

Numerical Study of Flow around Flat Tube between Parallel Walls

Hamidreza Bayat, Arash Mirabdollah Lavasani, Meysam Bolhasani, Sajad Moosavi

Abstract—Flow around a flat tube is studied numerically. Reynolds number is defined base on equivalent circular tube and it is varied in range of 100 to 300. Equations are solved by using finite volume method and results are presented in form of drag and lift coefficient. Results show that drag coefficient of flat tube is up to 66% lower than circular tube with equivalent diameter. In addition, by increasing L/D from 1 to 2, the drag coefficient of flat tube is decreased about 14-27%.

Keywords—Laminar flow, flat-tube, drag coefficient, cross-flow, heat exchanger.

I. INTRODUCTION

FLOW around bluff bodies in cross-flow has many applications such as heat exchangers, air conditioning and so on. Circular cylinders due to its ease of preparation are used in most of the industrial equipment. Zukauskas and Ziugzda [1], Zdravkovich [2] published book about flow and heat transfer from cylinder.

Rocha et al [3] numerically investigated elliptical and circular section in one and two row tubes and plate fin heat exchanger. Their results indicate that compare to circular tubes plate fin heat exchangers, elliptic one have performed better due to lower pressure drop and higher fin efficiency. Wilson and Bassiouny [4] simulate laminar and turbulent flow field around tube bank with finite volume method. They found that by increasing longitudinal pitch friction factor increases. Furthermore, they suggested that it is better to choose longitudinal pitch ratio $a \leq 3$, in order to have best performance and compactness.

Nouri-Borujerdi and Lavasani [5]-[9] experimentally studied flow and heat transfer from a cam-shaped tube in cross-flow of air. Their results indicate that single cam-shaped tube performed better than circular tube. Lavasani and Bayat [10]-[12] numerically studied flow and heat transfer characteristics from two cam-shaped tubes in side-by-side and tandem arrangement.

Mirabdollah Lavasani et al. [13] experimentally studied

H. Bayat is member of Young Researchers and Elite Club, Central Tehran Branch, Islamic Azad University, Tehran, Iran (e-mail: hrb.mech@gmail.com).

A. M. Lavasani is Assistant Professor of Department of Mechanical Engineering, Central Tehran Branch, Islamic Azad University, Tehran, Iran, (e-mail: arashlavasani@iauctb.ac.ir).

M. Bolhasani is Graduate student at Department of Energy Systems Engineering, Faculty of Engineering, Islamic Azad University-South Tehran Branch, Tehran, Iran (e-mail: bolhasani.me@gmail.com)

S. Moosavi is Graduate student at Department of Mechanical engineering, Takestan Branch, Islamic Azad University, Takestan, Iran (e-mail: moosavi.eng.90@gmail.com)

convective heat transfer from cam-shaped tube bank with in-line arrangement and Bayat et al. [14] experimentally studied thermal-hydraulic performance of cam-shaped tube bank in staggered arrangement. Their results show that thermal-hydraulic performance of cam-shaped tube bank in both in-line and staggered arrangement is about 5-6 times greater than circular tube bank.

There are several studies about flow and heat transfer around flat tube bank in cross-flow of air [15], [16]. These tubes due to lower air-side pressure drop compared to circular tube performed better in heat exchangers. In this study flow around single flat tube is studied numerically.

II. GOVERNING EQUATIONS

The cross section profile of the flat tube is represented in Fig. 1. These tubes are comprised of two circles with two line segments tangent to them. Characteristic length for these tubes is the diameter of an equivalent circular cylinder, $D_{eq} = P/\pi$, whose circumferential length is equal to that of the flat tube. In this study flow characteristics of two different tubes is studied. The tube distance between center to center of circle is 10 mm and 20 mm and their diameter is 10 mm for tube number 1 and 2, respectively. Therefore the equivalent diameter of tube number 1 and 2 is 16.37 mm and 22.73 mm, respectively.

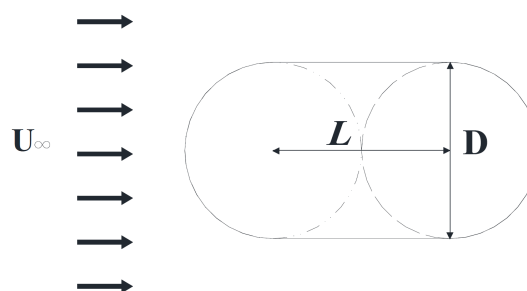


Fig. 1 Solution domain

The typical solution domain and the cylinder boundary definition and nomenclature used in this work are shown in Fig. 1.

Equations are written for conservation of mass and momentum in two dimensions. Cartesian velocity components U and V are used, and it has been assumed that the flow is unsteady and laminar, while the fluid is incompressible and Newtonian with constant transport properties. Furthermore, the effect of viscous dissipation is neglected. The governing equations consist of the following three equations for the dependent variables U , V and P :

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \quad (1)$$

$$\frac{\partial u}{\partial t} + \frac{\partial(uu)}{\partial x} + \frac{\partial(uv)}{\partial y} = -\frac{\partial P}{\partial x} + \frac{1}{\text{Re}} \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \quad (2)$$

$$\frac{\partial v}{\partial t} + \frac{\partial(uv)}{\partial x} + \frac{\partial(vv)}{\partial y} = -\frac{\partial P}{\partial y} + \frac{1}{\text{Re}} \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \quad (3)$$

Equations (1) to (3) are the conservation of mass, x and y direction momentum equations, respectively.

The dimensionless variables of present studies are:

$$\begin{aligned} u &= \frac{\bar{u}}{u_\infty}, & x &= \frac{\bar{x}}{D_{eq}} \\ v &= \frac{\bar{v}}{v_\infty}, & y &= \frac{\bar{y}}{D_{eq}} \\ p &= \frac{\bar{p}}{\rho u_\infty}, & t &= \frac{u_\infty \bar{t}}{D_{eq}} \end{aligned} \quad (4)$$

The boundary conditions used for the solution domain are uniform inlet velocity, fully developed outflow and no-slip on tube surface where at tube surface the boundary condition is:

$$u = v = 0 \quad (5)$$

The inlet flow has a uniform velocity. The velocity range considered only covers laminar flow conditions.

$$u = 1, \quad v = 0 \quad (6)$$

In order to decrease the effect of entrance and outlet regions, the upstream and downstream lengths are 15D and 50D, respectively. Outflow boundary condition is considered at outlet:

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial y} = 0 \quad (7)$$

The total drag coefficient is calculated from:

$$C_D = C_{DP} + C_{DV} = \frac{2F_D}{\rho U_\infty^2 D_{eq}} \quad (8)$$

where C_{DP} and C_{DV} are drag coefficient due to pressure and viscous forces of passing fluid. C_{DP} and C_{DV} are calculated from:

$$\begin{aligned} C_{DP} &= \frac{1}{0.5} \int_0^1 (p_{front} - p_{back}) dy \\ C_{DV} &= \frac{1}{0.5 \text{Re}} \int_0^1 \left[\left(\frac{\partial u}{\partial y} \right)_{bottom} + \left(\frac{\partial u}{\partial y} \right)_{top} \right] dx \end{aligned} \quad (9)$$

The total lift coefficient is defined by:

$$C_L = C_{LP} + C_{LV} = \frac{2F_L}{\rho U_\infty^2 D_{eq}} \quad (10)$$

where C_{LP} and C_{LV} are lift coefficient of pressure and viscous forces, respectively, and calculated from:

$$\begin{aligned} C_{LP} &= \frac{1}{0.5} \int_0^1 (p_{bottom} - p_{top}) dx \\ C_{LV} &= \frac{1}{0.5 \text{Re}} \int_0^1 \left[\left(\frac{\partial v}{\partial x} \right)_{front} + \left(\frac{\partial v}{\partial x} \right)_{back} \right] dy \end{aligned} \quad (11)$$

III. NUMERICAL METHOD

This problem considers a 2D section of flat tube. For the simulations presented here, depending on the geometry used, fine meshes of 35370 to 48234 elements were used. A sample of the mesh for the flat tube is shown in Fig. 2. In this domain quadrilateral cells are used in the regions surrounding the tube wall and the rest of the domain. In all simulation, a convergence criterion of 1×10^{-6} was used for all variables.

The computational grid is shown in Fig. 2. The second order upwind scheme was chosen for interpolation of the flow variables. The SIMPLE algorithm [17] has been adapted for the pressure velocity coupling. In all simulation, a convergence criterion of 1×10^{-6} was used for all variables.

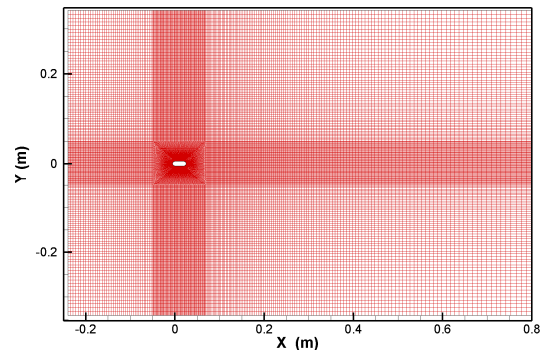


Fig. 2 Computational grid

IV. RESULT AND DISCUSSION

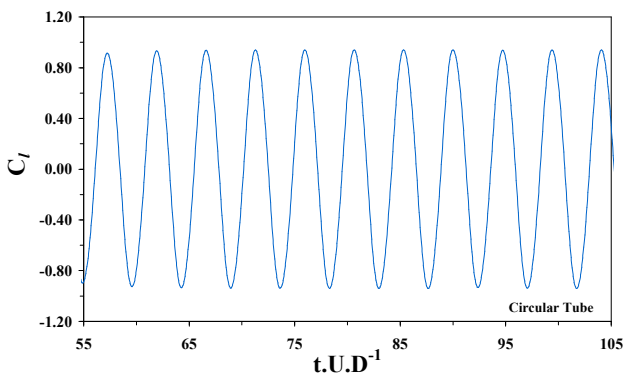
For the purpose of the validation of the solution procedure, it is essential that numerical simulations be compared with experimental data. Table I compares results of flow parameter of present work with other work on literature. As it is clear from there is a good agreement between Strouhal number, drag and lift coefficient and of present study with others.

TABLE I
 COMPARISON OF FLOW PARAMETERS FOR A SINGLE CIRCULAR CYLINDER AT
 $Re = 200$

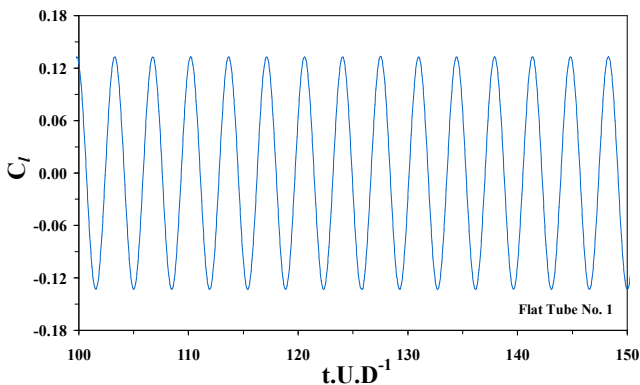
Author	Type of Data	C_D	C_l	St
Present work	Numerical	1.38 ± 0.04	± 0.694	0.199
Meneghini et al. [18]	Numerical	1.30 ± 0.05	± 0.659	0.196
Ding et al [19]	Numerical	1.348 ± 0.05	± 0.75	0.196
Mahir and Altac [20]	Numerical	1.376	---	0.192
Williamson [21]	Experimental	---	---	0.196

Oscillation of lift coefficient of flat tube at $Re=300$ is presented in Fig. 3. It is clear from this figure that by increasing l/D from 1 to 2, the amplitude of oscillation of lift coefficient decreases from 0.133 to 0.04. Compare to circular tube the amplitude of oscillation for flat tube is about 86% and 96% lower for $l/D=1$ and 2, respectively.

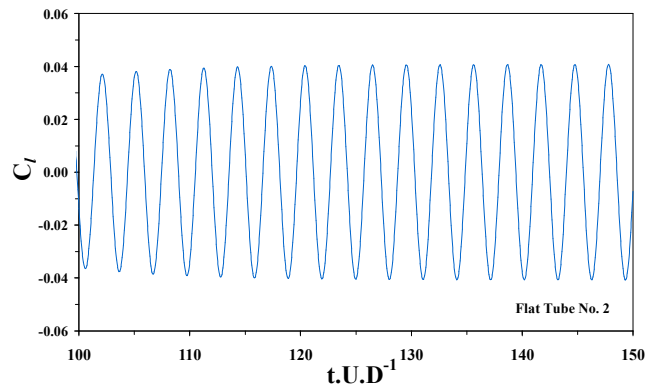
Fig. 4 shows variation of drag coefficient of both flat tubes with time and Fig. 5 represents variation of mean drag coefficient with Reynolds number. Results show that by increasing Reynolds number from 100 to 300 drag coefficient decreases about 25% and 36% for tube No.1 and 2, respectively. Moreover, for a fixed value of Reynolds number increasing l/D from 1 to 2 leads to 14%, 17% and 27% decrease in the value of drag coefficient for Reynolds number 100, 200 and 300, respectively. In all range of Reynolds number drag coefficient of flat tube with $l/D=1$ and 2 is about 41-54% and 50-66% lower than circular tube, respectively.



(a)

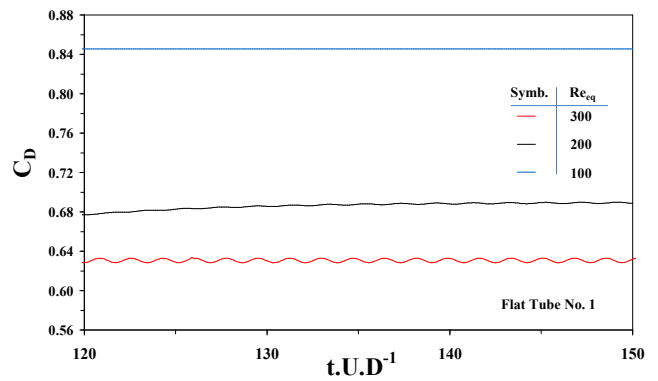


(b)

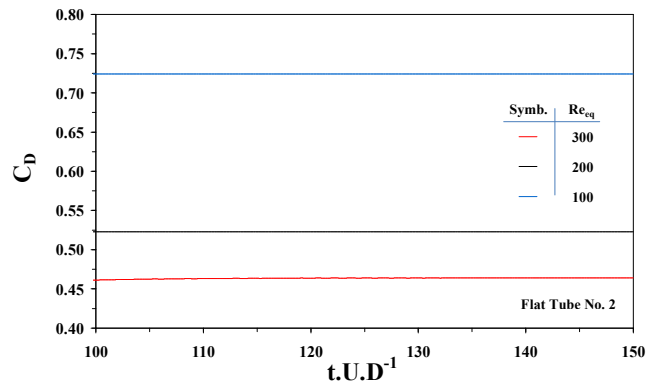


(c)

Fig. 3 Times-histories of lift coefficient at $Re=300$: (a) circular tube (b) flat tube No.1 (c) flat tube No. 2



(a)



(b)

Fig. 4 Times-histories of drag coefficient of flat tube: (a) tube No.1 (b) tube No.2

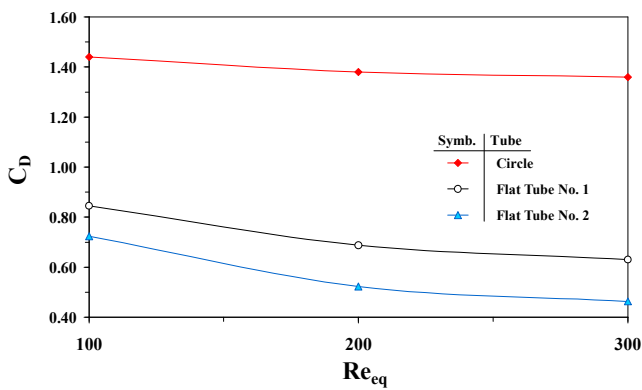


Fig. 5 Variation of drag coefficient of flat tube with Reynolds number

V. CONCLUSION

Unsteady flow from flat tube between parallel walls for $100 \leq Re_{eq} \leq 300$ is investigated by using finite volume method. Results showed that the amplitude oscillation of lift coefficient for flat tube is about 86-96% lower than circular tube. By increasing l/D from 1 to 2 value of drag coefficient is decreased up to 27%. Furthermore, in all range of Reynolds number drag coefficient of flat tube is about 41-66% lower than circular tube.

NOMENCLATURE

C_D	Drag coefficient
C_L	Lift coefficient
D	Diameter, (m)
l	Distance between centers, (m)
P	Pressure, circumferential length
Re	Reynolds number, $U_\infty D/\nu$
t	time (s)
u	x-direction velocity, (m/s)
v	y-direction velocity, (m/s)

A. Greek

ρ	Density, (kg.m ⁻³)
ν	fluid kinematic viscosity, (m ² .s ⁻¹)

B. Subscripts

flat	flat tube
Cir	Circular tube
eq	Equivalent
∞	Free stream

REFERENCES

[1] A. Zukauskas, J. Ziugzda, *Heat transfer of a cylinder in crossflow*, Hemisphere Publishing Corp., Washington, DC., 1985.
 [2] M. Zdravkovich, *Flow around circular cylinders*, vol. 1. Fundamentals, Oxford University Press, 1997.
 [3] L. Rocha, F. Saboya, J. Vargas, "A comparative study of elliptical and circular sections in one-and two-row tubes and plate fin heat exchangers," *Int. J. Heat Fluid Flow*, vol. 18, 1997, pp. 247-252.
 [4] A. Safwat Wilson, M. Khalil Bassiouny, "Modeling of heat transfer for flow across tube banks," *Chem. Eng. Process. Process Intensif.*, vol. 39, 2000, pp. 1-14.
 [5] A. Nouri-Borujerdi, A. Lavasani, "Flow Visualization Around a Non-Circular Tube," *International Journal of Engineering Transactions B: Applications*, vol. 19, 2006, pp. 73.
 [6] A. Nouri-Borujerdi, A. Lavasani, "Drag of a single noncircular cylinder," *Proc. 18th Int. Symp. Transport Phenomena*, 2007, pp. 27-30.

[7] A. Nouri-Borujerdi, A.M. Lavasani, "Pressure Loss and Heat Transfer Characterization of a Cam-Shaped Cylinder at Different Orientations," *ASME Journal of Heat Transfer*, vol. 130, 2008, pp. 124503-124503.
 [8] A. Nouri-Borujerdi, A. Lavasani, "Experimental study of forced convection heat transfer from a cam shaped tube in cross flows," *Int. J. Heat Mass Transfer*, vol. 50, 2007, pp. 2605-2611.
 [9] A.M.A. Lavasani, "The Effect of Angle of Attack on Pressure Drag from a Cam Shaped Tube," *World Academy of Science, Engineering and Technology*, vol. 5, 2011, pp. 204-208.
 [10] A.M.A. Lavasani, H. Bayat, "Heat Transfer from Two Cam Shaped Cylinders in Tandem Arrangement," *World Academy of Science, Engineering and Technology*, International Science Index 61, 6(1), 2012, pp. 1095 - 1098.
 [11] A.M.A. Lavasani, H. Bayat, "Heat Transfer from Two Cam Shaped Cylinders in Side-by-Side Arrangement," *World Academy of Science, Engineering and Technology*, International Science Index 67, 6(7), 2012, pp. 1008 - 1011.
 [12] A.M.A. Lavasani, H. Bayat, "Flow around Two Cam Shaped Cylinders in Tandem Arrangement," *World Academy of Science, Engineering and Technology*, International Science Index 67, 6(7) 2012, pp. 1079 - 1082.
 [13] A. Mirabdollah Lavasani, H. Bayat, T. Maarefdoost, "Experimental Study of Convective Heat Transfer from In-Line Cam Shaped Tube Bank in Crossflow," *Appl. Therm. Eng.*, vol. 65, 2014, pp. 85-93.
 [14] H. Bayat, A. Mirabdollah Lavasani, T. Maarefdoost, "Experimental Study of Thermal-Hydraulic Performance of Cam-Shaped Tube bundle with Staggered Arrangement," *Energy Conversion and Management*, vol. 85, 2014, pp. 470-476.
 [15] H.M. Bahaidarah, N. Anand, H. Chen, "A numerical study of fluid flow and heat transfer over a bank of flat tubes," *Numerical Heat Transfer, Part A: Applications*, vol. 48, 2005, pp. 359-385.
 [16] N. Benarji, C. Balaji, S. Venkateshan, "Unsteady fluid flow and heat transfer over a bank of flat tubes," *Heat and mass transfer*, vol. 44, 2008, pp. 445-461.
 [17] S. Patankar, *Numerical heat transfer and fluid flow*, CRC Press, 1980.
 [18] J. Meneghini, F. Saltara, C. Siqueira, J. Ferrari Jr, "Numerical simulation of flow interference between two circular cylinders in tandem and side-by-side arrangements," *Journal of fluids and structures*, vol. 15, 2001, pp. 327-350.
 [19] H. Ding, C. Shu, K. Yeo, D. Xu, "Numerical simulation of flows around two circular cylinders by mesh - free least square - based finite difference methods," *International journal for numerical methods in fluids*, vol. 53, 2007, pp. 305-332.
 [20] N. Mahir, Z. Altaç, "Numerical investigation of convective heat transfer in unsteady flow past two cylinders in tandem arrangements," *International Journal of Heat and Fluid Flow*, vol. 29, 2008, pp. 1309-1318.
 [21] C. Williamson, "Oblique and parallel modes of vortex shedding in the wake of a circular cylinder at low Reynolds numbers," *Journal of Fluid Mechanics*, vol. 206, 1989, pp. 579-627.