International Journal of Thermal Technologies
E-ISSN 2277 – 4114

Available at http://inpressco.com/category/ijtt/

Research Article

Comparison of Heat Transfer between a Circular and Rectangular Tube Heat Exchanger by using Ansys Fluent

Priyanka Bisht[Ȧ*], Manish Joshi[Ȧ] and Anirudh Gupta[Ȧ]

[Ȧ]Department of Mechanical Engineering, Bipin Tripathi Kumaon Institute of Technology, Dwarahat, Almora, Uttarakhand (India) 263653

Accepted 10 June 2014, Available online 30 June 2014, **Vol.4, No.2 (June 2014)**

Abstract

Heat exchangers are the important equipments with a variety of industrial applications including power plants, chemical, refrigeration and air conditioning industries. Circular tube heat exchangers are used in order to obtain a large heat transfer area per unit volume and to enhance the heat transfer coefficient on the inside surface. This paper deals with the CFD simulation of circular tube heat exchanger used for cooling water under constant wall temperature conditions. CFD results are compared with the results obtained by the simulation of rectangular tube heat exchanger of the same length under identical operating conditions. The results are validated by the results obtained by the numerical correlations used by different researchers. Results indicated that circular pipe heat exchangers showed 2.5% increase in the heat transfer rate over the rectangular tube. Simulation results also showed 8.5% increase in Nusselt number for the circular tube whereas pressure drop in case of circular tube is higher when compared to the rectangular tube.

Keywords: *Heat Exchanger, Ansys Fluent etc.*

1. Introduction

The analysis of pipe flow is very important in engineering point of view due to rigorous engineering application and implications dealt with it. In recent years considerable emphasis has been placed on the development of various augmented heat transfer surfaces and devices. Energy and material saving considerations, space considerations as well as economic incentives have led to the increased efforts aimed at producing more efficient heat exchanger equipment through the augmentation of heat transfer. When the performance of heat exchanger is enhanced, the heat transfer improvement enables the size of the heat exchanger to be decreased. Flow of real fluid exhibits viscous effects in pipe flow, here this effect is identified for turbulent flow condition. Also, due to the wide range of applications, In the study of thermodynamics the average heat transfer coefficient, is used in calculating the convection heat transfer between a moving fluid and a solid. This is the single most important factor for evaluating convective heat loss or gain. Knowledge of h is necessary for heat transfer design and calculation and is widely used in manufacturing processes, oil and gas flow processes and air-conditioning and refrigeration systems. The heat transfer coefficient is critical for designing and developing better flow process control resulting in reduced energy consumption and enhanced energy conservation. It is also influenced by flow velocity and surface geometry. It may be noted that the physical or thermal properties of the surface material play no part in the process of convective heat transfer. As the fluid properties vary with the temperature and locations the value of convective heat transfer coefficient vary from

Point to point, this leads to the situation that analytically derived equation are applicable only to a limited extent. This paper has three objectives the first objective is to present a general approach for accurately predicting the nusselt number in circular and non circular duct for fully developed flows. Secondly demonstrate that the square root of cross sectional flow area as an alternative to the hydraulic diameter leads to better correlation of the result in non circular duct. Finally developed a comparison between the circular and non circular duct which will predict the result for the uniform wall temperature condition.

2. Literature Review

Result for other common duct configuration encountered in heat transfer problem, namely the parallel plate, circular duct, rectangular duct geometries may also be found in literature. These geometries have received much attention in primarily as a result of their case to be solved using analytic technique many common geometries which have been analyzed for fully developed and thermally developing flow condition. Thermally fully developed and thermally developing laminar flow heat transfer in circular duct is discussed in most heat transfer texts (Incopera, 1990; Bejan, 1993) and in all convective heat transfer texts (Burmeister, 1993; Bejan, 1995). A comprehension review of this problem was compiled by (Shah and London, 1978)

*Corresponding author **Priyanka Bisht** and **Manish Joshi** are M.Tech Scholars; **Dr. Anirudh Gupta** is working as Associate Professor

while shorter reviewer has appeared in handbooks (John wiley and sons, 1987). (Gessner and Jones, 1976) examined the turbulent flow in square cross section they conducted a series of experiments using hotwire anemometry to analyze fully developed turbulent flow in a square duct at a Reynolds number of 150,000. They also carried out computations by a finite difference method with an algebraic stress model to predict qualitatively the major feature of the flow field namely, eight-vortex secondary flow structure. (Melling and Whitelaw, 1976) performed detailed experiments for fully-developed flow using laser- droppler anemometry and where the first to describe the axial velocity field and the Reynolds stress distribution in detail. (Nakayama et al, 1983) on the other hand, analyzed the fully-developed flow field in ducts of rectangular and trapezoidal cross-sections computationally using a finite-difference method based on the algebraic turbulence stress model of (Launder and Ying, 1972). They were able to obtain a flow field in good agreement with the available experimental measurements for a number of selected cross-sections. Improved calculations were conducted by (Gessner and Po, 1977) and (De Muren and Rodi, 1984) using the nonlinear algebraic stress model of Rodi.

3. Mathematical Formulation

The system consists of water flow moving through a circular and rectangular channel. The geometric model of the circular tube and rectangular tube were constructed using workbench in ANSYS 14 environment. In order to numerically establish the heat transfer coefficient of circular tube heat exchanger the parameters were assumed to be same that of rectangular tube. Tube diameter was considered to be 0.015 m and length considered was 3 m. The three dimensional computational domain modeled using quad mesh for both models are as shown in fig. 3.1. The flow is assumed to be steady and turbulent. In this numerical investigation, the following hypotheses are adopted.

(i) Physical properties of water are constant.
(ii) A profile of velocity is uniform at the inlet.
(iii) The radiation heat transfer is negligible.
(iv) The flow is assumed to be steady.

4. Governing Equation

4.1. Continuity Equation

Continuity Equation also called conservation of mass. Consider fluid moves from point 1 to point 2. The overall mass balance is Input – output = accumulation. Assuming that there is no storage the Mass input = mass output. However, as long as the flow is steady (time-invariant), within this tube, since, mass cannot be created or destroyed then the above equation. According to continuity equation, the amount of fluid entering in certain volume leaves that volume or remains there and according to momentum equation tells about the balance of the momentum. The momentum equations are sometimes also referred as Navier-Stokes (NS) equation. They are most commonly used mathematical equations to describe flow. The simulation is done based on the NS equations and then K-Epsilon model.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho v_x) + \frac{\partial}{\partial y}(\rho v_y) + \frac{\partial}{\partial z}(\rho v_z) = 0$$

4.2. Kappa-Epsilon Model

The K-epsilon model is most commonly used to describe the behavior of turbulent flows. It was proposed by A.N Kolmogrov in 1942, then modified by Harlow and Nakayama and produced K-ε model for turbulence. The Transport Equations for K-ε model are for k, Realizable k-ε model and RNG k-ε model are some other variants of K-ε model. K-ε model has solution in some special cases. K-ε model is only useful in regions with turbulent, high Reynolds number flow.

K **Equation**

$$\rho[\bar{u}\tfrac{\partial k}{\partial x} + \bar{v}\tfrac{\partial k}{\partial r}] = \tfrac{\partial}{\partial x}[(\mu_l + \tfrac{\mu_t}{\sigma_k})\tfrac{\partial k}{\partial x}] + \tfrac{1}{r}\tfrac{\partial}{\partial r}[r(\mu_l + \tfrac{\mu_t}{\sigma_k})\tfrac{\partial k}{\partial r}] + \rho g - \rho\varepsilon$$

Where, G is the production term and is given by

$$G = \mu_t[2\{(\tfrac{\partial \bar{v}}{\partial r})^2 + (\tfrac{\partial \bar{u}}{\partial x})^2 + (\tfrac{\bar{v}}{r})^2\} + (\tfrac{\partial \bar{u}}{\partial r} + \tfrac{\partial \bar{v}}{\partial x})^2]$$

The production term represents the transfer of kinetic energy from the mean flow to the turbulent motion through the interaction between the turbulent fluctuations and the mean flow velocity gradients.

ε **Equation**

$$\rho[\bar{u}\tfrac{\partial \varepsilon}{\partial x} + \bar{v}\tfrac{\partial \varepsilon}{\partial r}] = \tfrac{\partial}{\partial x}[(\mu_l + \tfrac{\mu_t}{\sigma_\varepsilon})\tfrac{\partial \varepsilon}{\partial x}] + \tfrac{1}{r}\tfrac{\partial}{\partial r}(r\mu_l + \tfrac{\mu_t}{\sigma_\varepsilon})\tfrac{\partial \varepsilon}{\partial r}] + C_{S1}G\tfrac{\varepsilon}{k} - C_{s2}\tfrac{\varepsilon^2}{k}$$

here C_{s1}, C_{s2}, σ_k and σ_ε are the empirical turbulent constant. The values are considered according to the Launder *et al.*, 1974. The values of Cμ, C_{s1}, C_{s2}, σ_k and σ_ε are 0.09, 1.44, 1.92, 1.0 and 1.3 respectively.

4.3 Boundary Condition

A turbulent flow is considered. The quantities *U, k, ε* are obtained by using numerical calculations based on the *k-ε* model for low Reynolds Number. The boundary conditions are listed below:

1) At the inlet of the channel:

$$u = U_{in}, v = 0$$

$$k_{in} = 0.005\, U_{in}^2$$

$$\varepsilon_{in} = 0.1 K_{in}^2$$

K_{in} stands for the admission condition for turbulent kinetic energy and ε_{in} is the inlet condition for dissipation.

2) At the walls:

$$u = v = 0$$
$$k = \varepsilon = 0$$

3) At the exit:

$$P = P_{atm}$$

The Reynolds number based on diameter of tube. D is the circular diameter of tube and D_h is hydraulic diameter of rectangular tube, analysis is carried out at different Reynolds number.

Reynolds number in case of circular tube

$$R_e = \frac{\rho u D}{\mu}$$

Reynolds number in case of rectangular tube

$$R_e = \frac{\rho u D_h}{\mu}$$

5. Solution Strategy & Convergence

The whole analysis is carried out with the help of software **"ANSYS Fluent 14.0".** ANSYS Fluent 14.0 is computational fluid dynamics (CFD) software package to stimulate fluid flow problems. It uses the finite volume method to solve the governing equations for a fluid Geometry and grid generation is done using GAMBIT which is the pre-processor bundled with FLUENT. The two dimensional computational domain modeled using hex mesh for 2-D models. The complete domain of 2-D circular tube in all three cases have element size is 0.005 m have 59925 nodes and 47138 element, and rectangular tube in all three cases have element size is 0.002 m, and 115577 nodes 90000 Elements. Grid independence test was performed to check the validity of the quality of the mesh on the solution. Further refinement did not change the result by more than 0.9% which is taken as the appropriate mesh quality for computation.

5.1 Sensitivity Analysis of the Mesh

A non-uniform mesh in both horizontal and vertical directions proved to be sufficient to model the system.

Table 5.1 Mesh sensitivity analysis

Element size (m)	Nodes	Element	U_{max}	Total pressure (pa)
0.003	422073	387220	1.94	1.47 e^{03}
0.004	88682	68760	1.96	1.49 e^{03}
0.005	59925	47138	1.97	1.50 e^{03}
0.006	138652	103938	2.00	1.515 e^{03}

The meshing size is comparatively small near the boundaries so a good estimate of the gradients can be obtained. The values in the table indicate the properties of circular duct for different meshing size.

6. Validation of Model

In the present paper a circular and rectangular tube was modeled and simulated using computational fluid domain for heating cold water by applying fixed wall temperature boundary conditions. Heat transfer parameters like temperature drop, heat transfer rate heat transfer coefficient, nusselt number skin friction coefficient and pressure drop were calculated. Simulation results were compared with the analytical results using the correlations developed by different researchers. Also the simulation results of the circular tube were compared with the results obtained for a rectangular tube of equal length and similar operating conditions in order to compare its performance related to heat transfer characteristics.

Table 6.1 Geometry description of circular and rectangular model

Length of the tube L (m)	3
Diameter of tube (m)	0.015
Working fluid	water
Inlet water temperature (K)	323
Outlet water temperature (K)	343
Constant wall temperature (K)	373

Table 6.2 Properties of working fluid water

Density (kg/m^3)	990
Specific heat C_p (J/kg-k)	4184
Thermal conductivity (W/m-k)	0.65
Kinematic viscosity (m^2/s)	0.516 e^{-06}
Prandtl number (Pr)	3.15

7. Results and Disscussion

CFD computations were done for three different mass flow rate of water 0.2624, 0.3498 and 0.5248 kg/s respectively for both circular and rectangular tube. Performance parameters adopted for comparison are Heat transfer coefficient, Nusselt number and pressure drop in both the cases. In order to validate the CFD results the important parameters like Nu and heat transfer coefficient were calculated by using the different correlations both for circular and the rectangular tube. Fig. 7.1 and Fig. 7.3 show the CFD simulated nusselt number plot vs. correlation values for circular and rectangular tubes for the different mass flow rates. The results have shown a good agreement between the correlation values used by different researchers and fluent results as the average error is within 3% and 4.5% for both cases. Similarly fig. 7.2 and Fig. 7.4 shows the plot of heat transfer coefficients plot vs. correlation for both cases.

Fig. 7.5 shows the variation of Nusselt number for circular and rectangular tubes. Nusselt number corresponding to the circular is higher than the rectangular tube for all mass flow rates. This is because of the shape in the circular tube which aids the heat transfer. For different mass flow rates there was significant increase in Nusselt number for circular tube under similar operating conditions was noticed. On an average Nusselt number increased by 8.5% when the mass flow rate was changed from 0.2624 kg/s to 0.5248 kg/s. In the circular tube at higher mass flow rates Reynolds number increases and also fluid turbulence increases. Higher turbulence increases the intensity of secondary flow and hence the Nusselt number.

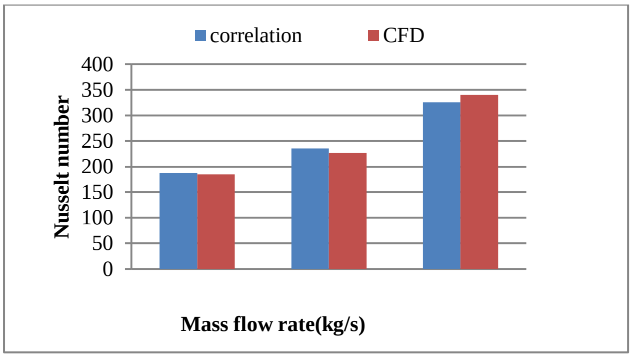

Fig. 7.1 Comparison of correlation and CFD values of Nusselt number for circular tube

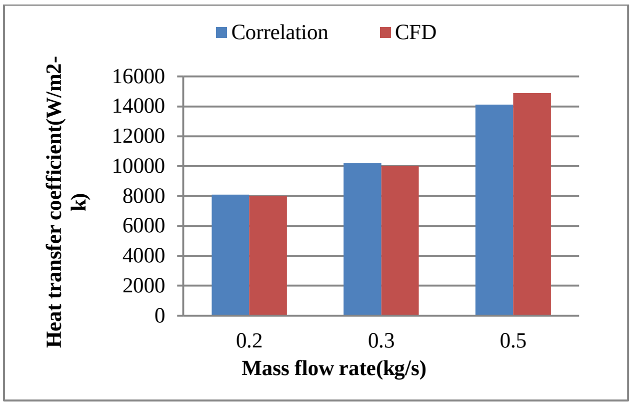

Fig. 7.2 Comparison of correlation and CFD values of Heat transfer coefficient for circular tube

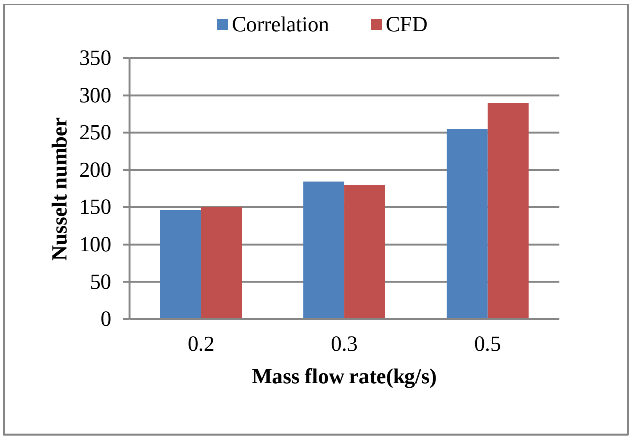

Fig. 7.3 Comparison of correlation and CFD values of Nusselt number for rectangular tube

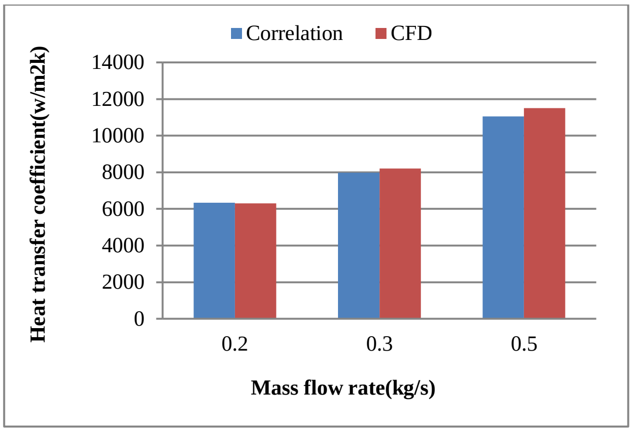

Fig. 7.4 Comparison of correlation and CFD values of Heat transfer coefficient for rectangular tube

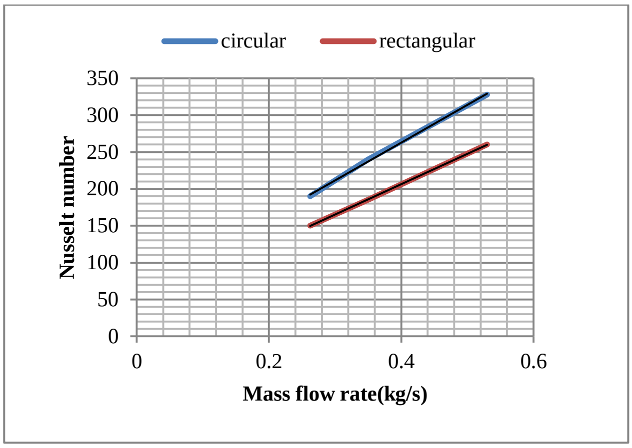

Fig. 7.5 Variation Nusselt number with mass flow rate for circular and rectangular tube

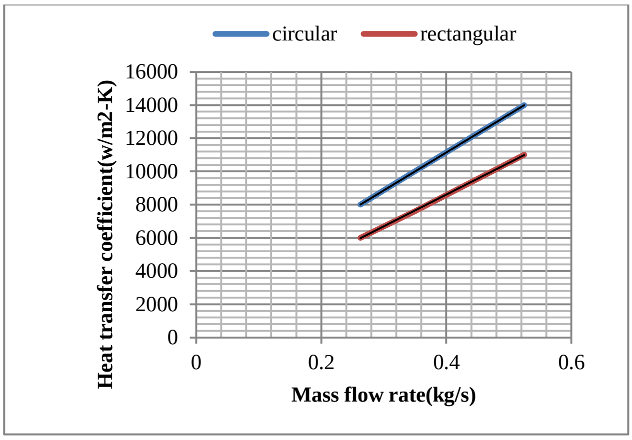

Fig. 7.6 Variation heat transfer coefficient with mass flow rate for circular and rectangular tube

Fig. 7.6 shows the variation of heat transfer coefficient for different mass flow rates. From the graph it is revealed that as the mass flow rate increases heat transfer coefficient also increases as expected since heat transfer rate is proportional to the mass flow rate. Further for circular tube heat transfer coefficient has increased by 10% when the mass flow rate is increased from 0.2624 to

0.5248 Kg/s. Fig. 7.7 show the comparison of pressure drops for the circular and rectangular tubes. Pressure drop for the circular tube is found to be more than the rectangular tube for all mass flow rates. Presence of secondary flow dissipates kinetic energy, thus increasing the resistance to flow. For lower mass flow rates pressure drop varies linearly whereas on increasing the mass flow rate pressure drop varies exponentially as seen in the graph.

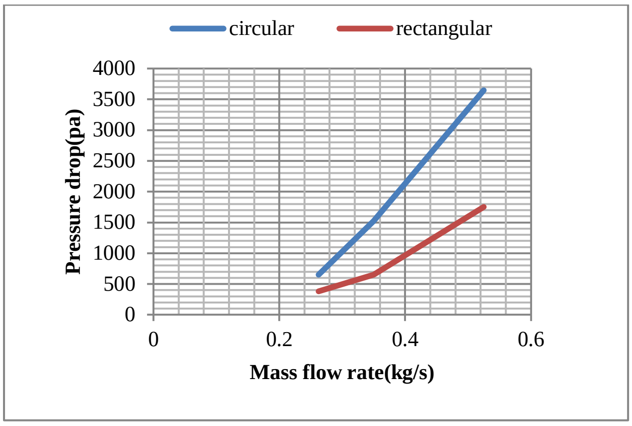

Fig. 7.7 Variation of pressure drop with mass flow rate for circular and rectangular tube

Conclusions

- In the present work CFD analysis for a circular tube heat exchanger was carried out and the results of heat transfer parameters have been compared with the rectangular tube under similar geometrical and operating conditions.
- CFD results are validated by the correlations used by the different researchers. There was a close agreement between the CFD predicted and correlation results.
- Simulation results indicated that nusselt number and heat transfer coefficient are higher in case of circular tube when compared with the rectangular tube.
- Pressure drop for circular tube is found to be more when compared with the rectangular tube for identical conditions.

References

Incopera, F.P and dewitt, D.P (1990) Fundamental of heat and mass transfer, john wiley and sons Inc., New York.

Bejan,(1993) A., heat transfer, john wiley and sons Inc., New York.

Burmeister,(1993) L.C, Convective heat transfer, john wiley and sons Inc., New York .

Bejan, (1993) A., Convective heat transfer, john wiley and sons Inc., New York., 1995.

Kays, W.M and craford, M.E., Convective heat transfer, McGraw-Hill., New York.

Shah, T.K and London. A.L, (1978) Laminar flow forced convection in ducts, Academic press, New York

Shah, R.K. and Bhatti, M.S, (1987) chapter 3: Laminar convective heat transfer in ducts, in handbook of single phase convective heat transfer, eds.S. Kakac, R. K Shah and W. Aung, john wiley and sons Inc., New York.

F.B. Gessner and A.F. Emery, (1976) A Reynolds stress model for turbulent corner flows – Part I: Development of the model, Journal Fluids Eng. 98, 261-268.

Melling, A. & Whitelaw, J.H. (1976) "Turbulent Flow in a Rectangular Duct," Journal of Fluid mechanics, Vol. 78, 289-315.

Nakayama, A., Chow, W. L., & Sharma, D. (1983) "Calculation of fully Developed Turbulent Flows in Ducts of Arbitrary Cross-Section," *Journal of Fluid Mechanics*, Vol. 128,199-217.

B.E. Launder and W.M.Ying., (1973) Prediction of flow and heat transfer in ducts of square cross section", Proc. Inst. Mech. Eng., 187, 455-461

F.B. Gessner, and J.K. Po, (1976) A Reynolds stress model for turbulent corner flows – Part II: Comparison between theory and experiment, Journal Fluids Eng. 98, 269-277.

Nomenclature

u : Inlet velocity of water (m/s)
P_r : Prandtl number
k : Thermal conductivity (w/m-K)
P : Pressure, Pa
D: Diameter of circular duct
D_h : Hydraulic diameter of rectangular duct
Re : Reynolds number
L: Tube length, m
G the flow production term
Nu : Nusselt number

Greek Symbols

ε:Dissipation rate of turbulence energy (*m^2/s*)
κ : Turbulent kinetic energy
ρ:Density of the water (*kg/m^3*)
μ : Kinematics viscosity (pl)
μ_l, μ_t : Laminar, turbulent viscosity (*Pa.s)*