Simulation of flow across complicated domain between tube bundles by the discrete vortex method*

CHEN Bin**, GUO Liejin and YANG Xiaogang***

(State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an 710049, China)

Received August 29, 2002; revised Octobor 11, 2002

Abstract On the basis of the analysis of numerical simulation methods for the complicated domain between tube bundles, an improved Lagragian discrete vortex method (DVM) and corresponding algorithm are put forward to solve the practical difficulties of flow across tube bundles. With this method the amount of vortices can be reduced considerably, which makes quick calculation possible. Applied to the practical configuration of horizontal tube bundles, the DVM simulation is carried out and compared with the experimental results. Both the transient flow field and the profile of mean velocity and fluctuations are in good agreement with experimental results, which indicate that the DVM is suitable for the simulation of single-phase flow across tube bundles.

Keywords: tube bundles, discrete vortex method, numerical simulation.

The flow across tube bundles is of three-dimensional unsteady turbulent shear flow which is applied widely in industry. Numerical simulation is a powerful tool to investigate this kind of problem. Up to now most of the engineering computations employ the Eulerian Reynolds equation, and turbulence models are introduced to close the equations. This method has fatal shortcomings: firstly, time averaging obliterates details of fluctuations and results in remarkable loss of turbulence fluctuation information, therefore excludes transient vortex structures and ignores the practical physical process of intensive eddy area1); secondly, each kind of turbulence model has its own limitation and defect, for example dependence on experimental data and imprecise prediction, so the achievement of instantaneously accurate turbulence characteristics becomes very difficult. In addition, in the numerical simulation the finite difference or finite element methods are employed. Then researchers have to confront with the additional work of grid generation to fit the complicated geometry domain in order to obtain satisfactory spatial resolution in different regions such as potential flow, boundary layer, separate flow and wake flow. Meanwhile the multitudinous grid number also considerably influences computation efficiency.

There are two ways to solve this problem. One is direct numerical simulation (DNS), the most accurate way to study turbulent flow. In this approach the flow field is solved directly from the Navier-Stokes equations and no averaging or turbulence modeling is applied. Thus only the numerical methods affect the accuracy of the solution. One of the drawbacks of DNS is that it requires high order numerical methods and a huge amount of computer capacity that it is not adaptable for engineering type flows. The main difficulty is that all dynamically significant scales of motion must be accounted for including everything from the large integral scale to dissipative scale (Kolmogorov scale). As Reynolds' number grows, the Kolmogoroff turbulent scales are much smaller. Thus smaller eddies will be present in the flow field and the affordable time-step and mesh-space should be able to capture these smallest scales of motion. In general, the computing resources required are proportional to the Reynolds number raised to the 9/4 power. When Reynolds number is higher, ordinary computers are not sufficient for the DNS[1].

Another promising approach to simulating unsteady separate flow is the discrete vortex method (DVM)[2]. With this method a number of point vortices are positioned into the potential flow field to

* Supported by the National Natural Science Foundation of China (Grant No. 59725616) and Major State Basic Research Development Program of China (Grant No. G2000048702)

** To whom correspondence should be addressed. E-mail: chenbin1971@yahoo.com.cn

*** Visiting Scholar of the Xi'an Jiaotong Uinversity, Paisley University, UK

1) Yang, X. Two-phase flow dynamics simulations and modeling. Ph.D. dissertation. University of Birmingham, UK, 1996

characterize the continuously distributed vorticity. Then the whole flow field can be obtained through calculating the complicated evaluation of vortex shedding, development and roll up. No turbulence model is adopted, so the loss of fluctuation information is avoided. Since the fluid movement is mimicked in a Lagrangian way, the variables are not transported by mesh, but advected through the changing of vortex positions. Hence, mesh is not necessary and no artificial boundary conditions are introduced. One of the advantages of the DVM is its ability to concentrate computational effort in the areas of high vorticity gradient, which is particularly suitable for simulations of vorticity dominated flow, such as flow across tube bundles. Besides the economization in computation, another advantage of vortex method is the way in which the initial vorticity is introduced. In Eulerian calculations usually the start conditions already introduce vorticity. A proper natural evaluation of the vorticity however should start from zero to the flight-velocity to simulate the time-dependent build-up of the flow. Within the vortex approach, this is performed by modeling the flow from the very beginning and tracking a continuously developing vortex wake, which is not bound by a computational domain or grid. This eliminates the vortex diminishing effects and does not require an initial introduction of vorticity as found in Euler codes. When simulating separated flows, we can introduce nascent vortices from separation points to the flow field conveniently. Therefore DVM has considerable advantages in simulation of high Reynolds separated flow. However, at the present time DVM is still limited to relatively simple shear flow, such as mix layer flow[3], jet flow[4] and flow across cylinder[5]. It is rather difficult to apply DVM to a complicated domain between tube bundles because of (1) Kelvin-Holmholtz instability and chaotic movement due to huge amount of vortices; (2) the difficulty for it to represent the bodies when multi-obstacle exists in the flow field; (3) the difficulty in accurate computation of separation points and nascent positions; and (4) the lack of comparison between simulation and experimental data. In the present study, DVM is investigated to seek a new way for simulating the transient flow between tube bundles.

1 DVM algorithm

1.1 Theory

In the discrete vortex method, unsteady incompressible and viscid flow is described by vorticity kinetic and dynamic equations as follows:

$$\frac{\partial \boldsymbol{\omega}}{\partial t} + (\boldsymbol{V} \cdot \nabla)\boldsymbol{\omega} = (\boldsymbol{\omega} \cdot \nabla)\boldsymbol{V} + v\,\nabla^2\boldsymbol{\omega}, \quad (1)$$

$$\nabla^2\psi = -\boldsymbol{\omega}, \quad (2)$$

where $\boldsymbol{\omega}$, $\boldsymbol{V}$, v and ψ represent vorticity, velocity, viscosity and stream function, respectively. According to Chorin[6], the vorticity equation (1) is divided into convection and diffusion parts and the two equations are solved sequentially:

Convection

$$\left(\frac{\partial \boldsymbol{\omega}}{\partial t}\right)_I = -(\boldsymbol{V} \cdot \nabla)\boldsymbol{\omega} \quad \text{and} \quad \nabla^2\psi = -\boldsymbol{\omega}, \quad (3)$$

Diffusion

$$\left(\frac{\partial \boldsymbol{\omega}}{\partial t}\right)_{II} = v\,\nabla^2\boldsymbol{\omega}. \quad (4)$$

The solution of the diffusion part is the famous Biot-Savart Law. Based on this inviscid solution, the viscous diffusion part can be simply simulated by the random walk method[6].

1.2 Application to tube bundles

The flow across tube bundles involves many complex phenomena such as flow separation, evolution of shear layer, wake instability and interaction between fluids and flow structure and so on. The transient large vortex and mentioned historical effects play very important roles on above phenomena in this unsteady flow. As mentioned above, DVM is a powerful tool to investigate this kind of transient flow but has not ever been applied to the flow across tube bundles. When DVM is implemented, the potential flow is acquired first. The surface vortex method can be used to approximate it[7]. With regard to the discretion of vorticity field, the Rankine vortex is selected as the vortex element model, meanwhile the radius of vortex element is calculated according to the Lamb-Oseen vortex. The mixed vortex model is convenient for the numerical computation, and viscous effect is considered in the separation region.

2 Algorithm improvement of DVM

Based on the Lagrangian frame, DVM uses the Biot-Savart law to determine the induced velocity of each vortex element. No spatial grid is needed, so numerical diffusion is avoided. However, the direct sum should consider the interaction between each vortex and the rest $N-1$ ones. Therefore, each loop of velocity computation requires an order of N^2 operations approximately. With the advance of time steps, the number of vortex elements will become extremely huge, which will lead to a heavy load to programming and computation. For instance, Van der Vegt et al.[1)] used 51000 vortex elements to calculate the flow across a single cylinder. Skomdedal and Vada[8] used 15000 ~ 100000 elements to simulate single and double cylinders. It can be imagined that the number of vortex elements will be much larger in the flow between tube bundles because of the increase of separation.

In order to improve computation efficiency, the vortex-in-cell (VIC) method is developed. The VIC method is a hybrid Eulerian-Lagrangian approach, in which the vorticity of the flow field is described by a cloud of moving vortex elements. Velocity field is obtained through the Poisson equation of stream function in the Eulerian way, and then vortex elements are tracked in the Lagrangian way[9]. The advantage of this approach is the high efficiency through the hybrid of Lagrangian and Eulerian superiority. However, to calculate the stream function, the vorticity of these discrete vortex elements has to be re-distributed on a numerical grid through the area-weighted method, with which the fundamental stream function-vorticity equation is solved (as shown in Fig. 1). Therefore, it makes an otherwise grid-free method once again grid-independent, and numerical diffusion will certainly be introduced. Furthermore, the traditional structured grid is not adapted to the complicated domain between tube bundles. If the unstructured grid is adopted, considerable error will be introduced when calculating velocity from stream function on the grid nod. At the same time, this error will be transferred to each vortex element through the area-weighted method. Thus it can be seen that the VIC is not promising to solve the problem with a huge number of elements existing in the flow between tube bundles.

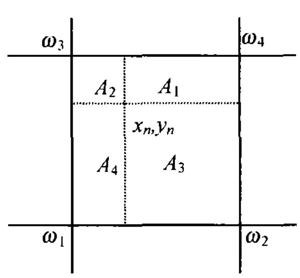


Fig. 1. Schematic of VIC.

From the literature about the application of DVM to flow across single cylinder, it is found that the reason for the huge number of vortex elements is the method to determine the shedding vorticity from the cylinder wall. At present the typical way is the nascent vortex method proposed by Chorin[6]. With the approach new vortices are introduced from the cylinder surface to satisfy the no-slip condition. The cylinder wall is divided into many small sections. One vortex is introduced on each section during each time step, and the resultant induced velocity exactly counteracts the tangential velocity of the wall. This method is very simple and with a clear physical meaning. It does not need to determine the separation point which appears automatically in advance. However, the obvious disadvantage of this approach is that many new vortices will be introduced to the flow field during each time step, so the total number of vortices will be very huge with time advancement.

Based on the analysis above, we can conclude that the way to decrease the number of vortex elements is the way to change shedding vorticity from the wall. If we can calculate the separation points accurately and determine the position and strength of nascent vortex, the number of vortex elements will be decreased considerably, making quick computation of the flow across tube bundles possible. In order to determine the variable separation point and the interaction between wake and mainstream, in the present paper the Pohlhausen quasisteady state approximation[10] is used to calculate the boundary layer during each time step, then the vorticity is continuously shed into the flow field from the separation point. The nascent vortex is placed at a distance, ε, from the cylinder surface (with its image at the inverse point) along the radial line passing through the separation point. Assuming the zero slip velocity at the separation point, we can use the velocity at that point to calculate the rate at which the vorticity is shed into the wake from

$$\Delta\Gamma = 0.5U_s^2\Delta t, \qquad (5)$$

where $\Delta\Gamma$ refers to the shedding vorticity, U_s is the velocity at the separation point. Apparently, the velocity, position and strength of nascent vortex, and time interval are dependent on each other and cannot be selected arbitrarily.

When this approach is adopted, only 400 ~ 500

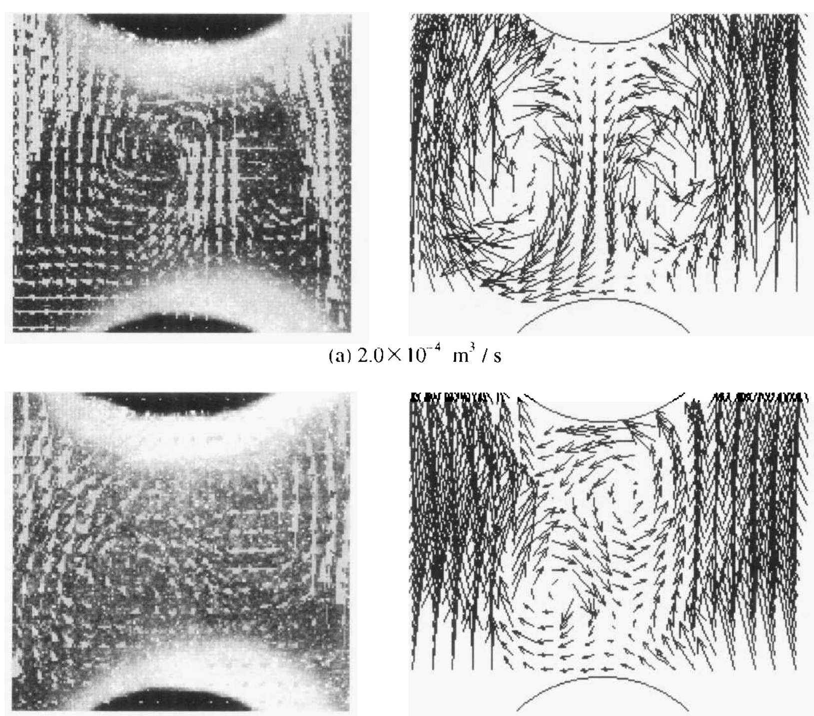
1) Van der Vegt, J. J. W. A variationally optimized vortex tracing algorithm for flows around solid bodies. Ph.D thesis, Maritime Research Institute, The Netherlands

vortex elements are needed to construct a satisfactory flow across a single cylinder. With the combination of vortex amalgamation and elimination, the total number of vortex elements of flow across tube bundles can be considerably decreased, and then quick computation is possible.

3 Results

The experiment was carried out on the transient flow across tube bundles by PIV. In order to verify the effectivity of the application of DVM to the numerical simulation of flow across tube bundles, a comparison between numerical simulation and experimental results was conducted according to the experimental conditions. The setup and parameters of the test section can be found in Ref. [11].

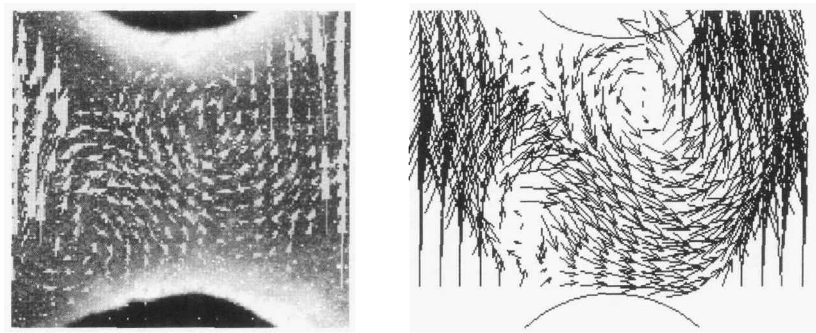
The comparisons of local flow field in the wake region of tube 1 and 2 at different flowrates are shown in Figs. 2 and 3, respectively. The left side of the figures show the experimental results and the right side indicate numerical simulation results. Since both of the experiment and simulation are aimed at transient situations, there are many results available for comparison. However, only some typical results are shown here. The instantaneous measurement technique of PIV can supply details of the flow, and the Lagrangian DVM can mimic transient flow well, which make the comparison possible. It can be seen that the experimental results of vortex movement, pairing and amalgamation agree well with those of the simulation, which approves the effectivity of DVM simulation of unsteady shear flow across tube bundles. It is shown that the wake region of each tube is similar to the flow across single cylinder, the vortices shed into the wake flow alternatively, which is determined by the interaction of two separated shear layers. The shear layer on one side evolves to vortex and it is enhanced by obtaining vorticity from the conjoint layer until it is strong enough to attract the shear layer of the other side. Then the vortex is cut down and shed into the flow because of the opposite sign of the two layers. In the wake region, the vortex structure cannot fully develop under the depression of cylinders existing downstream. Vortices pairing still appears occasionally but is not continuous like a mixing layer. The amalgamation of vortices is the main reason of vortex development. The amalgamation means the redistribution of strength and position of wake vorticity.



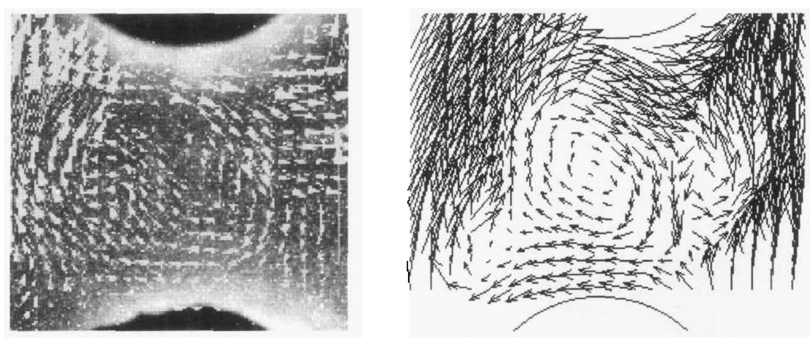
(a) 2.0×10^{-4} m^3/s

(b) 2.8×10^{-4} m^3/s

Fig. 2 Comparison of the local flow field in the wake region of tube 1.



(a) 2.0×10^{-4} m^3/s



(b) 2.8×10^{-4} m^3/s

Fig. 3. Comparison of the local flow field in the wake region of tube 2.

Both of our experiment and simulation are transient, which means it is rather difficult to determine the time correspondence to the experiment and simulation. Therefore, the results are averaged to get the mean velocity and fluctuations. Figs. 4 and 5 show the distribution of the mean velocity and fluctuations on cross sections behind tubes 1 and 2, where it is two times the tube radius away from the tube center at different flowrates. In the wake region, $u'u'$ is symmetrical about the centerline of the wake with two peaks. $u'v'$ is antisymmetrical about the centerline of the wake with two peaks. The distribution of $v'v'$ is symmetrical about the centerline at which the only peak appears. In the figure solid lines present the results of numerical simulation, and the dotted lines are experimental results. It can be seen that the numerical simulation agrees well with the experimental results. Because of the limitation of experimental condition, only 100 data were collected and the minimum time interval was 0.1 s. Then average error cannot be avoided.

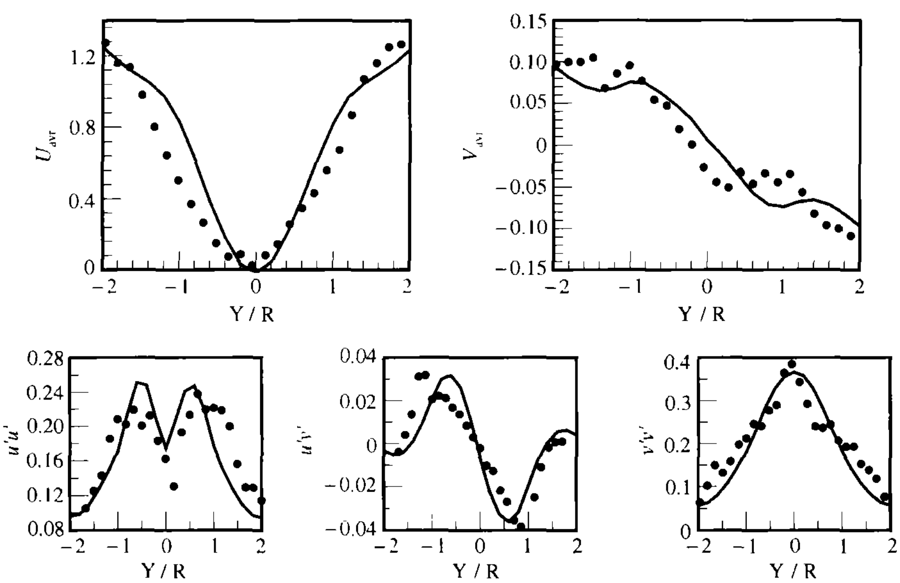


Fig. 4. Distribution of mean velocity and fluctuations of tube 1.

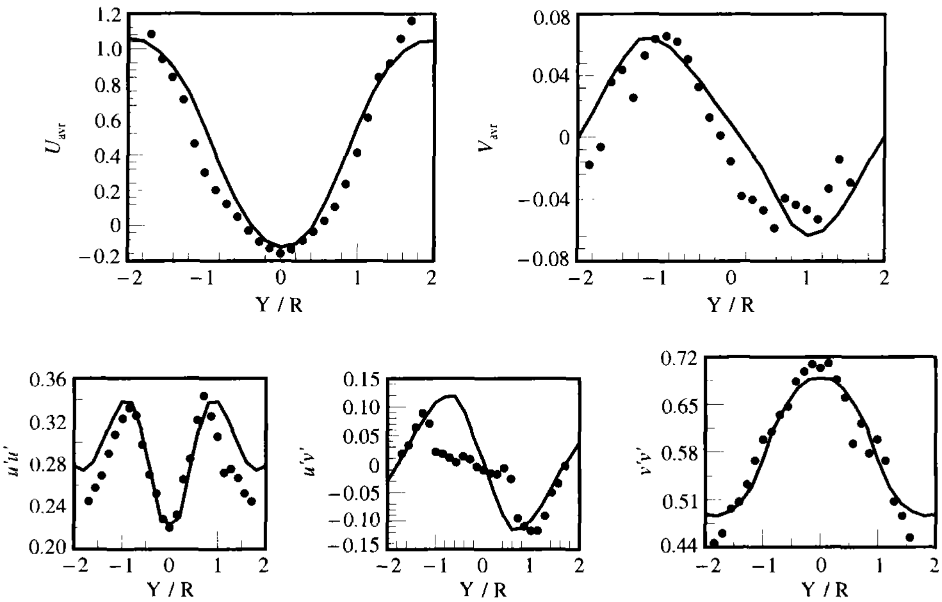


Fig. 5. Distribution of mean velocity and fluctuations of tube 2.

4 Conclusions

(1) A new model is put forward to apply the Lagrangian DVM to the complicated domain between tube bundles, which overcomes the disadvantages of the Eulerian model and provides a new method to simulate the transient shear flow across tube bundles.

(2) The Lagrangian DVM is improved to solve the practical difficulties of flow across tube bundles. With the new approach, the amount of vortices is reduced considerably, which makes quick computation possible.

(3) Applied to the practical configuration of horizontal tube bundles, the DVM simulation is carried out and compared with the experimental results. With this approach the evolution of shear layer, phenomena of vortices shedding, paring, amalgamation, and development of the Karman vortex street can be visualized clearly. Both the transient flow field and the profiles of mean velocity and fluctuations are in good agreement to the experimental results, which indicates that the DVM presented by this paper is suitable for the simulation of single-phase flow across tube bundles.

References

1 Shi, X. G. Turbulence (in Chinese). Tianjin: Press of Tianjin University, 1994.

2 Chen, B. et al. A review on the numerical methods of the two-phase flow across a tube bundle. J. of Basic Science and Engineering, 1999, 7(1): 60.

3 Chen, B. et al. Simulation of flow across circular cylinders with discrete vortex method. In: 1st International Conference on Heat Transfer, Fluid Mechanics, and Thermodynamics, Kruger Park, South Africa, 8~10 April, 2002.

4 Lin, J. Z. et al. Influence of vortex structure on the particle diffusion in gas-solid two-phase jet. Applied Mathematics and Mechanics (in Chinese), 1999, 20(5): 470.

5 Sene, K. et al. The role of coherent structures in bubble transport by turbulent shear flows. J. Fluid Mech., 1994, 259: 219.

6 Chorin, A. J. Numerical study of slightly viscous flow. J. Fluid Mech., 1973, 57: 785.

7 Stansby, P. K. et al. Simulation of vortex shedding including blockage by the random-vortex and other methods. Int. J. Numerical Methods in Fluids, 1993, 17: 1003.

8 Skomedal, N. G. et al. Numerical stimulation of vortex shedding induced oscillations of a circular cylinder. In: Proc. 2nd Int. Conf. on Advances in Numerical Methods in Eng. Theory and Applications, Swansea, Wales, 1987, 1.

9 Tong, B. G. et al. Theory of Vortex Movement (in Chinese). Hefei: Press of China Scientific and Technologic University, 1994.

10 Schlichting, H. Boundary Layer Theory. Beijing: Science Press, 1988.

11 Chen, B. et al. Particle image velocimetry measurement of flow across tube bundle in waste heat boiler. J. Thermal Science, 2000, 9(3): 249.