

Optimizing Flow Conditions in Baffled Channels-A CFD Simulation Analysis

Abhinav^{1*}, Sujith Kumar S.G.², K. N. Vishwanath³, Jayatheertha H. S.⁴

¹Assistant Professor, Department of Mechanical Engineering, Dayananda Sagar Academy of Technology and Management, Bangalore Karnataka, Affiliated to VTU-Belagavi, India

²Associate Professor and Head, Department of Mechanical Engineering, T. John Institute of Technology, Bangalore Karnataka, Affiliated to VTU-Belagavi, India

³Professor and Head, Department of Civil Engineering, Dayananda Sagar Academy of Technology and Management, Bangalore Karnataka, Affiliated to VTU-Belagavi, India

⁴Assistant Professor, Department of Civil Engineering, Dayananda Sagar Academy of Technology and Management, Bangalore Karnataka, Affiliated to VTU-Belagavi, India

Abstract: This study explores the pressure, velocity and fluid flow features within a channel when a rectangular baffle is present, focusing on laminar flow conditions. Computational Fluid Dynamics (CFD) code Fluent is employed to analyze flow characteristics. The comparison includes total pressure and velocity drop, with the analysis conducted under 2D, single-phase, and steady-state conditions. An observation is made on a rectangular-shaped baffle channels. In general a rectangular-shaped baffle channels found to generates a recirculation zone at its tips, leading to the entrainment of fluid particles in contact with the baffles. Also, the pitch length in a channel with baffles has a significant impact on pressure and velocity profiles. Increasing the pitch length from 60 cm to 70 cm tends to reduce flow resistance and pressure drop, leading to a more uniform pressure distribution along the channel. A detailed study has been conducted, and the results may be beneficial for CFD engineering in the design of thermo-structural members.

Keywords: CFD, Boundary layer, Pressure gradient, Velocity gradient, Single phase etc.

1. Introduction

One of the most efficient approaches for augmenting heat transfer rates in a smooth channel involves incorporating baffles along the channel walls in either in-line or staggered arrays. Baffles play a crucial role in disrupting hydrodynamic and thermal boundary layers, creating recirculation zones or vortices downstream [1]. The reattachment of flow post-baffle induces a sweeping motion along the channel walls, leading to an increased heat transfer rate. This method is widely employed in engineering applications like compact heat exchangers, air-cooled solar collectors, and electronic packages, resulting in extensive literature on the subject. While the baffle arrangement enhances heat transfer, it also raises the pressure drop in the channel flow due to reduced flow area effects [2]. Consequently, the design of channel heat exchangers involves critical parameters such as baffle spacing and height. Closer spacing or greater baffle height yields a higher heat transfer rate but compromises stream distribution and increases pressure drop. Conversely, greater baffle spacing or smaller baffle height reduces pressure drop but promotes more longitudinal flow, resulting in decreased heat transfer [3]. Balancing these factors to realize the advantages of baffle arrangements or geometry is challenging. Various researchers have studied the impact of baffle spacing on heat transfer and pressure drop. However, there is currently no precise criterion for determining optimal baffle shapes in existing literature. Many previous works introduce the use of baffle height and pitch spacing at 0.5 and 2 times the channel height, respectively. The initial exploration into numerically investigating flow and heat transfer attributes within a duct under periodically fully developed flow conditions was undertaken [4]. In one of the research work the authors proposed that laminar behavior occurs in channels with baffles at Reynolds numbers below 600, and under such conditions, the flow is free from vortex shedding [5]. In another research work reported a numerical investigation of fluid flow and heat transfer features in a smooth channel with staggered baffles, building upon the periodically fully developed flow concept introduced [6]. Kellar et al.[7] conducted heat transfer computations in channels with staggered baffles, observing an increase in heat transfer with higher baffle height and reduced baffle spacing.

Cheng and Huang [8] explored asymmetrical baffles, emphasizing a significant influence of baffle location, especially with increased baffle height. They further presented laminar forced convection in the entrance region of a horizontal channel with one or two pairs of baffles [9]. Habib et al. [10] reported turbulent flow and heat transfer characteristics within a periodic cell formed by segmented baffles staggered in a rectangular duct, noting an increase in pressure drop with baffle height. Amiri et al. [11] investigated laminar flow and heat transfer in two-dimensional channels with packed bed porous media, employing a two-phase equation model. Lopez et al. [12] explored into numerical investigations of laminar forced convection in a three-dimensional channel with baffles, specifically focusing on periodically fully developed flow and uniform heat flux in the top and bottom walls. Guo and Anand [13] studied three-dimensional heat transfers in a channel with a single baffle in the entrance region. Investigations on both solid and porous baffles in two-dimensional channels for turbulent and laminar flow regimes were conducted, yielding similar thermal performance results for both cases [14]. In one of the experimental investigations it was examined that turbulent channel flow with porous baffles, exhibit comparable flow behavior to solid baffles [15]. Amit Garg [16] numerically explored heat transfer behavior in the entrance region of a channel with staggered baffles, considering Reynolds numbers from 50 to 500 and varying baffle heights. They highlighted the influence of the Prandtl number on the precise location of the periodically fully developed region.

In the current study, numerical computations focus on two-dimensional laminar periodic channel flows over a pair of staggered rectangular baffles mounted on the channel walls. The primary objective is to analyze changes in flow patterns in terms of pressure and velocity performance in a baffled rectangular channel. Also, an attempt is made to identify the most effective baffling configuration.

2. Flow Configuration and Mathematical Foundation

2.1 Physical Model

The system under investigation comprises a horizontal plane channel with a pair of baffles arranged in a staggered configuration on the upper and lower channel walls, as illustrated in the Fig.1. The anticipated flow in this system is expected to reach a periodic flow state, wherein the velocity field repeats itself from one cell to the next. The concept of periodically fully developed flow and its corresponding solution procedure have been elucidated. The fluid enters the channel at an inlet temperature (T_{in}), and traverses over a staggered baffle pair, where the baffle height (denoted as b) is set to 0.03 m. The channel height is represented as H , and the ratio b/H is referred to as the blockage ratio (BR). The spacing between the baffles is designated as s , with s/H defining the spacing ratio (S).

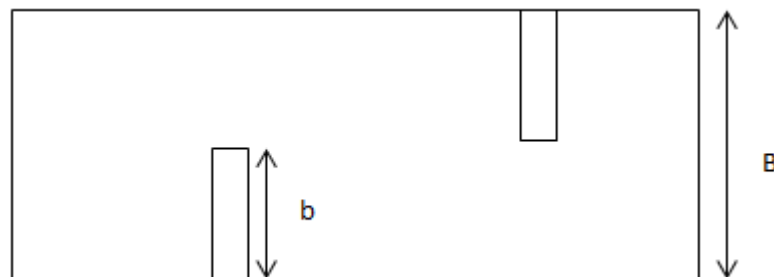


Fig.1 Schematic of 2D baffled channel.

2.2 Numerical Method

The numerical simulations were conducted utilizing the Fluent-6 version of the FLUENT CFD Software package, which employs the finite-volume method to solve the governing equations. The simulation focused on water flow within a copper channel. The geometry was designed, and meshing in the CFD model employed quad/tri shapes. The numerical model for fluid flow and heat transfer in the channel was established based on the following assumptions.

Assumptions:

- Steady two-dimensional fluid flow and heat transfer.
- The flow is laminar and incompressible.

- Constant fluid properties.
- Body forces and viscous dissipation are ignored.
- Negligible radiation heat transfer.

Derived from the aforementioned assumptions, the channel flow is regulated by the continuity equation, the Navier-Stokes equations, and the energy equation. In the Cartesian tensor system, these equations can be expressed as follows:

Continuity Equation:

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0$$

Momentum Equation:

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]$$

Energy Equation:

$$\frac{\partial(\rho u_i T)}{\partial x_i} = \frac{\partial}{\partial x_j} \left(\tau \frac{\partial T}{\partial x_j} \right)$$

Where τ is thermal diffusivity and is given by $\tau = \frac{\mu}{Pr}$

The governing equations underwent discretization through the second-order upwind scheme, decoupling via the SIMPLE algorithm, and resolution using a finite volume approach. Convergence of the solutions was deemed achieved when the normalized residual values dropped below 10^{-6} for energy and 10^{-3} for momentum variables. Table 1 shows the properties of water at 27 °C.

Table 1: Properties of water at 27 °C

Properties	Value
Density, ρ	998.2 kg/m ³
Specific heat capacity C_p	4182 J/kg K
Thermal conductivity, k	0.6W/m K
Viscosity, μ	0.001003 kg/m s

2.3 Boundary Conditions

Periodic boundaries are used for the inlet and outlet of the flow domain. Constant mass flow rate of air with 300 K ($Pr=5.976$) is assumed in the flow direction rather than constant pressure drop due to periodic flow conditions. The inlet and outlet profiles for the velocities must be identical. The physical properties of the water have been assumed to remain constant at average bulk temperature. Impermeable boundary and no-slip wall conditions have been implemented over the channel wall as well as the baffle. The constant temperature of the bottom and upper plates is maintained at 300 K while the baffle is assumed at adiabatic wall conditions. It should be noted that the dimensionless temperature must be identical between the inlet and outlet. Outlet is taken as Pressure outlet

3.Results and Discussion

The results are compared for the three scenarios with different numbers of baffles. Pressure drop and velocity distribution along the channel length are evaluated refer Fig.1 to 10. The variation in pressure and velocity in laminar single-phase water flow through rectangular channels with different numbers of baffles found to arises from the intricate interplay of flow obstruction, redirection, and recirculation induced by the baffles. Baffles act as obstacles, increasing resistance and altering the flow path, leading to higher pressure drops. The redirection of flow around baffles creates varying velocity profiles along the channel observed. Recirculation zones and vortex formation, particularly in the wake of baffles, further contribute to localized pressure variations understood. Changes in channel geometry found to influence velocity gradients, and the interaction between baffles introduces complexities, especially in cases with multiple pairs. The presence of baffles impacts boundary layer development on channel walls, influencing velocity distributions and pressure gradients. In essence, the observed variations result from the complex fluid dynamics influenced by the obstruction and redirection effects introduced by the baffles in the channel.

The pitch length in a channel with baffles has a significant impact on pressure and velocity profiles. Increasing the pitch length from 60 cm to 70 cm tends to reduce flow resistance and pressure drop, leading to a more uniform pressure distribution along the channel. Longer pitch lengths facilitate smoother flow between baffles, diminishing the intensity and size of recirculation zones. Moreover, a shorter pitch may induce higher velocity gradients and localized variations, while a longer pitch promotes a more even velocity distribution, dampening secondary flow effects refer Fig. 11 (a) and (b). The pitch length also influences boundary layer development, with longer pitches contributing to thicker and more stabilized boundary layers, impacting velocity gradients near the channel walls. Overall, the choice of pitch length plays a crucial role in optimizing flow conditions, affecting pressure, and velocity characteristics within the baffled channel.

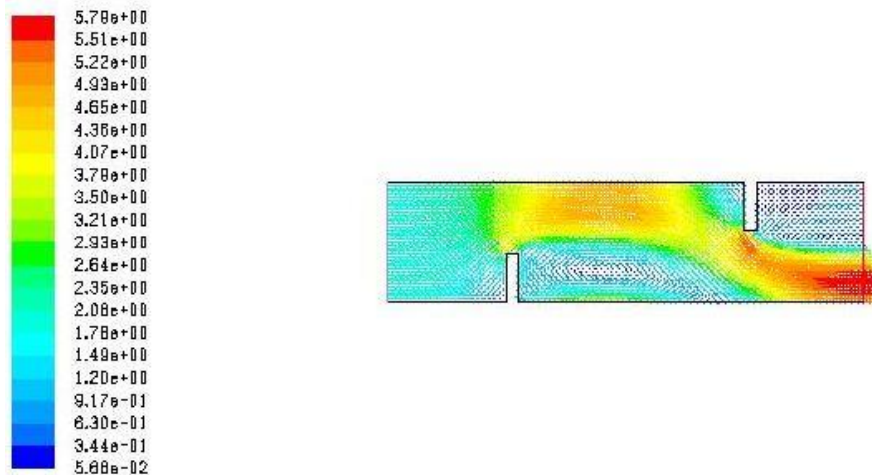


Fig-1 Velocity vector for one baffle

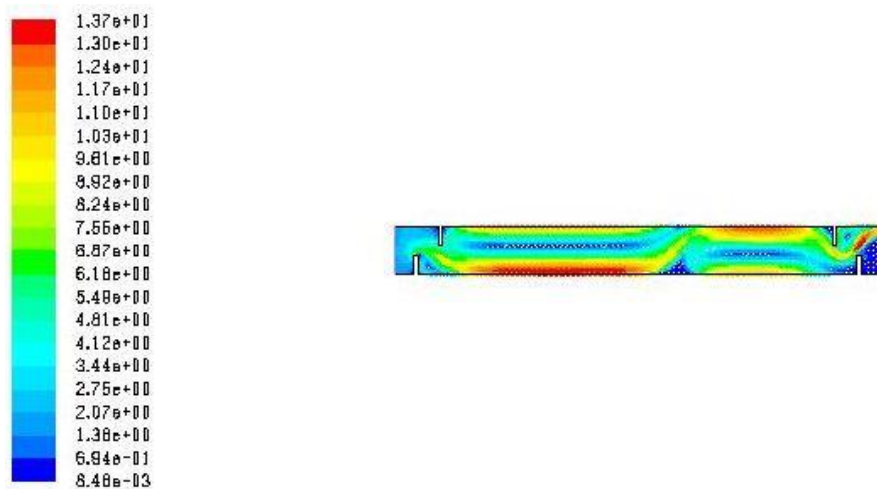


Fig-2 Velocity vector for two pair of baffles

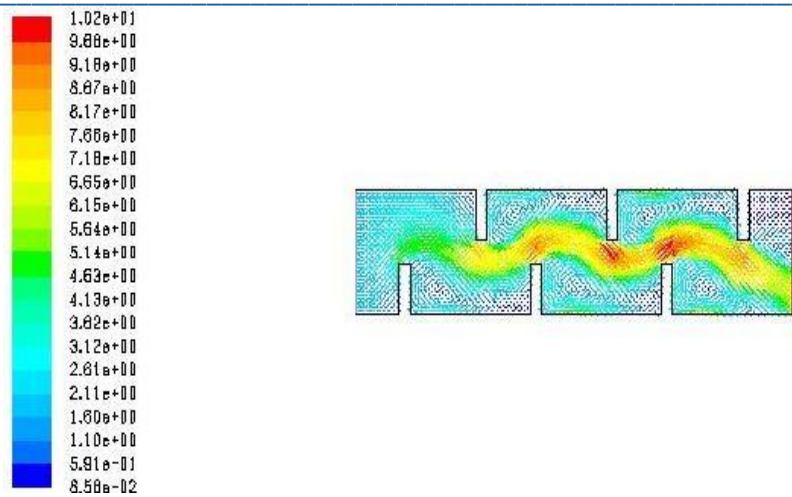


Fig-3 Velocity vector with three pair of baffle

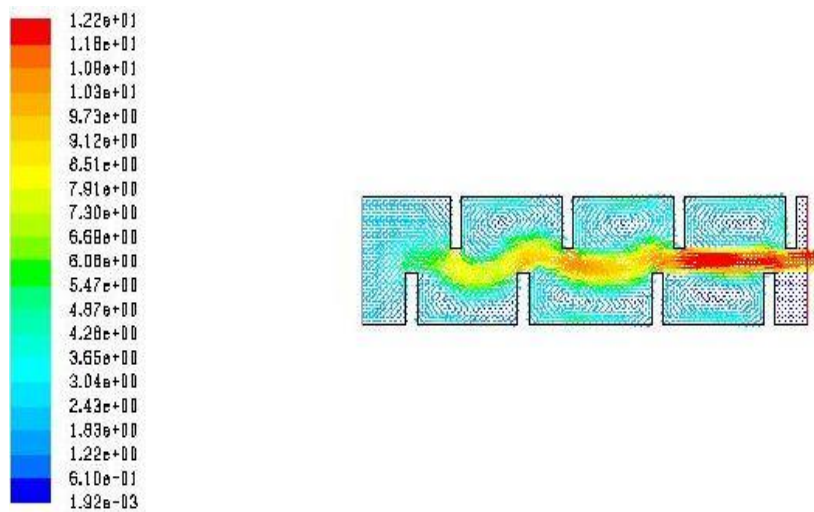


Fig-4 Velocity vector with four pair of baffle

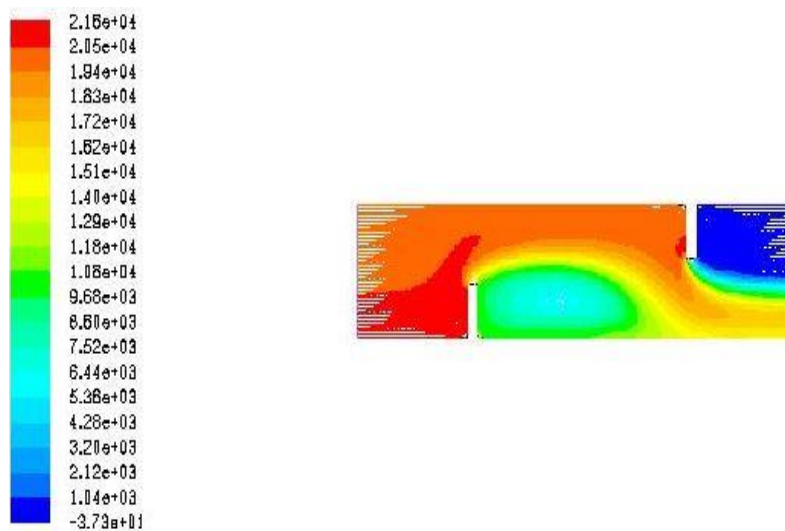


Fig-5 Pressure variation for one pair baffle

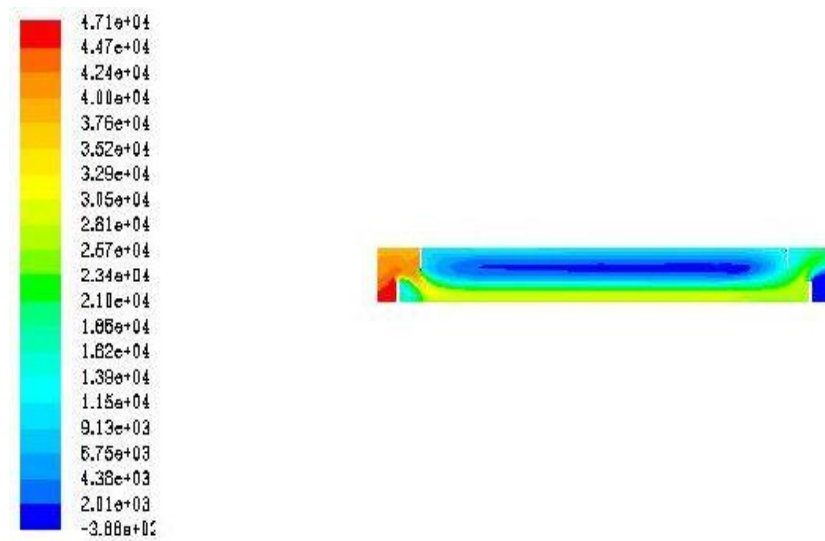


Fig-6 Pressure variation for two pair of baffles

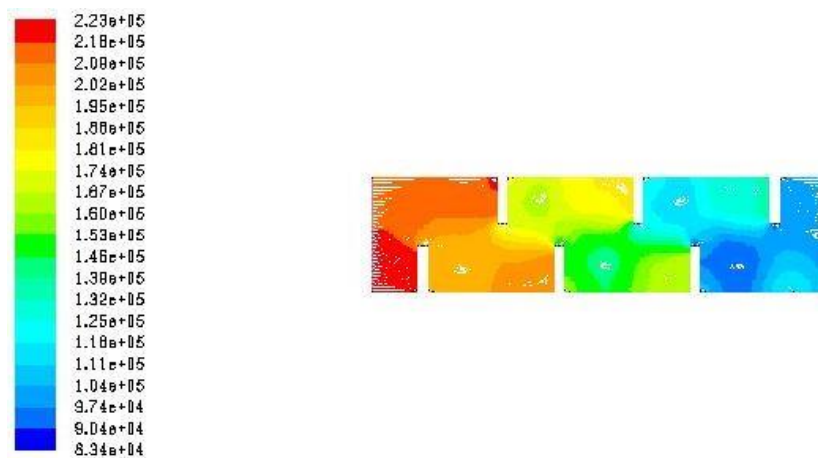


Fig-7 Pressure variation for three pair of baffles

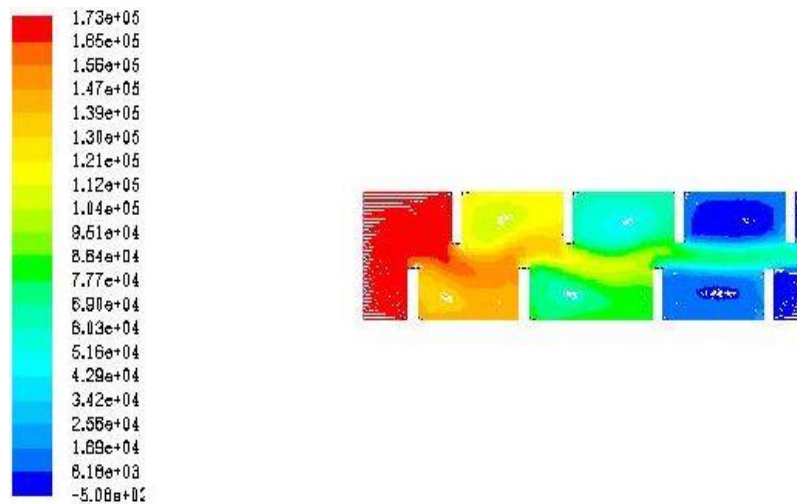


Fig-8 Pressure variation for four pair of baffles

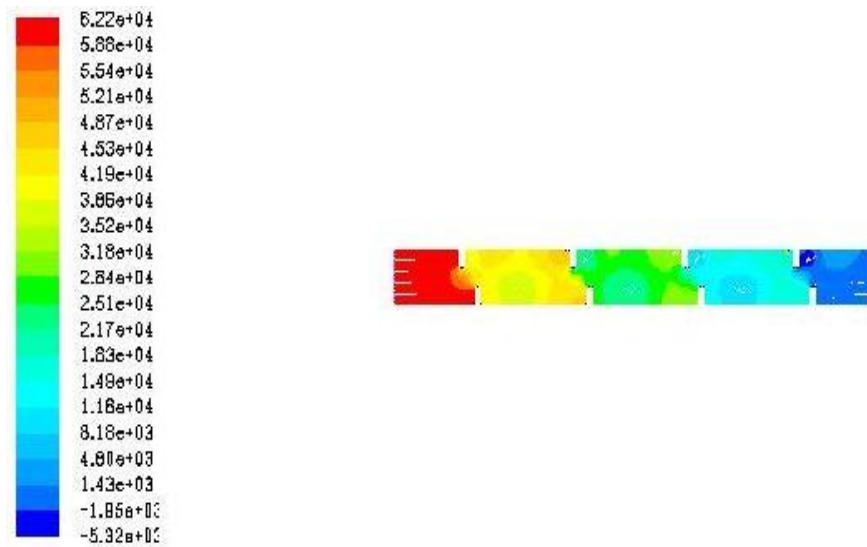


Fig-9 Pressure variation for the same pitch having four pair of baffle, Pitch length 60 cm

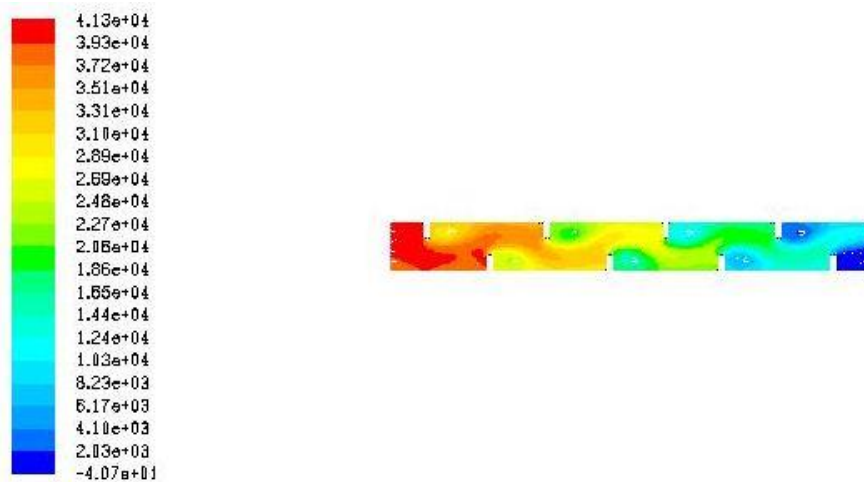


Fig-10 Pressure variation for the same pitch having four pair of baffle, Pitch length 70 cm

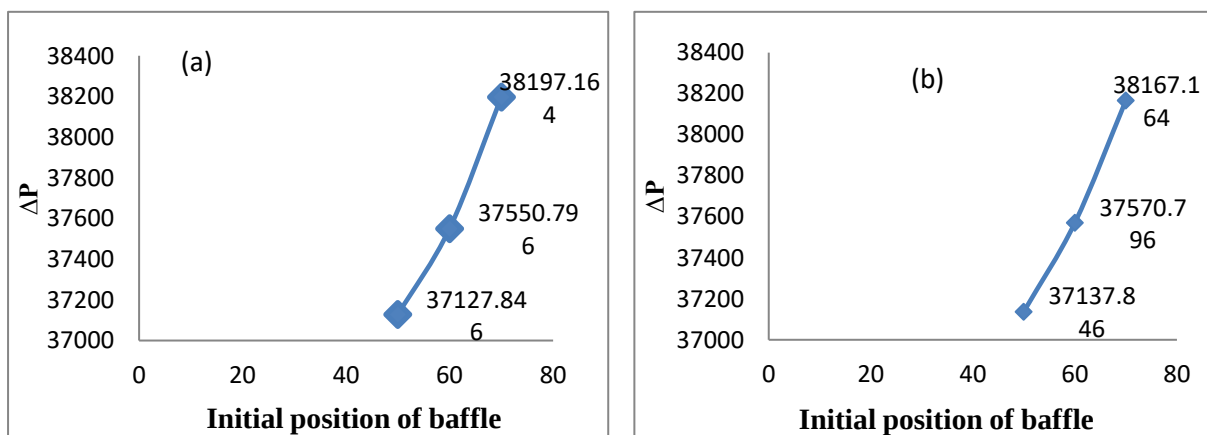


Fig.11 Pressure variation v/s pitch for pitch length (a) 60 and (b) 70 cm

2. Conclusion

Through simulation, the pressure and velocity variations in a baffled rectangular channel helped in identifying the most effective baffling configuration. Simulation outcomes can be crucial for optimizing heat transfer, minimizing pressure drop, and improving overall fluid flow characteristics. Results obtained on pressure and velocity variations can be further used to optimize the distribution of fluid flow within the complex channel. This is particularly important in applications where uniform flow is critical for achieving desired results, such as in certain chemical processes or cooling systems. Understanding the pressure variation helps engineers and researchers assess the energy requirements of the system and make informed decisions about pump sizing and power consumption. By analyzing velocity variations, the simulation can reveal how different baffle configurations impact fluid mixing and turbulence within the channel. This information is valuable in applications where efficient mixing is essential, such as in chemical reactors or heat exchangers. Pressure and velocity patterns found to reveal the presence of recirculation zones in the baffled rectangular channel. Understanding these flow phenomena is crucial for minimizing dead zones and improving the overall efficiency of the system. Over all, from the CFD investigations it has been found that pitch length in a channel with baffles has a significant impact on pressure and velocity profiles. Increasing the pitch length from 60 cm to 70 cm tends to reduce flow resistance and pressure drop, leading to a more uniform pressure distribution along the channel. Longer pitch lengths facilitate smoother flow between baffles, diminishing the intensity and size of recirculation zones. The simulation outcomes can be used to establish design guidelines for practical applications of baffled rectangular channels, assisting engineers and researchers in making informed decisions for specific industrial or scientific purposes.

References

- [1] Amnart Boonloi, Withada Jedsadaratanachai, "Effects of Baffle Height and Baffle Location on Heat Transfer and Flow Profiles in a Baffled Duct: A CFD Analysis", *Modelling and Simulation in Engineering*, vol. 2022, Article ID 3698887, 19 pages, 2022. <https://doi.org/10.1155/2022/3698887>
- [2] S. Eiamsa-ard, A. Suksangpanomrung, P. Promthaisong, Enhanced heat transfer mechanism and flow topology of a channel contained with semi-circular hinged V-shaped baffles, *International Journal of Thermal Sciences*, Volume 177, 2022, 107577, ISSN 1290-0729. <https://doi.org/10.1016/j.ijthermalsci.2022.107577>.
- [3] Yu J-S, Kim J-H, Kim J-T. Effect of Triangular Baffle Arrangement on Heat Transfer Enhancement of Air-Type PVT Collector. *Sustainability*. 2020; 12(18):7469. <https://doi.org/10.3390/su12187469>
- [4] S.V. Patankar, C.H. Liu, E.M. Sparrow, Fully developed flow and heat transfer in ducts having streamwise-periodic variations of cross-sectional area, *ASME J. Heat Transfer* 99 (1977) 180–186.
- [5] C. Berner, F. Durst, D.M. McEligot, Flow around baffles, *Trans. ASME J. Heat Transfer* 106 (1984) 743–749.
- [6] B.W. Webb, S. Ramadhyani, Conjugate heat transfer in a channel with staggered ribs, *Int. J. Heat Mass Transfer* 28 (1985), 1679–1687.
- [7] K.M. Kelkar, S.V. Patankar, Numerical prediction of flow and heat transfer in a parallel plate channel with staggered fins, *ASME J. Heat Transfer* 109 (1987) 25–30.
- [8] C.H. Cheng, W.H. Huang, Laminar forced convection flows in horizontal channels with transverse fins placed in entrance regions, *Int. J. Heat Mass Transfer* 20 (1991) 1315–1324.
- [9] C.H. Cheng, W.H. Huang, Numerical prediction for laminar forced convection in parallel-plate channels with transverse fin arrays, *Int. J. Heat Mass Transfer* 34 (11) (1991) 2739–2749.
- [10] M.A. Habib, A.M. Mobarak, M.A. Sallak, E.A. Abdel Hadi, R.I. Affify, Experimental investigation of heat transfer and flow over baffles of different heights, *Trans. ASME J. Heat Transfer* 116 (1994) 363–368.
- [11] A. Amiri, K. Vafai, T.M. Kuzay, Effects of boundary conditions on non-Darcian heat transfer through porous media and experimental comparison, *Numer. Heat Transf. Appl.* 27 (1995) 651–664.
- [12] J.R. Lopez, N.K. Anand, L.S. Fletcher, Heat transfer in a three-dimensional channel with baffles, *Numer. Heat Transf., A Appl.* 30 (1996) 189–205.
- [13] Z. Guo, N.K. Anand, Three-dimensional heat transfer in a channel with a baffle in the entrance region, *Numer. Heat Transf., A Appl.* 31 (1) (1997) 21–35.
- [14] M.E. Nimvari, M. Maerefat, M.K. El-Hossaini, Numerical simulation of turbulent flow and heat transfer in a channel partially filled with a porous media, *International Journal of Thermal Sciences*, Volume 60, 2012, Pages 131–141, ISSN 1290-0729,

- <https://doi.org/10.1016/j.ijthermalsci.2012.05.016>.
- [15] M. Anil Kizilaslan et al., Experimental and Numerical Evaluation of a Porous Baffle Design for Contact Tanks, Journal of Environmental Engineering Volume 146, Issue 7, May 7 2020.
[https://doi.org/10.1061/\(ASCE\)EE.1943-7870.0001747](https://doi.org/10.1061/(ASCE)EE.1943-7870.0001747)
- [16] Amit Garg et al., CFD analysis of laminar heat transfer in a channel provided with baffles: comparative study between two models of baffles: diamond-shaped baffles of different angle and rectangle, International Journal of Enhanced Research in Science Technology & Engineering, Volume. 3 Issue 7, July-2014, pages 267-276), ISSN: 2319-7463