CFD Time Evolution of Heat Transfer Around A Bundle of Tubes In Staggered Configuration

G.S.T.A. Bangga[1*], W.A. Widodo[2]

[1,2]Department of mechanical engineering
Field of study energy conversion
Institut Teknologi Sepuluh Nopember (ITS), Surabaya, Indonesia
*Corresponding author : probability.schrodinger@gmail.com

Abstract

The necessity to include unsteady effects on the heat transfer phenomenon over shell and tubes heat exchanger becomes increasingly important. The induced vibration by fluctuating forces coefficient could affected the capability of heat exchanger to transfer the energy to the fluid flows, and in other hand, this phenomenon could also endanger the structure of heat exchanger itself. This paper presents a numerical study of the heat transfer on the flow through a bundle of heated tubes at $Re_w = 6.8x10^4$ with constant heat flux 1000 W/m^2. The finite volume method is used in the equations of the mathematical model and unsteady RANS simulation were performed using *SST-kω* turbulence model. The qualitative and quantitative data revealed that there is occurs periodic oscillation on the force acting in a bundle of tubes. It is found that there is exist a *beat* phenomenon which is shown by lift coefficient history by multiple Sthrouhal number $St_1 \approx 0.045$ and $St_2 \approx 0.25$. The tubes which is located on the third row give higher contribution on the fluid heating compared with the others.

Keywords: Heat transfer, Bundle of tubes, Staggered configuration, CFD, Unsteady flows, Time evolution.

Introduction

The study of flow normal to a bank of tubes continues to attract interest because of the importance of this flow configuration in the design of heat exchangers. Many related engineering applications of the heat transfer and flow characteristics of tubes in staggered or in-line tube banks have been presented [1]. Buyruk [2] studied the forced convection heat transfer for tandem, in-line and staggered cylinders configurations. The computations were carried out for constant fluid properties, incompressible fluid, laminar flow and steady state conditions. The ANSYS/Flotran CFD software was used. For equal size cylinders in tandem, the influence of the center-to-center distance on the heat transfer rate was analysed at Re = 400 and Pr = 0.71. Grimison [3] obtained an empirical equation for various spacing ratios of tube bundles from experimental data in the range of $400 \leq Re \leq 27000$, where Re is the Reynolds number based on the uniform velocity far upstream and the tube diameter. Zhukauskas [4] has studied the heat transfer from tubes in cross flow, proposed an equation applicable for a wide range of the Reynolds number of $2 \leq Re \leq 1.2 \text{ x}10^6$ and a few spacing ratios of the tube bundle. Mittal et al. [5] numerically studied incompressible flows past a pair of cylinders at Reynolds numbers 100 and 1000 in tandem and staggered arrangements using a stabilized finite element formulation. Yukio Takemoto et al. [6] studied Heat transfer in the flow through a bundle of tubes and transitions of the flow. It was found that physical quantities such as the Nusselt number and the pressure loss exhibit discontinuous jumps with continuous change in the Reynolds number which mean that to enhance the heat transfer, Reynolds number should be raised larger than the targeted value and then reduce Re to the desired value. Contrarily, to suppress the heat transfer, Reynolds number should be increased monotonously from a small value. Achenbach [7] studied the effect of surface roughness on the heat transfer rate in inline tube bank at high Reynolds number. The roughness parameter is varied from $k/d = 0$ to $9 \text{ x } 10^{-3}$. The results from the local heat transfer measurements contribute to the understanding of the complicated flow around the tubes. The roughness parameter determines the value of the critical Reynolds number from which the improvement of heat transfer starts. The maximum enhancement to be obtained already

occurs for k/d ≈ 3 x 10^{-3}. A further increase of the roughness parameter does not result in a higher heat transfer.
This paper will discuss numerical simulation of the heat transfer and fluid flow over tubes bank in staggered configuration with specified dimension. Unsteady RANS analysis were performed using SST-kω turbulence model. The qualitative and quantitative data would be present on this paper. The main purpose of this research is to contribute in the development of the heat exchanger system related to its capability to transfer energy.

Numerical Method

During the generation of a computational analysis, mesh are the important factor for the accuracy that could be achieved. The influence of the domain size on the solution should always be born in mind and hence a careful selection of the boundary locations becomes vital. In this study the computational boundaries at the upstream, downstream and sides were placed at 8.5D, 21.5D, and 2.75m from the upper and lower side. The configuration of the heat exchanger system and the code number of each cylinder could be seen on figure 1. The meshgrid on a bundle of tubes that being observed is shown on figure 2, in general, mesh profile should represent the flow over cylinder itself, mesh profile should growth follow the contour of the cylinder bodies. The height of the first row of cells is set at a distance to the wall of 10-4D and this corresponds to y+ < 1. For the purpose of accurately resolving the boundary layer behaviour, and thus the aerodynamic loads on the bodies, no wall function is employed and hence the values of y+ should be less than 1.0 [8].
Simulations were performed on the cylinder by uniform velocity and temperature upstream to the bodies is 1 m/s and 303K, at Rew = 6.8x104, and it were used to the models in order to test the numerical convergence of the solution. A bundle of tubes is heated by constant heat flux by value of 1000 W/m2. Convergence is achieved when the residuals of the turbulent transport, momentum transport and pressure-correction equations reach a predetermined value. A finite-volume method was employed with a segregated algorithm to solve the Navier–Stokes and the Turbulent Transport equations [9]. The coupling between the pressure and velocity fields was achieved by using the pressure-correction-based SIMPLE technique and SST-kω turbulence model with viscous heating is used as viscous turbulence modelling of the simulation. Whilst the spatial discretization of the convective terms of the RANS equations and turbulent transport was achieved with discretization using second order upwind. Simulation were performed using unsteady analysis. Unsteady effect is captured on the analysis and time step 0.01s is suffeciently small for the purpose to predict shedding vortex over downstream body of the cylinder.

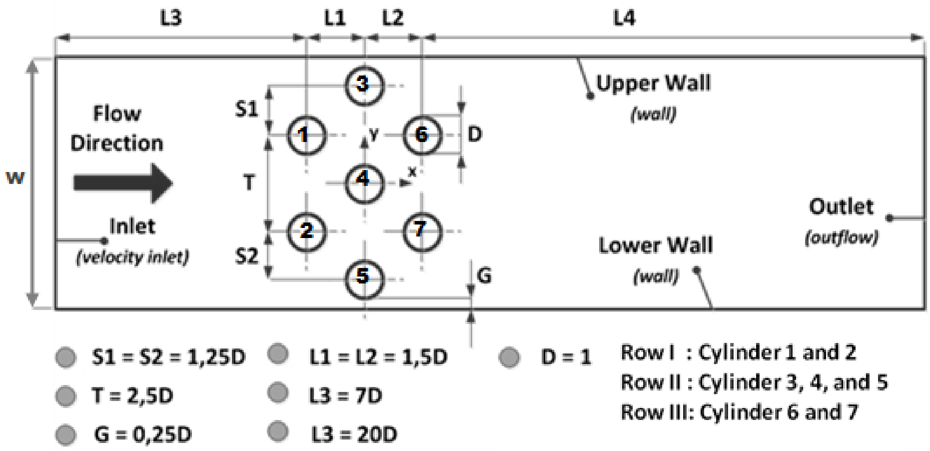

Figure 1. Configuration of the heat exchanger system.

Table 1. Calculated drag and lift coefficient on each mesh at 6.25 s.

Mesh	Number of Grid	Total CL	CL Error	Total CD	CD Error
A	131596	0.005094	-	11.98834	-
B	134956	0.008806	0.7287	11.93049	0.004826
C	138316	0.00284	0.677493	11.88737	0.003614
D	141676	0.002755	0.02993	11.8484	0.003278

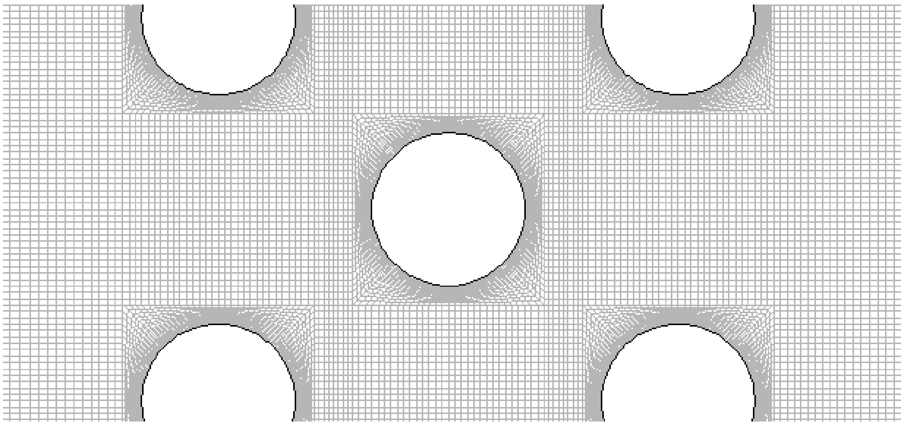

Figure 2. Meshgrid on a bundle of tubes.

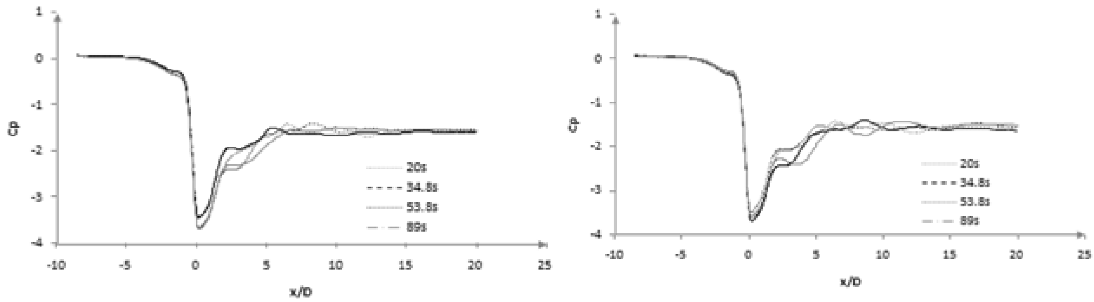

Figure 3. Pressure coefficient distribution at different time on (a) upper wall and (b) lower wall.

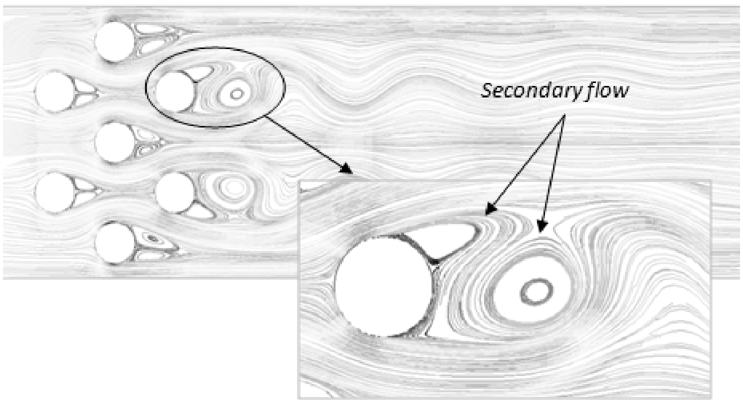

Figure 4. Velocity pathline of a bundle of tubes at 20s.

Results and Discussion

In order to demonstrate the fact that the computational results were independent from the grid density, grid dependence analysis were performed with four different mesh in conjunction with unsteady RANS using SST-kω turbulence model and second order upwind as the discretization on the convective terms and also for the turbulence transport equation. Calculation were performed up to 6.25s in time iteration. Total drag and lift coefficient of a bundle of tubes were used in this analysis. As shown in table 1. Mesh density is increase from the mesh A to mesh D and all of this computational results were simulated using numerical RANS calculation. Mesh C is choosen in this analysis to be used by determining its value of lift and drag coefficient is relatively does not change when the number of grid is increased with error to mesh D is 2.99% in lift coefficient and 0.32% in drag coefficient.

Figure 3 shows pressure coefficient on the upper and lower wall by unsteady analysis at 20s. From the figure, it could be seen that on the position upstream a bundle of tubes has high pressure and this pressure decreases significantly at point x/D = 0 (Centre cylinder) as seen quantitatively on figure 3.

This is agree with Buryuk [10], that the pressure drops with increasing distance from the stagnation point. Unstable region is found at at the region $0 < x < 10$ as seen on wake region on figure 4 and fluctuating pressure coefficient on figure 3. As time increase, pressure loss due to the addition of tubes does not change in significant value and remainin stable by average pressure coefficient value $Cp \approx -1.6$ at the outlet section. A significant change on the pressure as time increase occurs at the wake region on a bundle of tubes ($0 < x < 10$). This phenomenon is caused by unstability of the flow at this region. The interaction among cylinders produce a complex-separated flow on the turbulent wake of downstream cylinder on the second row. Secondary flow which is produced in this region changes periodically with certain frequency. Figure 4 gives the flow interpretation on this wake region by velocity pathline.

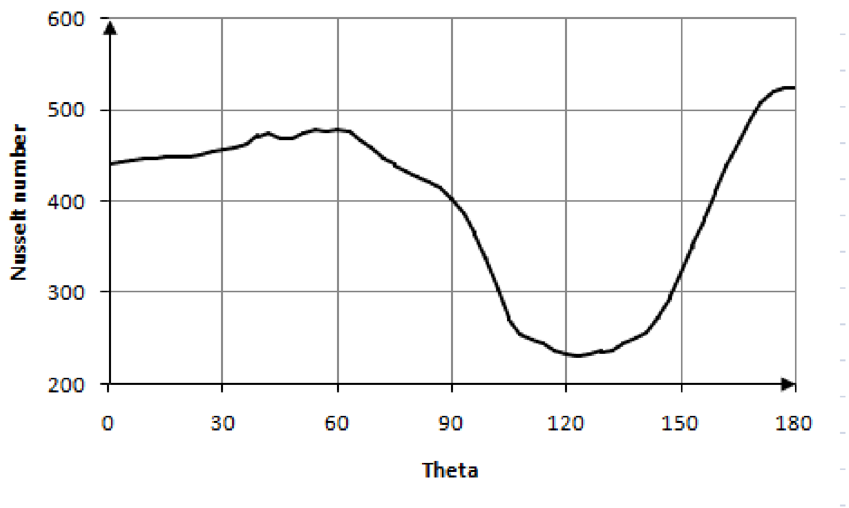

Figure 5. Sectional Nusselt number on cylinder 4 at 53.8s.

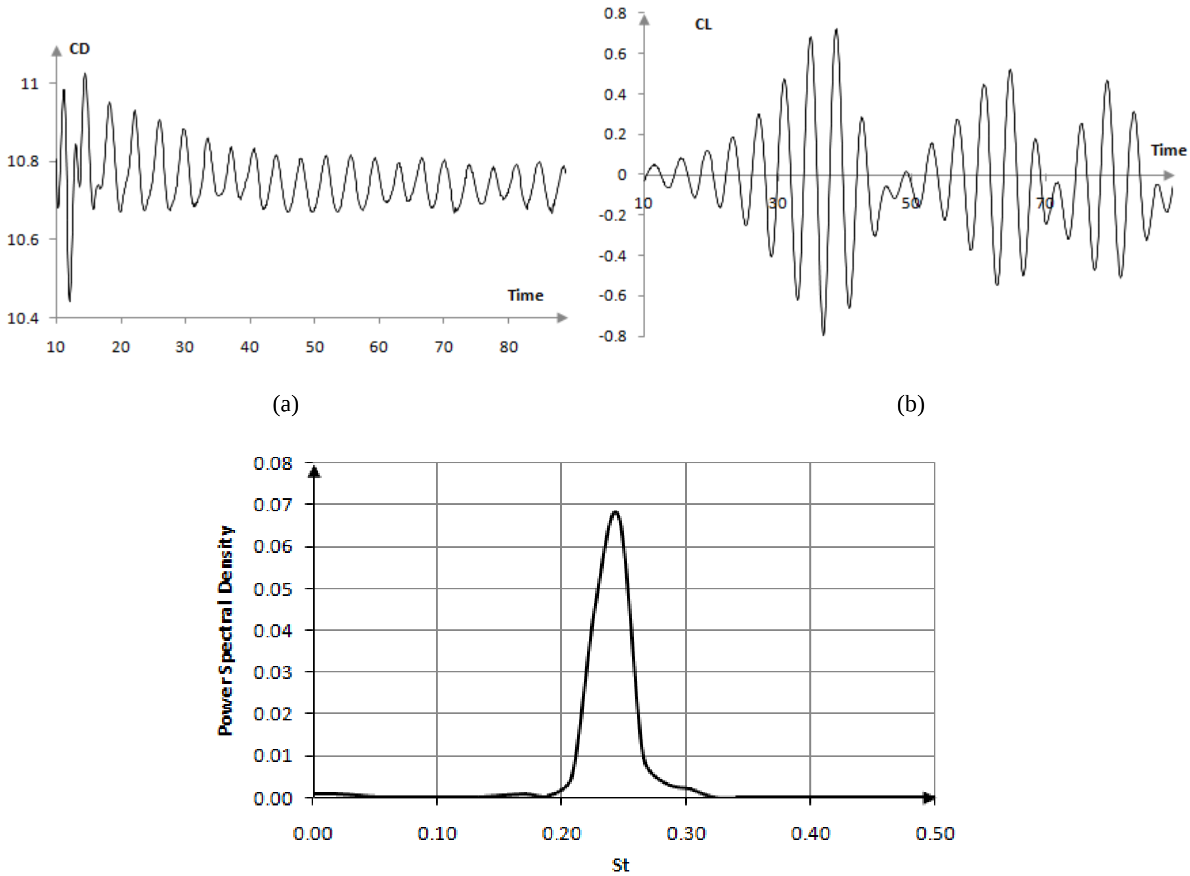

Figure 6. Calculated forces history (a) drag coefficient, (b) lift coefficient, and (c) FFT of lift coefficient.

Table 2. Calculated average total and sectional Nusselt number of a bundle of tubes on different time.

Row	cylinder/time	20s	34.8s	53.8s	89s
I	Cylinder 1	340.161	340.4672	339.8058	340.5213
	Cylinder 2	340.1102	339.7783	340.3262	339.5122
II	Cylinder 3	382.2016	382.8254	382.0746	384.8959
	Cylinder 4	392.9679	393.3662	392.7029	391.9545
	Cylinder 5	382.0449	381.2158	383.3332	380.9084
II	Cylinder 6	452.7258	456.87	442.1643	441.3924
	Cylinder 7	450.7536	441.8181	450.6014	428.043
	$\overline{Nu}$	395.8351	395.1419	394.4378	390.978

Due to unsteady effects, the sectional nusselts number on cylinder 4 is slightly different with single cylinder, when the lift coefficoent values zero, the pattern would be close to single cylinder with maximum value of Nusselt number on the upstream is located in the stagnation point. However, when the lift coefficient is larger or lower than zero, the location of maximum Nusselt number on the upstream cylinder 4 is delayed towards downstream as seen on figure 5.

Fluid flow is heated by a bundle of tubes as the time increases. Tubes which is located on the third row produces higher temperature increment more than any other row from this analysis. It could be happen because this tubes are heated two times by first and second row tubes which is located on the upstream and this is agree with the average sectional Nusselt number on table 2 by value 441.3924 in cylinder 6 and 428.043 in cylinder 7 at 89s. The capability to heat the fluid is followed by the second row of the tubes bank. The cylinder which placed near wall produce lowest contribution related to the fluid heating on this row. It is caused by the effect of the wall which make the blockage effect on the fluid flow. This phenomenon would deflect the flow keep away from the wall which produces lower Nusselt number compared with the centre cylinder on the second row. As a results, fluid flow over the centre cylinder would have higher Reynolds number, this phenomenon was also discussed by Buryuk [10]. This results agree with the calculation of average sectional Nusselt number on table 2. Finally the tubes which is located on the first row give the lowest contribution on the fluid heating, this results agree with Buryuk [10]. The increasing Nusselt number start from cylinder which is located on the first row to third row is discussed by Baughn [11], that this could be due to the turbulence being more effective on the inner rows, and at the third row, the fluid flow is fully turbulent causing the increase of the average Nusselt number at this section. Aiba et al. [12] have found that the average heat transfer rate was the lowest for the first cylinder and the highest for the third one. Zukauskas [4], Murray and Fitzpatrick [13] have observed that the highest heat transfer takes place in the third row of a tube bank. Knudsen and Katz [14] found that the average Nusselt number increases up to the third row, decreases slightly and then remains essentially constant beyond in the fifth row.

From drag and lif coefficient history on figures 6, shedding vortices has 4s and 22s as the periods of the cycle. Sthrouhal number of the shedding vortices could be calculate using 1m as cylinder diameter and 1m/s as the inlet velocity, resulting the value $St1 \approx 0.045$ and $St2 \approx 0.25$, and it is agree with CFD calculation using FFT on figures 6(c). From lift coefficient history, it could be seen that there is exists oscillation pattern which exhibit beat phenomenon [15]. It could be infered that some of the tubes produces nearly similar natural frequency among others and its frequency amplified when it is combined which shown by lift coefficient history. A careful attention should be taken by the existence of this phenomenon, this would endanger the structure of heat exchanger itself by induced vibration. However, when lift and drag force reach its peak, the average Nusselt number of a bundle of tubes reach its highest value that could be seen on table 2 at 20s and 34.8s. The high Nusselt number which is calculated at this time shows the indication of high capability to transfer energy to the fluid flow. Related with this problem, a method to reduce its vibration at transient condition should be developed in order to increase the lifetime of this equipment without reduce its capabilty to transfer energy

Conclusion

Using RANS numerical simulation, flow past a bundle of tubes with specified dimension has been done. From this simulation, it could be seen that pressure loss due to fluid flow over a bundle of tubes does not change in significant value as time increases to achieve its steady condition. It could be infered that cylinder which placed on the third row of the tubes bank give significant contribution on fluid heating, followed by second and first row. Lift coefficient history which is produced by this simulation shows beat phenomenon, and this is very dangerous to the structure of heat exchanger itself. However, higher lift coefficient results on the higher Nusselt number, which mean higher capability to heat the fluid flow. More investigation in needed to design a heat exchanger system with stable lift forces without reducing its capability to transfer energy to the fluid flow.

References

[1] Chen, C. K., Wong, K. L., Cleavert, J. W., Finite element solutions.of laminar flow and heat transfer of air in a staggered and an in-line tube bank, Int. J. Heat and Fluid Flow 86 (1986) 0142-727.

[2] Buyruk, E., Numerical study on heat transfer characteristics on tandem cylinders, inline and staggered tube banks in cross-flow of air, Int. Commun. Heat Mass Transfer 29 (2002) 355–366.

[3] Grimison, E. D., Correlation and utilization of new data on flow resistance and heat transfer for cross flow of gases over tube banks, Trans. ASME 59 (1937) 583–594.

[4] Zukauskas, A., Heat transfer from tubes in cross-flow, Adv. Heat Transfer 8 (1972) 93–160.

[5] Mittal, S., Kumar, V., Raghuvanshi, A., Unsteady incompressible flows past two cylinders in tandem and staggered arrangements. Int. J. Numer. Meth. Fluids 25 (1997) 1315–1344.

[6] Takemoto, Y., Kawanishi, K., Mizushima, J., Heat transfer in the flow through a bundle of tubes and transitions of the flow, International Journal of Heat and Mass Transfer 53 (2010) 5411–5419

[7] Achenbach, E., Heat transfer from smooth and rough in-line tube banks at high Reynolds number, In,. J. Hear Mass Transfer 34 (1991) 199-207.

[8] Fluent I. Fluent 6. 3 user’s guide. Fluent documentation.

[9] Niemann, H. J., Holscher, N., A review of recent experiments on the flow past circular cylinders, Wind Engineering and Industrial Aerodynamics, 33, pp. 197-209, (1990).

[10] Buryuk, E., Heat transfer and flow structures around circular cylinders in cross-flow, Tr. J. of Engineering and Environmental Science 23 (1999) 299 - 315.

[11] Baughn, J.W., Elderkin, M. J., and McKillop, A. A., Heat transfer from a single cylinder, cylinders in tandem and cylinders in the rntrance region of a tube bank with an uniform heat flux, Trans. ASME, J. Heat Transfer 108 (1986) 386-391.

[12] Aiba, S., Tsuchida, H., and Ota, T., Heat transfer around tubes in staggered tube banks, Bulletin of the JSME 25 (1982) 927-933.

[13] Murray, D. B., Fitzpatrick, J. A., Local heat transfer coeffcients for a tube array using a micro-foil heat flow sensor, Proc. 2nd U.K. Nat. Conf. on Heat Transfer 2 (1988) 1635-1649.

[14] Knudsen, J. G., and Katz, L. D., Fluid dynamics and heat transfer, McGraw-Hill Co., (1958) 514-520.

[15] Singiresu, S. R., Mechanical vibration, Prentice Hall (2005) Singapore.