Analysis of Flow-Induced Vibrations in Piping Systems and Circular Cylindrical Structures*

Noriyuki SADAOKA**, Keiji KOBASHI**
and Kikuo UMEGAKI**

A new approach to simulate flow-induced vibrations in piping systems and circular cylindrical structures is introduced. In the developed method, three-dimensional unsteady flow is solved by a direct simulation method. Based on the fluid dynamic calculation, fluid forces on structure surfaces (piping inner wall or cylinder surface) are estimated by integrating fluid pressure and shear stress. The calculated fluid forces are translated to finite element nodes in structure surfaces and treated as external forces acting on these surfaces. Then dynamic response of the structures is calculated by using modal analysis or the Newmark- β method. For a full coupling analysis, the moved surfaces become the new coordinate boundaries at the next time step. As an application of the developed method, flow-induced vibrations in an elbow and an array of tubes were evaluated. Turbulent intensity agreed well with experimental data in the unsteady flow analysis and vibration of the elbow was successfully simulated in the dynamic response analysis. For a 3×3 square array, the occurrence of fluid elastic instability could be simulated.

Key Words: Flow Induced Vibration, Computational Fluid Dynamics, Unsteady Flow, Coupled Vibration, Vortex, Pipe Flow, Turbulent Flow

1. Introduction

An unsteady fluid force on structures in a turbulent flow can cause their vibration. This phenomenon is the most important among various flow-induced vibrations (FIVs) and, consequently, it is an important subject in designing nuclear plant components like heat-exchangers. In recent years, in order to improve the functions of components, such as their response to a thermal effect, fluid velocity in the components has tended to be higher and component structures have tended to be more slender in design. These conditions combine to raise the probability for occurrence of FIVs. This is particularly so when the frequency of periodic vortex shedding around structures coincides with the natural frequency of the structures, since synchronization occurs and the vibration amplitude becomes large. Furthermore when the velocity is increased to a critical value, fluid energy may feed

into the structures, resulting in large structure vibrations. This phenomenon is called fluid elastic instability and it is very important when predicting critical velocity⁽¹⁾.

Customarily, empirical equations derived from experimental data are mainly used to evaluate FIVs⁽²⁾. But the mechanism of FIVs is not clearly understood because of the complexity of the dynamic interactions of the structures and fluid. Considering this, the authors have been developing a new approach to simulate FIVs in piping systems and circular cylindrical structures⁽³⁾⁻⁽⁴⁾.

The purpose of the present study is to develop a new method which can predict FIVs in turbulent flow. First, the developed method is introduced and then calculation results are given for an elbow and an elastically supported 3×3 square array.

2. Developed Method

2.1 Algorithm

In the method, FIVs in incompressible single phase flow are calculated. Figure 1 shows the algorithm for piping systems. Curvilinear coordinates are generated by the boundary-fitted method (BFM)⁽⁵⁾.

* Received 13th July, 1997

** Hitachi, Ltd., Power & Industrial Systems R & D Division, 7-2-1 Omika-cho, Hitachi-shi, Ibaraki-ken 319-12, Japan

Fig. 1 Algorithm and features of the developed method for a piping system

Then based on the generated coordinates, turbulent flow is calculated. In this flow calculation a Navier-Stokes equation for incompressible viscous flows is solved by the direct simulation method. In the next step unsteady fluid force on the piping inner wall is calculated from the flow field obtained at each time step. At this point, fluid force on the piping inner wall at BFM grid points is translated to fluid-force at FEM nodal points. Then dynamic responses of the piping are analyzed by using modal analysis or the Newmark- β method.

For an array of cylinders, the fluid-structure coupling effect becomes large, so a full coupling method as shown in Figure 2 is developed. In this case, to follow structure motions, new curvilinear coordinates are regenerated at each calculation step by the BFM. In this calculation a Navier-Stokes equation for incompressible viscous flow, in which the moving boundary effects are taken into consideration, is solved by the direct simulation method and arbitrary Lagrangian-Eulerian method⁽⁶⁾. The moved structure surfaces become the new coordinate boundaries in the next time step. By repeating these calculation steps, the fluid-structure coupled effect in turbulent flow can be evaluated.

In the following sections, the details of each calculation step are explained.

2.2 Time-dependent curvilinear coordinates generation

The BFM is used in regenerating coordinates at each time step against the deformed flow space. The

Fig. 2 Algorithm and features of the developed method for an array of tubes

BFM can generate coordinates automatically when boundary conditions are given. Therefore, when structures in the flow field move at each calculation step, if the coordinates of the structure surfaces are known, the coordinates for the whole flow field can be generated by this method.

The basic equations of coordinate transformation are given as follows.

$$\alpha_{im} \frac{\partial^2 x_i}{\partial \xi_1 \partial \xi_m} + J^2 P_1 \frac{\partial x_i}{\partial \xi_1} = 0 \quad (1)$$

where

$$\alpha_{im} = \beta_{i1} \cdot \beta_{im}, \quad \beta_{i1} = \frac{\partial x_j}{\partial \xi_m} \cdot \frac{\partial x_k}{\partial \xi_n} - \frac{\partial x_j}{\partial \xi_n} \cdot \frac{\partial x_k}{\partial \xi_m}$$

By solving these equations in the successive over relaxation (SOR) method, the curvilinear coordinates in a physical space can be generated, corresponding to orthogonal coordinates in the computational space. When the structures move or deform, their surfaces in the physical space also move and the flow area changes.

2.3 Method for turbulent flow calculation coupled with structure vibration

2.3.1 Basic equations for flow considering the moving boundary In the calculation step for turbulent flow, the equation of continuity and the Navier-Stokes equation for incompressible viscous flow are solved. However, taking into account structure movement in the flow field, a new convection term for change of physical quantities due to the moving

boundary should be added to the ordinary Navier-Stokes equation. Therefore, the basic equations become the following in the two-dimensional case.

$$\frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0 \quad (2)$$

$$\begin{aligned} \frac{\partial U}{\partial t} + \frac{\partial(U - U_G)}{\partial x} U + \frac{\partial(V - V_G)}{\partial y} U \\ = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial x} + f_x \end{aligned} \quad (3)$$

$$\begin{aligned} \frac{\partial V}{\partial t} + \frac{\partial(U - U_G)}{\partial x} V + \frac{\partial(V - V_G)}{\partial y} V \\ = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{yy}}{\partial y} + f_y \end{aligned} \quad (4)$$

Equations (2)–(4) are transformed to equations in the computational space.

The modified simplified marker and cell (SMAC) method⁽⁷⁾ is used for solving Eqs. (2)–(4). For time integration of the Navier-Stokes equation, the Adams-Bashforth method is used. The SOR method and multi-grid method are combined to reduce calculation time for iteration in solving the pressure Poisson equation⁽⁷⁾ and the third-order upwind finite difference (QUICK) is used in discretization of the convective term.

2.3.2 Boundary conditions In the method, velocity conditions at structure surfaces are changed between the initial time step and other time steps. At the initial time step, the velocity at the structure surfaces is zero (non-slip condition) and at other time steps it becomes the structure moving velocity (Dirichlet condition). On the other hand, the pressure condition at the structure surfaces is that its difference in the normal direction is zero (Neumann condition) if the inner grids to the structures are generated within a boundary layer. The general equation for the boundary conditions is

$$a_1 \phi + a_2 \frac{\partial \phi}{\partial \eta} = a_3 \quad (5)$$

At each condition, a_1 , a_2 , a_3 have the following values.

Non-slip condition: $a_1=1$, $a_2=0$, $a_3=0$

Dirichlet condition: $a_1=1$, $a_2=0$, $a_3=U_{GB}$ or V_{GB}

Neumann condition: $a_1=0$, $a_2=1$, $a_3=0$

where U_{GB} , V_{GB} : moving boundary velocities of structures.

2.3.3 Calculation of fluid force on structures

The fluid force added on structures can be estimated by integrating fluid pressure and shear stress on structure surfaces at each calculation step as shown in Eq. (6)

$$F(t) = \int_s P_w(t) dS + \int_s \tau_w(t) dS \quad (6)$$

where $F(t)$: unsteady fluid force, $P_w(t)$: pressure on structure surfaces, $\tau_w(t)$: shear stress on structure surfaces.

2.4 Calculation of structure deformation

By turning the obtained fluid force into an external force on the structures, the dynamic response of structures is calculated and the amplitude and frequency of the vibrations are determined. The structures are treated as isotropic elastic bodies and the material linearity and geometric linearity are postulated. When there is no deformation on structures, they are treated as rigid bodies. When structures are deformed, the equation of motion which includes the damping term is solved using the finite element method (FEM). The calculated structure surfaces become new coordinate boundaries at the next time step.

3. Calculated Results

3.1 Calculated results for three-dimensional unsteady flow in a piping system

In order to verify the accuracy of calculated unsteady fluid force on piping inner wall, the flow field in an elbow was calculated. In this calculation, piping inner walls were stationary. A time-averaged velocity profile at several points on cross sections was compared with the experimental data obtained by Kawamura and Nakao⁽⁸⁾.

Figure 3 shows the calculation domain and generated coordinates. The calculation region at the front of the elbow was $3D$ long, and at the rear of the elbow the region was $10D$ long (D : tube diameter). The whole region was divided into 3 blocks and there were 339 233 grid points. Inlet velocity was 7.8 (m/s) and the fluid was air at room temperature. Pipe diameter D was 100 (mm) and the Reynolds number based on inlet velocity was 50 000. These conditions are the

Fig. 3 Calculational domain and conditions for an elbow

same as in the experiment⁽⁸⁾. The time-averaged and fluctuating velocities downstream from the elbow were measured using a backscatter laser probe. As shown in Fig. 3, measured positions were two cross sections at $0.55D$ and $2.0D$.

3.2 Comparison between calculated flow field and experiment

In Fig. 4, calculated instantaneous velocity vectors at vertical cross section ($X/D=1.0$) are shown. In this figure, blue indicates low pressure regions, which coincide with centers of the vortices, and red indicates high pressure regions. Unsymmetrical secondary flow vortices were generated. From comparison of other flow patterns at other time steps, sizes and shapes of these vortices varied unsteadily.

In order to compare the calculated results with experimental ones, velocity and pressure in the whole calculation domain were averaged over time. Figure 5 compares calculated and experimental results for the time-averaged velocity profile on a Y -axis for the two vertical cross sections. The calculated velocity profile agreed well with the experimental one. Figure

Fig. 4 Instantaneous velocity vectors at the vertical cross section ($X/D=0.55$)

Fig. 5 Comparison of time-averaged velocity profile for two cross sections

6 compares calculated and experimental results for the fluctuation profile of velocity. The tendency for the profile to change from the end of the curve section was successfully simulated, although there were some differences between the calculated profile and experimental data. These were caused by making a two-dimensional calculation and the calculation method of turbulence.

3.3 Calculated results for flow-induced vibration of an elbow

3.3.1 Calculation domain and conditions

Based on the calculated results for unsteady flow at the elbow, the dynamic response of the whole elbow was calculated by using the modal method. In this calculation step, the elbow was modeled with 8-node solid elements and unsteady fluid forces on the inner wall were treated as external forces. In the modal analysis, both ends of the elbow were clamped and there was no support between the inlet and outlet.

3.3.2 Calculated results of dynamic response

Figure 7 shows the calculated results of dynamic response of an elbow. Because the turbulence intensity at the curve section was larger than that at other sections, deformation at the curve section was larger than that at other sections. In the results, there were not only vibration modes toward the Y -direction, but also vibration modes toward the X -direction. The cause of the latter was periodic fluctuation of a vortex generated at the inner curve wall. Because of air flow, the maximum deformation of the elbow was very small and its value was 2.1×10^{-3} (mm).

3.4 Calculated results for flow-induced vibrations of an array of tubes

3.4.1 Calculation domain and conditions

Figure 8 shows the calculation domain and generated coordinates for an elastically supported 3×3 square array. The pitch ratios P/D , T/D were set at 1.5 to allow comparison with the experiment of Weaver and

Fig. 6 Comparison of turbulence intensity profile for two cross sections

Fig. 7 Dynamic response of an elbow induced by unsteady fluid force

Fig. 8 Calculational domain and conditions for an array of tubes

Abd-Rabbo⁽⁹⁾. The mass ratio and the natural frequency of tubes were set at 1.9 and 25.5(Hz), respectively. The reduced damping parameter was set to the same value, 0.037, for all tubes. These values were the same as in the experiment.

The whole region was divided into 40 blocks and

Fig. 9 Instantaneous pressure contours ($U_{in}=0.2$ m/s)

the total number of grid points was 35 866. The upper and lower boundary conditions were periodic condition. Support conditions of every tube were the same and tubes were supported elastically by a spring and damper. All tubes could be oscillated in both X - and Y -directions. Therefore, vibrations of tubes could be expressed by the equation of motion for two degrees of freedom. The time step interval for re-generation of coordinates was $\Delta t' = 2.0 \times 10^{-1}$ and all coordinates were re-generated 20 times for a cycle.

3.4.2 Calculated results Figure 9 shows calculated pressure contours for the center region of the tube array when inlet velocity U_{in} was set at 0.2 (m/s). At this inlet velocity, amplitudes of each tube were less than 1.0 % of the tube diameter. But because of vibrations of all tubes, the vortex pattern at the back of the tubes was changed from that at the stationary condition. As shown in Fig. 9, separated vortices at the back of the first row tubes (Nos. 1, 4, 7) became symmetrical to the horizontal center lines of tubes. This characteristic vortex pattern has been observed experimentally during flow visualization⁽⁹⁾.

In Fig.10, the orbits of all 9 tubes when inlet velocity U_{in} was set at 0.3(m/s) are shown. After flow began, all tubes vibrated in both the X - and Y -directions for a moment and the orbits became elliptical. These amplitudes were 2-3% of the tube diameter and the vibration pattern was the same as that at $U_{in}=0.2$ (m/s). But at $t \geq 0.15$ (s), vibration amplitude of tube 6 rapidly became more than 10% of the tube diameter. Following the vibration of tube 6, the amplitude of tube 3 also became large.

Figure 11 shows the dependence of RMS amplitude of tube 6 on inlet velocity. The experimental

Fig. 10 Orbits of each tube ($U_{in}=0.3$ m/s)

Fig. 11 Comparison of amplitude between experiment and calculation for tube 6

data⁽⁹⁾ are plotted in order to compare the critical velocity at which amplitude rapidly became large. Although the calculated critical velocity was smaller than that in the experiment, the occurrence of fluid elastic instability in a 3×3 square array could be successfully simulated. From the detailed analysis of the calculated results, when fluid elastic instability occurred, there was some phase lag between tube displacements and unsteady fluid forces. This phase

lag could not be seen when fluid elastic instability did not occur.

4. Conclusion

A new approach to simulate flow-induced vibrations in piping systems and circular cylindrical structures was introduced. Flow-induced vibrations in an elbow were evaluated and the dynamic response of the elbow caused by unsteady flow was successfully simulated. As another application, flow-induced vibrations for an array of tubes subject to fluid cross flow were simulated. The occurrence of fluid elastic instability in a 3×3 square array was also simulated successfully.

References

- (1) Blevins, R.D., Flow-Induced Vibration - second edition, (1990), p. 153-193, Van Nostrand Reinhold, New York.
- (2) Chen, S.S., Flow-Induced Vibration of Circular Cylindrical Structures, (1987), p. 386-437, Hemisphere Publishing Corporation, Washington.
- (3) Sadaoka, N. and Umegaki, K., A Numerical Method to Calculate Flow-Induced Vibration in a Turbulent Flow (1st Report, Derivation and Analysis of Oscillating Circular Cylinder), Trans. Jpn. Soc. Mech. Eng., (in Japanese), Vol. 60, No. 570, B (1994), p. 59-66.
- (4) Sadaoka, N. and Umegaki, K., A Numerical Method to Calculate Flow-Induced Vibration in a Turbulent Flow (2nd Report, Analysis of Vortex-Induced Vibration of Elastically Supported Cylinder and Investigation of Vibration Mechanism), Trans. Jpn. Soc. Mech. Eng., (in Japanese), Vol. 60, No. 577, B (1994), p. 48-55.
- (5) Thompson, J.F., Warsi, Z.U.A. and Mastin, C.W., Numerical Grid Generation, (1985), North-Holland, Amsterdam.
- (6) Hurt, C.W., Amsden, A.A. and Cook, J.L., An Arbitrary Lagrangian-Eulerian Computing Method for All Flow Speeds, J. Comp. Phys., Vol. 14 (1974), p. 227-231.
- (7) Umegaki, K. and Miki, K., A Numerical Method for the Solution of Three-Dimensional Incompressible Viscous Flow Using the Boundary-Fitted Curvilinear Coordinate Transformation and Domain Decomposition Technique, Trans. Jpn. Soc. Mech. Eng., (in Japanese), Vol. 56, No. 523, B (1989), p. 133-139.
- (8) Kawamura, T. and Nakao, T., Measurement of Turbulent Flow Downstream from Elbow Pipe, Trans. Jpn. Soc. Mech. Eng., (in Japanese), Vol. 62, No. 593, B (1996), p. 230-233.
- (9) Weaver, D.S. and Abd-Rabbo, A., A Flow Visualization Study of a Square Array of Tubes in Water Crossflow, J. of Fluids Eng., Vol. 107 (1985), p. 354-363.