IJIST

International Journal
of
Integrated Sciences & Technology (IJIST)

http://www.cuet.ac.bd/IJIST/index.html

International Journal of Integrated Sciences & Technology **1** (2015) 53-57

Numerical Simulation of The Heat Transfer Phenomenon for a Fluid in a Circular Tube without Insert and with Insert

S. Hossain[1]*, U. K. Deb[2] and K. A. Rahman[1]

[1]Department of Mechanical Engineering, [2]Deparment of Mathematics, Chittagong University of Engineering and Technology Chittagong 4349, Bangladesh

Abstract

In this study heat and mass transfer phenomena for a fluid in a circular tube has been investigated. We simulate the fluid flow and the temperature distributions by putting inserts and with no insert for a non-isothermal laminar flow. A circular 800mm long pipe with 26 mm inner diameter is considered in our simulation. We have used four rectangular boxes (25mm×5mm×5mm) as inserts which are fitted at equal distance from the leading edge perpendicular to the fluid flow. The inserts are being arranged horizontally and vertically along the length of the tube respectively. A constant heat flux is generated at the boundary of the tube close to the outer wall of the water domain. It is observed more vortices in the fluid flow after using inserts comparing with the domain without insert. The heat transfer rate is also increased after using inserts inside the tube.

Keywords: Mass Transfer; Heat Transfer; CFD; Simulation.

1. Introduction

The heat transfer enhancement technology has been progressed and widely applied to the heat exchanger applications including refrigeration, automobiles, power plants, process industries etc. A large number of attempts have been made to minimize the size and the costs of the heat exchangers. The heat transfer augmentation techniques play a vital role for laminar flow since the heat transfer coefficient is normally low in plain tubes [1, 2]. A number of experiments related to the heat transfer enhancement have performed in the heat transfer laboratory is in mechanical engineering department. Bergles developed an exhaustive survey on heat transfer enhancement by various techniques. Among many techniques examined for augmentation of heat transfer rates inside the circular tubes, the tube fitted with full length twisted tape inserts using water has been shown to be very effective[3, 4]. Furthermore, a good number of literature are available on computational fluid dynamics modeling of the heat transfer and convective heat transfer augmentation process using different types of inserts.

The important attempt on utilizing different methods is to increase the heat transfer rate by using different kinds of inserts. Meanwhile, it is found that this way can reduce the manufacturing cost and save up the energy. Experimental investigation was performed to study the fluid flow and the heat transfer enhancement in a tube with rectangular cut twisted tape inserts [5]. The heat transfer enhancement efficiencies were increased with the increase of Reynolds number and found to be in the range of 1.9 to 2.3. An experimental investigation of heat transfer and friction factor was presented of a smooth tube for the laminar flow [6]. The maximum improvement of the Nusselt number obtained in the range of 50% to 100% for the aluminium tape, 40% to 94% for stainless steel tape, 40% to 67% for the insulated tape. By using twisted tape and wire coil in heat transfer enhancement for determining the Nusselt number, the friction factor, the thermal enhancement index had been investigated experimentally [7].

The Finite Element Method (FEM) software is used for numerical analysis which is stressed and analyzed for approximate results. It is used for existing product refinement and new product design. The finite element based simulation gives the more accurate results by discretizing the domain into finite number of elements. The technique developed in heat transfer technology by using FEM software becomes popular in recent years [8].

In case of the laminar flow, heat is transferred by

* Corresponding author. Mobile: +880-1674932993
E-mail: sabbirhossain.leads@gmail.com
For color version visit:http://www.cuet.ac.bd/IJIST/index.html

ISSN : 0000-0000

conduction, convection and molecular diffusion since there is no cross-mixing of fluids. Also, a small amount of heat is transferred by the fluid in the laminar flow [9, 10]. Therefore, the augmentation of the heat transfer enhancement is necessary by using different kinds of inserts. In this simulation, we have shown a technique of the heat transfer enhancement in a circular tube by using rectangular boxes inside the tube perpendicular to the flow of the fluid considering the model as non-isothermal flow. The water domain and the tube domain of copper material are taken into account for this simulation. It is a two-way coupling problem between the fluid flow and the heat transfer. A constant heat flux is generated at the wall of the tube.

The rest of our paper is organized as follows. In section 2, the mathematical model and the mesh design of the domain is presented. In section 3, the numerical results for this model are shown. Finally, concluding remarks are presented in section 4.

2. Mathematical Model

The mathematical model of this study describes the behavior of the flow in a circular tube along the length. To predict the dynamics behavior inside the circular tube Computational Fluid Dynamics (CFD) holds the greatest potential for long term application. By using computer and numerical method, CFD solve problems involving the movement of fluid inside the reactor as well as a prediction of the fluid dynamics and the related physical phenomena can be determined.

2.1 Governing Equations

In the simulation, we have chosen the physics as non-isothermal laminar flow. In this case, heat flows by conduction, convection and via motion of the medium of interest is a flowing fluid referred as mass transport which is considered here for the one-dimensional case. The governing equations which describe the flow are given as follows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \quad (1)$$

$$\rho \mathbf{v} \cdot \nabla \mathbf{v} + \rho \frac{\partial \mathbf{v}}{\partial t} = -\nabla p + \nabla \cdot \left(\mu (\nabla \mathbf{v} + \nabla \mathbf{v})^{T} - \frac{2}{3} \mu (\nabla \cdot \mathbf{v}) I \right) + \mathbf{F} \quad (2)$$

where, $\mathbf{v}$ is the velocity vector (m/s), ρ is the density (kg/m^3), p is the pressure (Pa), **F** is the body force vector (N/m^3), μ is the dynamic viscosity (Pa•s). It also solves the heat transfer through a fluid and governed by the following equation:

$$\rho C_p \left(\frac{\partial T}{\partial t} + (\mathbf{v} \cdot \nabla) T \right) = \boldsymbol{\tau} \cdot S - (\nabla \cdot \mathbf{q}) - \frac{T}{\rho} \frac{\partial \rho}{\partial T} \mid p \left(\frac{\partial \rho}{\partial t} + (\mathbf{v}.\nabla) p \right) + Q \quad (3)$$

where, C p is the specific heat capacity at constant pressure (J/(kg•K)), T is absolute temperature (K), q is the heat flux by conduction (W/m^2), τ is the viscous stress tensor (Pa), S is the strain-rate tensor (1/s), Q is the amount of heat.

The pressure work term is

$\frac{T}{\rho} \frac{\partial \rho}{\partial T} \mid p(\frac{\partial \rho}{\partial t} + (\mathbf{v} \cdot \nabla) p)$ and the viscous heating term τ·s are not included by default as they are commonly negligible. The interface also describes the heat transfer in solids:

$$\rho C_p \frac{\partial T}{\partial t} = -(\nabla \cdot q) - T \frac{\partial E}{\partial t} + Q \quad (4)$$

where, E is the elastic contribution to entropy [J/(m^3•K)]. In the case of fluids, the pressure work term is $T\frac{\partial E}{\partial t}$.

2.2 Boundary conditions

In the simulation, the boundary conditions are assumed as a uniform flow velocity i.e $u = u_{in}$ = 0.08 m/s at the inlet, no slip condition i.e u=0 at the inner wall of the tube and zero normal stress at the outlet of the domain governed by the following equation,

$$\left[-p\mathbf{I} + \mu \left(\nabla \mathbf{u} + (\nabla \mathbf{u})^{T} - \frac{2}{3} \mu (\nabla \mathbf{u}) \mathbf{I} \right) \right] \mathbf{n} = -f_0 \mathbf{n} \quad (5)$$

Constant heat flux is subjected of 32087 W/m^2 to the wall of the tube closes to the water domain, inlet water of the fluid is $T=T_{in}$=293.15 K

2.3 Computational Domain and Mesh Analysis

At the outset, we have designed the domain combinations such as without insert and with four inserts. The length of the pipe is assumed 800 mm with 26 mm inner diameter. The inserts are fitted maintaining equal distance along the length. As the geometry of the domain is more complex, the configuration of the computer is also an important factor in simulation. We have chosen coarse mesh for water domain and fine mesh for tube domain. Therefore, the mesh element of the domain with four inserts domain generally more than the domain without insert. For without insert, the number of edge elements, boundary elements and elements are 900, 8100 and 25438 for the water domain and 1600, 25584, 37147 for the tube domain respectively. For with inserts, the number of edge elements, boundary elements and elements are 900, 8100, 25438 for water domain and 1824, 29792, 39441 for tube domain respectively. A transparent view of the computational domain for both the tube and water with inserts is shown in Fig.2.1(a). The inserts are placed equally with each other as depicted in this figur. The mesh design of the tube domain is shown Fig.2.1(b) and the mesh design for the water domain is shown in Fig.2.1(c).

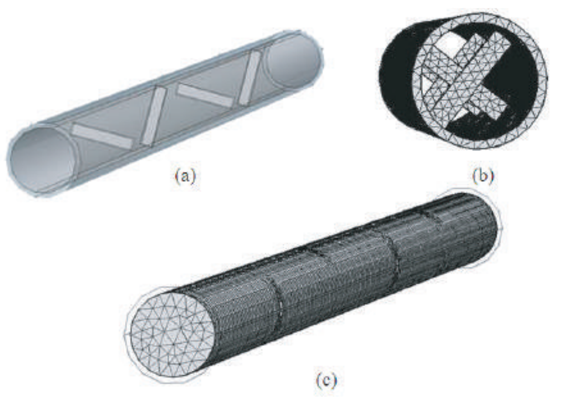

Fig.2.1: (a) Computational domain with inserts; (b) mesh design for the tube; (c) mesh design for water.

3. Results and Discussion

In the study, we run our simulation using COMSOL multiphysics version 4.2a. The aim of this study is to observe the heat transfer phenomenon of a fluid through a circular tube for the non-isothermal laminar flow. The resuts of CFD analysis accomplished with the flow phenomenon which causes an effect on the heat teransfer by putting inserts inside the tube normal to the flow direction. The function of the insert is to separate the fluid layer of the laminar flow close to the inner wall of the tube which results increasing the heat transfer. For this purpose, differents characteristics of the flow e.g temperature distribution, velocity field and shear rate have been observed.

There are three cross-sectional planes are taken to represent the temperature distribution at 150 mm, 330 mm, 490 mm along the pipe. From Fig.3.1, it is observed that the heat transfer is divided into five different layers of water and the distribution remains almost similar along the pipe. The temperature of the layer close to the wall of the tube becomes higher. As, the simulation has been performed for laminar flow, the water particles do not change their position due to layer to layer friction while carrying heat. As a result, heat does not transfer to the water particles layer at center. Four Inserts have been fitted at 160mm, 320 mm, 480 mm and 640 mm from the leading edge to overcome this problem. The temperature distribution plot for with inserts has been shown in Fig 3.2. The plots were drawn at the same positions like without insert. The plot at 150 mm that is just before the insert has shown similar heat transfer characteristics as without inserts.

In the plot [Fig 3.2.(b)] at 330 mm the flow characteristic has been changed after the first insert which is arranged horizontally. In this case, the fluid particles is scattered in different direction and there is no fixed layer in the fluid. Therefore, most of the fluids particles are able to carry heat from the tube wall which causes maximum heat transfer. In Fig 3.2 (c), the flow characteristic has been shown at 490 mm after second insert which is fitted vertically.

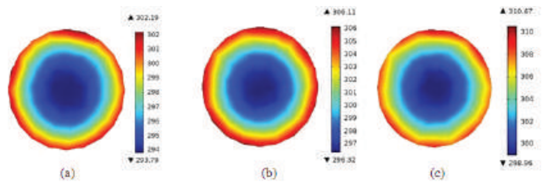

Fig 3.1: Temperature distribution plots for without insert at (a) 150 mm, (b) 330 mm, (c) 490 mm.

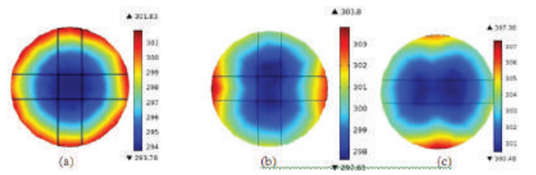

Fig 3.2: Temperature distribution (Cross-sectional) for with inserts at (a) 150 mm, (b) 330 mm, (c) 490 mm.

The streamline of water flow with temperature distribution has shown in Fig 3.3(a) for without insert. It is observed that, the direction of fluid flow does not change along the pipe and the final temperature at the outlet is 313.62 K. The streamline of the water flow with temperature distribution has also shown in Fig 3.3 (b) for the domain using inserts. The streamline has changed its direction and created vortices at the inserts positions which causes the increase of outlet water to 315.46 K.

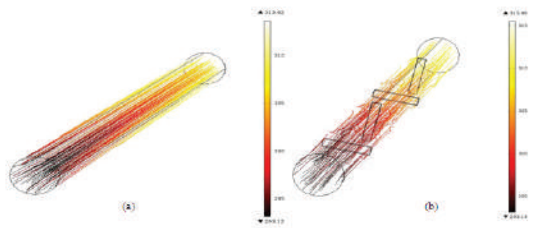

Fig 3.3: Streamline plot of temperature distribution for (a) without insert; (b) with insert.

Fig 3.3: Streamline plot of temperature distribution for (a) without insert; (b) with insert.

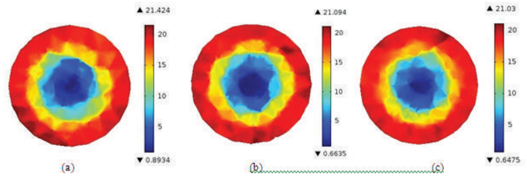

Fig 3.4: Shear rate (1/s) plots for without insert (a) 150 mm; (b) 330 mm; (c) 490 mm.

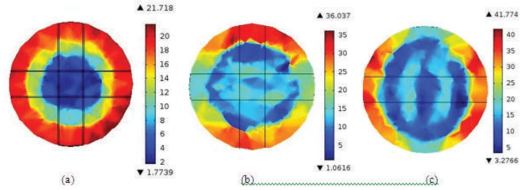

Fig 3.5: Shear rate (1/s) plots for with insert (a) before insert at 150 mm; after insert at (b) 330 mm and (c) 490 mm.

In Fig 3.4, the shear rate of different layer of the flow has shown for without insert. The layer of the fluid close to the tube wall is represented by red color shows higher shear rate region. In this simulation, we have used laminar flow which is introduced high shear rate among the fluid layer. The fluid particles move in a layer and the interaction among the layers are negligible, a small amount of heat is transferred from one layer to another. After using inserts, the fluid layers are broken down and the shear rate among the layers is decreased. In Fig 3.5 (b), it is noticed that the shear rate close to the wall decreases that represented by blue color region while the red color region represents high shear rate which is very small.

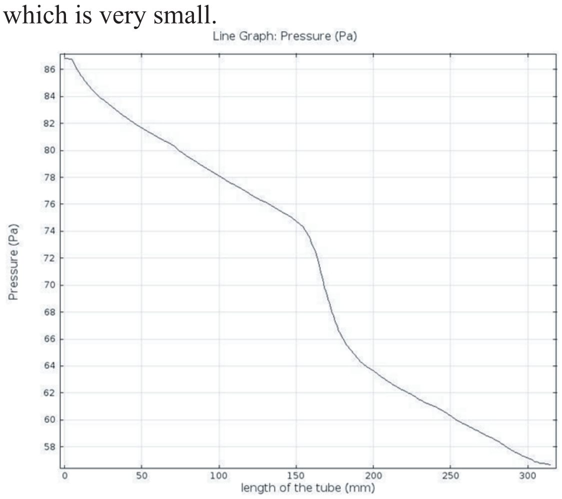

Fig 3.6: Pressure distribution profile.

The pressure distributions plot has shown in above Fig 3.6. The plot is drawn for pressure distribution along the tube length. In our geometry, the position of the inserts at 160 mm and a non-uniform pressure drop is observed from the inlet to the outlet of the tube. It is also notable that around the insert positions a sharp pressure drop is found.

4. Conclusion

Numerical simulation of heat transfer phenomenon of a fluid flowing through a circular tube with insert and without insert for a non-isothermal laminar flow has been presented. The temperature distributions, the velocity distribution, the shear rate distributions, the pressure drop at insert positions have been shown. A comparison of the effect of heat transfer phenomenon without insert and after using the inserts is also depicted. A remarkable change of the heat transfer rate, the shear rate is obtained after using the inserts. The regular sharp pressure drop is formed around the insert positions. In manufacturing, this technique of heat transfer enhancement should consider in the applications of such thermal processes in molding, mixing, curing, and drying. Also, in vehicle design, studies of thermal analysis can be applied by finite element methods to many components from brakes to exhaust systems.

Acknowledgements

The authors gratefully acknowledges for the technical support to Prof. Dr. Benchawan Wiwatanapataphee, Department of Mathematics, Mahidol University, Bangkok-10400, Thailand and The Simulation Lab, Department of Mathematics, Chittagong University of Engineering and Technology, Chittagong-4349.

References

[1] D. V. Hutton. "Fundamental of Finite Element Analysis (1st Edition), The McGraw Hill Companies".
[2] Y. A. Çengel. "Heat transfer and Mass Transfer".
[3] J. P. Hartnett, W. M. Rohsenow, and E. N. Ganic, Eds., chapter 1, McGraw-Hill, New York, NY, USA, 2nd edition, (1985).
[4] R. J. Yadav, A. S. Paalkar. "CFD Analysis for Heat Transfer Enhancement inside a Circular Tube with Half-Length Upstream and Half-Length Downstream Twisted Tape".
[5] B. Salam, S. Biswas, S. Saha and M. M. K. Bhuiya, 56, 96 (2013).

[6] A. V. N. Kapatkar, B. Dr. A. S Padalkar, C. Sanjay Kasbe, 1, 1 (2011).
[7] A. G. Matani, S. A. Dahake."Experimental study of heat transfer enhancement in a pipe using twisted tapes".
[8] J. N. Ready, "An introduction to the Finite Element Method", Third Edition, McGraw-Hill Education Private Limited.
[9] John H. Lienhard IV, John H. Lienhard V. "A Heat Transfer Text Book (Third Edition)".
[10] Frank P. Incoropera, David P. DeWitt, Bergman, Lavine. "Fundamentals of Heat and Mass Transfer (5th Edition)".