International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:8, No:8, 2014

Designing a Low Speed Wind Tunnel for Investigating Effects of Blockage Ratio on Heat Transfer of a Non-Circular Tube

Arash Mirabdolah Lavasani, Taher Maarefdoost

***Abstract*—**Effect of blockage ratio on heat transfer from non-circular tube is studied experimentally. For doing this experiment a suction type low speed wind tunnel with test section dimension of 14×14×40 and velocity in rage of 7-20 m/s was designed. The blockage ratios varied between 1.5 to 7 and Reynolds number based on equivalent diameter varies in range of 7.5×10^3 to 17.5×10^3. The results show that by increasing blockage ratio from 1.5 to 7, drag coefficient of the cam shaped tube decreased about 55 percent. By increasing Reynolds number, Nusselt number of the cam shaped tube increases about 40 to 48 percent in all ranges of blockage ratios.

***Keywords*—**Wind tunnel, non-circular tube, blockage ratio, experimental heat transfer, cross-flow.

I. Introduction

FLOW around bluff bodies mounted near solid surfaces in cross-flow has many applications such as pipelines in seabed, cross-flow heat exchanger, and electric heating elements. 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.

In industrial heat exchangers and in most experimental rigs, the tubes are placed in channels. The effect of channel walls is expressed by the blockage factor, which is the ratio of the cylinder diameter to the channel height, D/H. An increase of the blockage factor involves basic changes of the pressure distribution with consequential changes in the velocity distribution outside the boundary layer and in the velocity gradient. Zukauskas [1] analyzed the study of Akilba'yev et al. [3] about the effect of blockage ratio on the flow and heat transfer of a cylinder in a restricted channel. Mohamed et al. [4] experimentally investigated forced convection heat transfer from the outer surface of triangular cylinder in cross flow of air. The blockage ratios varied in ranges of 0.066 to 0.263. Their results showed that using triangular cylinder instead of circular and square cylinders at high Reynolds number enhances the heat transfer. Chakrabarty and Brahma [5] carried out experimental investigation in fluid flow and heat transfer at Reynolds number Re = 49×10^3 and blockage ratios from 0.1 to 0.4 with aspect ratio varied in range of 1.5 to 0.333, to study the effect of wall proximity around rectangular cylinder. They found that the value of average Nusselt number decreases as the prism moves in the direction of the upper wall of wind tunnel. Singha and Sinhamahapatra [6] studied numerically flow around a circular cylinder placed centrally inside a channel with Reynolds number and blockage ratio varying in range of 45 to 250 and 2 to 8, respectively. Their results showed that the mean drag coefficient and Strouhal number decreases with increasing channel height.

Nouri-Borujerdi and Lavasani [7]-[11] experimentally studied flow and heat transfer from cam-shaped tube in cross-flow of air. Their results indicate that single cam-shaped tube performed better than circular tube. Lavasani and Bayat [12]-[14] numerically studied flow and heat transfer characteristics from two cam-shaped tubes in side-by-side and tandem arrangement. Mirabdolah Lavasani et al. [15] experimentally studied convective heat transfer from cam-shaped tube bank with in-line arrangement and Bayat et al. [16] 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.

A. M. Lavasani is Assistant Professor of Department of Mechanical Engineering, Central Tehran Branch, Islamic Azad University, Tehran, Iran, (phone: +98 9381299907; e-mail: arashlavasani@iauctb.ac.ir).
T Maarefdoost is member of Young Researchers and Elite Club, Central Tehran Branch, Islamic Azad University, Tehran, Iran (e-mail: t.marefdoost@gmail.com).

II. Design Parameter of Low Speed Wind Tunnel

A. Settling Duct and Nozzle

The settling duct is located at the entrance of the wind tunnel and contains the honeycomb. It used to moderate longitudinal variations in the flow. In this study settling duct has a square cross section with 56 cm width by 56 cm height and 28 cm length. Honeycombs are located in the settling duct and are used to reduce no uniformities in the flow. For optimum benefit, honeycombs should be 0.6-1.2 mm cell diameters and the length of the honeycombs should be 6-8 times the hydraulic diameter of the honeycomb holes [17]. In this work a plastic honeycomb with square cross section and length of 8 cm and cell diameters of 1 cm has been designed.

Contractions sections (Nozzle) are located between the settling chamber and the test sections and serve to both increase mean velocities at the test section inlet and moderate inconsistencies in the uniformity of the flow. Small tunnels typically have contraction ratios between 8 and 12 and appropriate length for nozzles varies between 1.2D to 2D [16]. In this work contraction ratios is 12:1 and the length of the nozzle is 68 cm.

B. Test Section

The test section should be designed to match the upstream contraction section and downstream diffuser section. A straight test section of 16 cm width by 16 cm height and 40 cm length has been fabricated.

C. Diffuser and Fan

To smoothly decelerate the airflow a diffuser should be located downstream of the test section. Diffusers are chambers that slowly expand along their length, allowing fluid pressure to increase and decreasing fluid velocity. Diffusers are located downstream of the test section and are used to recover pressure from kinetic motion of the fluid thereby reduced the power required to drive the tunnel. Wide angle diffuser design should provide completely attached flow so the divergence angle should be restricted to 3 degrees [17]. In this work the diffuser angel is 3 degree.

An adjustable fan is needed to suck down the airflow in the wind tunnel. An adjustable Speed Drive, 750 W, 3-phase variable frequency inverter / controller to regulate the fan shaft RPM is needed. The fan generates the 5-14.1 m/s for test section.

D. Flow Simulation inside Wind Tunnel

In this study before making the wind tunnel simulations is done by using finite volume method. The computational grid is shown in Fig. 1. The second order upwind scheme was chosen for interpolation of the flow variables. The SIMPLE algorithm has been adapted for the pressure velocity coupling. In all simulation, a convergence criterion of 1×10^{-6} was used for all variables. Velocity Contours inside wind tunnel and test section is shown in Fig. 1. The velocity contours in wind tunnel and test section is represented in Fig. 2.

III. Experimental Setup

Fig. 3 shows the cross section profile of the cam shaped tube that comprised of four parts, two circles which two arcs segments are tangent to them. The tube has identical diameters are equal to d=8 mm and D=16 mm where distance between their centers is l=15.75 mm. Characteristic length for this tube is the diameter of an equivalent circular cylinder, $D_{eq}=P/\pi=22.44$ mm, whose circumferential length is equal to that of the cam-shaped cylinder.

As it is shown in Fig. 3 the test tube is mounted horizontally perpendicular to the flow direction in test section of open loop wind tunnel. For creating different blockage ratios, two plates with thickness of 2 mm were mounted in test section where the distance between plates, H, changes in order to have several H/D_{eq} values.

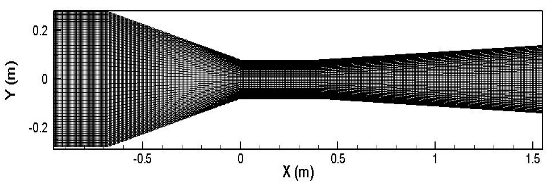
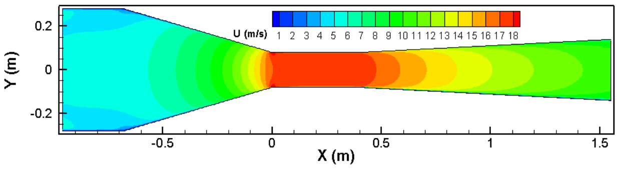
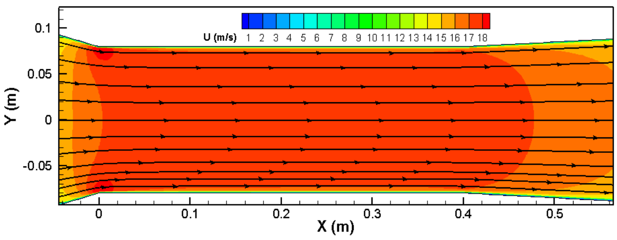


Fig. 1 Computational Grid



(a)



(b)

Fig. 2 Velocity Contours: (a) Wind Tunnel (b) Test section

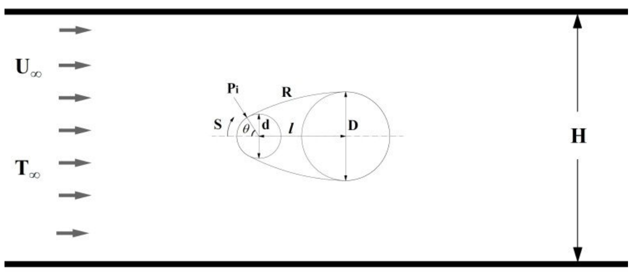


Fig. 3 Schematic of cam-shaped tube between parallel walls

A pitot static tube is used to measure the free stream velocity in front of the test section. The air velocity varies from 5.9 to 14.1 m/s by controlling a variable speed motor.

The pressure drag coefficient C_D is determined experimentally from pressure distribution over the cam shaped-tube surface, including the large and small circles as well as two tangent arcs between them as follows:

$$C_D = \frac{1}{D_{eq}}\left\{\sum_{i=1}^{14} C_{p,i} Cos\,\theta_i \Delta S_i\right\} \quad (1)$$

θ is different for each of the holes. This angle denotes the angle between the normal vector on the tube surface and free stream. S denotes the surface distance from leading edge of the cylinder and ΔS_i represents a length on the tube perimeter belong to each hole. The pressure distribution on the cam shaped is expressed in dimensionless form by the pressure coefficient, $C_{p,i}$.

$$C_{P,i} = \frac{p_i - p_\infty}{0.5\rho_{air}U_\infty^2} \quad (2)$$

where P_i is the static pressure measured by a differential pressure meter at the location of the holes drilled on the tube surface. P_∞, U_∞ and ρ_{air} are the pressure, velocity and density of the free stream, respectively. Drag coefficient of circular tube was measured and its results are compared White [18], in order to assure that current setup is working properly for measuring drag coefficient of cam shaped tube.

The rate of heat transfer from the tube to the air is obtained by measuring mass flow rate and temperature of water at inlet and outlet of tube:

$$\dot{Q}_w = \dot{m}_w C_{p,w} \left(T_{wi} - T_{wo}\right) \tag{3}$$

where $\dot{m}_w = \rho_w \dot{V}_w$ which $C_{p,w}$, ρ_w and $\dot{V}_w$ are specific heat, density and volume flow rate of water, respectively.

The mean Nusselt number is determined as follows:

$$\overline{N}u_{eq} = \frac{\overline{h}D_{eq}}{k} = \frac{\dot{Q}_w}{\pi Lk\left(T_s - T_\infty\right)} \tag{4}$$

where temperature of tube surface is defined by $T_s = \left(T_{wi} + T_{wo}\right)/2$.

IV. Result and Discussion

A single circular tube with diameter of 2.73 cm and length of 30 cm is tested before testing the cam shaped-tube, to verify the data-taking process and to check the related equipment setup. Figs. 4 and 5 compare the present results with the results of White [18] and Zukauskas [1], respectively. The difference between drag coefficients of present work with White [18] is about 3 percent and the difference between Nusselt numbers of present work with Zhukauskas is about 2 to 5 percent. Therefore, it can be concluded that the present experimental set up can be used for measuring pressure drag and heat transfer of a cam shaped tube.

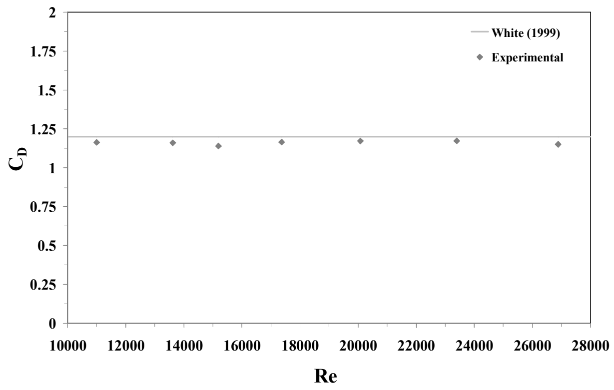


Fig. 4 Drag coefficient of circular tube in cross-flow

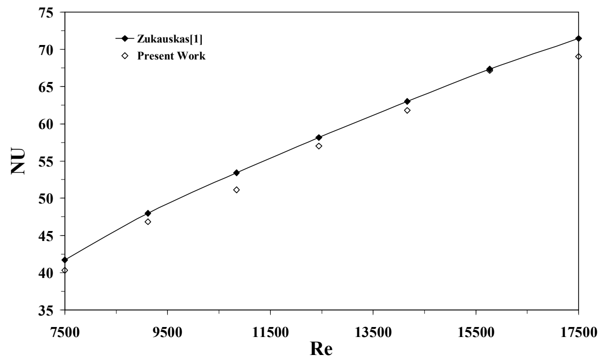


Fig. 5 Heat transfer circular tube in cross-flow

The drag coefficient was obtained by integrating pressure distribution around the circumference of the cam shaped tube in the direction of air flow. The variation of drag coefficient for blockage ratios 1.5 and 7 is presented in Fig. 6. Result shows that as the value of blockage ratio increases, the drag coefficients continuously decreases for all ranges of Reynolds numbers. As the blockage ratio increases from 1.5 to 7 the mean value of C_D decreases from 1.054 to 0.456.

The variation of the averaged Nusselt number of the cam shaped tube with blockage ratio and Reynolds number is shown in Fig. 7. It can be concluded from this figure that as the value of blockage ratio decreases, Nusselt number continuously increases for all ranges of Reynolds numbers. For H/Deq = 1.5 and 7 by increasing Reynolds number from 7000 to 17500 Nusselt number for cam shaped tube increases about 40 and 48 percent, respectively. Moreover, results show that by decreasing blockage ratio from 7 to 1.5, heat transfer increases about 8 to 23 percent for Reynolds number of 7000 to 17500.

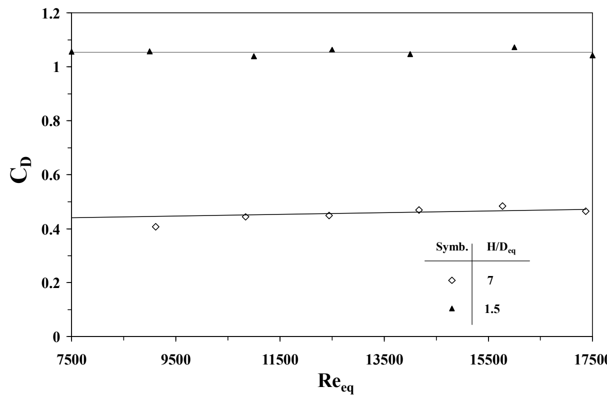


Fig. 6 Drag coefficient of cam-shaped tube

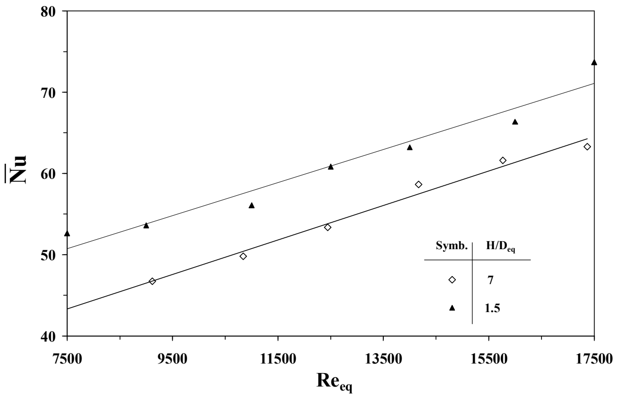


Fig. 7 Heat transfer from cam-shaped tube

Comparison of Nu/C_D of cam shaped tube with circular tube is presented in Fig. 8. Due to aerodynamic shape of cam shaped tubes, its drag coefficient is much lower than circular tube. Therefore, for all rang of blockage ratios and Reynolds number Nu/C_D of cam shaped tube is about 52 to 115 percent greater than circular tube with equivalent diameter.

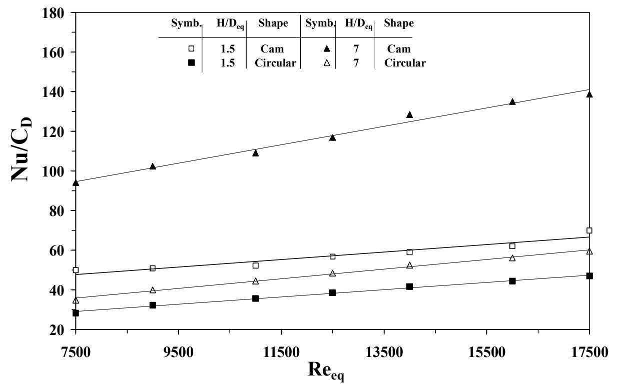


Fig. 8 Comparison of Nu/C_D of cam shaped tube with circular tube

V. Conclusion

In this study flow and heat transfer from a cam shaped tube between two parallel walls were investigated experimentally. The blockage ratios varied between

$1.5 \leq H/D_{eq} \leq 7$ and Reynolds number based on equivalent diameter varies in the range of $7500 \leq Re_{eq} \leq 17500$. The dependency of the Pressure coefficient, pressure drag and Nusselt number for cam shaped tube on the blockage ratio is quite clear from the results. By increasing blockage ratio from 1.5 to 7, drag coefficient decreases about 57% and by decreasing blockage ratio from 7 to 1.5 Nusselt number increases up to 23%.

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] Z.S. Akilba'yev, S. Isata'yev, P. Krashtalev, N. Masle'yeva, "The Effect of Channel Blockage on the Local Heat Transfer Coefficient of a Uniformly Heated Cylinder," *Problemy Teploenergetyki i Prikladnoi Teplofiziki*, vol. 3, 1966, pp. 179-198.

[4] M. Ali, O. Zeitoun, A. Nuhait, "Forced convection heat transfer over horizontal triangular cylinder in cross flow," *International Journal of Thermal Sciences*, vol. 50, 2011, pp. 106-114.

[5] D. Chakrabarty, R.K. Brahma, "Effect of wall proximity in fluid flow and heat transfer from a rectangular prism placed inside a wind tunnel," *International Journal of Heat and Mass Transfer*, vol. 51, 2008, pp. 736-746.

[6] S. Singha, K. Sinhamahapatra, "Flow past a circular cylinder between parallel walls at low Reynolds numbers," *Ocean Engineering*, vol. 37, 2010, pp. 757-769.

[7] A. Nouri-Borujerdi, A. Lavasani, "Flow Visualization Around a Non-Circular Tube," *International Journal of Engineering Transactions B: Applications*, vol. 19, 2006, pp. 73.

[8] A. Nouri-Borujerdi, A. Lavasani, "Drag of a single noncircular cylinder," Proc. 18th Int. Symp. Transport Phenomena, 2007, pp. 27-30.

[9] 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.

[10] 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.

[11] 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.

[12] 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.

[13] 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.

[14] 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.

[15] A. Mirabdolah 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.

[16] H. Bayat, A. Mirabdolah 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.

[17] W.H. Rae, A. Pope, *Low-speed wind tunnel testing*, John Wiley, 1984.

[18] F.M. White, *Fluid mechanics*, McGraw-Hill, Boston, 1999.