. International Journal of Heat and Mass Transfer 2019, vol. 130, 1053-1069. https://doi.org/10.1016/j.ijheatmasstransfer.2018.10.125

[19]. Tamura, T., Miyagi, T. and Kitagishi, T. 1998. Numerical Prediction of Unsteady Pressures on a Square Cylinder with Various Corner Shapes. Journal of Wind Engineering and Industrial Aerodynamics 1998, vol. 74, 531-542. https://doi.org/10.1016/S0167-6105(98)00048-8

[20]. Tamura, T. and Miyagi, T. The Effect of Turbulence on Aerodynamic Forces on a Square Cylinder with Various Corner Shapes. Journal of Wind Engineering and Industrial Aerodynamics 1999, vol. 83, no. (1-3), 135-145. https://doi.org/10.1016/S0167-6105(99)00067-7

[21]. Hu, J., Zhou, Y. and Dalton, C. Effects of the Corner Radius on The Near Wake of a Square Prism. Experiments in Fluids 2006, vol. 40, 106-118.

[22]. Zhang, W. and Samtaney, R. Effect of Corner Radius in Stabilizing the Low-Re Flow Past a Cylinder. Journal of Fluids Engineering 2017, vol. 139, no. 12, 121202. https://doi.org/10.1007/s00348-005-0052-2

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