International Academic Institute for Science and Technology

International Academic Journal of Science and Engineering
Vol. 6, No. 1, 2019, pp. 67-71.
ISSN 2454-3896

International Academic Journal of Science and Engineering

www.iaiest.com

Investigation of Thermal Efficiency Tubes with Porous Baffles

Meysam Saboor[a], Hamidreza Khakrah[b]

Department of Enginnering, Islamic Azad University, Yazd Branch, Yazd, Iran.
Department of Enginnering, Islamic Azad University, Shiraz Branch, Fars, Iran.

Abstract

Use of baffles in channels is a common technique for enhancing heat transfer in single-phase flows. The present study aimed to achieve a comprehensive simulation of a pipe with interior porous baffles to investigate the effects of different parameters such as baffle size, baffle porosity, and fluid properties on pressure drop, heat transfer enhancement, flow field, and temperature field. A horizontal, rectangular channel was considered, on the upper and lower walls of which staggered porous baffles were installed. Various heights and porosity levels were assumed for the baffles. The heights were in the range of $1/3D_h$ to $2/3D_h$, where $D_h$is the hydraulic diameter, and baffle porosities were 10, 20 and 40 PPI. In addition, the Reynolds number of the flow ranged from 20000 to 50000. The commercial software package ANSYS FLUENT 15 was used to solve the governing equations. The effects of different parameters including baffle heights, the Reynolds number, and porosity were investigated on flow field, temperature and pressure drop in terms of streamline diagrams, velocity vectors, and temperature as well as pressure contour lines.

Keywords: Heat transfer, baffle, channel, incompressible flow, temperature contour lines, ANSYS FLUENT

Introduction:

Heat transfer is an important issue observed in various phenomena. It is sometimes used positively to cool down flows, machines and engines. On the other hand, sometimes we are in search of ways of preventing heat transfer, e.g. in thermal insulation of hot water pipes, doors and windows. In industries, we are typically looking for different techniques to control and increase the heat transfer rate.

Use of baffles in channels is a common, passive method for increasing heat transfer in single-phase flows. Passive methods are widely used for increasing heat transfer in numerous industrial equipment including

DOI:10.9756/IAJSE/V6I1/1910006

Received: 12 March 2019/Revised: 10 April 2019/Accepted: 14 May 2019/Published: 28 June 2019

shell and tube heat exchangers, cooling of electrical equipment, thermal amplifiers, and blade cooling systems in gas turbines.

Two configurations can be assumed for baffles, namely inline and staggered. In the staggered configuration, the flow is deviated and collides against the front wall, leading to an increased washing action. The staggered pattern was used in our study [14]. The channel was rectangular with a square cross-sectional area. The spacing between baffles was considered constant, and 40 baffles were used. In order for a developed flow to enter the baffle region, a particular length of the channel was removed, along which the flow may fully develop. In addition to a steady state condition with constant fluid properties along the channel, the flow was considered single-phase and incompressible. Boundary conditions at the channel walls were constant heat flux and adiabatic in the baffle and entrance regions,

respectively. The gravity effects were neglected, rendering buoyancy forces ineffective in the heat transfer process.

The height and the porosity of baffles varied. The dimensionless height ranged from $1/3D_h$ to $2/3D_h$, where D_h is the hydraulic diameter of the channel. Porosities in the medium were 10, 20 and 40 PPI. Additionally, the Reynolds number of the flow varied between 20000 and 50000. The total number of baffles at both upper and lower walls was 40.

The commercial software package ANSYS FLUENT 15 was used to solve the governing equations, and ANSYS MESHING was used for creating meshes. In addition to constant heat flux for the channel walls, the channel was assumed horizontal, and the effects of gravity forces were neglected. The governing equations included continuity, momentum and energy equations, and a steady state, incompressible solution was adopted for the problem. Simulation results were obtained for both cases of a channel with and without baffles. The results from the baffle-free channel were used to verify the present simulation along with results from other empirical and numerical studies. The effects of different parameters including baffle size, i.e. thickness and height of the baffles, the Reynolds number, and porosity were investigated on the flow and temperature fields and ultimately on the thermal efficiency.

1- Materials and methods

ANSYS FLUENT 15 was used for simulation and solving the governing equations and Gambit was used to create the geometric shapes as well as meshes. We used the existing geometric shapes to create and mesh a rectangular channel with porous baffles. By selecting appropriate physical models and discretization, the flow and temperature fields are solved. In order to verify the results, the channel was simulated with no baffles and the results were compared with other studies. The effect of different parameters including baffle height, baffle thickness, porosity, and the Reynolds number on heat transfer was investigated.

The main objective of the present study was to calculate the heat transfer rate; therefore, the heat transfer equations were the principal equations in our theory. To this end, the fluid behavior inside a pipe should be investigated to take into consideration such conditions as flow incompressibility/stability as well as flow laminarity/turbulence. In this regard, different equations such as Bernoulli, continuity, and Navier–Stokes were used.

The main part of the problem to be solved needs to be determined first. The following steps should be taken in order in doing so:

- Determining the final model and the modeling objective.
- Creating geometric shapes and meshing.
- Determining the solver and the physical model.
- Calculating and displaying the results.

ANSYS FLUENT software package: This software is widely used in the industry due to its enormous capacity. When fluid analysis of a body is required, instead of practical experiments, which include building of the body in different sizes and placing it in a wind tunnel, significantly more accurate results

could be obtained at considerably lower costs by simulating the body in this software. ANSYS FLUENT is capable of identifying meshes with high levels of flexibility, which includes solving flow problems using unstructured meshes and complex geometries. Different identifiable meshes include 2 dimensional, e.g. triangular and quadrilateral, and 3 dimensional, e.g. tetrahedral, hexahedral, pyramid, wedge, polyhedral and hybrid. The software allows users to modify meshes according to their intended flow solutions. Once the mesh is imported into ANSYS FLUENT, the remaining operations are carried out there. The operations involve determining boundary conditions, defining fluid properties, solving process, mesh corrections and viewing results.

There are generally three main approaches for solving differential equations such as Navier-Stokes which govern physical problems, namely Finite Difference Method (FDM), Finite Element Method (FEM) and Finite Volume Method (FVM). FVM is the most recent one as compared with the other two and is used in most of today's commercial computational fluid dynamic (CFD) packages, including FLUENT, to solve differential equations. The prominent feature of this method compared to the other two is the conservation of quantities in Navier-Stokes equations, which is critical from the perspective of fluid flow phenomena. In this approach, governing equations are first integrated over the solution domain, and FDM or FEM is then used for the final discretization. The approach is capable of simulating flows in complex geometry shapes. Flow simulation in our study was carried out using this method.

2- Findings and conclusions

3-1- Investigating fluid behavior in a channel with baffles: The simulated computational domain is demonstrated. The cross-sectional area of the channel was a square with a side length of 12 mm. This length was also the hydraulic diameter. As can be seen, the computational domain was made up of 3 regions. There were no baffles in the first 120 mm of the channel. Walls were considered adiabatic in this region. In the second region, 20 baffles were installed on each of the lower and upper walls, i.e. 40 baffles in total. Baffle porosity values were considered 40, 20 and 10 percent. The heat flux ratio for all walls in this region was assumed 100000 w/m^2. There was a baffle-free region at the end of the channel whose walls were also considered adiabatic. The reason for considering adiabatic walls at the two ends of the channel was the fully developed flow in these regions. As can be seen in the demonstrated computational grid, finer grids were used near walls to better model the effects of the boundary layer. This computational grid consisted of 370000 cells.

In the first solved model, the porosity values and the Reynolds number were considered 40 percent and 20000, respectively. Baffle heights and spacing were both 1/3 H. As can be observed, porosity is inversely proportional to the Nusselt number. It should be noted that this pressure is relative, and is

measured relative to the operating pressure adjusted in the FLUENT. The operating pressure in our study was set to 101325 Pa. As is demonstrated, the pressure drops as the flow passes through the channel and reaches the minimum values of -56823, -72223 and -58358 for porosities of 10, 20 and 40 percent, respectively. In other words, the decreases in the pressure were 0.56, 0.72 and 0.58 Bar. As the porosity increases, a greater portion of the flow passes through the porous medium, causing decreased mass flux at the center of the channel. As a result, flow velocity at the center of the channel is lower than a less porous medium. In fact, decreases in porosity affect the flow in two ways, i.e. increasing both the heat transfer and the pressure drop, with the former being desirable and the latter undesirable.

3-3 Investigating the effects of the Reynolds number on heat transfer and pressure drop: In this section, the solution results of the baffle-containing channel were compared for the Reynolds numbers of 20000 and 50000, as well as for the baffle-free case, to investigate the effect of baffle on heat transfer and pressure drop in different Reynolds numbers. The diagrams demonstrate the Nusselt number for two Reynolds numbers of 20000 and 50000 in the baffle-containing channel, and for Reynolds numbers of 10000, 20000, 30000, 40000 and 50000 in the baffle-free channel. As illustrated in the figures, increases in the Reynolds number results in increases in the Nusselt number. This increase is higher in the baffle-containing channel. For the Reynolds numbers of 20000 and 50000, the results are available for both channels with and without baffles, and as they indicate, the Nusselt number and the heat transfer rate are higher in the baffle-free channel.

3-4- Investigating the effect of baffle shape on heat transfer and pressure drop: Two baffles with the heights of 1/3 H and 2/3 H were considered to investigate the effect of baffle shape in a channel. According to the results, computational domain for a baffle of 2/3 H height was 8 mm, and the diagram of the Nusselt number for two baffles of 1/3H and 2/3H heights are demonstrated in Fig. … . The results were obtained for a porosity value of 40 percent and the Reynolds number of 20000. As can be seen, the Nusselt number increases with increasing baffle heights. The diagram shows the streamlines in the channel for a baffle height of 2/3H, porosity of 40 percent and Reynolds number of 20000. The pressure drops as the flow passes through the channel, reaching -58358 and -1652127 Pa at the outlet for the heights of 1/3H and 2/3H, respectively, equaling pressure drops of 0.58 and 16.5 Bar. The pressure drop for the baffle height of 2/3H is very significant, and the use of such a baffle is not reasonable to increase heat transfer. Fig. … illustrates vector velocities on Plane 1 for a baffle of 2/3H height, Reynolds number of 20000 and porosity of 40 percent. Changes in the flow pattern can clearly be observed as the baffle height changes.

3- Conclusion and results

In the present study, use of baffles in channels for increasing heat transfer was investigated. Repeated baffle configurations in channels changed the flow pattern. Traveling the distance between the two baffles, the flow was separated, underwent a rotation, formed a vortex and collided against the walls. The collision as well as fin-like behavior of the baffle led to heat transfer enhancement and pressure drop. The process took place by creating disturbances in the hydrodynamic and thermal boundary layers. The effects of porosity on heat transfer and pressure drop in a channel were investigated. It was observed that decreasing porosity of the porous medium resulted in heat transfer enhancement as well as pressure drop. It is suggested that porosity be decreased to a degree at which the porous baffle does not display a similar behavior to a solid baffle, since this may cause significant increases in pressure drop. In addition to increasing the heat transfer rate, use of porous baffles maintains pressure drop at its minimum. The effect of variations in the Reynolds number was also investigated on the heat transfer. Increased Reynolds numbers resulted in increased velocities at the center of the channel. In fact, increased Reynolds numbers

affect the flow in two ways: increasing the heat transfer rate and causing major pressure drops in baffle-containing channels. The effects of changes in the baffle shape were also investigated. By increasing baffle heights, heat transfer increased as well; however, the use of such baffles is not recommended as it causes major pressure drops.

5- **References**

[1] A. A. Mohamad, "Heat transfer enhancements in heat exchangers fitted with porous media Part I: constant wall temperature," Int. J. Therm. Sci., vol. 42 (4), pp. 385-395, 2003.

[2] Z. F. Huang, A. Nakayama, K. Yang, C. Yang, and W. Liu, "Enhancing heat transfer in the core flow by using porous medium insert in a tube," Int. J. Heat Mass Transfer, vol. 53 (5-6), pp. 1164-1174, 2010.

[3] B. I. Pavel and A. A. Mohamad, "An experimental and numerical study on heat transfer enhancement for gas heat exchangers fitted with porous media," Int. J. Heat Mass Transfer, vol. 47 (23), pp. 4939-4952, 2004.

[4] Y.-T. Yang and M.-L. Hwang, "Numerical simulation of turbulent fluid flow and heat transfer characteristics in heat exchangers fitted with porous media," Int. J. Heat Mass Transfer, vol. 52 (13-14), pp. 2956-2965, 2009.

[5] Y. T.-. Yang and C.-Z. Hwang, "Calculation of turbulent flow and heat transfer in a porous-baffled channel," Int. J. Heat Mass Transfer, vol. 46, pp. 771-780, 2003.

[6] P. MG and G. SM, "Convective heat transfer and thermal dispersion duirng laminar pulsating flow in porous media," Int J Therm Sci, vol. 50, pp. 440-8, 2011.

[7] P. MT and O. M, "Heat transfer in porous media of steel balls under oscillating flow," Exp Therm Fluid Sci, vol. 42, pp. 79-92, 2012.

[8] M. A. Teamah, W. M. El-Maghlany, and M. M. K. Dawood, "Numerical simulation of laminar forced convection in horizontal pipe partially or completely filled with porous material," Int. J. Therm. Sci., vol. 8, pp. 1512-1522, 2011.

[9] T. V. Morosuk, "Entropy generation in conduits filled with porous medium totally and partially," Int. J. Heat Mass Transfer, vol. 48 (12), pp. 2548-2560, 2005.

[10] M. Mahdavi, M. S. -Avval, S. Tiari, and Z. Mansoori, "Entropy generation and heat transfer numerical analysis in pipes partially filled with porous medium," Int. J. Heat Mass Transfer, vol. 79, pp. 496-506, 2014.

[14] Heat Transfer: A Practical Approach Hardcover – November 1, 2002 by Yunus A. Cengel (Author)