Numerical analysis of heat transfer and fluid flow around circular and non-circular tubes

R Deeb¹,² and D V Sidenkov²

¹ Department of general mechanics, Faculty of Mechanical and Electrical Engineering, Damascus University, Damascus, Syria

² Department of theoretical basis of heat engineering, National Research University, 111250 Moscow, Russia 14 Krasnokazarmennaya St.

Abstract. A two-dimensional study of turbulent flow and heat transfer in a channel with circular and non-circular tubes has been carried out using the CFD Fluent software package. Four non-circular tubes were investigated: flat, ellipse, cam, and drop-shaped. Reynolds number varied in the range of 7300 ≤ Re ≤ 14600. All tubes were investigated under similar operating conditions. Local heat transfer, pressure, and friction coefficients over a surface of the tubes were presented. The thermal-hydraulic performance was used to estimate the efficiency of the non-circular tubes. The results indicated that the drop-shaped tube has the best thermal-hydraulic performance, which was about 5.6, 2.6, 1.7, 1.3 times higher than that of the circular, flat, ellipse, cam tube, respectively.

1. Introduction

An important economic challenge today is the rational and efficient use of energy resources in various industries. The improvement of heat exchangers is a technical and economic imperative. Improving power plant performance requires the use of efficient heat exchange surfaces in heat exchangers, which have high thermo-hydraulic efficiency, low cost, and ease of manufacture.

Many investigations were carried out to study the heat transfer and fluid flow characteristics around circular tubes bundle in crossflow. Zukauskas and Ulinskas [1, 2] suggested correlations for heat transfer and pressure drop for staggered and in-line of circular tubes bundles. Lavasani [3] experimentally investigated the flow around cam shaped tube bank with inline arrangement for both longitudinal pitch ratios 1.5 and 2. It was noted that by increasing longitudinal pitch ratios from 1.5 to 2, heat transfer increases about 7-14 %. Furthermore, friction factor of cam shaped tube bank is approximately 95 % lower than circular tube bank. Ibrahim and Gomma [4] experimentally and numerically investigated the turbulent flow over a bundle of elliptical tubes. The axis ratios and the flow angles of attack were varied from 0.25 to 1 and 0° to 150° respectively. For fixed pumping power, the best flow angle of attack was 0°. Toolthaisong and Kasayapanand [5] studied the effect of attack angles on heat transfer and pressure drop characteristics of the crossflow heat exchangers with staggered flat-shaped tubes bundles. The results indicated that the angle of attack of 90° gives the maximum heat transfer rate. Moreover, the impact of changing the angle of attack on heat transfer surfaces of the lower aspect ratios is more than it on the surfaces of higher aspect ratios. Deeb and Sidenkov [6-9] numerically studied hydrodynamics and heat transfer characteristics of a drop-shaped tubes bundle of various configurations. Their results indicate that the hydrodynamic resistance of the drop-shaped tubes bundles was better than the circular ones at angles of attack θ = 0°, 180°. They proposed a correlation for heat transfer in terms of Re, θ, and axis ratios with taking into account the stress-strain state of the tubes.
As shown above, experimental and numerical investigations of local heat transfer and flow characteristics around non-circular tubes in crossflow are very limited. Therefore, the aim of the present study was to investigate the properties of local and overall flow and heat transfer characteristics of non-circular tubes and compare them with the circular tubes for \(7.3 \times 10^3 \leq \text{Re} \leq 14.6 \times 10^3\).

2. Problem definition and boundary conditions

2.1. Problem formulation

Two dimensions steady-state calculations for five tubes: circular, flat, ellipse, cam, drop-shaped tubes (figure 1) in crossflow were performed to clarify heat transfer and fluid flow characteristics around the studied tubes using ANSYS Fluent [10].

![Figure 1. Studied tubes: a) circular, b) flat, c) ellipse, d) cam, e) drop-shaped.](image)

The tubes are placed in a square cross-section channel (305 × 305 mm) and a length of 780 mm. The diameter of the equivalent standard circular \(D_{eq}=22.44 \text{ mm}\) was used to compare the heat transfer and friction factor from each tube. As an external flow, the air flow was used. The velocity of the incoming airflow \((u)\) was varied in the range of \(6 \sim 12 \text{ m/s}\), which corresponds to \(\text{Re}=7.3 \times 10^3 \sim 14.6 \times 10^3\), at a temperature of 26 ºC and atmospheric pressure. The tube was maintained at a constant temperature of 82 ºC.

The thermo-physical properties of the fluid were calculated at the mean inlet air temperature. The nonslip boundary conditions were applied at the tube. The effect of radiation, gravity and buoyancy are neglected.

2.2. Governing equations

The fluid flow and heat transfer characteristics are described by the transport equation for the conservation of mass, momentum, and energy:

\[
\frac{\partial}{\partial x_i} (\rho U_i) = 0
\]

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

\[
\rho C_p \frac{\partial (u_i T)}{\partial x_j} = \frac{\partial}{\partial x_j} \left( \mu C_p \frac{\partial T}{\partial x_j} - \rho C_p u_j' T' \right)
\]

where: \(i\) is a tensor indicating 1 and 2; \(u_i, u_j\) are the air velocity in \(i\) and \(j\) directions, respectively; \(\rho\) is the air density; \(P\) is the air pressure; \(\mu\) is the dynamic viscosity; \(C_p\) is the specific heat at constant pressure, \(T\) is the air temperature, \(Pr\) is the Prandtl number.

The velocities \(u\) and the temperature \(T\) consist of the sum of mean \(\bar{u}, \bar{T}\) and fluctuating values \(u', T'\). Thus, RNG k-\(\varepsilon\) turbulent model with the “Enhanced Wall Treatment” function was used to solve this set of equations. This model improves the ability to model highly strained flows, vortices, separation, and recirculation of the fluid [10].

2.3. Mesh generation and discretization

Figure 2 shows the configuration of the two-dimensional grid employed in the present computation with the boundary conditions and the real dimensions of the test channel. The mesh was generated using ICEM CFD. The domain is a symmetrical rectangle, includes half of the tube. The working fluid area was meshed with quadrilateral mesh elements with refining the mesh near walls, so as to capture the boundary layer
over the tube surface. The location of the first node away from a wall were kept at $1<y^+<5$. The mesh quality of 0.926 was maintained for the entire simulation.

**Figure 3.** Mesh-sensitivity analysis.

**Figure 2.** Computational mesh used in numerical calculations with boundary conditions: (a) circular, (b) flat, (c) ellipse, (d) cam, (e) drop-shaped.

In this study, the governing equations for momentum and turbulent kinetic energy are discretized with the second-order upwind scheme. The simulation used the segregated solver, with which continuity and momentum equations were solved in a decoupled fashion during the outer iteration loop, besides using SIMPLE pressure-based solution algorithm of the velocity–pressure coupling. The solution was considered converged when the scaled residual of the energy and other equations reach $10^{-8}$.

The mesh-sensitivity analysis was carried out in figure 3. Nodes number varied from 9584 to 236206. It is seen that the computational results of the average Nusselt number ($\text{Nu}_{\text{av}}$) become independent from the mesh for the mesh of about 63824 nodes. Hence, the mesh of 63824 nodes was considered here-onwards to optimize the time and the accuracy of the solution.

**2.4. Numerical results validation**

The numerical results of the present study were validated with the corresponding experimental results [2, 9] of $\text{Nu}_{\text{av}}$ and friction factor ($f$) in the same range of values of Reynolds numbers. Figure 4 shows that good agreements were obtained between the experimental and numerical results. Thus, the model and the method of the CFD simulation presented in this study is reliable.

**Figure 4.** Validation of (a) $\text{Nu}_{\text{av}}$ versus $Re$; (b) $f$ versus $Re$.

**3. Results and discussion**

**3.1. Characteristics of air flow**

Figure 5 shows the streamlines contours for the studied tubes at $Re=11000$. It is noticed that a pair of vortexes appears behind the tubes. The vortex area gradually decreases for the circular, cam, and drop-shaped tube, respectively, which affects heat transfer and fluid flow characteristics. It is also seen that the wake zone in the case of the circular tube is larger compared to the rest of the tubes.
3.2. Heat transfer characteristics

The temperature of the air increases by gaining the heat from the tube surface. The maximum temperature of 82.85 °C is observed at the surface of the tubes (boundary condition). Figure 6 demonstrates contours of static temperature for different shapes of the tube for selected Reynolds numbers. It is seen that at Re=7300, the thermal boundary layer is thicker than that for Re=14600. This is attributed to the separation of the flow over the tube surfaces, which enhances the heat transfer with increasing the Re.

The local heat transfer is affected by the thickness of the boundary layer over the surface of the tube. Since the distribution of a local heat transfer coefficient $\alpha$ over the tube is symmetrical, the distribution of $\alpha$ over a half surface of each tube at Re=11000 given in figure 7. For all tubes, the local heat transfer coefficient has a maximum value at the forward stagnation point ($L=0$ m) and then it decreases until reach the separation point of the boundary layer. The minimum value in figure 7 indicates the separation point of the curves, and then the heat transfer coefficient increases again due to the occurrence of the vortex.

The average Nusselt number is determined from the computational results as:

$$Nu_{av} = \frac{\overline{\alpha} D}{\lambda}$$

(4)

where $\overline{\alpha} = \frac{1}{F} \int_0^F \alpha dF$ is the heat transfer coefficient averaged over entire tube surface.

The variation of the average Nusselt number $Nu_{av}$ against the Reynolds number is illustrated in figure 8. The average Nusselt number increases with the increase in Re. The highest and lowest values of $Nu_{av}$ are achieved for drop-shaped, cam, flat, circular and ellipse tubes, respectively.

3.3. Pressure coefficient distribution and friction factor

The pressure over the tube surface can be expressed through a pressure coefficient as:

$$C_p = \frac{2(P_i - P_{\infty})}{\rho U_{\infty}^2}$$

(5)

where $P_i$ is the local static pressure; $P_{\infty}$ and $U_{\infty}$ are the pressure and the free-stream velocity, respectively.

Distribution of pressure coefficient over a half surface of the tube is demonstrated in figure 9. It is clear that all tubes have the maximum value of the pressure coefficient at the forward stagnation and then $C_p$ increases up to the rear stagnation point of the tube. In the case of flat tube, there are two minimum values of $C_p$. However, for the rest tubes, there is one minimum value of $C_p$. The minimum value of $C_p$ is the point of change from a favorable ($dP/dx < 0$) to an adverse pressure gradient ($dP/dx > 0$) over the surface of the tube. Unlike the circular tube, non-circular tubes have a high favorable pressure gradient at the front of the tube and a low adverse pressure gradient on sides of the tube, which prolongs the separation of the fluid from the wall surface.

To find the location of the boundary layer separation point from the tube surface, the distributions of the skin friction coefficient $C_f$ over a half surface of the studied tubes are plotted in figure 10 at Re=11000.

The skin friction coefficient could be expressed as:

$$C_{f,j} = \frac{2\tau_{w,j}}{\rho U_{\infty}^2}$$

(6)

where $\tau_{w,j}$ is a local skin shear stress on a tube surface.

The value of zero for $C_f$ (figure 10) indicates that the flow is separated from the surface of the tube. For different tubes, it is seen that a first zero value of $C_f$ exists at the forward stagnation point. The location of the second zero value of the skin friction coefficient is shifted towards the rear stagnation point of the tube for a circular tube compared to the rest tubes. The maximum value of $C_f$ shifts toward the forward stagnation
point for the case of non-circular tubes as compared to that of the circular one. The highest values of $C_f$ were 0.159, 0.121, 0.114, 0.104, 0.052 for the flat, drop-shaped, ellipse, cam, circular tube, respectively.

Re= 7300  
Re= 14600  
Re= 7300  
Re= 14600

Figure 6. Temperature contours for: a, b) circular; c,d) flat; e-f) ellipse; g, h) cam; i, j) drop-shaped tube.

Figure 7. The distribution of the local $\alpha$.

Figure 8. Variation of average Nu versus Re.

Figure 9. Pressure coefficient distribution.

Figure 10. Skin friction coefficient distribution.

The fluid flow is always associated with the friction factor over which the flow is established. The friction factor is determined as follows:

$$f = 2\Delta P \left[ \frac{(L/D)(\rho U_{\text{max}}^2)}{\left( \frac{L}{D} \right)} \right]^{-1}$$  \hspace{1cm} (7)

where $\Delta P$ is the pressure drop across the channel obtained from the results of the computational simulation.

Figure 11 presents the friction factor for different shapes of tube. For all studied tubes, the friction factor for the fluid decreases by increasing Reynolds number. This is usually due to the dominant pressure force, which reduces the friction. The friction factor is maximum in case of the circular tubes. There is a significant reduction of 81%, 76%, 72.5% and 52.6% in the friction factor for drop-shaped, cam, ellipse and flat tubes, respectively, as compared to that of the circular.
3.4. Overall thermal performance

It is necessary to evaluate the combined effect of heat transfer along with friction factor associated with the flow over the tubes. The thermal-hydraulic performance shaped tube base on a circular tube was proposed by Webb [11] as:

\[ \eta = \left[ \frac{Nu_{av,drop}}{Nu_{av,circ}} \right] \left[ \frac{f_{drop}}{f_{circ}} \right]^{-1} \]  

Figure 12 illustrates the thermal-hydraulic performance for the studied tubes for the entire range of the Reynolds number. It is clear from the plot that \( \eta \) decreases as Reynolds number for the flow increases. The thermal-hydraulic performance of drop-shaped, cam, ellipse, flat tubes is about 5.6, 4.4, 3.3, 2.2 times, respectively, greater than circular tube. This is due to the decrease in the friction factor along with an increase in the Nusselt number with the increase in the Reynolds number. This means that non-circular tubes perform better compare to circular tube.

![Figure 11. Friction factor versus Re.](image)

![Figure 12. \( \eta \) versus Re](image)

4. Conclusions

In this study flow around circular, flat, ellipse, cam and drop-shaped tubes have been investigated numerically. Results show that non-circular tubes have a high favorable pressure gradient at the front of the tube and a low adverse pressure gradient in the rear of the tube, which prevent an early flow separation.

For all studied tubes, the results indicated that the average Nusselt number increases, while the friction factor decreases with an increase of the Reynolds number.

The values of the friction factor of the drop-shaped tube were about 81%, 59.8%, 36.9%, 20% lower than those obtained for circular, flat, ellipse, cam-shaped tube, respectively.

The highest and the lowest values of the thermal–hydraulic performance were achieved for the drop-shaped, cam, ellipse, flat, circular tube, respectively.

5. References

[1] Zhukauskas A., Ulinskas R.V. 1985 *Heat Transfer Engineerin* 6 (1) 19–25.
[2] Zhukauskas A. 1972 *Heat Transfer Engineerin* 18 87–159.
[3] Lavasani A. M., Bayat H., Maarefdoost T. 2016 *Applied thermal engineering* 65 (1-2) 85–93.
[4] Ibrahim T.A., Gomma A. 2009 *Int. J. of Thermal Sciences* 48 2148–2158.
[5] Toolthaisong S., Kasayapanand N., 2013 *Int. J. of Heat and Mass Transfer* 34 417 – 429.
[6] Deeb R., Sidenkov D.V. 2020 *IOP Conf. Series: Journal of Physics* 1683 042082.
[7] Deeb R., Sidenkov D.V. 2020 *Inforino* DOI: 10.1109/Inforino48376.2020.9111775.
[8] Deeb R., Sidenkov D.V. 2019 *IOP Conf. Series: Journal of Physics* 1359 012135.
[9] Deeb R. 2021 *Physics of Fluids* 33 065110.
[10] ANSYS Inc. 2019 *Ansys Fluent User’s Guide*. In Ansys Aim Student 19.0, 19.2.
[11] Webb R. L. 1981 *Heat and Mass Transfer*. 24 715–726.