ANL/RE/CP-101082

# Coupled CFD/CSM Vibration Design Methodology for Generation IV
## Long-Life Fuel and Component Design

**Contact Person:**


**David P. Weber***
**Associate Division Director**
**Reactor Engineering Division**
**9700 S. Cass Ave., RE/208**
**Argonne, IL 60439**
**(630) 252-8175**
**(630) 252-4780 FAX**
**dpweber@anl.gov**







*This work was performed under the auspices of the U.S. Department of Energy, Office of Technology Support Programs, under Contract No. W-31-109-ENG-38.

## DISCLAIMER

# Coupled CFD/CSM Vibration Design Methodology for Generation IV Long-Life Fuel and Component Design

D. P. Weber, S. S. Chen, C. Y. Wang, and T. Y. C. Wei
*Reactor Engineering Division*
*Argonne National Laboratory*
*Argonne, IL 60439 (USA)*
*dpweber@anl.gov; sschen88@aol.com; cywang@anl.gov; tycwei@anl.gov*

S. Jansson
*adapco (Analysis and Design Application Co., Ltd.)*
*Melville, New York 11747 (USA)*
*stefan@adapco.com*

## Abstract

Fluid-structure interaction is a cause of failures experienced in fuel rods and is of particular importance for all reactor components in Generation IV reactors which unanimously adopt a pool configuration. A Nuclear Energy Research Initiative (NERI) project proposal has been submitted to United States Department of Energy (USDOE) to develop an advanced design methodology to model fluid-structure interaction, predict its consequences, and guide the design of reactor components. The proposed design methodology is an integrated experimental/theoretical/numerical technique. Preliminary studies have been performed utilizing the CFD code STAR-CD coupled to a first-order structural mechanics model to explore the issues of coupled dynamic fluid/structure interactions - flow field, fluid forces, and instability of tubes - in the cross flow regime. The coupled tool has been used to predict the characteristics of complex dynamic fluid/structure interactions. It includes flow field in the wake of a tube or tube array, motion-dependent fluid forces for a tube, and fluid elastic instability of tube arrays. Specifically, the following calculated quantities have been compared with published experimental data, (a) Flow Field: flow velocity, fluid pressure, and fluid forces (steady and fluctuating components) of steady flow across a circular cylinder at subcritical and critical regions. (b) Fluidelastic Forces: a tube is excited at a given frequency and amplitude. The resulting flow field and fluid forces acting on the tube are calculated. (c) Fluidelastic Instability of a Tube Row: coupled vibration of a tube row is analyzed as a function of flow velocity. Those predicted quantities which have been compared, agree well with experimental data.

## 1 Introduction

Recent renewed collaborative international interest in the development of nuclear reactor power plants, which embody the lessons of the current fleet, have focused on the required features of the Generation IV plants [Carelli 2000, Greenspan 2000]. One of the major desired features of the Generation IV plants will be proliferation resistance. The common concept behind the design solutions proposed to incorporate this feature is long-life fuel which reduces tremendously, refueling needs and the opportunities for diversion. No leakers during a 15 year fuel cycle implies very tight specifications on fuel rod cladding fretting and wear caused by flow-induced vibrations during the in-core burn-up period. Vibrations may be caused by axial flow, assembly-to-assembly and sub-channel to sub-channel cross flow and vortex-shedding, wakes, etc. from fuel bundle internals such as spacer grids, etc. However, just as equally important as fuel bundle long life, in the design of Generation IV plants are the major reactor components which will be subject to similar fluid forces while immersed in pool configurations. The supercomputing simulation technology need is to produce an advanced design methodology accurate enough and sensitive enough to predict vibrations (amplitude and frequency) in the fuel bundle designs and in major reactor components immersed in pool configurations for the proposed Generation IV reactor. A NERI project has been proposed to USDOE to resolve this need. The objective of the NERI project is to produce a generic design methodology/coupling interface capable of coupling state-of-the-art Computational Fluid Dynamics (CFD) design tools with Computational Structure Mechanics (CSM) tools to perform design analyses of fluid-structure interaction and flow-induced vibrations which lead to structural fretting and wear for Generation IV long-life fuel and component designs. Two state-of-the-art design tools, the STAR-CD CFD code and the PSTRUC CSM code have been selected to develop and demonstrate this design methodology. However, the proposed advanced design methodology also incorporates experimental information together with the CFD results in an integrated experimental/theoretical/numerical technique for the design evaluation of fluid structure interaction phenomena.

In the past, flow field, fluid forces, and tube responses were often measured experimentally and the filtered results were used as input in theoretical-based models to perform design analysis. This preliminary study paper utilizing STAR-CD shows that alternatively, supercomputing analysis methods have the potential to provide this input in a cost-effective manner. Preliminary studies reported in this paper, have been performed utilizing STAR-CD coupled to a first-order structural mechanics model to explore the issues of coupled dynamic fluid/structure interactions - flow field, fluid forces, and instability of tubes - in the cross flow regime. A methodology is also presented to show how these results would be used in the proposed integrated experimental/theoretical/numerical technique. The coupled tool has been used to predict the characteristics of complex dynamic fluid/structure interactions. It includes flow field in the wake of a tube or tube array, motion-dependent fluid forces for a tube, and fluid elastic instability of tube arrays. Specifically, the following analyses are performed and compared with published experimental data, (a) Flow Field: flow velocity, fluid pressure, and fluid forces (steady and fluctuating components) of steady flow across a circular cylinder are presented at subcritical and critical regions. The predicted Strouhal number using two different turbulence models agree well with experimental data. (b) Fluidelastic Forces: a tube is excited at a given frequency and amplitude. The resulting flow field and fluid forces acting on the tube are calculated. Based on this method, fluid added mass, fluid damping, and fluid stiffness can be predicted. Once these are obtained from CFD calculations, the response of the tubes can be predicted using the unsteady flow theory. (c) Fluidelastic Instability of a Tube Row: coupled vibration of a tube row is analyzed as a function of flow velocity. Once flow velocity is increased to a certain value, fluidelastic instability occurs. Different instability mechanisms are seen depending on system parameters. The predicted critical flow velocity agrees well with experimental data. These results impart confidence to the proposed advanced vibration design methodology.

Although additional development is needed, the preliminary results are very encouraging. Once the proposed NERI work is completed, it can be applied to the design assessment of nuclear reactor system components such as steam generators as well as core fuel bundle designs. In addition, the method can also be applied in the evaluation of existing plants to identify the root cause of flow-induced vibration problems, and the prediction of the remaining life of current nuclear power plants from the standpoint of flow-induced vibration.

## 2 State-of-the-Art Review

Since the advent of commercial nuclear energy, nuclear power engineers have been faced with flow-induced vibration (FIV) problems [Chen 1987, Paidoussis 1980]. Although FIV problems are not overwhelming and generally not considered as a major aspect of the design, there is no doubt that flow-induced vibration problems in nuclear power stations will continue to receive considerable attention because of economic loss, safety, and past costly incidents [Blevins 1976, Yamaguchi 1997]. The core of nuclear reactors consists of slender cylindrical rods subjected to external axial flow of the primary coolant and its vibration due to axial flow is normally not a concern. However, for a fuel assembly designed for 15 year operation, long-term fretting wear problems cannot be ignored. Incidental situations associated with various excitation mechanisms such as baffle jetting and leakage flow induced instability of fuel assemblies which can be very detrimental have led to failure of the fuel rods.

The use of numerical simulations in understanding the complex interactions of a coupled fluid-rod system can be divided into two groups; those specifically focusing on new numerical techniques, and those specifically focusing on the fluid/rod physics for specific applications. Various methods can be used to solve the Navier-Stokes equations: finite element approach; time-integration approach based on a time-dependent coordinate; ALE (Arbitrary Lagrangian-Eulerian) finite difference method; finite volume model; vortex-in-cell method, parallel spectral element/Fourier method, etc. These involve the solution of Navier- Stokes equations with moving boundary conditions and their interactions with fuel rods. In formulations such as these, where the unsteady flow induces changes in the rod, a second independent set of equations governing the rod motion is used. An economical and reliable computer program for dynamic interactions of fuel rods and flow is needed. This problem should be addressed by coupling the fluid-rod systems by reducing iteration between the two codes.

At this time, for practical applications, the flow field is calculated based on rigid structures; i. e., the coupling effect of structural motions and flow is not considered in calculating the flow field. There are CFD codes, which can be applied to study the flow field and are then applied to fluid-structure interaction problems. Coupled fluid/structure codes have also been developed for different applications. However, at this time, it appears that there is no single code available for predicting completely and accurately the dynamic interactions of fuel rods with flow.

At present, an assessment of flow-induced vibrations is considered early in the design process and is incorporated with other design procedures [Au-Yang 1984]. In many cases, model testing is used to supplement the design process to assure that detrimental behavior, from a flow-induced vibration point of view, will not occur after the component is fabricated. While these procedures attempt to minimize the chance of adverse performance of components, the ability to extrapolate analytical/numerical design techniques and/or model testing, to actual operation of the plant remains a concern. Therefore, test of components and/or vibration measurements of components in the reactor system are used to provide additional assurance. There are no definite licensing requirements for vibration response, as long as a design can be provided to meet the safety operational limits, except that preoperational and component testing is required for first-of-a-kind reactor system [NRC 1971].

Quite significant progress has already been achieved in several aspects of predictive methods for assessing FIV problems in the past quarter of a century. Indeed, numerical tools for computing the linear and nonlinear vibrations of complex components have been developed and then used by the designers. It has demonstrated the interest of adapting the modern sophisticated numerical techniques of turbulent flow calculation to FIV problems. Furthermore, a large class of the FIV problems that are of practical importance in nuclear reactors can be modeled reasonably well using the sophisticated tools, such as direct or large eddy simulations. However, the prediction method is not sufficiently accurate and cost effective and can not be used to predict all possible excitation mechanisms. At present, there is no coupled CFD/CSM code available to be used for the design and assessment of fuel assembly for the operational requirement of 15 years.

The development of a coupled fluid/structure code based on new numerical techniques for the assessment of such nuclear fuel designs will provide the needed answers: a design tool for the fuel assembly and optimization of the fuel assembly with respect to flow-induced vibration, and an operational tool for plant operation monitoring. STAR-CD is powerful commercial CFD tool [Computational Dynamics 1998] for fluids and thermal analysis and has been designed for use in a CAE environment. It has many advantageous attributes and contains provisions for all necessary modeling aspects of the proposed application especially: allowance for compressible viscous flow, a wide selection of turbulence models including Large Eddy Simulation (LES), ability to deal with moving boundary problems, an open architecture for interface with other codes, and efficient parallel processing capability. STAR-CD was, therefore, selected as the CFD code of choice for this preliminary study of coupled CFD/CSM design methodology development for analyzing Generation IV reactor fluid-structure interaction issues.

## 3 Integrated Experimental/Theoretical/Numerical Design Methodology

The proposed integrated experimental/theoretical/numerical technique for the advanced design methodology relies significantly on, the proper interpretation and extraction of parameters from CFD results. For this preliminary study, experimental data were therefore chosen for the evaluation of CFD results based on the consideration of mathematical models, motion-dependent fluid forces, available test results, and future needs dictated by the proposed design methodology which is presented below.

### Mathematical Models

Although different terms have been used in different disciplines for dynamic fluid/structure interactions, mathematically, different types of fluid/structure response can be described by a standard equation [Chen 1987]:

$$[M_s + M_f]\{\ddot{Q}\} + [C_s + C_f]\{\dot{Q}\} + [K_s + K_f]\{Q\} = \{G\} \tag{1}$$

where $\{Q\}$, $\{\dot{Q}\}$, and $\{\ddot{Q}\}$ are generalized structural displacement, velocity, and acceleration, respectively; mass matrices include structural mass $[M_s]$ and added mass $[M_f]$; damping matrices include structural damping $[C_s]$ and fluid damping $[C_f]$; stiffness matrices include structural stiffness $[K_s]$ and fluid stiffness $[K_f]$; and $\{G\}$ is fluid excitation forces. The dynamic characteristics of structural components in different flow conditions can be analyzed on the basis of the unified model given in Eq. 1. Structural responses include lock-in resonance, turbulence-induced vibration, fluid-damping-controlled instability, fluid-stiffness-controlled instability, and parametric and combination resonance. Their motions may be periodic oscillations, random vibrations, or chaotic motions.

There are several approaches to these types of problems. In one approach, system components are tested to determine whether they are acceptable. In another approach, the detailed interaction of two fields, fluid flow and structure component, is ignored and only the resultant effect is considered. In these two approaches, the effect of fluid forces is not completely accounted for. The third approach is to consider the dynamic interaction of fluid and structure based on the Navier-Stokes equation and theory of elasticity. The third method will provide the most complete solution to the problem. Some simplified models based on these approaches have been used.

Quasistatic-Flow Theory: At any instant in time, the fluid-dynamic characteristics of a structure oscillating in a flow are equal to the characteristics of the same stationary structure, whose configuration is identical to the actual instantaneous configuration. The fluid forces depend on the deviation from a reference state of steady flow [Connors 1970, Blevins 1976].

Quasisteady-Flow Theory: At any instant in time, the fluid-dynamic characteristics of a structure moving in flow are equal to the characteristics of the same structure moving with constant velocities equal to the actual instantaneous values. The fluid forces depend on structure configuration and are proportional to its motion [Price and Paidoussis 1984, Granger and Paidoussis 1996].

Unsteady-Flow Theory: The unsteady fluid forces acting on a structure are the same as those acting on a structure that is performing periodic oscillations. In general, the fluid-force components are functions of structure displacements, velocities, and accelerations [Tanaka and Takahara 1981; Lever and Weaver 1984; Chen 1987].

The critical elements of these theories, which the proposed integrated experimental/theoretical/numerical technique is based on, are fluid-excitation forces and motion-dependent fluid forces. Two methods have been used to measure those fluid forces: (1) Indirect Method: fluid forces are calculated from structure responses, such as tube displacements and accelerations [Granger 1990]. (2) Direct Method: fluid forces are measured directly [Tanaka and Takahara 1981, Scanlan and Simiu 1978, Chen et al 1987].

## Motion-Dependent Fluid Forces

Motion-dependent fluid forces are the key components in determining structural stability in fluid flow. Consider a group of tubes vibrating in a flow, as shown in Fig. 1. The axes of the tubes are parallel to one another and perpendicular to the x-y plane. The radius R of each tube is the same, and the fluid is flowing with a gap flow velocity U. The displacement components of Tube j in the x and y directions are $u_j$ and $v_j$, respectively. The motion-dependent fluid-force components acting on Tube j in the x and y directions are, respectively, $g_j$ and $h_j$, and are give by Chen (1987) as

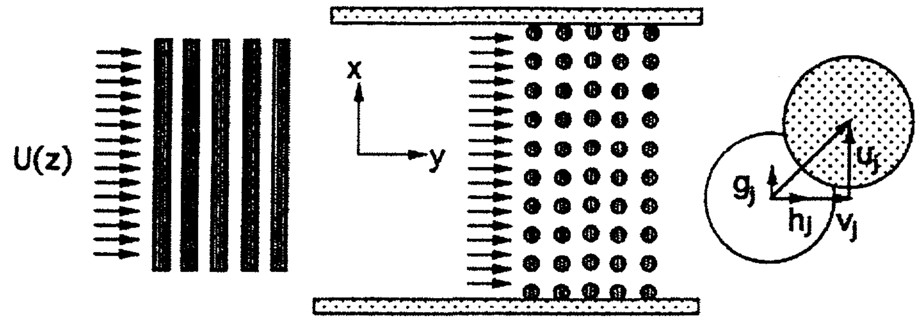


Figure 1. Tube arrays in crossflow (two views) and schematic representation of displacement components of Tube j and motion-dependent fluid-force components acting on Tube j.

$$g_j = -\rho\pi R^2 \sum_{k=1}^{n}\left(\alpha_{jk}\frac{\partial^2 u_k}{\partial t^2} + \sigma_{jk}\frac{\partial^2 v_k}{\partial t^2}\right) + \frac{\rho U^2}{\omega}\sum_{j=1}^{n}\left(\alpha'_{jk}\frac{\partial u_j}{\partial t} + \sigma'_{jk}\frac{\partial v_k}{\partial t}\right)$$
$$+\rho U^2 \sum_{j=1}^{n}\left(\alpha''_{jk}u_k + \sigma''_{jk}v_k\right) \tag{2}$$

and

$$h_j = -\rho\pi R^2 \sum_{k=1}^{n}\left(\tau_{jk}\frac{\partial^2 u_k}{\partial t^2} + \beta_{jk}\frac{\partial^2 v_k}{\partial t^2}\right) + \frac{\rho U^2}{\omega}\sum_{k=1}^{n}\left(\tau'_{jk}\frac{\partial u_j}{\partial t} + \beta'_{jk}\frac{\partial v_k}{\partial t}\right)$$
$$+\rho U^2 \sum_{j=1}^{n}\left(\tau''_{jk}u_k + \beta''_{jk}v_k\right) \tag{3}$$

where $\rho$ is fluid density; t is time; $\omega$ is circular frequency of tube oscillations; $\alpha_{jk}$, $\beta_{jk}$, $\sigma_{jk}$, and $\tau_{jk}$ are added-mass coefficients; $\alpha'_{jk}$, $\beta'_{jk}$, $\sigma'_{jk}$, and $\tau'_{jk}$ are fluid-damping coefficients; and $\alpha''_{jk}$, $\beta''_{jk}$, $\sigma''_{jk}$, and $\tau''_{jk}$ are fluid-stiffness coefficients. Motion-dependent fluid forces depend on deviation from a reference state of steady flow, which have been grouped into three theories: quasi-static flow theory, quasi-steady flow theory, and unsteady flow theory. Each theory has been studied in the last three decades.

Motion-dependent fluid forces can be measured or calculated for tube arrays. Fluid-force coefficients can be determined by measuring or calculating the fluid forces acting on the tubes that are due to oscillations of a particular tube. The particular tube can be excited in the x and y directions. For example, if Tube k is excited in the x direction, its displacement in the x direction is given by

$$u_k = u \cos \omega t. \tag{4}$$

The fluid-force components acting on Tube j in the x and y directions are

$$g_j = \frac{1}{2}\rho U^2 c_{jk} \cos(\omega t + \phi_{jk}) u \tag{5}$$

and

$$h_j = \frac{1}{2}\rho U^2 d_{jk} \cos(\omega t + \psi_{jk}) u , \tag{6}$$

where $c_{jk}$ and $d_{jk}$ are the fluid-force amplitudes and $\phi_{jk}$ and $\psi_{jk}$ are the phase angles by which the fluid forces acting on Tube j lead to the displacement of Tube k.

With Eqs. 2 to 6, the fluid force coefficients are

$$\begin{aligned} \alpha''_{jk} &= \frac{1}{2} c_{jk} \cos \phi_{jk} - \frac{\pi^3}{Ur^2} \alpha_{jk} , \\ \tau''_{jk} &= \frac{1}{2} d_{jk} \cos \psi_{jk} - \frac{\pi^3}{Ur^2} \tau_{jk} , \\ \alpha'_{jk} &= \frac{1}{2} c_{jk} \sin \phi_{jk} , \end{aligned} \tag{7}$$

and

$$\tau'_{jk} = \frac{1}{2} d_{jk} \sin \psi_{jk} , \tag{8}$$

where Ur is the reduced flow velocity $(Ur = \pi U / \omega R)$.

The added-mass coefficients $\alpha_{jk}$ and $\tau_{jk}$ in Eqs. 7 can be calculated by applying the potential-flow theory [Chen and Chung 1976]. Then $\alpha'_{jk}, \alpha''_{jk}, \tau'_{jk}$, and $\tau''_{jk}$ can be calculated from Eqs. 7 and 8. If Tube k is excited in the y direction, using the same technique, we can obtain force coefficients $\beta'_{jk}, \beta''_{jk}, \sigma'_{jk}$, and $\sigma''_{jk}$ in the same manner. This above method is used to obtain motion-dependent fluid force coefficients both experimentally and numerically from the CFD flow field results. These coefficients can then be used together with the model of Eq. (1) in the proposed design methodology. To the authors' knowledge, this method has not been combined with CFD calculations.

Tests

Many experiments have been performed at ANL including basic research, development of design guidelines, assessment of new designs, diagnosis of flow-induced vibration mechanism and identification of root causes of damage, and development of a cost effective quick fix of flow-induced vibration problems. Three series of experiments were selected which would best test the three important phenomena essential to the application of the proposed design methodology. Specifically, the following phenomena in the related tests were compared with STAR-CD results.

a. Flow Field: Flow velocity, fluid pressure, and fluid forces (steady and fluctuating components) of steady flow across a circular cylinder. Extensive data are available in the literature.

b. Fluidelastic Forces: A tube in crossflow is excited at a given frequency and amplitude. The resulting fluid forces acting on the tube are measured.

c. Fluidelastic Instability of a Tube Row: Coupled fluid/structure vibration analysis was performed in order to predict the critical onset velocity and tube response for a row of five flexible tubes. A parametric study of tube oscillation amplitudes and frequencies was measured based on gap flow velocity.

These experimental data are summarized by the following information:

- Vortex shedding frequencies as a function of Reynolds number and tube geometry, such as different tube arrays [Chen 1987]. The data used in this paper is for a single cylinder only.

- Motion-dependent fluid forces are available for tube row, square and triangular arrays. In this study, only the data for a single tube is used [Chen, Cai, and Zhu, and Cai 1995].

- Fluidelastic instability of tube arrays have been tested for different tube arrays. A classical case is a tube row in cross flow. Depending on mass-damping parameter, different stability mechanisms, fluid-damping-controlled and fluid-stiffness-controlled instabilities, are observed and there are hystersis and multiple stability regions. This case is chosen as the benchmark case for evaluation. The instability map is given in Fig. 2 [Chen and Jendrzejczyk 1982 and 1983].

## 4 Numerical CFD Model and Comparison Results

### Numerical Model

The governing partial differential equations for momentum, pressure and turbulent quantities were solved in order to describe the fluid flow motion. The simulations were carried by adopting STAR-CD, which is a general-purpose commercial CFD computer code suitable for predicting a large variety of industrial fluid flow problems. The flows studied in this paper were assumed to be statistically two-dimensional flows, although the unsteady behavior of the flow is taken into account in the calculations. Two different non-linear turbulence models, based on the turbulent (or eddy) viscosity concept, were tested in this study; a quadratic and a cubic approach. These models were used in combination with a two-layer near-wall treatment.

The analysis of the tube row motions was modeled as a linear spring and dashpot system, where the tubes were assumed to have no deformation. Given the tube material, dimensions and natural frequency in air, the stiffness coefficient and the effective mass were obtained from a finite element

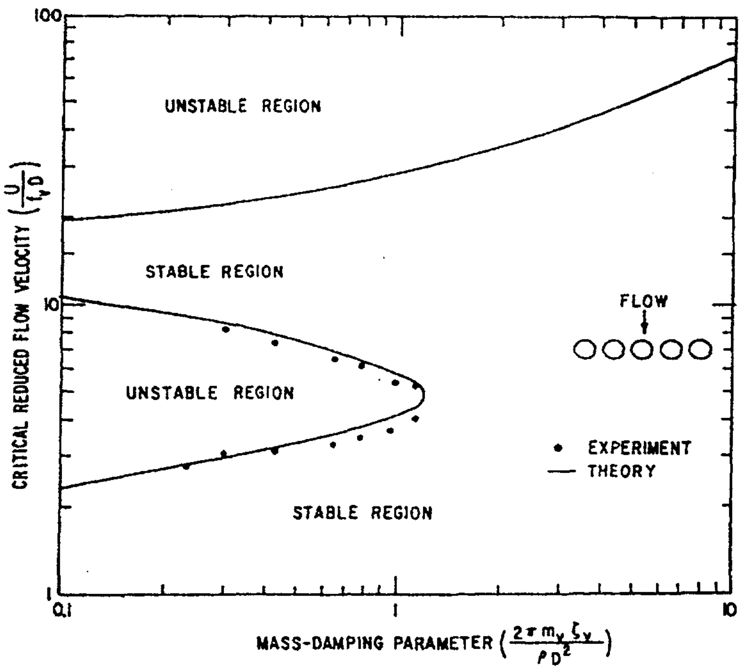


Figure 2. Stability map for a tube row in crossflow (U = flow velocity, $f_v$ = natural frequency, D = tube diameter, ρ = fluid density, $m_v$ = tube mass per limit length, $\zeta_v$ = tube damping ratio).

model. At the end of each time step, the pressure distribution on the tube surfaces was integrated in order to obtain the magnitude and direction of the forces acting on the tubes. Once the forces and their direction were known, a linear equation describing the tube motion in the drag- and lift direction was solved. It should be mentioned that an explicit coupling was assumed between the flow field and the tube motions. Due to the calculated tube motions, the mesh needs to be modified at the end of each time step, where the cell vertices are moved according to the new positions of the tubes. This moving mesh capability was performed by PROSTAR, which is the pre- and post processor included in the STAR-CD software package.

## Results and Discussion

As discussed in Section 3, three sets of experiments were analyzed with the numerical model described above; (1) Rigid single tube in a cross flow, (2) Cross flow past a single tube with forced motion, (3) Fluid/structural analysis of a tube row in a cross flow.

For experiment set (1), numerical simulations were performed of the flow field around a fixed rigid single tube according to the experimental setup given by Chen (1995). Predicted Strouhal numbers, drag- and lift coefficients were compared to corresponding experimental observations. The predicted instantaneous velocity field at $Re \approx 3.5 \times 10^3$ is shown in Fig. 3, clearly revealing the unsteady behavior of the wake flow. According to the experimental results given by Chen (1987), the Strouhal (Sr) number should be around 0.20-0.23 for this Reynolds number. The quadratic model results in a slightly higher value of the Strouhal number (0.23) than the cubic model (0.21) but a conclusion is that both models are capable of predicting the Strouhal number fairly accurate. Additional simulations show good agreement between predictions and experiments for $Re \approx 10^3$-$10^6$. In Fig. 4, the lift and drag coefficients for $Re \approx 3.5 \times 10^3$ are shown.

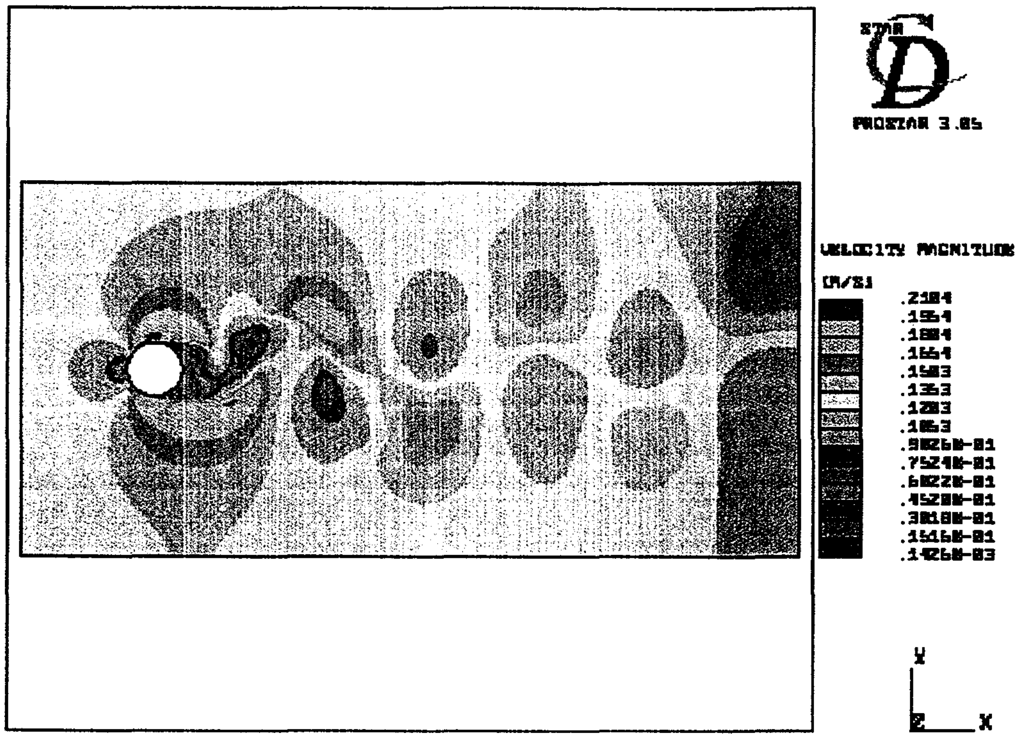


Figure 3. Instantaneous velocity field at Re=3,500.

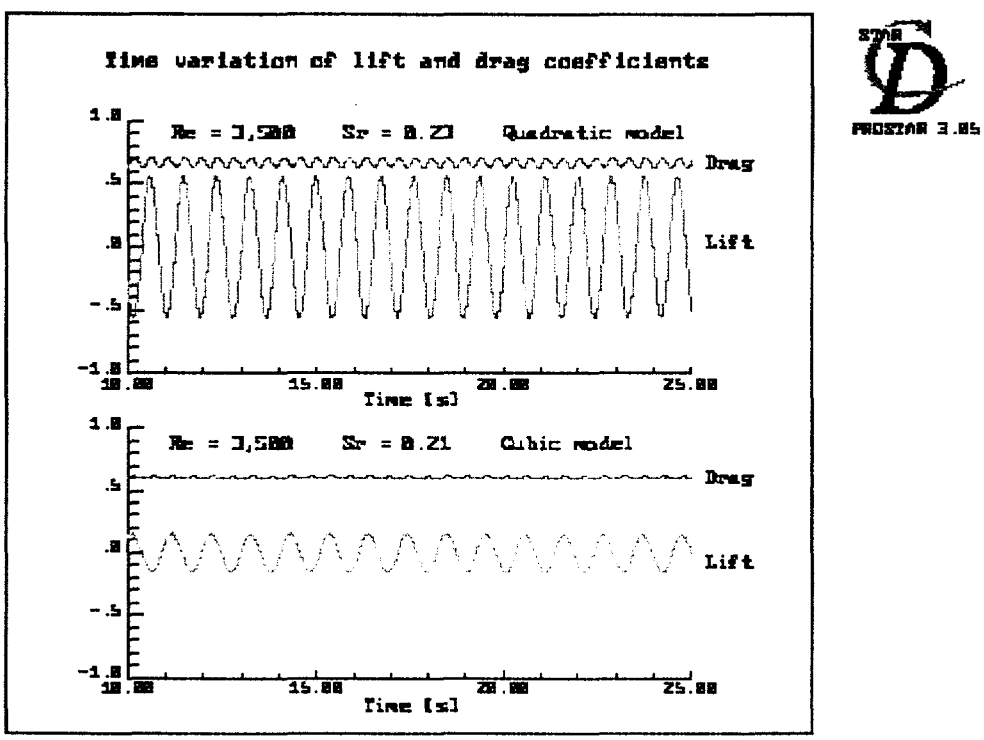


Figure 4. Fluctuating lift and drag coefficients.

According to experiments by Norberg et al. (1987), the time mean value of the drag coefficient at this Reynolds number should be around 1 (0.98 ± 0.05). Hence, both the quadratic (≈ 0.67) and the cubic (≈ 0.61) model seem to underpredict the drag, which indicates an underestimation of the recirculation zone downstream of the tube. However, the predicted frequency of the drag coefficient variation is approximately twice that of the lift coefficient, which is in good agreement with experimental observations.

For experiment set (2), predictions of the flow field around a rigid single tube with a prescribed motion in the lift (y) direction were performed. Parametric studies with different tube motion frequencies and amplitudes were performed and compared to the experimental results reported by Chen (1995). The tube was excited in the lift (y) direction, where the tube motion was described by $y = d_o \cdot \cos(\omega \cdot t)$, where $d_o$ is the amplitude, $\omega$ is the circular frequency and t is time. In each case, the inlet velocity, $U_{in}$ was set to 0.127 m/s and the tube was excited at three different values of the reduced velocity, Ur = 5, 6 and 7, respectively (the corresponding oscillation frequencies, $f_{cyl} = \omega/2\pi$ being 1.0, 0.83 and 0.71 Hz). For each value of Ur, three different oscillation amplitudes were applied, $d_o$ = 1.2, 2.4 and 3.6 mm. The time evolution of the excitation amplitude, lift coefficient and the cross-stream mean velocity component for Ur = 5 and $d_o$ = 1.2 mm is shown in Fig. 5. Various frequencies of interest can be extracted from these data. The prescribed oscillation amplitude is scaled by the tube diameter, D and the vortex shedding frequency, $f_{vortex}$ is calculated from the variation of the cross stream mean velocity component, V in the wake, scaled by the inlet velocity $U_{in}$. Further details of the predicted frequencies for each case are given in Table 1 below, where $\Phi_1$, is the phase angle. These are based on detailed tabulated data similar to those shown in Fig. 5. In future work, equations (2) to (8) of the theoretical model proposed for the integrated experimental/theoretical/numerical design methodology, will be used to determine the added-mass coefficient, fluid-damping coefficients and fluid stiffness coefficients for the motion dependent forces. These values will then be compared to the experimental values measured and presented for this series of experiments, experiment set (2), in [Chen, Zhu and Cai 1995].

For experiment set (3), a coupled fluid/structure vibration analysis was performed in order to predict the critical onset velocity and tube excitation for a row of five flexible tubes. The setup is that given in the experimental study by Chen (1982). A parametric study of tube oscillation amplitudes and frequencies was performed based on gap flow velocity. The transient behavior of the wake flow downstream of the tube row can be seen in Fig. 6, clearly indicating the complex irregularity of the flow downstream of the tubes. The tubes are referred to as no. 1–5, starting from the bottom one.

As already mentioned, one objective was to predict the critical velocity, $U_{cr}$ at which the tubes start to oscillate. The gap velocity, $U_g$ is defined as:

$$U_g = \frac{U_{in}}{1 - \frac{D}{T}} \tag{9}$$

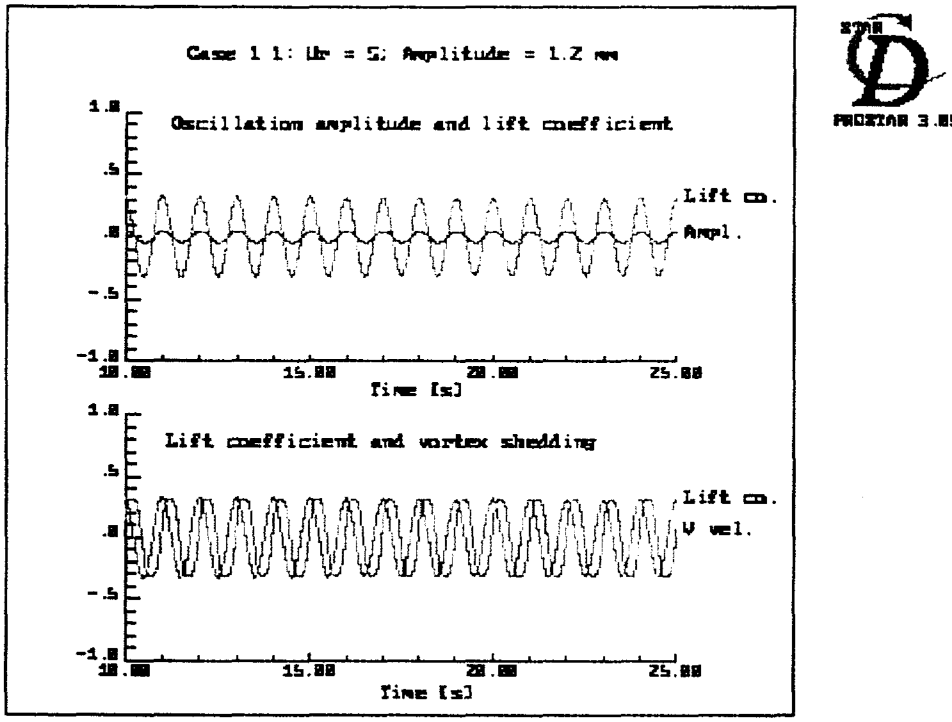


Figure 5. Results used to derive frequencies of cylinder oscillation, lift coefficient and vortex shedding.

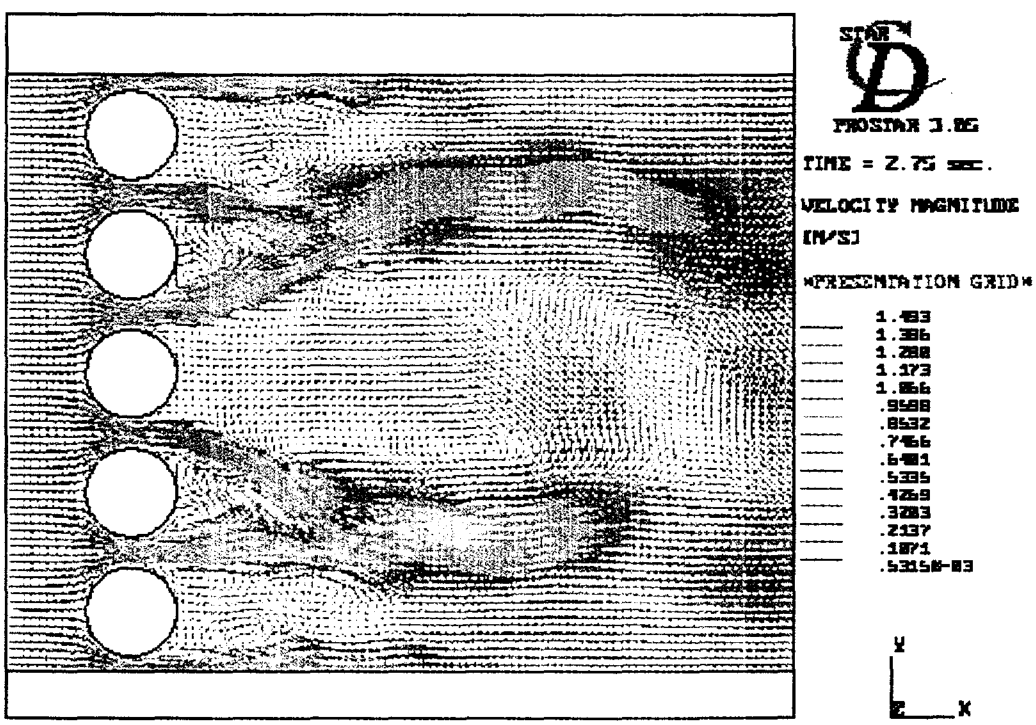


Figure 6. Instantaneous velocity vectors at T = 2.75 seconds.

Table 1. Frequencies of Vortex Shedding and Lift Force Versus Tube Motion

| Ur | $d_o$ [mm] | $f_{cyl}$ [Hz] | $f_{lift}$ [Hz] | $f_{vortex}$ [Hz] | $\Phi_l$ [°] |
|---|---|---|---|---|---|
| 5.0 | 1.2 | 1.0 | 1.0 | 1.0 | 0 |
| | 2.4 | 1.0 | 1.0 | 1.0 | 101 |
| | 3.6 | 1.0 | 1.0 | 1.0 | 0 |
| 6.0 | 1.2 | 0.83 | 0.97 | 0.97 | N/A |
| | 2.4 | 0.83 | 1.1 | 1.1 | N/A |
| | 3.6 | 0.83 | 0.83 | 0.83 | 57 |
| 7.0 | 1.2 | 0.71 | 0.97 | 0.97 | N/A |
| | 2.4 | 0.71 | 1.1 | 1.1 | N/A |
| | 3.6 | 0.71 | 0.97 | 0.98 | N/A |

where T/D is the pitch-to-diameter ratio (in this case T/D =1.35). The gap velocity was varied between 1.08–1.20 m/s in the calculations. The simulations indicate that the value of the critical velocity, $U_{cr}$ is around 1.19 m/s. A small increase of the velocity from this value results in dramatically increased oscillation amplitudes and finally in tube impact, as is shown in Fig. 7, where $U_g$ = 1.20 m/s. Tube No.'s 1, 3 and 5 oscillate in phase, while there is a phase difference, $\Phi$ of 170 degrees compared to Tube No. 2 and Tube No. 4. Tabulated values of tube oscillation amplitudes and frequencies as a function of gap velocity are given in Table 2 below. The instability onset and oscillating frequency is very well captured by the numerical method, in spite of the simplifications made in the computations. The predicted critical gap velocity and tube oscillation frequency was 1.19 m/s and 23 Hz, respectively. The corresponding experimental values obtained in the study by Chen et al. (1982) were 1.19 m/s and 24.3 Hz. The lift coefficient $C_L$ for each tube at $U_g$ = 1.16 m/s. is shown in Fig. 8 and it shows that there is a strong coupling between lift coefficient and tube oscillation, as expected.

The conclusion from this study is that accurate information regarding fluid/structure interaction may be obtained by adopting rather robust numerical methods, such as finite-difference technique, simple turbulence models and a single degree of freedom system for the description of tube motions. This is encouraging for the use of the proposed integrated experimental/theoretical/numerical design methodology to solve practical flow problems, relevant for industrial applications. This study presents the results for structures in cross flow. The approach is applicable to other flow conditions such as axial flow and skewed flow. In a fuel bundle, the dominant flow is axial flow in which the dominant structural axis is parallel to flow direction. The method presented here can be applied to fuel bundles similar to the cross flow cases. It should be pointed out that the method presented has several advantages: (1) Two or three dimensional simulations can be performed for coupled fluid/structural interactions in flow; this is similar to those by other researchers, e.g. Yamaguchi et al (1997) and Kassera and Strohmeier (1996). (2) Motion-dependent fluid forces can be calculated using a CFD code similar to experiments to obtain fluidelastic force coefficients which can then be applied to a mathematical model of practical coupled fluid-structure systems. (3) By using the integrated analytical/experimental/numerical technique, flow-induced vibration and instability of complicated practical systems can be predicted to satisfy the need in design assessment while only using CFD may not resolve practical issues such as how many rods in fuel bundles and tubes in steam generators should be modeled. At this time, in some

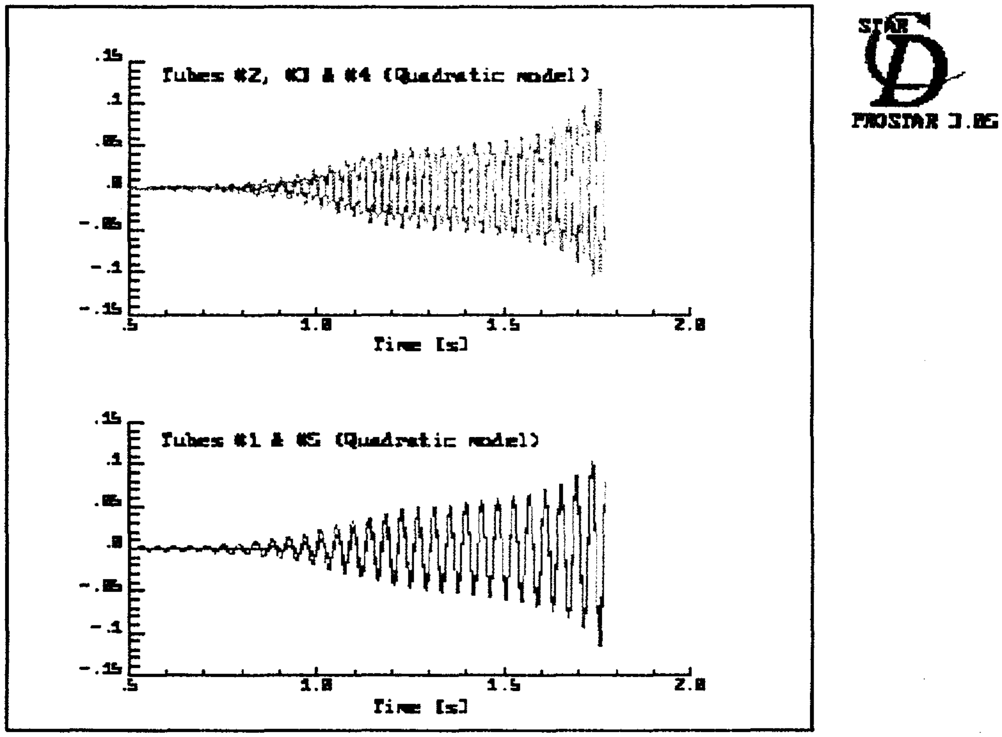


**Figure 7. Tube displacements in the cross stream direction vs. time. Ugap = 1.20 m/s (Uin = 0.31 m/s).**

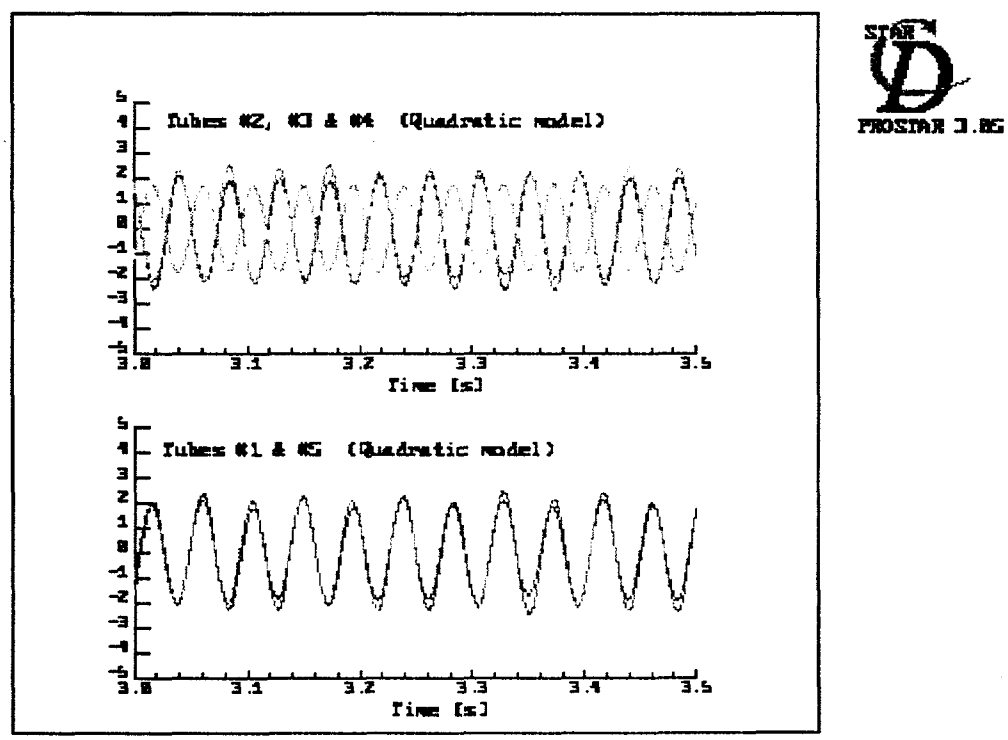


**Figure 8. Lift coefficient vs. time. Ugap = 1.16 m/s (Uin = 0.30 m/s).**

practical problems, it appears only the integrated approach will provide the most effective way in the resolution of the difficulties associated with flow-induced vibration and instability.

Table 2. Tube Oscillation Amplitudes and Frequencies as a Function of Gap Velocity

| | $U_g$=1.08 m/s | | $U_g$=1.16 m/s | | $U_g$=1.18 m/s | |
|---|---|---|---|---|---|---|
| | d/D [%] | f [Hz] | d/D [%] | f [Hz] | d/D [%] | f [Hz] |
| Tube 1 | 0.025 | 20 | 1.5 | 22.7 | 2.0 | 23 |
| Tube 2 | 0.05-0.07 | 10 | 1.5 | 22.7 | 2.0 | 23 |
| Tube 3 | 0.1 | 10 | 1.2 | 22.7 | 1.5 | 23 |

## 5 Conclusion

The 20th Century has seen important developments in the field of fluid-structure interactions. Various design guides are available from professional societies, research and development institutions, regulatory agencies, and private companies. Even though there are significant gaps in codified knowledge, designers have been able to develop many system designs that have provided useful service without significant problems. From the practical point of view, questions of immediate concern are; what is the critical flow velocity and how much vibration is too much? A complete answer to these questions is not always available for general cases. Furthermore, interactions of fluidelastic instability with vortex shedding, flow separation, or turbulence have not been seriously studied in the past. More extensive measurements and development of CFD methods for motion-dependent fluid forces should be performed for various geometries. Based on the characteristics of experimental data and CFD results on fluid-force coefficients, some questions and unresolved issues about fluidelastic instability can be resolved.

This paper presents an integrated experimental/theoretical/numerical technique for the evaluation of fluid-structure interaction phenomena. In the past, flow field, fluid forces, and tube responses were often measured experimentally. Using CFD methods, these can be analyzed in a cost-effective manner. Preliminary studies have been performed utilizing STAR-CD coupled to a first-order structural mechanics model to explore the issues of coupled dynamic fluid/structure interactions - flow field, fluid excitation forces, motion-dependent fluid forces, and instability of tubes - in the cross flow regime. The tool has been used to predict the characteristics of complex dynamic fluid/structure interactions. It includes flow field in the wake of a tube or tube array, motion-dependent fluid forces on a tube, and fluid elastic instability of tube arrays. Although additional development is needed, the preliminary results are very encouraging. This is the first study using the method for the calculation of motion-dependent fluid forces. Once this work is completed, it can be applied to design assessment of various system such as nuclear power plants, process plants, aerospace, offshore and undersea technology. In addition, the method can also be applied in the evaluation of existing plants to identify the root cause of flow-induced vibration problems, and the prediction of the remaining life of current plants from the standpoint of flow-induced vibration.

STAR-CD is a powerful CFD tool for fluids and thermal analysis and has been designed for use in a CAE environment, because it contains provisions for all necessary modeling aspects of various applications. Future research is required to continue the development of CFD codes capable of predicting motion-dependent fluid forces, structural response to high Reynolds number, separated flows with moving boundaries and developing these into practical engineering tools. Given the complexity of the problems and the limitations of current computer technology, it seems that such codes will be difficult to predict all cases for some time. In the meantime, careful experimentation and CFD development is required to improve our understanding of excitation mechanisms, establish more precise design guidelines and provide benchmark data for evaluating CFD codes.

**Acknowledgments**

This work was performed under the auspices of the U.S. Department of Energy, Office of Nuclear Energy, Science and Technology under Contract W-31-109-ENG-38.

**References**

M. K. Au-Yang, 1984, "Flow Induced Vibration: Guidelines for Design, Diagnosis, and Troubleshooting of Common Power Plant Components," ASME Symposium on Flow-Induced Vibration, Vol. 3, pp. 119-138.

R. D. Blevins, 1976, Flow-Induced Vibration, John Wiley & Sons, Inc., New York.

M. D. Carelli, et al.,2000, "IRIS, International New Generation Reactor," Proceedings of ICONE-8 Conference, April 2-6, 2000, Baltimore, USA.

S. S. Chen, and H. Chung, 1976, "Design Guide for Calculating Hydrodynamic Mass, Part I: Circular Cylindrical Structures," ANL-CT-76-45, Argonne National Laboratory, Argonne, IL.

S. S. Chen and J. A. Jendrzejczyk, 1982, "Experiment and analysis of instability of tube rows subject to liquid crossflow," Journal of Applied Mechanics, Transactions of the ASME, Vol. 49, pp. 704–709.

S. S. Chen, and J. A. Jendrzejczyk, 1983, "Stability of Tube Arrays in Crossflow," Nuclear Engineering and Design, Vol. 75, pp. 351-374.

S. S. Chen, 1987, Flow-Induced Vibration of Circular Cylindrical Structures, Hemisphere Publishing Corp., New York.

S. S. Chen, S. Zhu, and Y. Cai, 1995, "An Unsteady Flow Theory for Vortex-Induced Vibration," Journal of Sound and Vibration, Vol. 184(1,), pp. 73-92.

Computational Dynamics, 1998, STAR-CD, V3.05 Methodology Manual.

H. J. Connors, Jr., 1970, "Fluidelastic Vibration of Tube Arrays Excited by Cross Flow," in Flow-Induced Vibration of Heat Exchangers, ed. D. D. Reiff, ASME, New York, pp. 42-56.

S. Granger, 1990, "A Global Model for Flow-Induced Vibration of Tube Bundles in Cross-Flow," ASME Publication, PVP-Vol. 189, New York, pp. 139-151.

S. Granger, and M. P. Paidoussis, 1996, "An Improvement to the Quasi-Steady Model with Application to Cross-Flow-Induced Vibration of Tube Arrays," J. Fluid Mechanics, Vol. 320, pp. 163-184.

E. Greenspan, et al, 2000, "The Encapsulated Nuclear Heat Source Reactor Concept," Proceedings of ICONE-8 Conference, April 2-6, 2000, Baltimore, USA.

V. Kassera and K. Strohmeier, 1996, "Simulation of Cross Flow Induced Tube Bundle Vibrations," ASME Publication, PVP-Vol. 328, Flow-Induced Vibration, Ed. by M. J. Pettigrew, ASME, New York, pp. 39-46.

J. H. Lever, and D. S. Weaver, 1984, "On the Stability Behavior of Heat Exchanger Tube Bundles: Part 1 - Modified Theoretical Model, Part 2 - Numerical Results and Comparison with Experiments," Proceedings ASME Symposium on Flow-Induced Vibrations, Vol. 2, ASME, New York, pp. 83-116.

C. Norberg and B. Sunden, 1987, "Turbulence and Reynolds numbers effects on the flow and fluid forces on a single cylinder," Journal of Fluids and Structure, Vol. 1, pp. 337-357.

M. P. Paidoussis, 1980, "A Flow-induced vibration in nuclear reactors and heat exchangers," In Practical Experiences with Flow-Induced Vibration. Ed. E. Naudascher and D. Rockwell, pp. 1-81, Springer Verlag.

NRC Safety Guide 20, 1971, - Vibration measurements on Reactor Internals, Dec. 29, 1971, Wash-1226-20, Directorate of Regulatory Standards AEC.

S. J. Price, and M. P. Paidoussis, 1984, "An Improved Mathematical Model for the Stability of Cylinder Rows Subject to Cross-Flow," J. Sound Vib., Vol. 97, pp. 615-640.

R. H. Scanlan, and E. Simiu, 1978, Wind Effects on Structures, John Wiley & Sons.

H. Tanaka, and S. Takahara, 1981, "Fluid Elastic Vibration of Tube Array in Cross Flow," J. Sound Vibration, Vol. 77, pp. 19-37.

A. Yamaguchi, et al., 1997, "Failure Mechanism of a Thermocouple Well Caused by Flow-Induced Vibration," ASME, 4th Int. Sym. on Fluid-Structure Interactions, Aeroelasticity, Flow-Induced Vibration and Noise, Vol. 1. pp. 139-148.