STM JOURNALS

International Journal of
Energy and Thermal Applications

Volume 1, Issue 2, 2023
July—December
DOI (Journal): 10.37591/IJETA

https://journals.stmjournals.com/ijeta

Research IJETA

Study and Thermal Analysis of a Rectangular Duct with Z-shaped Baffle Using Ansys

Abinav Suresh[1,*], Amal Johnson[2], Aromal Asok A.R.[3], Sibin P.[4]

Abstract

This project aims to enhance heat transfer in a rectangular duct by utilizing Z-shaped baffles. It involves creating a 3D CAD model using SolidWorks and validating it in ANSYS Fluent 2019 R2. The study focuses on analyzing the effect of baffle arrangement, height, and angle on heat transfer rate. Comparative analysis is conducted between rectangular ducts with a 60° angle and heights of 4 mm, 6 mm, and 8 mm. The obtained data is then compared with values from relevant academic sources. The project's findings will contribute to improving heat transfer in various engineering applications, leading to more efficient and sustainable industrial processes. It also explains the meshing, boundary conditions and thermal analysis techniques employed in ANSYS Fluent to ensure accurate simulation and prediction of fluid flow and heat transfer behavior. By optimizing heat transfer within rectangular ducts, this project aims to advance thermal management systems and develop improved heat transfer technologies.

Keywords: Z-shaped baffles, thermal analysis, solid work, ANSYS, CAD, DUCTS

INTRODUCTION

Enhancing heat transfer is critical in many engineering applications, including power generation, electronics cooling and chemical processing. Using flow disturbance devices such as baffles to increase turbulence in the flow is one way to improve heat transfer. Z-shaped baffles have been shown to be effective in improving heat transfer in channels due to their ability to create a secondary flow that promotes mixing and turbulence.

The goal of this project is to use Ansys, a popular simulation software in the field of engineering, to analyse the heat transfer enhancement in a channel with Z-shaped baffles. The primary goal of this project is to investigate the effect of Z-shaped baffles on the rate of heat transfer in the channel. We will specifically look at the effect of baffle arrangement, baffle height, and baffle angle.

The project is implemented in stages. In the first stage, we used Solid Works to create a 3D cad model. The solid works programme is used to design the baffle angle, height, and experiment apparatus. The model is then uploaded into ANSYS and validated by comparing simulation results to experimental data from the literature. We will also conduct a parametric study to investigate how the different height used in the apparatus affect the heat flow. Finally, we will analyse the data and draw conclusions about the effectiveness of the Z-shaped baffles in improving heat transfer in the channel [1].

This project's findings will help to improve understanding of heat transfer enhancement using Z-shaped baffles in channels, as well as provide insights for the design of more efficient heat exchangers and other thermal management systems. This project's knowledge can be applied to a wide

***Author for Correspondence**
Abinav Suresh
E-mail: abinavsuresh77@gmail.com

[1-4]Student, Department of Mechanical Engineering, APJ Abdul Kalam Technological University, Thiruvananthapuram, Kerala, India

Received Date: February 28, 2024
Accepted Date: March 19, 2024
Published Date: April 19, 2024

Citation: Abinav Suresh, Amal Johnson, Aromal Asok A.R., Sibin P. Study and Thermal Analysis of a Rectangular Duct with Z-shaped Baffle Using Ansys. International Journal of Energy and Thermal Applications. 2023; 1(2): 14–30p.

range of engineering applications, helping to improve the efficiency and sustainability of various industrial processes.

TYPES OF BAFFLES USED IN DUCTS

Baffles are used in thermal analysis of a duct to improve heat transfer and fluid flow characteristics within the duct. Depending on the specific requirements of the application, various types of baffles can be used in a duct. Some of the most common types of baffles used in duct thermal analysis are as follows:

- *Straight Baffles:* The most basic type of baffle used in a duct. Straight baffles are commonly used to direct the flow of fluid within the duct and promote fluid mixing.
- *Curved baffles:* are used to increase swirl and turbulence in the fluid, which improves heat transfer and mixing. Curved baffles are especially useful in laminar fluid flow applications.
- *Perforated Baffles:* These baffles have a series of holes or perforations in them. By allowing the fluid to pass through the baffles, they promote mixing and heat transfer [2].
- *Z-shaped Baffles:* These baffles are shaped like a Z and are used to direct the fluid in a tortuous path. This lengthens the time the fluid spends in the duct and promotes mixing and heat transfer.
- *Helical Baffles:* These helical baffles are used to create a swirling flow pattern in the fluid. Helical baffles can improve duct heat transfer and mixing.

The type of baffle chosen will be determined by the application's specific requirements, such as the fluid flow rate, temperature, and pressure, as well as the desired heat transfer and mixing characteristics.

Z-SHAPED BAFFLES IN RECTANGULAR DUCTS

Z-shaped baffles increase the surface area of the fluid in contact with the duct wall, resulting in a more efficient heat transfer process. The use of Z-shaped baffles can reduce duct pressure drop, resulting in a more energy-efficient system. The baffle causes the fluid to flow around the plates, promoting mixing and improving fluid temperature distribution uniformity. By promoting mixing and reducing stagnant areas, Z-shaped baffles can help to reduce fouling within the duct [4].

AIM And Objective

In many industries, such as HVAC, aerospace, and automotive, rectangular ducts are widely used for the transfer of air or gas streams. Z-shaped baffles are a type of flow control device that is used to create a tortuous path for the fluid passing through the rectangular duct.

Objective of the project is to:

- To modify the design of the rectangular duct by adding new angle and different height of baffle using solid works and analyse the structure in ANSYS.
- The effect of baffle arrangement, baffle height, and baffle angle will be examined by comparing with journal [5].

LITERATURE REVIEW

P. Promvonge [3] "Experimental and numerical study on heat transfer enhancement in a channel with Z-shaped baffles" says about the influence of baffle turbulators on heat transfer augmentation in a rectangular channel and the same is investigated experimentally. The main data's taken for the experiment is from the reference of this journal. In this experiment the Reynolds number is varied from 4400 to 20,400 and the z baffle is inclined at an angle of 45°.Also the baffle- to channel-height ratios (e/H=0.1, 0.2 and 0.3) and baffle pitch ratios (P/H=1.5, 2 and 3) are also referred. In this journal the experimental results show a significant effect of the presence of the Z-baffle on the heat transfer rate and friction loss over the smooth channel with no baffle.

"Heat transfer and friction in solar air heater duct with V-shaped rib roughness on absorber plate" says that Nusselt number increases whereas the friction factor decreases with an increase of Reynolds number. The values of Nusselt number and friction factor are substantially higher as compared to those obtained for smooth absorber plates. This is due to distinct change in the fluid flow characteristics as a result of roughness that causes flow separations, reattachments and the generation of secondary flows. It also says that the maximum enhancement of Nusselt number and friction factor as a result of providing artificial roughness has been found to be respectively 2.30 and 2.83 times that of smooth duct for an angle of attack of 60°.

P. Promvonge [3] "Periodic laminar flow and heat transfer in a channel with 45° staggered V-baffles" says that it is apparent that in each of the main vortex flows, a pair of streamwise twisted vortex (P-vortex) flows can induce impinging flows on a sidewall and a wall of the inter baffle cavity leading to drastic increase in heat transfer rate over the channel. In addition, the rise in the V-baffle height results in the increase in the Nusselt number and friction factor values (Table 1).

DESIGN CRITERIAS

The primary design goal is to accelerate heat transfer within the channel. This is achieved by employing Z-shaped baffles, which generate turbulence and increase the contact area between the fluid and the channel walls. The effect of various Z-shaped baffle geometries on heat transfer enhancement is investigated in this study. The height and pitch of the baffles are varied to determine their impact on heat transfer (Tables 2 and 3). The Reynolds number, a dimensionless parameter that characterises fluid flow, is changed to see how it affects heat transfer enhancement [6]. The effect of Z-shaped baffles on heat transfer enhancement is investigated in this study. The number of baffles varies between one and twelve (Figure 1).

Table 1. Literature review.

Author(s)	Journal	Observation
Parkpoom Sriromreun, Chinaruk Thianpong, Pongjet Promvonge [1].	"Experimental and numerical study on heat transfer enhancement in a channel with Z-shaped baffles"	This journal the experimental results show a significant effect of the presence of the Z-baffle on the heat transfer rate and friction loss over the smooth channel with no baffle.
Abdul-Malik Ebrahim Momin, J.S. Saini, S.C. Solanki [2].	"Heat transfer and friction in solar air heater duct with V-shaped rib roughness on absorber plate"	The thermo-hydraulic performance parameter improves with increasing the angle of attack of flow and relative roughness height and the maxima occurs with an angle of attack of 60°.
Pongjet Promvonge, Sutapat [1] EQ Kwankaomeng	"Periodic laminar flow and heat transfer in a channel with 45° staggered V-baffles"	Rise in the V-baffle height results in the increase in the Nusselt number and friction factor values.

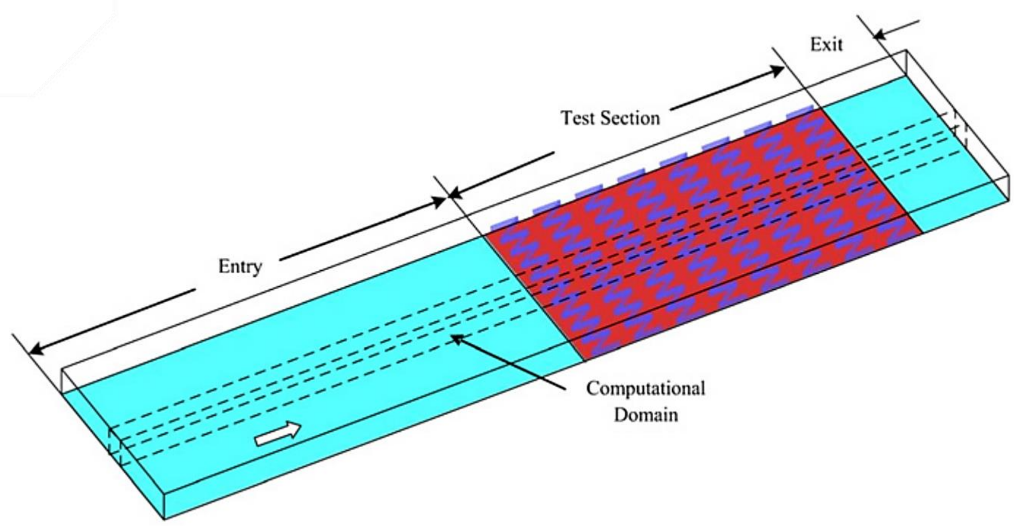


Figure 1. Sample design.
Table 2. Requirements for 45 deg. Z-shaped baffles.

Parameters	Range
Reynolds number range	5000, 10000, 15000, 20000
Height range	3, 6, 9mm
Channel width W	300mm
Channel length	450mm inlet 380mm test section – taken as *L* 90mm outlet
Channel height h	30 mm

Table 3. Requirements for 60 deg. Z-shaped baffles.

Parameters	Range
Reynolds number range	5000, 10000, 15000, 20000
Height range	4,6,8mm
Channel width W	300mm
Channel length	450mm inlet 380mm test section – taken as *L* 90mm outlet
Channel height h	30mm

FLOW RATE

The Z-shaped baffles are inserted into the channel to improve heat transfer between the fluid and the channel walls, and the flow rate is an important parameter that influences the channel's heat transfer performance. The flow rate is incorporated into the model that simulated fluid flow and heat transfer in the Z-shaped baffled channel. The numerical simulations are used to predict the channel's heat transfer performance for various flow rates and baffle configurations [7].

TEMPERATURE

The temperature of the fluid at the baffles is higher than that of the fluid in the channel's centre. The increased turbulence caused by the Z baffles causes the temperature difference. The temperature of the fluid varies with the Reynolds number. The temperature of the fluid near the baffles is slightly lower than the temperature in the centre of the channel at lower numbers. At higher Reynolds numbers, the temperature near the baffles is higher than the temperature in the channel centre [8].

REFINEMENT OF DATA

The dimensions of the rectangular cross-section channel were selected based on the size of the apparatus. The channel's walls were made of aluminium to facilitate heat transfer. Z baffles are incorporated into the channel to improve heat transfer. The baffles were placed at regular intervals along the length of the channel, perpendicular to the flow direction. The baffles' z-shaped design was chosen to create turbulence in the fluid flow, which would aid in heat transfer. The duct is designed to have a rectangular cross-section with two opposite walls containing Z-shaped baffles. The baffles are placed perpendicular to the direction of fluid flow and are arranged in a staggered pattern [9]. The height, spacing, and angle of the baffles are varied in the study to investigate their effect on heat transfer enhancement (Figures 2-4).

The experimental apparatus contains three main sections they are:

1. Entry
2. Test section
3. Exit

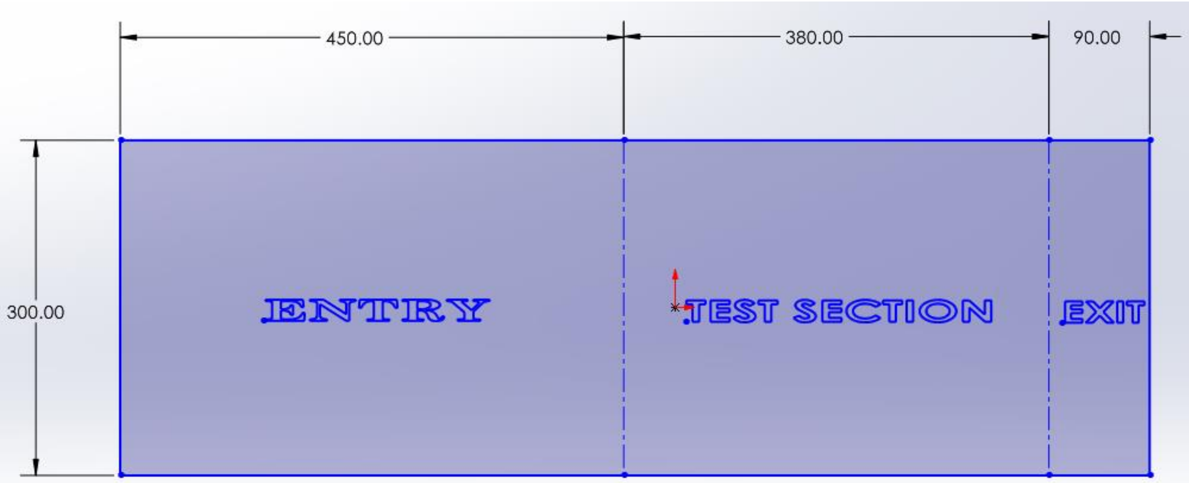


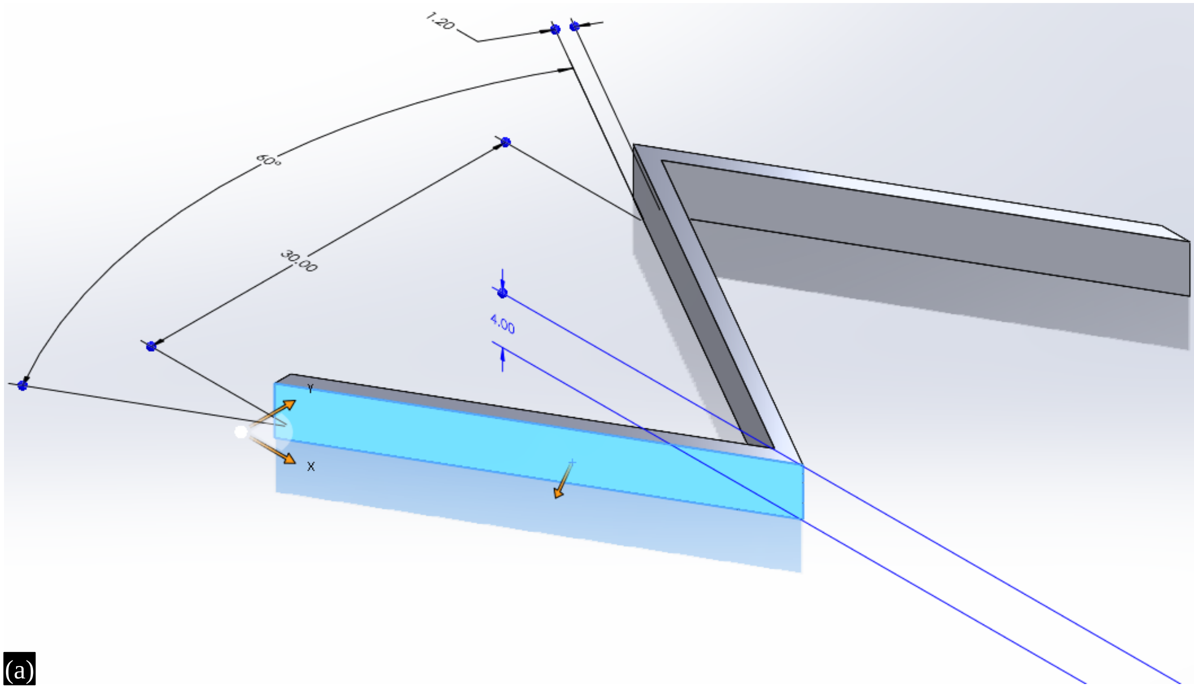
Figure 2. Shows the entry, test section and exit of the duct, entry with 450 mm, test section with 380 mm and exit with 90.

(a)

(b)

(c)

Figure 3. Side view of duct (a) duct with 8 mm baffle height (b) duct with 6 mm baffle height (c) duct with 4 mm baffle height.



(a)

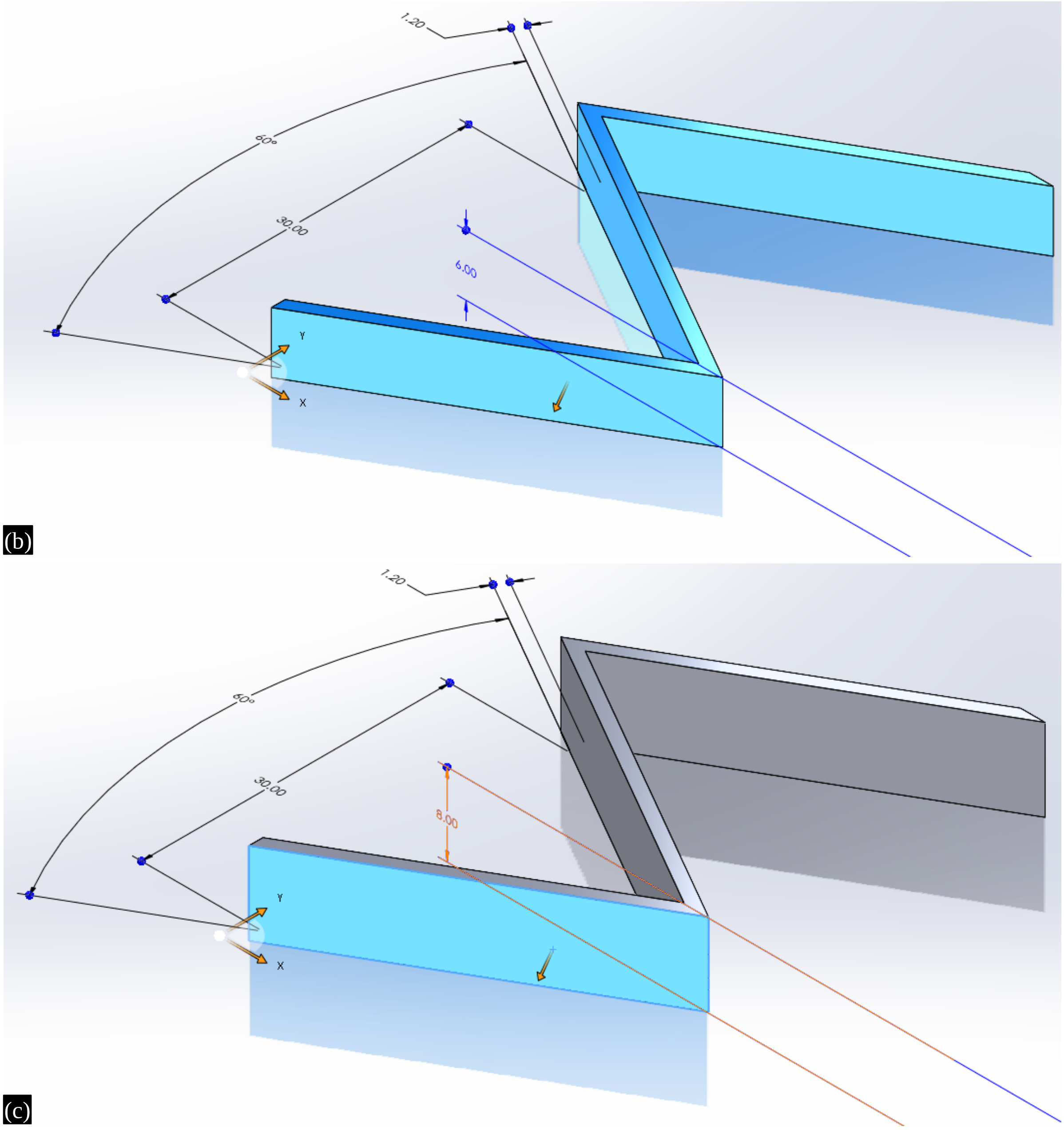


Figure 4. Isometric view of baffle (a) Z-shaped baffle with height 4 mm (b) Z-shaped baffle with height 6 mm (c) Z-shaped baffle with height 8 mm.

CALCULATION

In the project, the velocity (V) is determined using the Reynolds number (Re) formula, which involves the hydraulic diameter (D), kinematic viscosity (v), and the characteristic length. The hydraulic diameter is calculated using the cross-sectional area (A) and perimeter (P) of the test section [10]. For the given values, the area of the test section is calculated as:

$$A = W \times h = 300mm \times 30mm = 0.009m^2$$

where W is the width and h is the height.

The perimeter of the test section is calculated as:

$$P = 2 \times (w + h) = 300mm + 30mm = 0.066m^2$$

Using the formula for hydraulic diameter

$$D = \frac{4 \times A}{P}$$

we can substitute the calculated values and find that the hydraulic diameter is 0.0545 m.

To determine the velocity, we can use the equation:

$$V = \frac{Re \times v}{D}$$

where Re is the Reynolds number and v is the kinematic viscosity.

By substituting the appropriate Reynolds number and kinematic viscosity values, the equation becomes:

$$v = \frac{Re \ \times v}{\left[\frac{4 \times A}{P}\right]}$$

Since the hydraulic diameter $D = \frac{4 \times A}{P}$, the equation simplifies to $V = \frac{Re \times v}{D}$.

Now, you can substitute the specific Reynolds number and kinematic viscosity values you have into the equation to calculate the velocity 9 Table 4.

For friction factor:

$$f = \frac{\Delta p}{\frac{\left[\frac{L}{D}\right] \times \rho U^2}{2}}$$

Where
L = duct length
D = hydraulic diameter
U = mean air velocity

ANALYSIS

Static and Periodic Analysis

Static and periodic analysis are two important techniques used in ANSYS Fluent, a powerful computational fluid dynamics (CFD) software. These analyses play a significant role in understanding and predicting the behavior of fluids in various engineering applications. Static analysis in ANSYS Fluent focuses on studying the flow characteristics of fluids under steady-state conditions. It is commonly used to analyse the pressure distribution, velocity profiles, and forces acting on structures or objects immersed in the fluid. In this analysis, the fluid properties, boundary conditions, and geometry are typically assumed to be constant over time. By solving the governing equations of fluid dynamics, such as the Navier-Stokes equations, ANSYS Fluent can provide valuable insights into the behavior of the fluid in a given system.

Table 4. Inlet velocity with respect to Reynolds number.

Reynolds number (RE)	Inlet velocity (M/S)
5000	1.44
10000	2.88
15000	4.31
20000	5.75

Periodic analysis, on the other hand, deals with flow phenomena that exhibit periodic behavior or periodic boundary conditions. This type of analysis is particularly useful for studying flows in systems with repeating structures, such as turbomachinery, heat exchangers, or periodic flow domains. ANSYS Fluent allows for the modeling of periodicity by defining periodic boundaries or specifying periodic conditions, which enables the simulation of one period of the flow and provides insights into the flow characteristics over the entire system.

In ANSYS Fluent, both static and periodic analyses involve several key steps. These include geometry creation or import, mesh generation, specifying fluid properties and boundary conditions, setting up the appropriate solver settings, and post-processing the results. The software provides a wide range of tools and features to facilitate each step of the analysis process.

During the analysis, ANSYS Fluent utilizes numerical methods, such as finite volume or finite element methods, to discretize the governing equations and solve them numerically. The software employs iterative solvers to obtain converged solutions, ensuring accuracy and reliability in the results. Additionally, ANSYS Fluent offers various turbulence models, multiphase flow models, and other advanced modeling capabilities to accurately capture complex flow phenomena and simulate real-world scenarios.

Once the analysis is complete, ANSYS Fluent provides comprehensive post-processing tools to visualize and analyse the results. Users can generate contour plots, vector plots, streamline plots, and other visual representations to gain insights into the flow behavior. Additionally, ANSYS Fluent allows for the extraction of quantitative data, such as forces, pressures, velocities, and other flow parameters, which can be used for further analysis or comparison with experimental data.

Overall, static and periodic analyses in ANSYS Fluent offer powerful capabilities for studying fluid flow behavior in a wide range of engineering applications. These analyses provide engineers and researchers with valuable insights into the performance, efficiency, and safety of various systems, aiding in the design, optimization, and troubleshooting processes. ANSYS Fluent's robust features, accurate solvers, and extensive post-processing capabilities make it a popular choice for fluid flow simulations and analysis in the engineering community.

Meshing

Meshing plays a crucial role in computational fluid dynamics (CFD) simulations as it affects the accuracy and reliability of the results. In ANSYS Fluent, a popular CFD software, various meshing techniques are available to generate meshes for different geometries. One commonly used technique is hexahedral meshing, which offers several advantages such as improved numerical accuracy and reduced computational cost. In this note, we will discuss the application of hexahedral meshing with the sweep method for a rectangular duct with a z-shaped baffle.

The rectangular duct with a z-shaped baffle geometry consists of a main duct with a rectangular cross-section and a baffle placed inside the duct in a z-shape pattern. This configuration is often encountered in heat exchangers, where the baffle is used to enhance heat transfer and fluid mixing. To perform accurate simulations of fluid flow and heat transfer in such systems, an appropriate mesh must be generated.

Hexahedral meshing is well-suited for this type of geometry because it provides a structured mesh with primarily hexahedral elements. The use of hexahedral elements offers advantages such as better control over the mesh quality, reduced numerical diffusion, and improved convergence characteristics.

The sweep method is commonly employed in ANSYS Fluent to generate hexahedral meshes for complex geometries. This method involves dividing the domain into layers and sweeping each layer along a specific direction to create the hexahedral elements. In the case of a rectangular duct with a z-shaped baffle, the sweep method can be applied to generate layers of hexahedral cells along the duct's length, width, and height.

To start the meshing process, the geometry of the rectangular duct with the z-shaped baffle is imported into ANSYS Fluent. The geometry can be constructed using CAD software or built directly within the software using its built-in modeling capabilities.

Next, the domain is divided into multiple layers along the desired directions. For example, the mesh can be divided into layers along the length, width, and height of the duct. Each layer represents a region where the mesh cells will be generated.

The sweep method is then applied to each layer, starting from an inlet face and moving towards the outlet face. This involves creating a base mesh layer and then sweeping it along the specified direction. The sweep operation creates hexahedral cells by connecting corresponding faces of adjacent layers. The process is repeated for each layer until the entire domain is meshed.

During the meshing process, several parameters can be adjusted to control the mesh quality and resolution. These include the size of the mesh elements, the number of layers, and the growth rate of the mesh along each direction. It is important to strike a balance between mesh resolution and computational cost, ensuring that the mesh is fine enough to capture the flow features accurately while maintaining reasonable computational requirements.

After the mesh generation is complete, it is essential to perform mesh quality checks and refinement if necessary. ANSYS Fluent provides tools to assess the quality of the mesh, such as checking for skewness, aspect ratio, and orthogonality of the elements. If any issues are identified, mesh refinement techniques, such as local mesh resizing or smoothing, can be applied to improve the mesh quality. Once the hexahedral mesh is finalized, it can be used for conducting CFD simulations in ANSYS Fluent. The mesh provides a discretized representation of the domain, enabling the solution of the governing equations for fluid flow, heat transfer, and other relevant phenomena (Figure 5).

Boundary Condition

During the analysis of a rectangular duct with Z-shaped baffles using ANSYS Fluent, several boundary conditions need to be specified to accurately model the fluid flow and heat transfer behavior. These boundary conditions include specific heat, density, thermal conductivity, kinematic viscosity, and temperature.

1. *Specific Heat:* The specific heat (Cp) is a material property that represents the amount of heat energy required to raise the temperature of a unit mass of the fluid. In this case, a specific heat value of 1006.43 J/kg K is applied, which indicates that the fluid has a high heat capacity.
2. *Density:* The density (ρ) of the fluid determines its mass per unit volume. For the analysis, a density value of 1.177 kg/m^3 is specified, representing the density of air in the rectangular duct with Z-shaped baffles.
3. *Thermal Conductivity:* The thermal conductivity (k) of a material describes its ability to conduct heat. It represents the rate at which heat is transferred through the material. In this analysis, a thermal conductivity value of 0.02638 W/m K is applied to represent the thermal properties of the fluid.
4. *Kinematic Viscosity:* The kinematic viscosity (v) is a measure of the fluid's resistance to flow under the influence of a shear stress. It is defined as the ratio of dynamic viscosity (μ) to density (ρ). A kinematic viscosity value of 1.57E-05 is specified, indicating the fluid's resistance to flow.
5. *Temperature:* The temperature boundary condition is crucial for modeling heat transfer processes. A temperature value of 300 K is applied to represent the initial or inlet temperature of the fluid in the rectangular duct with Z-shaped baffles.

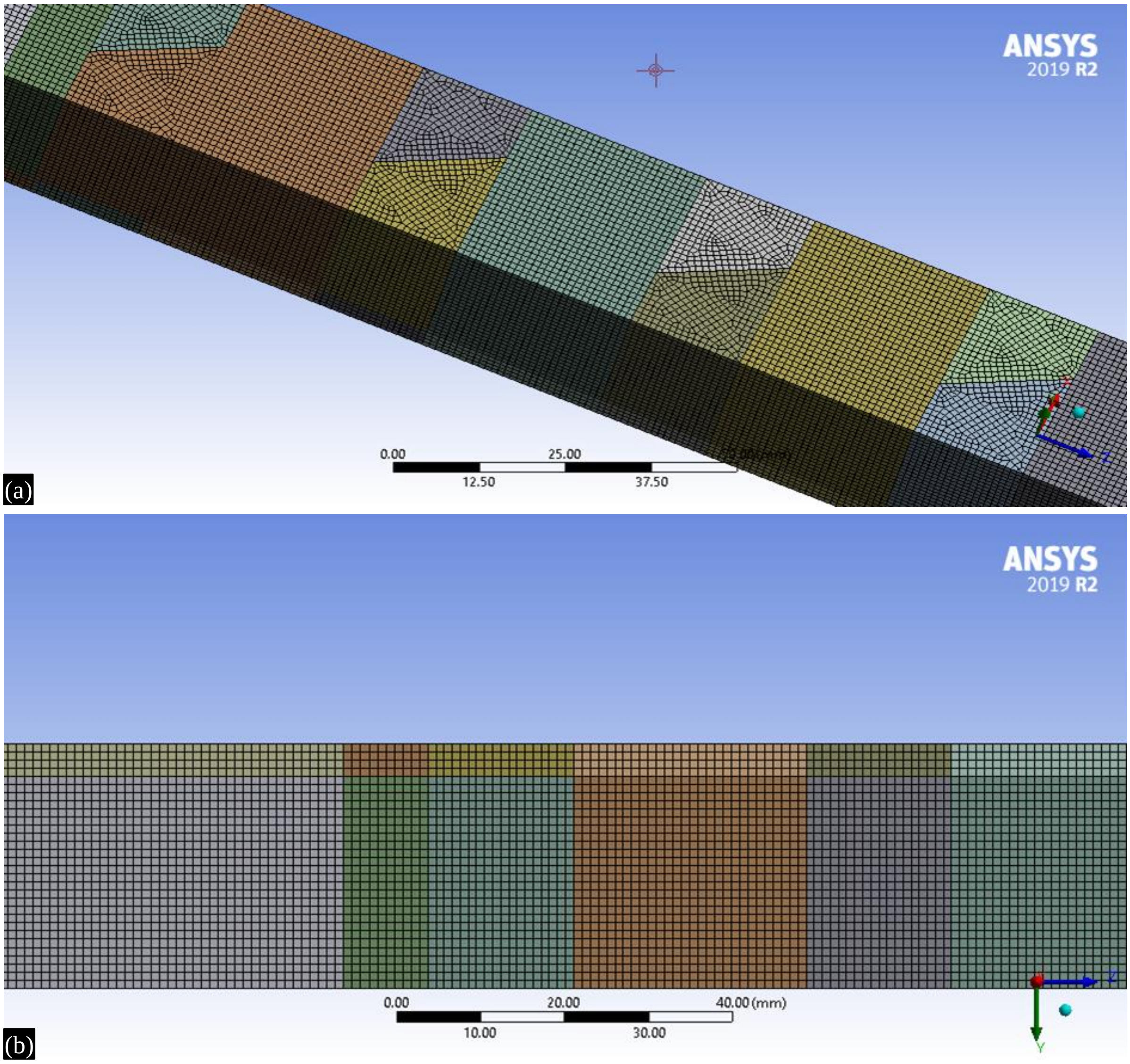


Figure 5. Hexahedral meshing in rectangular duct.

These boundary conditions, along with the geometry and mesh of the duct and baffles, allow ANSYS Fluent to simulate the fluid flow and heat transfer within the system accurately. The software uses the specified boundary conditions and solves the governing equations of fluid dynamics and heat transfer, such as the Navier-Stokes equations and energy equation, to predict the flow and temperature distribution within the duct.

Thermal Analysis

Thermal analysis of a rectangular duct with Z-shaped baffles using ANSYS Fluent involves studying heat transfer and temperature distribution. The process includes creating the geometry, generating the mesh, defining material properties, setting boundary conditions, configuring solver settings, solving the equations, and post-processing the results. Both static and periodic analysis techniques are employed.

In static analysis, the steady-state temperature distribution is examined, while periodic analysis simulates the cyclic thermal behavior. ANSYS Fluent utilizes boundary conditions, fluid properties, and numerical methods to calculate temperature distributions accurately. Post-processing tools allow visualization and analysis of temperature contours, heat flux, and velocity profiles, aiding in design optimization and evaluating heat transfer within the system.

The thermal analysis is performed in three different baffle heights of 4 mm, 6 mm and 8 mm. Each baffle height is run with varying Reynolds number from 5000 to 20000. The angle of the baffle is set to 60° (Figures 6-8).

Case 1: Analysis with 4mm baffle.

Case 2: Analysis with 6 mm baffle.

Case 3: Analysis with 8 mm baffle.

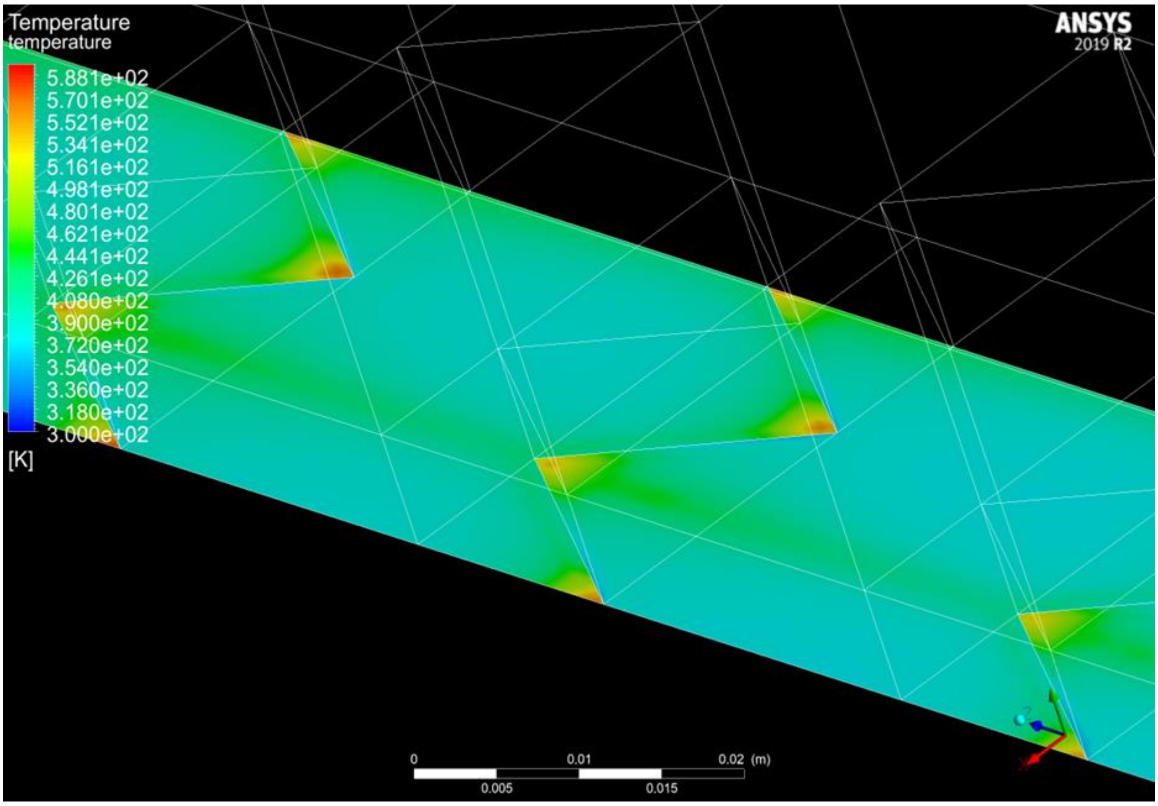


Figure 6. Temperature gradient of baffle with 4 mm height.

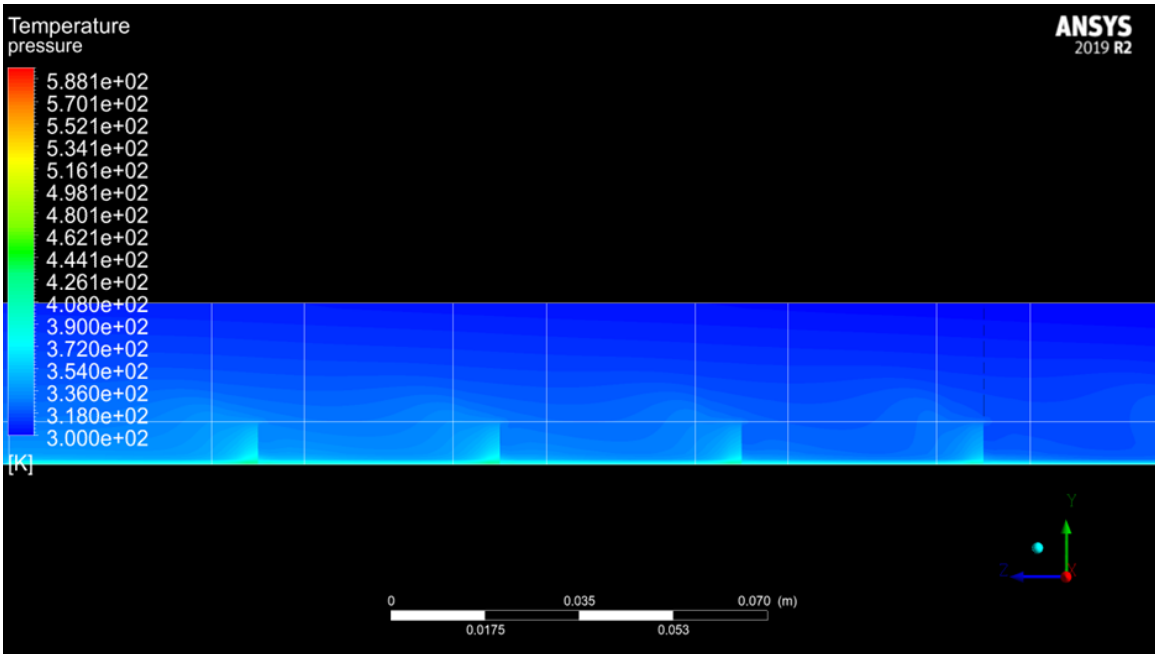


Figure 7. Temperature gradient of Baffle with 6 mm height

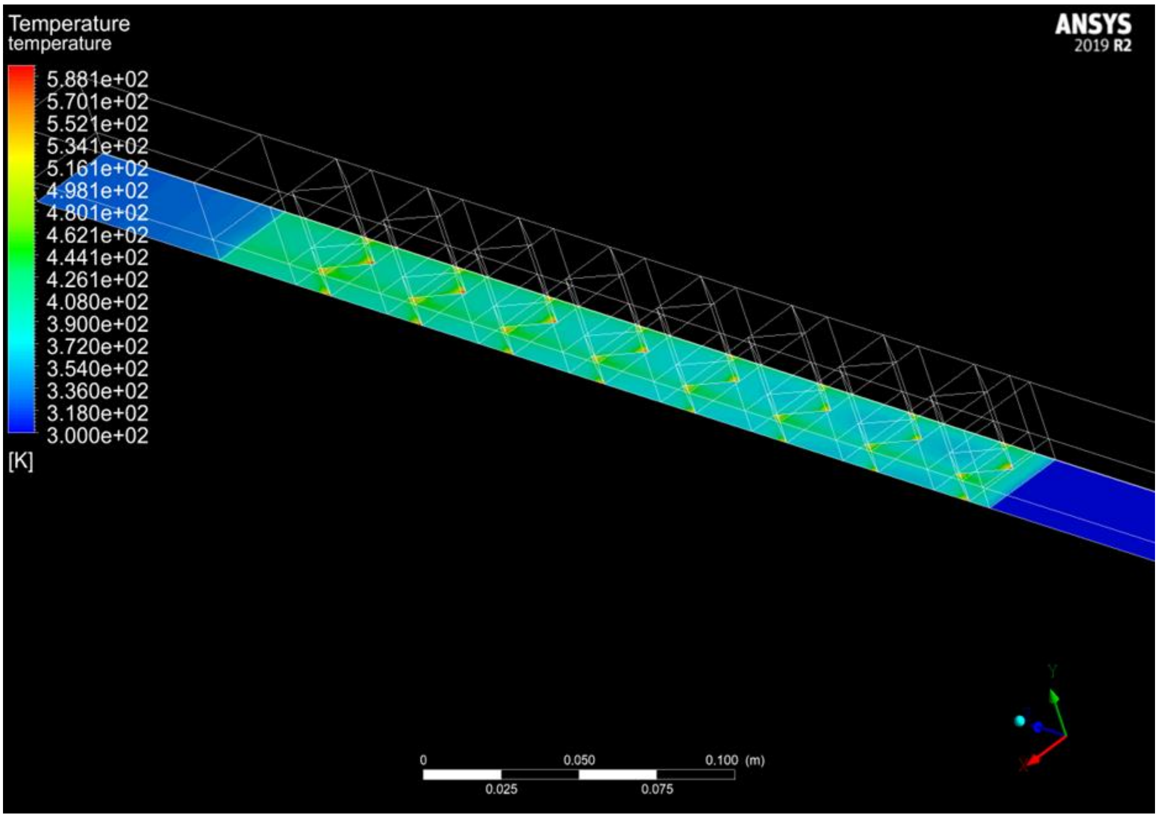


Figure 8. Temperature gradient of Baffle with 8 mm height.

RESULT

Attack Angle Analysis

The analysis to find out which attack angle was the most efficient a comparison between three different angles were conducted, the angles taken was 55°, 60° and 65° respectively. The following graph shows the difference in the Nusselt number and Friction factor in comparison with Reynolds number.

Comparison of Various Attack Angle

Different attack angles, namely 55 degrees, 60 degrees (reference angle), and 65 degrees, were analysed for their impact on heat transfer efficiency and fluid flow resistance. Nusselt Number: When considering the Nusselt number, a 55-degree attack angle showed a decrease of 4.03% compared to the reference angle of 60 degrees, while a 65-degree attack angle showed a decrease of 3.42%. This indicates that the 60-degree attack angle effectively promotes heat transfer within the system compared to the other two angles. Friction Factor: When considering the friction factor, a 55-degree attack angle exhibited a decrease of 3.97% compared to the reference angle of 60 degrees, while a 65-degree attack angle showed an increase of 3.86%. This suggests that the 55-degree angle has less resistance to flow, but it has a lower Nusselt number compared to both the 65-degree and 60-degree angles. Overall, the 60-degree attack angle was found to be the most efficient among the three angles analysed (Figures 9 and 10).

Baffle Height Analysis

We have found out the friction factor and Nusselt number for the modified baffle geometry and the journal geometry. The friction factor measures how much resistance there is to fluid flow in pipes or channels. Higher values indicate more resistance to flow. The Nusselt number describes how efficiently heat is transferred from a solid surface to a fluid. Higher values mean better heat transfer (Tables 5-7).

For 4 mm baffle:

For 6 mm baffle:

For 8 mm baffle:

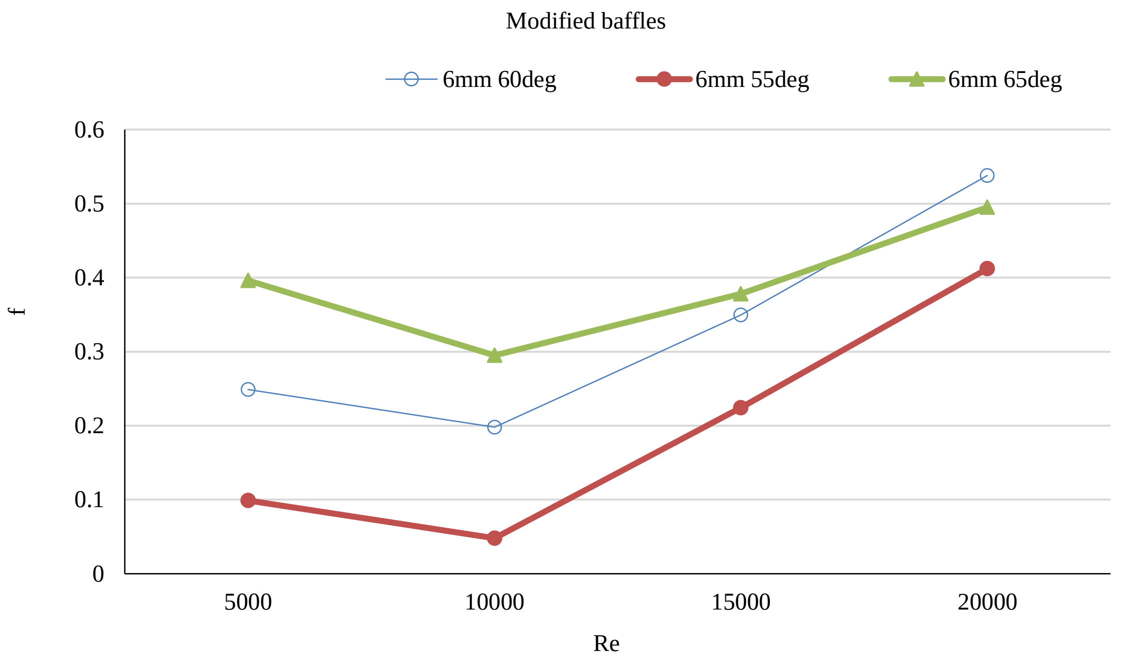


Figure 9. Friction factor vs Reynolds number graph of baffle with 6 mm height and 60°, 55° and 65° attack angles.

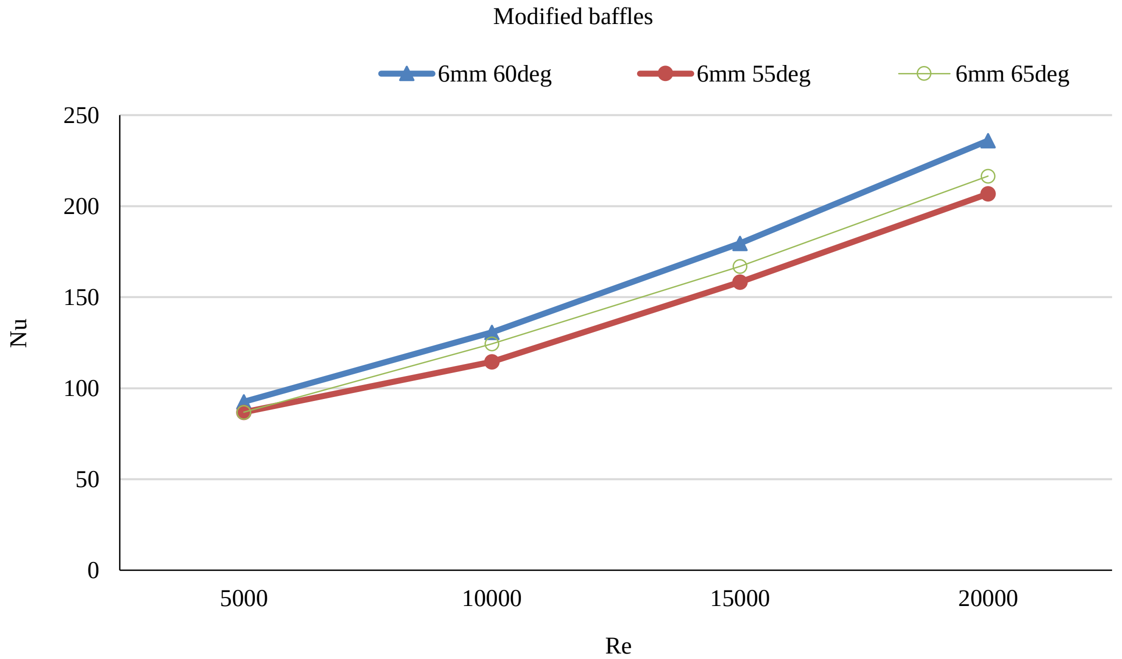


Figure 10. Nusselt Number vs Reynolds number graph of baffle with 6 mm height and 60°, 55° and 65° attack angles.

Table 5. Values of 4 mm baffle.

Velocity (m/s)	Temp outlet (K)	Pressure differential (Pa)	Nu	Friction factor
1.44	319.1941	3.1857777	73.98285	0.37473
2.88	309.6694	11.912867	96.80087	0.350316
4.31	306.4242	26.774212	116.9074	0.351553
5.75	304.7813	49.078075	137.1769	0.36206

Table 6. Values of 6 mm baffle.

Velocity (m/s)	Temp outlet (K)	Pressure differential (Pa)	Nu	Friction factor
1.44	319.0908	4.4648724	89.44641	0.525185309
2.88	309.8135	17.612833	129.1118	0.517931999
4.31	306.6246	41.048172	179.9449	0.538973959
5.75	304.9703	76.592682	236.0016	0.565041166

Table 7. Values of 8 mm baffle.

Velocity (m/s)	Temp outlet (K)	Pressure differential (Pa)	Nu	Friction factor
1.44	318.9513	5.7442279	108.3479	0.675670844
2.88	309.8044	22.799541	176.9407	0.670454994
4.31	306.6521	53.615158	255.3277	0.703981994
5.75	305.0214	99.733673	351.7815	0.735757378

Comparison

1. Nusselt Number vs Reynolds number
2. Friction Factor vs Reynolds number

Increasing the baffle height was found to result in an increase in the Nusselt number, indicating enhanced heat transfer efficiency within the duct. Specifically, at Reynolds numbers ranging from 5000 to 10000, the Nusselt number exhibited a significant increase of 2.08%. This suggests that the baffle design effectively promotes heat transfer within the system. However, it is important to note that the friction factor also increased with higher baffle heights, indicating increased resistance to fluid flow within the duct. This can have implications for system performance and energy consumption (Figures 11-14).

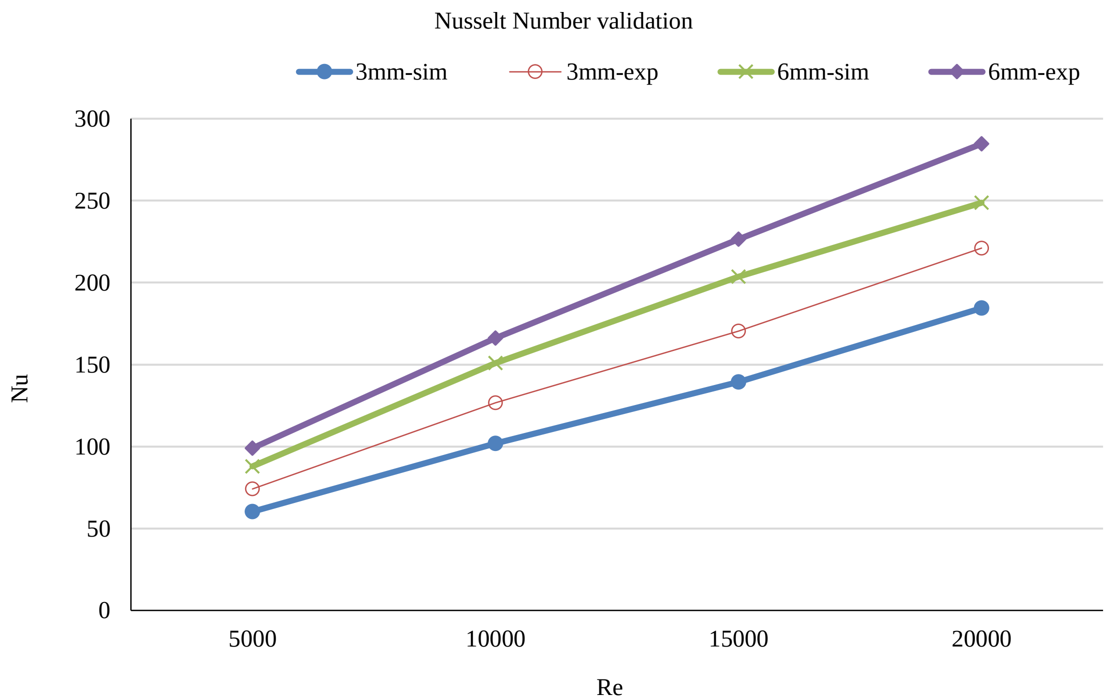


Figure 11. Nusselt number vs Reynolds number graph of journal.

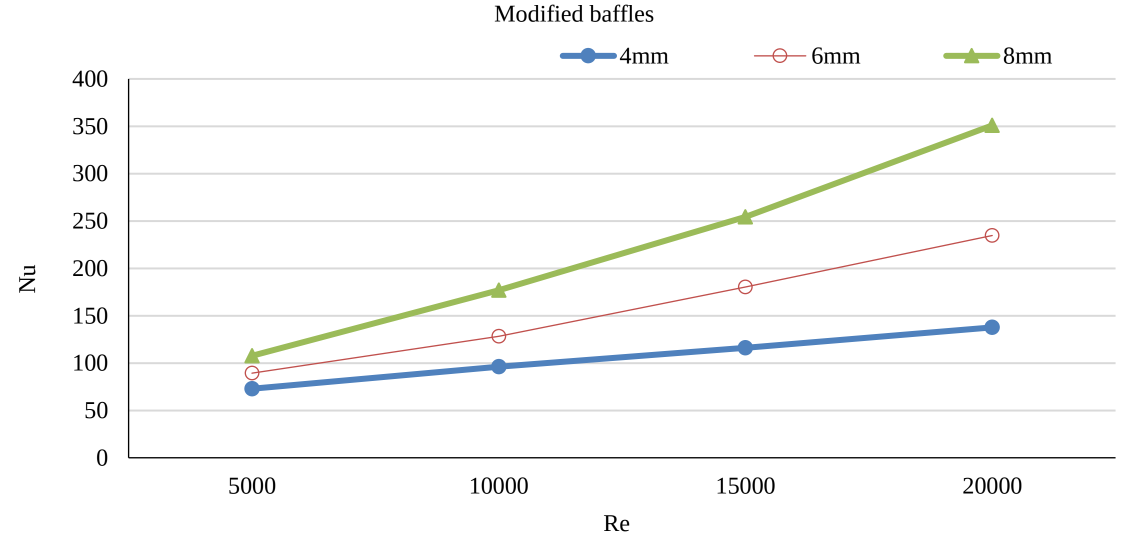


Figure 12. Nusselt number vs Reynolds number graph of modified geometry.

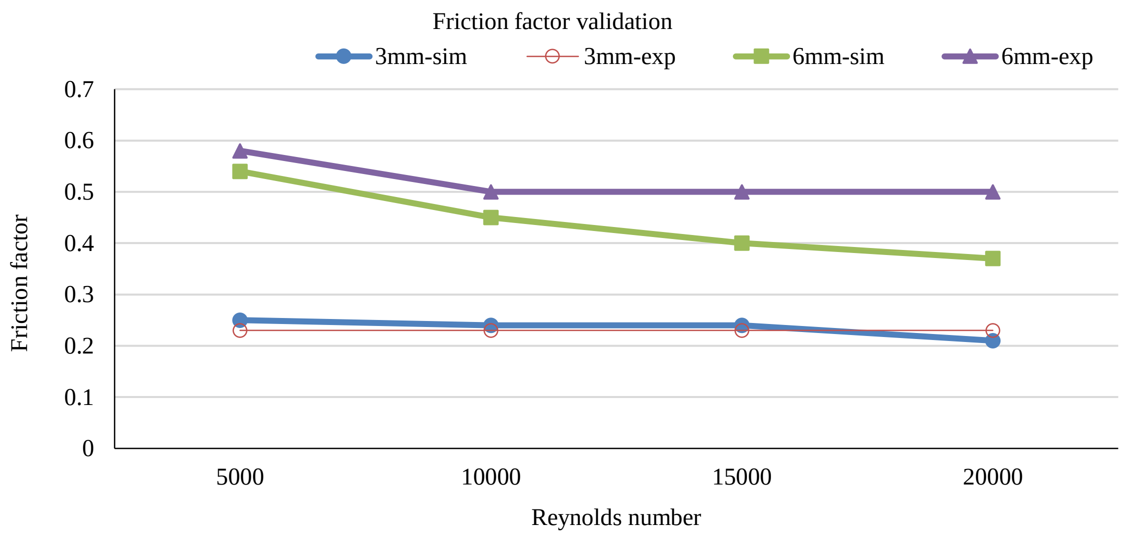


Figure 13. Friction factor vs Reynolds number graph of journal.

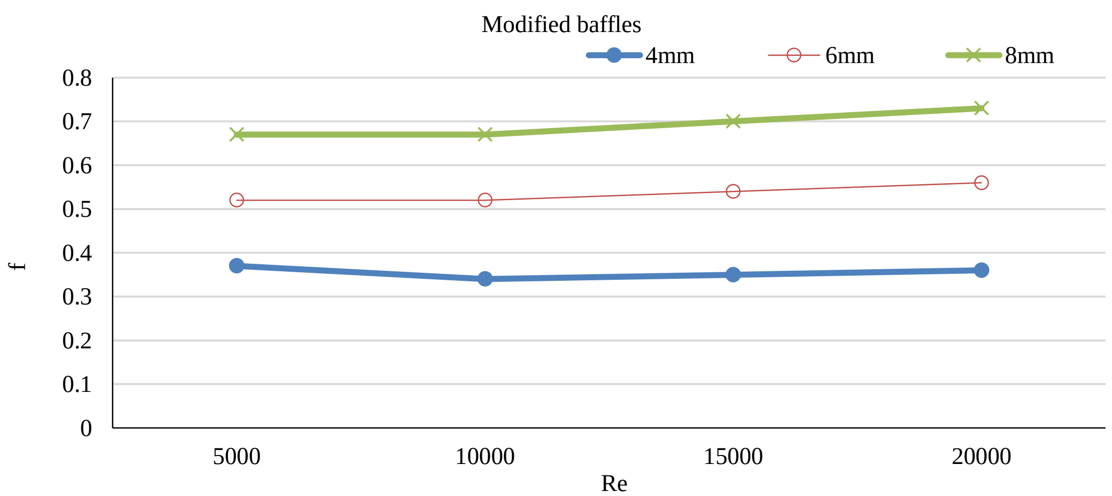


Figure 14. Friction factor vs Reynolds Number graph of modified geometry.

CONCLUSION

The baffle geometry was changed from 45° to 60° and the height of the baffle was changed from 3 mm, 6 mm and 9 mm to 4 mm, 6 mm and 8 mm. A varying Reynolds number was used from 5000 Re to 20000 Re. The thermal analysis was performed using ANSYS Fluent 2019 R2 software, while the design modifications were carried out using SolidWorks 2022 software.

One key finding is that increasing the baffle height resulted in an increase in the Nusselt number. This indicates enhanced heat transfer efficiency in the duct. Specifically, at Reynolds numbers ranging from 5000 to 10000, the Nusselt number showed a significant increase of 2.08%. This suggests that the baffle design effectively promotes heat transfer within the system. However, it is important to note that the friction factor also increased as the baffle height increased. This implies a restriction in fluid flow within the duct. The increase in friction factor indicates higher resistance to flow, which can have implications for system performance and energy consumption. Also, the choice of attack angle affects both heat transfer and flow characteristics, with the 60-degree angle demonstrating superior performance in terms of heat transfer efficiency.

During the calculations, it is worth mentioning that there was an 11% error in determining the friction factor and a 15% error in calculating the Nusselt number. These errors may arise from various factors, including assumptions made during modeling, limitations of the software used, or uncertainties in the input parameters. In conclusion, the project study and thermal analysis of the rectangular duct with a Z-shaped baffle demonstrated that increasing the baffle height leads to improved heat transfer efficiency, as indicated by the increase in the Nusselt number. However, this comes at the cost of increased friction factor and potential flow restrictions. The observed errors in calculating the friction factor and Nusselt number highlight the need for further analysis and refinement in future studies.

Future Scope

The current modified angle is apt to use between the Reynold number 5000-10000.For using the same design for higher Reynolds number, the baffles have to be modified. The possible modifications are:

- *Tapered Baffles:* Introducing a tapering effect to the baffles, where they gradually reduce in width along the flow direction, can promote better fluid distribution and mixing at higher Reynolds numbers.
- *Baffle Inclination:* Adjusting the inclination angle of the baffles can enhance flow disruption and mixing. Experimenting with different inclinations can help optimize the design for higher Reynolds numbers.
- *Adding perforations:* It can potentially help improve mixing at higher Reynolds numbers. The presence of perforations allows fluid to pass through the baffles, increasing the contact area between the fluid and the baffles. This increased contact area can enhance turbulence and promote better mixing.

REFERENCES

1. Parkpoom Sriromreun, Chinaruk Thianpong, Pongjet Promvonge "*Experimental and numerical study on heat transfer enhancement in a channel with Z-shaped baffles*". International Communications in Heat and Mass Transfer 39 (2012) 945–952.
2. Abdul-Malik Ebrahim Momin, J.S. Saini, S.C. Solanki "*Heat transfer and friction in solar air heater duct with V-shaped rib roughness on absorber plate*". International Journal of Heat and Mass Transfer 45 (2002) 3383–3396.
3. Pongjet Promvonge, Sutapat Kwankaomeng "*Periodic laminar flow and heat transfer in a channel with 45° staggered V-baffles*" International Communications in Heat and Mass Transfer 37 (2010) 841–849.
4. Dogan Engin ALNAK "*Thermohydraulic performance study of different square baffle angles in cross-corrugated channel*" Journal of Energy Storage, Volume 28, April 2020, 101295.

5. Liu XP, Niu JL. Effects of geometrical parameters on the thermohydraulic characteristics of periodic cross-corrugated channels. International Journal of Heat and Mass Transfer. 2015 May 1;84:542-9.
6. Sakr M. Convective heat transfer and pressure drop in V-corrugated channel with different phase shifts. Heat and Mass Transfer. 2015 Jan;51(1):129-41.
7. Mehta SK, Pati S. Analysis of thermo-hydraulic performance and entropy generation characteristics for laminar flow through triangular corrugated channel. Journal of Thermal Analysis and Calorimetry. 2019 Apr 15;136:49-62.
8. Mahmoud IA, Saleh MA, Mesalhy OM, Mohamed EF, Abdelatief MA. Hollow trapezoidal baffles in a rectangular channel: Thermal/hydraulic assessment with ANN numerical approach. International Communications in Heat and Mass Transfer. 2022 Dec 1;139:106505.
9. Islam MS, Saha SC. A thermo-hydraulic characteristics investigation in corrugated plate heat exchanger. Energy Procedia. 2019 Feb 1;160:597-605.
10. Choudhary A, Kumar M, Patil AK, Chamoli S. Enhanced thermal and fluid flow performance of cross flow tube bank with perforated splitter plate. Experimental Heat Transfer. 2021 Jun 7;34(4):329-41.

