World Academy of Science, Engineering and Technology
International Journal of Aerospace and Mechanical Engineering
Vol:9, No:12, 2015

Numerical Simulation of Turbulent Flow around Two Cam Shaped Cylinders in Tandem Arrangement

Arash Mir Abdolah Lavasani, Meghdad Ebrahimi Sabet

Open Science Index, Aerospace and Mechanical Engineering Vol:9, No:12, 2015 publications.waset.org/10003024/pdf

***Abstract*—**In this paper, the 2-D unsteady viscous flow around two cam shaped cylinders in tandem arrangement is numerically simulated in order to study the characteristics of the flow in turbulent regimes. The investigation covers the effects of high subcritical and supercritical Reynolds numbers and L/D ratio on total drag coefficient. The equivalent diameter of cylinders is 27.6 mm The space between center to center of two cam shaped cylinders is define as longitudinal pitch ratio and it varies in range of 1.5< L/D<6. Reynolds number base on equivalent circular cylinder varies in range of 27×10^3< Re <166×10^3 Results show that drag coefficient of both cylinders depends on pitch ratio. However, drag coefficient of downstream cylinder is more dependent on the pitch ratio.

***Keywords*—**Cam shaped, tandem, numerical, drag coefficient, turbulent.

I. Introduction

IT is well known that two circular tubes in tandem arrangement are prevalent in engineering applications, such as tube bundles in heat exchanger. In the tandem arrangement, the flow field depends highly on the configuration and the spacing of the cylinder pair due to both wake and proximity-induced interference effects. There are many experimental studies [1]-[10] devoted to the flow over two circular cylinders with different arrangement. Similar experiment was extended by [1] up to L/d = 7.5 (Re = 1.05×10^5). They measured only C_D of the individual cylinders; therefore, more investigation was needed to clarify the other parameters, such as, C_D rms, C_L rms, wakes, St, boundary layer characteristics around the cylinders. Reference [2] carried out an experimental study on two rotating circular cylinders for $13\times10^3\leq$ Re $\leq 40\times10^3$, 1.6< L / D < 9 They measured the drag coefficient of two cylinders at various mutual distances. Reference [3] investigated experimentally interference between1.6< L / D < 6. Reference [4] studied two cylinders in turbulent flow at $20\times10^3\leq$ Re $\leq 80\times10^3$.They observed that at subcritical Reynolds numbers, the reattachment of separated flow from the upstream cylinder to the downstream one occurred in the critical spacing range of 3 < L / D < 4. Reference [5] measured the combined drag force acting on the two parallel circular wires in tandem arrangement for *L/d* < 4.5. They observed that the minimum drag on two wires in contact is only 40 percent of the drag on one wire alone. This was resulted from the fact that the existence of the downstream wire improved the streaming of the upstream wire. In most cases, the studies were performed at subcritical Reynolds numbers(less than 1.5×10^5) and focused on fundamental supercritical Reynolds numbers, one can mention the studies of [6] at Re = 166×10^3. The reported studies were concerned with two circular cylinders or elliptic cylinders the flow pattern for tandem arrangement was studied numerically by [11]-[16] for both laminar and turbulent regimes. References [17]-[20] numerically studied flow and heat transfer from single and two no-circular tube with different arrangements in laminar flow regime. Reference [21] investigated numerically the flow over a two-dimensional (2D) circular cylinder at a turbulent Reynolds number of 20×10^3 and its control by air blowing from several slots located on the surface of the cylinder, computationally. Non-circular tubes perform better compare to circular tubes [22]-[30]. References [22]-[29] studied flow and heat transfer around non-circular tube bank in cross-flow in in-line and staggered arrangements. They found that cam-shaped tube performs much better than circular tube bank. There has not been extensive research on forces and flow around two cam shaped cylinders in turbulent flow. In the present article, the turbulent flows around two cam-shaped cylinders in a tandem arrangement are numerically simulated. In turbulent regime, simulations are performed at $27\times10^3\leq$ Re $\leq 166\times10^3$ for a range of gaps between $1.5\leq$ L / D ≤6.

II. Problem Description and Governing Equations

The cross section profile of the cylinder comprised some parts of two circles with two-line segments tangent to them. The cylinder have identical diameters equal to d=11 mm and D=22mm with distance between their centers, *l*=13 mm, (Fig. 1). Characteristic length for this tube is the diameter of an equivalent circular cylinder, $D_{eq}=P/\pi$ =27.6 mm, whose circumferential length is equal to that of the cam-shaped cylinder The typical solution domain and the cylinder boundary definition and nomenclature used in this work are shown in Fig. 2. The inlet flow has a uniform velocity U_∞. The velocity range considered covers turbulent flow conditions. The solution domain is bounded by the inlet, the outlet, and by the plane confining walls, AB and CD. These are treated as solid walls, while AC and BD are the flow inlet and outlet planes. In order to decrease the effect of entrance and outlet regions, the upstream and downstream lengths are $15D_{eq}$ and 50 D_{eq}, respectively and for neglecting the wall effects on cylinders the distance between walls is $30D_{eq}$.

Arash Mir Abdolah Lavasani, Assistant Professor, and M. Ebrahimi Sabet. Ms. Student, are with the Department of Mechanical Engineering, Islamic Azad University Central Tehran Branch, Tehran, Iran (Corresponding author; phone: +98-2144600078; fax: +98-21 44600071; e-mail: arashlavasani@iauctb.ac.ir, ebrahimis1365@gmail.com).

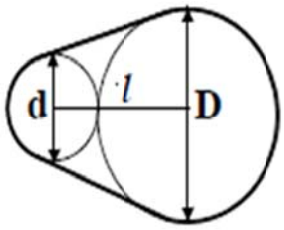


Fig. 1 Schematic of a cam shape

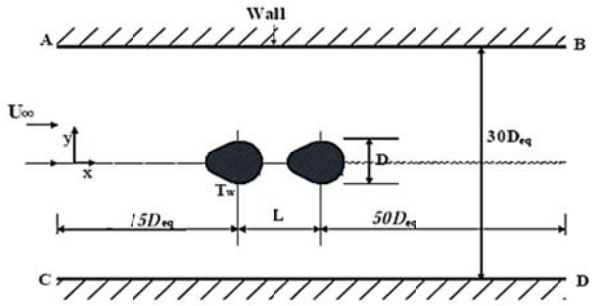


Fig. 2 Solution domain

Equations are written for conservation of mass and momentum in two dimensions. Cartesian velocity components U are used, and it has been assumed that the flow is unsteady and turbulent, while the fluid is incompressible and Newtonian with constant transport properties. Consider the unsteady, two-dimensional, incompressible and turbulent flow past two cam shaped cylinders in tandem arrangement in cross flow in the simulations. Conservation of mass and momentum equations solved are as:

$$\rho \frac{\partial ui}{\partial xi} = 0 \tag{1}$$

$$\rho\left[\frac{\partial u}{\partial t} + \frac{\partial (uiuj)}{\partial xj}\right] = -\frac{\partial p}{\partial xi} + \frac{\partial}{\partial xj}\left[\mu\left(\frac{\partial ui}{\partial xj} + \frac{\partial uj}{\partial xi}\right)\right] + \frac{\partial\left(-\rho\overline{u'iu'j}\right)}{\partial xj} \tag{2}$$

Boussinesq approximation given by (3) is utilized to model Reynolds stress term in (2).

$$\rho\overline{u'iu'j} = \mu\left(\frac{\partial ui}{\partial uj} + \frac{\partial uj}{\partial xi}\right) \tag{3}$$

Equations (1)-(3) are solved computationally by using CFD simulation.

III. Numerical Method

CFD simulations are performed by using RANS equations and Spalart-Allmaras turbulence modeling. Incompressible turbulent flow is modeled with double precision in the simulations Spalart-Allmaras turbulence model is used to model turbulent structures since it is very efficient for external airflows with separations. Two circular cylinders are tested before two cam shape cylinders, because it was necessary to verify the data taking process and to check turbulence model. In Table I, mean drag coefficients for two circular cylinders at Re = $40\times10^3$and L/D = 2.3 computed with different turbulence models are compared with experimental results. The non-dimensional time step for this flow regime is set to 0.002. It is clear from Table I that Spalart-Allmaras model offer more accurate results than the other models.

TABLE I
CFD Results Obtained for Two Circular Cylinders at RE=40×10^3 and L/D = 2.3

Turbulence Mode	C_{d1}	C_{d2}	Diff. C_{d1}(%)	Diff. C_{d2}(%)
Standard k −ε	0.81	0.28	15.6	20
RNG k −ε	0.87	0.33	9.4	5.7
Spalart-Allmaras	0.93	0.38	3.1	8.5
Standard K-ω	0.9	0.29	6.25	17.1
SST K-ω	0.91	0.31	5.2	11.4

TABLE II
Verification of Mesh Independency for Two Cam Shaped Cylinders at RE=40×10^3 and L/D = 2.5

Number of cells	C_{d1}	C_{d2}
62000	0.42	0.14
96000	0.42	0.15
130000	0.43	0.14

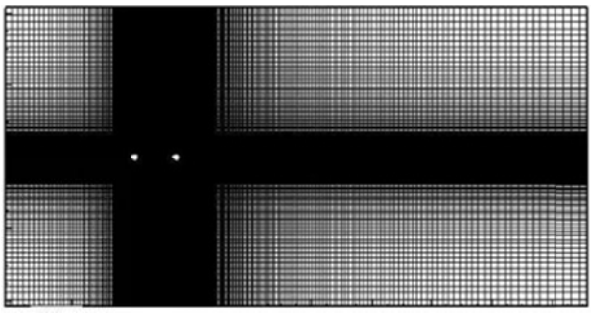
TABLE III
Verification of Time Steps Independency for Two Cam Shaped Cylinders at RE=100×10^3 and L/D = 2.5

$\Delta t\times10^{-4}$ (sec.)	C_{d1}	C_{d2}
1.5	0.38	0.12
2	0.38	0.12

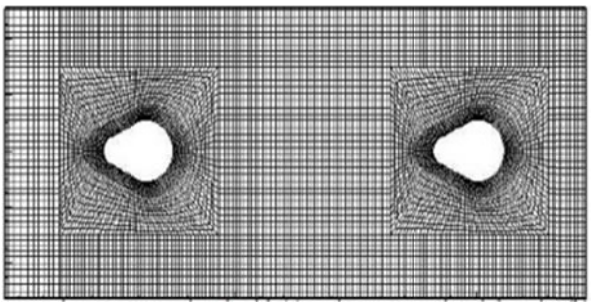
For the simulations presented here, depending on the geometry used, fine meshes of 62000 to 130000 elements for L/D=1.5 to 6 were used (Table II). The computational grids used in this work were generated using the set of regions shown in Fig. 3. In this domain quadrilateral cells are used in the regions surrounding the cylinder walls and the rest of the domain. In all simulation, a convergence criterion of 1×10^{-6} was used for all variables. The governing equations with appropriate boundary conditions are solved using finite volume approach based in Cartesian and coordinate systems. The second order upwind scheme was chosen for interpolation of the interpolation of the flow variables. The SIMPLEC algorithm has been adapted for pressure-velocity coupling and Second-order upwind method for spatial discretization. The thickness of the first cell in the boundary layer is 0.02mm.To demonstrate that the results are grid-independent; we carried out numerical simulations for several grids with different sizes. Furthermore, to show the independency of results from the grid size, mesh independency is verified through different grids at Re = $40\times10^3$and L/D = 2.5. There is 2.3 percent difference between these grids. So the solution is sufficiently grid independent and hereinafter other simulations of the article are carried out by using this mesh type. For temporal discretization, it is employed a time-step of $\Delta t = 3.0\times10^{-3}$ s or $\Delta t^* = \Delta t u_\infty/D = 3\times10^{-1}$ two different time-step sizes (Δt) are tested for two cam shaped cylinders at Re=$100\times10^3$and L/D = 2.5and a vortex shedding period is modeled with $\Delta t=2\times10^{-3}$ seconds and $\Delta t=1.5\times10^{-3}$ seconds, respectively. Table III

Open Science Index, Aerospace and Mechanical Engineering Vol:9, No:12, 2015 publications.waset.org/10003024/pdf

shows drag coefficients (C_d) value obtained as a result of performed simulations with Grid-1. A time step of $\Delta t=1.5\times10^{-3}$ seconds is selected for the rest of the calculations.



(a) Entire computational domain



(b) Closer view around cylinders

Fig. 3 Computational grid

IV. Results and Discussion

For the purpose of the validation of the solution procedure, it is essential that CFD simulations be compared with experimental data. Fig. 4 compares the present results for two circular cylinders in the range of $2.7\times10^4< Re < 1\times10^5$ and L/D =1.6 for drag coefficient with the results of [1]-[3]. For supercritical Reynolds numbers in Table IV drag coefficients for two circular cylinders at $Re=1.6\times10^5$ and L/D=1.435 compared with the results of [6]. There is a difference of about 5-9 percent between the present results and the results of experimental studies. It can therefore be concluded that the CFD code can be used to solve the flow fields for similar geometries and conditions.

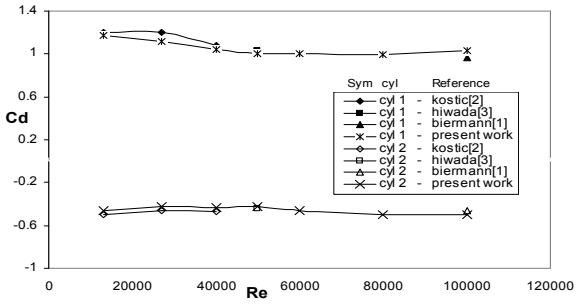
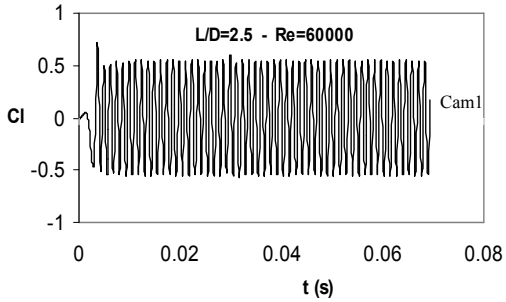


Fig. 4 Comparison of experimental and numerical Variation of drag coefficient with Reynolds number for two circular cylinders

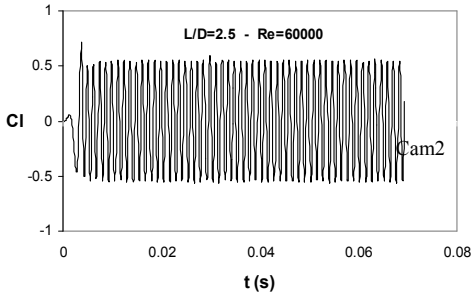
TABLE IV
Comparison of Drag Coefficients for Two Circular Cylinders at Supercritical Reynolds Number

L/D =1.435and $Re=166\times10^3$	C_d	
	Jenkins[14]	present work
Cyl. 1	0.6	0.61
Cyl.2	-0.17	-0.15

Figs. 5 and 6 show time variation of lift and drag Coefficient at $Re=60\times10^3$ and L/D = 2.5 for two cam shaped cylinders.



(a) Cam1



(b) Cam2

Fig. 5 Time Variation of lift Coefficient at $Re=60\times10^3$ and L/D = 2.5 for two cam shaped cylinders

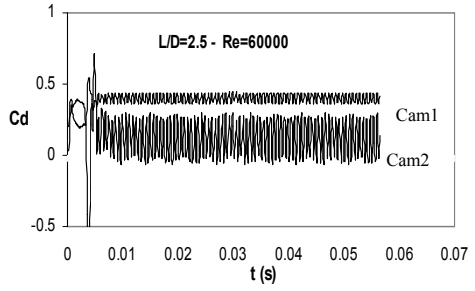
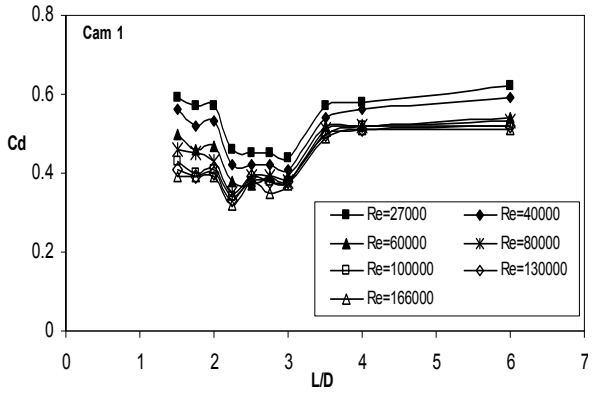


Fig. 6 Time Variation of drag Coefficient at $Re=60\times10^3$ and L/D = 2.5 for two cam shaped cylinders

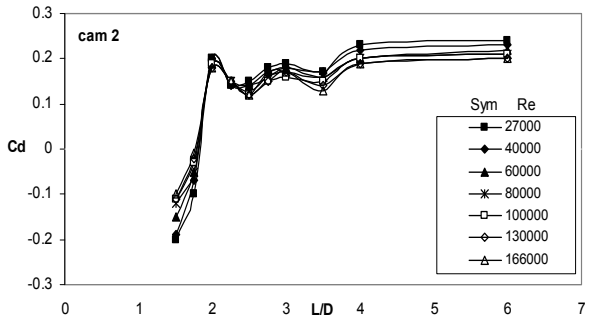
The effect of the L/D ratio from 1.5 to 6 over total drag coefficient for cam shaped cylinders presented in Fig. 7. However, the drag coefficient for the downstream cylinder increases about 100 to 200 percent as the L/D ratio increase from 1.5 to 6 but the drag coefficient for upstream cylinders will be more like single cylinder in cross flow.

Open Science Index, Aerospace and Mechanical Engineering Vol:9, No:12, 2015 publications.waset.org/10003024/pdf

Open Science Index, Aerospace and Mechanical Engineering Vol:9, No:12, 2015 publications.waset.org/10003024/pdf



(a) Cam1



(b) Cam2

Fig. 7 Variation of drag coefficient with Reynolds number and pitch ratio for two cam shaped cylinders

Fig. 8 represents changing of cylinder shape from circular to cam shaped with same L/D ratio lead (L/D=1.5) to decrease drag coefficient about 50-100 percent.

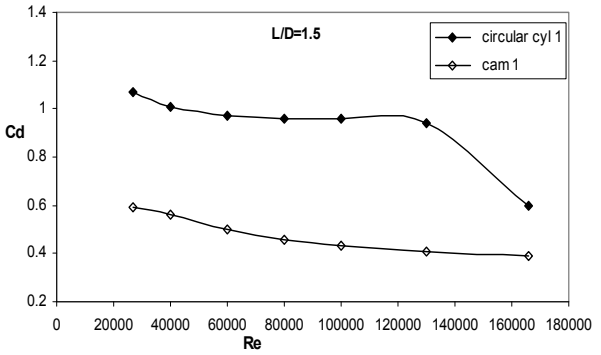


Fig. 8 Comparison drag coefficient for the cam and circular cylinder

V. Conclusion

In this study, flow around two cam shaped cylinders had been investigated. The investigation covers the effects of high subcritical, supercritical, and transcritical Reynolds numbers and L/D ratio on total drag coefficient. Controlled flow simulations are performed further with the same grid and numerical methods.

The drag coefficient increases about 100 to 200percent respectively when L/D ratio increases from 1.5 to 6.Changing the cylinders shape from circular to cam shaped with same transverse pitch and longitudinal pitch reduces drag coefficient about 50 to 100 percent.

Nomenclature

A	Area, m²
C_d	Drag coefficient, $2F_d/(\rho u_\infty^2 A)$
C_L	Lift coefficient, $2F_L/(\rho u_\infty^2 A)$
d	Small diameter
D	Large diameter
L	Distance between centers
P	Pressure, circumferential length
Re	Reynolds, number [ρUD/μ]
SL	Longitudinal pitch
ST	Transverse pitch
U	x-direction velocity
Δt	Time step
x	x coordinate
y	y coordinate

Greek

ρ	Fluid density
μ	Fluid dynamic viscosity
μ_t	Turbulent viscosity

Subscripts

Cam	Cam-shaped cylinder
Cyl	Cylinder
eq	Equivalent
∞	Free stream

References

[1] Biermann D. and Herrnstein, Jr., "The Interference Between struts in Various Combinations", *National Advisory Committee for Aeronautics* Tech. Rep. 468, 1933.

[2] Kostic,G., Oka, S. N., "Fluid flow and heat transfer with two cylinders in cross-flow International Journal of Heat and Mass Transfer",Vol.15,1972, 279–299.

[3] Hiwada, M., Mabuchi, I.,Yanagihara, H., "Fluid flow and heat transfer around two circular cylinders", Bulletin of the JSME 25, 1982,pp.1737–1745.

[4] Zhang H., Melbourne W. H. "Interference between two circular cylinders in tandem in turbulent Journal of Wind Engineering and Industrial Hydrodynamics", Vol.41, 1992, pp. 589-600.

[5] Pannell, J. R., Grffiths, E. A. and Coales, J. D., "Experiments on the nterference between Pairs of Aeroplane Wires of Circular and Rectangular Cross-Section", (British) Advisory Committee for Aeronautics , Reports and memoranda No. 208, 7, 1915, pp,219-221 .

[6] Jenkins L. N., Neuhart D. H., McGinley C. B., Choudhari M. M., Khorrami M. R., "Measurements of Unsteady Wake Interference Between Tandem Cylinders". 15 th AIAA/CEAS Aero acoustics Conference, 11-13 May 2009, Miami, Florida.

[7] Lee K., Yang K. S., "Flow patterns past two circular cylinders in Proximity", International journal Computers & Fluids, Vol. 38, 2009.

[8] Sumner. D, "Two circular cylinders in cross-flow: A review," Journal of Fluids and Structures, Vol.26, 2010, pp.849–899.

[9] Moriya. M., Alam. M. and Takai K. et al., "Fluctuating fluid forces of two circular cylinders in tandem arrangement at close spacing", Transactions of the Japan Society of Mechanical Engineers,Vol.68, 2002, pp.1400-1406.

[10] Zdravkovich M. M. "Flow Around Circular Cylinders Volume: Fundamentals". Oxford University Press: Oxford, 1997.

[11] Ota. T., Nishiyama. H., "Flow around Two Elliptic Cylinders in Tandem Arrangement," *Journal Fluids Eng*. Vol. 108, 1986, Issue 1, pp.98-103.

[12] Mittal S., Kumar V., Raghuvanshi A. Unsteady incompressible _ows past two cylinders in tandem and staggered arrangements. International Journal for Numerical Methods in Fluids, Vol.25, 1997, pp.1315 –1344.

[13] Meneghini J. R., Saltara F., Siqueira C. L. R., Ferrari Jr. J. A. "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.
[14] Dehkordi, B., Moghaddam. H., Jafari. H., "Numerical Simulation of Flow over two Circular Cylinders in Tandem Arrangement," Journal of Hydrodynamics, Vol. 23, 2011, pp.114-126.
[15] Ding H., Shu C. and Yeo K. S. et al. "Numerical simulation of flows around two circular cylinders bymesh-free least square-based finite difference methods", International Journal for Numerical Methods in Fluids,Vol.53, 2007, pp. 305-33.
[16] Kitagawa. T, Ohta. H, "Numerical investigation on flow around circular cylinders in tandem arrangement at a subcritical Reynolds number", Journal of Fluids and Structures,Vol.24 (5), 2008, pp. 680-699.
[17] Lavasani, A. M., Bayat, H., "Flow around a Cam-Shaped Cylinder between parallel walls", 7th International Chemical Engineering Congress & Exhibition, 2011, Kish Island, Iran.
[18] Lavasani, A. M. A., Bayat, H., "Heat Transfer from Two Cam Shaped Cylinders in Side-by-Side Arrangement", World Academy of Science, Engineering and Technology, International Science Index 67, 2012. 6(7): pp. 1008 - 1011.
[19] Lavasani, A. M. A., Bayat, H., "Flow around Two Cam Shaped Cylinders in Tandem Arrangement", World Academy of Science, Engineering and Technology, International Science Index 67, 2012. 6(7): pp. 1079 - 1082.
[20] Lavasani, A. M. A., Bayat, H., "Heat Transfer from Two Cam Shaped Cylinders in Tandem Arrangement", World Academy of Science, Engineering and Technology, International Science Index 61, 2012. 6(1): pp. 1095 - 1098.
[21] Apacoglu. B, Paksoy. A and Aradag. S, "Effects of air blowing on turbulent flow over a circular cylinder"J. of Thermal Science and Technology, Vol.32 (2), 2012 pp.107-119.
[22] Mirabdolah Lavasani, A., Bayat, H., Maarefdoost, T., "Experimental Study of Convective Heat Transfer from In-Line Cam Shaped Tube Bank in Crossflow", Appl. Therm. Eng., 2014. 65 (1–2): pp. 85–93.
[23] Mirabdolah Lavasani, A., Bayat, H., Eftekhari Yazdi, M., Maarefdoost, T., "Experimental investigation of Flow around Non- Circular Tubes in Cross-flow", The 22nd Annual International Conference on Mechanical Engineering, 2014, Shahid Chamran University of Ahvaz, Iran.
[24] Mirabdolah Lavasani, A., T. Maarefdoost, and H. Bayat. "Experimental Study of Blockage ratio Effects on Flow around a Non-circular Tube in Cross-flow", The 22nd Annual International Conference on Mechanical Engineering, 2014, Shahid Chamran University of Ahvaz, Iran.
[25] Mirabdolah Lavasani, A. and H. Bayat. "Heat transfer and Fluid Flow in Developing Region of Elliptical Cylinder", First Iranian Conference on Heat and Mass Transfer, September 2012, Zahedan, Iran.
[26] Bayat, H., A. Mirabdolah Lavasani, and T. Maarefdoost, "Experimental Study of Thermal-Hydraulic Performance of Cam-Shaped Tube bundle with Staggered Arrangement", Energy Conversion and Management, 2014. 85: pp. 470-476.
[27] Bayat, H., Mirabdolah Lavasani, A., Eftekhari Yazdi, M., Maarefdoost, T., "Experimental Study of Heat transfer from Non-Circular Tubes", in The 22nd Annual International Conference on Mechanical Engineering, 2014, Shahid Chamran University of Ahvaz, Iran.
[28] Bayat, H., Mirabdolah Lavasani, A., Bolhasani, M., Moosavi, S., "Numerical Study of Flow around Flat Tube between Parallel Walls", International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering, World Academy of Science, Engineering and Technology, 92, 2014. 92: pp. 1463 - 1466.
[29] Bayat, H., Mirabdolah Lavasani, A., Bolhasani, M., Moosavi, S., "Numerical Investigation of Thermal-Hydraulic Performance of a Flat Tube in Cross-Flow of Air", International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering, World Academy of Science, Engineering and Technology, 92, 2014. 92: pp. 1444 - 1447.
[30] Lavasani, A.M.A., T. Maarefdoost, and H. Bayat, "Effect of blockage ratio on pressure drag and heat transfer of a cam-shaped tube", Heat and Mass Transfer, 2015, DOI: 10.1007/s00231-015-1711-3.

Open Science Index, Aerospace and Mechanical Engineering Vol:9, No:12, 2015 publications.waset.org/10003024/pdf