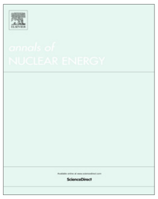
Annals of Nuclear Energy 124 (2019) 198–210

Contents lists available at ScienceDirect

Annals of Nuclear Energy

journal homepage: www.elsevier.com/locate/anucene



On the number of tubes required to study oscillating vortices and frequency spectrums of tube arrays in cross flow

Di Tang [a,*], Shiyi Bao [a], Min Xu [b], Lijia Luo [a], Binbin Lv [c], Li Yu [c], Hao Cui [d]

[a] *College of Mechanical Engineering, Zhejiang University of Technology, Hangzhou 310014, China*
[b] *New Mexico State University, Las Cruces, NM 88001, United States*
[c] *High Speed Aerodynamic Institute, China Aerodynamics Research and Development Center, Mianyang 621000, China*
[d] *Shenyang Aircraft Design and Research Institute of AVIC, Shenyang 110035, China*

ARTICLE INFO

Article history:
Received 27 May 2018
Received in revised form 27 September 2018
Accepted 1 October 2018

Keywords:
Tube arrays
Oscillating vortices
Unsteady performances
Cross flow
Frequency spectrum

ABSTRACT

The oscillating vortices are of much practical importance in the analysis of the fluid-induced vibrations in cross-flow of a steam generator. However, the investigations on the vortex shedding, unsteady lifts, frequency spectrums, and the required number of tubes to study the phenomenon are very limited. In this work, a series of numerical experiments of a triangular array of tubes with a pitch ratio of 1.633 have been performed. The lift coefficients of the tubes were monitored to examine the effects of the numbers of tube rows and columns on the flow and fluid dynamics. The Fast Fourier Transform (FFT) method was used to figure out frequency spectrums of lifts of the cylinder bundle. Results show that the oscillating vortices generated by the tube arrays have greatly enriched the frequency spectrums. On the basis of the investigation of the unsteady lift performances, vortex shedding of tube arrays, eleven columns and eight rows were recommended. Furthermore, the fluid induced vibrations of a cylinder were investigated, and a fluid dynamics enhancement was observed.

1. Introduction

The heat exchange tubes of the steam generator (SG) are the key factor in flow induced vibrations (In-Cheol et al., 2011). When fluid flows across U-tubes, each tube will suffer from strong cross forces with oscillating vortices shed downstream which greatly enriches the frequencies of the cross forces. These cross forces with abundant frequencies can make the tubes vibrate, triggering the problems of friction, wear, fatigue and so on. The prediction of the oscillating vortex behavior contributes to be one of the most important processes in the tube fluid dynamics and structural design as well as fatigue life design (Shoki and Hironobu, 2017).

The investigation of oscillating vortices is a primary task in the design of heat exchangers. The key feature is the periodic vortex shedding phenomenon and how it can lead to better understanding of flow induced vibrations (Mittal et al., 2017). Due to practical significance of oscillating vortices, much effort has been contributed to reveal the reason mechanism of formation and development of vortices (Xu et al., 2015; Xu and Wei, 2016; Xu et al., 2016). The oscillating vortices or Karman vortices of a single tube have been widely investigated because of its simplicity. It has been reported that when the steam or water flowed across a U-tube, a typical cross flow was formed and an adverse pressure gradient was raised up followed by separations in shear layer of a heat exchanger tube. Small scale vortices generated in the corresponding shear layer very close to the near wall surface are coalescing with each other (Alex et al., 2014). The resulting stronger vortices will lead to a pair of co-rotating line vortices shed from the tube, which gives rise to the oscillating vortices (Govardhan and Williamson, 2000). However, if there is no tube or other solid body in the wake region, the line vortices will develop until Karman vortices are generated. Then the heat exchanger tube will suffer from strong cross forces finally. The flow characteristics of a single tube (such as Strouhal number) time averaged drag coefficients have been studied and some good results have been obtained by Computational Fluid Dynamics (CFD) method, compared to the experimental results. However, the wholly unsteady turbulent flow existing in the separation region is still difficult to be predicted, which will lead to the misestimate of pressure coefficients around cylinder surface. In addition, it is difficult to simulate each vortex in the whole SG for oscillating vortices generated by each tube will suffuse the whole SG (Price et al., 2007). Thus, a decoupling method of unsteady predictions of some particular tubes from

---

[*] Corresponding author.
E-mail address: tangdi@zjut.edu.cn (D. Tang).

https://doi.org/10.1016/j.anucene.2018.10.008
0306-4549/© 2018 Published by Elsevier Ltd.

thousands of tubes should be an alternative and effective way. In general, a simple case to study the unsteady flow of multiple cylinders will be two cylinders in tandem or stagger arrangement by both numerical and experimental methods (Gustavo et al., 2006). It was found that oscillating vortices will separate from the upstream cylinder and reattach to the downstream cylinder, and the flow structures are quite sensitive to the gap between two cylinders. For the flow of two cylinders in tandem arrangement (Sumner et al., 2005), three regimes (reattachment regime, co-shedding regime and extended-body regime) have been concluded by the space ratio *L/C* (Zdravkovich, 1987). The wake galloping phenomenon of a cylinder was studied in a water tunnel in which a cylinder was arranged upstream the target cylinder.

It is difficult to investigate the unsteady flow of the exact tube bundle due to the huge number of tubes. However, the flow of a heat exchanger can be simplified and restricted to a small region according to similarities in the flow (Xiaowei et al., 2014; Tian et al., 2016). Early works about tube bundle were done since 1981, Weavera and Elkashlana (1981) investigated the cross flow induced vibrations in tube banks, and recommended that six rows of tube should be used. Zukauskas and Ulinskas recommended 4 rows for a heat transfer, Duan and Jiang (2008) used 5 rows of tubes in a bundle to simulate the flow distribution of a 3D helical tube bundle with RANS method. Several rows of cylinders were used in the cross-flow research so far. For instance, the unsteady flow of normal triangular cylinder with one single tube undergoing oscillations was studied by Beatriz et al. (2016) based on 2D URANS methods. Abramov et al. (2015) studied turbulent flow and heat transfer of liquid metal over inline tube bundles based on the SST turbulence model. A series of cylinder bundle with different columns and rows were studied in their research. By comparing the computed and experimental data, it is found that an acceptable level of agreement is achieved when three or more tube rows are included in the cross direction.

For the development of CFD method, the N-S equations are computed to replay the cross flow and predict the unsteady loads (Beatriz et al., 2016). The numerical simulation based on the unsteady N-S equations is considered as the most promising area (Tang et al., 2017). Generally, the Reynolds-Averaged Navier-Stokes (RANS) method and vortex resolving method are two common approaches. In the RANS method, the SST turbulent model is commonly used in the flow around circular cylinders. The modified SST turbulent, especially transition SST model improves the simulation accuracy in the separated flow to some extent (Lin et al., 2014). However, the eddy viscosity may be overestimated and some important vortex structures will be expunged in unsteady flow using the RANS method. Thus, some vortex resolving methods with filters have been developed, in which the detached eddy simulation (DES) and large eddy simulation (LES) are the two most important methods (Cheng et al., 2017; Thomas and Marion, 2016). However, the corresponding huge computational resources have limited their applications in engineering greatly, especially in three dimensional simulations of tubes.

Much work, as mentioned above, has been devoted to identify the unsteady flow with several tubes. However, little work has been conducted in statistics of vortices. In this work, the main objective is to understand the oscillating vortices and unsteady fluid dynamics of tubes in crossflow and obtain reliable and comprehensive frequency spectrum data, furthermore, give a decoupling method of frequency analysis of cylinder bundles. Section 2 describes the CFD method and decoupling method. The general flow, vortices characteristics and frequency spectrum of a cylinder bundle will be discussed in Section 3, the decoupling results are shown in Section 4.

2. Numerical and decoupling method

2.1. CFD method and fluid induced vibration of a cylinder

In this work, the flow was governed by incompressible Navier-Stokes equations:

$$\frac{\partial u_i}{\partial x_i} = 0, \tag{1}$$

$$\rho\left(\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j}\right) = \frac{\partial}{\partial \mathrm{x_j}}\left[(\mu + \mu_{\mathrm{t}})\frac{\partial u_i}{\partial x_j}\right] - \frac{\partial \mathrm{p}}{\partial \mathrm{x_j}}, \tag{2}$$

where t is the time, u is the velocity, p is the pressure, ρ is the fluid density, μ denotes the coefficient of dynamic viscosity, and u_t denotes the turbulence viscosity. The Transition SST model was used for the turbulence modeling considering its high accuracy in the simulation of vapor flow (Yang et al., 2017). The transport equations of the intermittency γ and the transition momentum thickness Reynolds number $\mathrm{Re}_{\theta t}$ are (Menter, 1994):

$$\frac{\partial(\rho\gamma)}{\partial \mathrm{t}} + \frac{\partial(\rho \mathrm{U_j}\gamma)}{\partial \mathrm{x_j}} = \mathrm{P}_{\gamma1} - \mathrm{E}_{\gamma1} + \mathrm{P}_{\gamma2} - \mathrm{E}_{\gamma2} + \frac{\partial}{\partial \mathrm{x_j}}\left[\left(\mu + \frac{\mu_{\mathrm{t}}}{\sigma_\gamma}\right)\frac{\partial\gamma}{\partial \mathrm{x_j}}\right], \tag{3}$$

$$\frac{\partial}{\partial \mathrm{t}}\left(\rho\widetilde{Re}_{\theta t}\right) + \frac{\partial}{\partial \mathrm{x_j}}\left(\rho \mathrm{U_j}\widetilde{Re}_{\theta t}\right) = \mathrm{P}_\theta + \frac{\partial}{\partial \mathrm{x_j}}\left[\sigma_{\theta t}(\mu + \mu_{\mathrm{t}})\frac{\partial\widetilde{Re}_{\theta t}}{\partial \mathrm{x_j}}\right], \tag{4}$$

where $\mathrm{P}_{\gamma1}$ and $\mathrm{E}_{\gamma1}$ are transition sources, $\mathrm{P}_{\gamma2}$ and $\mathrm{E}_{\gamma2}$ are destruction/relaminarization sources, P_θ is the source term in the transport equation for the $\widetilde{Re}_{\theta t}$, and σ_γ is the constant for the intermittency equation.

The simulation was conducted via ANSYS Fluent release 16.0. The pressure-based solver was chosen to simulate the transient flow. We used SIMPLEC algorithm for the pressure and velocity coupling, Green-Gauss Cell Based method for the gradient calculation, second order scheme for the pressure discretization, Quick scheme for the momentum discretization, and first order scheme for the turbulence equations discretization.

To validate the simulation, a benchmark study of a flow passing a circular cylinder at $Re = 2.6 \times 10^5$ was undertaken. A rectangle computational domain of $40D \times 50\mathrm{D}$ was adopted, where D denotes the cylinder diameter. An O-block was used to construct the mesh of the cylinder, as shown in Fig. 1. The cylinder was discretized uniformly with 181 nodes. The radial size of the first layer was set according to the rule of non-dimensional wall distance y^+ less than 1. The mesh at the downstream was discretized at a higher resolution to resolve the flow at the near wake. A small time-step of 3×10^{-5} was used for the time advancement, based on the characteristic of oscillating vortices. A uniform total pressure was specified at the inlet boundary. A pressure outlet of 0 Pa is specified at the outlet. The symmetry boundary conditions were applied at both the top and the bottom boundaries. No-slip wall boundary condition was applied on the cylinder surface.

Fig. 2 compares the distribution of the time-averaged pressure coefficient on the cylinder surface, which was obtained from the current numerical simulation and experiment measurement by Achenbach (1968). The simulation gave a maximum pressure coefficient of 1.0, which matches the theoretical value, and a minimum value of −2.16, which fits well with the experimental value of −2.11. Despite of a small deviation between 80° and 120°, where flow separation occurs, the computational results agree well with the experimental data.

One circular cylinder was mounted as a spring-damper-mass system to study the fluid-elastic performances of the cylinder bundle according to the assumption in Beatriz et al. (2016). The