# FLOW-INDUCED VIBRATION OF CYLINDRICAL STRUCTURES

**Sabine Upnere[1,2], Normunds Jekabsons[2]**
[1]Riga Technical University, Latvia; [2]Ventspils University College, Latvia
upnere@gmail.com, normundsj@venta.lv

**Abstract.** Two-dimensional numerical simulations of the cross-flow around circular cylinders in staggered arrangements have been performed. The part of the water-cooling system rod bundle is investigated. One of the cylinders is free to oscillate in the transverse and longitudinal direction, in response to the fluid loads. The moving cylinder is modelled as a simple mass-spring-damper system. Cylinders are rigid objects. The Navier-Stokes equations and continuity equation are used to describe a fluid. Dynamics of the structure are described by equations of motion. Turbulent incompressible flow with Reynolds number 10 000 is simulated using the Unsteady Reynolds Averaged Navier-Stokes type Low-Re turbulence model. Fluid-structure interaction is modelled by Arbitrary Lagrangian Eulerian framework, which combines the fluid flow formulated using an Eulerian description and a spatial frame with solid mechanics formulated using the Lagrangian description and a material frame. Commercial solver is used for analyzed the process simulation. The numerical results of benchmark cases are compared with literature to check the used methodology.

**Keywords:** fluid-structure interaction, flow-induced vibrations, circular cylinder.

## Introduction

The high velocity coolant flowing through the water-cooling system of different industrial and nuclear plants is a source that can induce component vibration and instability, which leads to the maintenance and operational problems, as well as security problems. In this paper the flow through circular cylinder rod bundle of the staggered arrangement is studied. Generally, there are four main flow-induced mechanisms causing structural vibration: fluid-elastic instability, periodic wake shedding, turbulence-induced excitation and acoustic resonance [1]. With the greatest potential for flow-induced vibration damages to the rod bundle has been fluid-elastic instability [1; 2].

Many preview studies have been devoted to enhancing understanding of processes in the rod bundle systems since the work of Connors (1970, identified and evaluated a basic fluid-elastic displacement mechanism), Belvins (1974, recast the observations in terms of coupled vibration of an infinite tube row) or Tanaka and Takahara (1980, carried out an experimental program to measure the unsteady fluid dynamic force coefficients for in-line tube arrays). However, numerical calculations of the multi-cylinder system is challenging due to complex interactions between the share layers and vortices, turbulence and other phenomena. Besides, fluid-structure interaction (FSI) problems have strong nonlinearity and multidisciplinary nature [3]. Modelling actuality of flow-induced vibrations is still on top and there are new publications, for example [4; 5].

In the present study a fluid has been described by the Navier-Stokes equations and continuity equation of mass in the Eulerian framework. Although cylinders are rigid bodies, due to software features moving cylinders are described as linear elastic objects governed by equations of motion in Lagrangian formulation. As a coupling algorithm between Eulerian and Lagrangian formulations the Arbitrary Lagrangian-Eulerian (ALE) method [6] has been used. In order to present large unsteady structures of vortices in the flow field and to consume computational cost the so-called Unsteady Reynolds Averaged Navier-Stokes (URANS) equations have been calculated. Finite Element analysis is done using commercial software Comsol Multiphysics.

This research contains a series of two-dimensional numerical studies that analyse turbulent single-phase flow in the part of a multi-cylinder system.

## Mathematical formulation

The fluid problem is governed by the Reynolds Averaged Navier-Stokes equations (1) and conservation of mass (2) in Eulerian form for incompressible flow. With subscripts $f$ and $s$ denoted variables are related with fluid and solid domain, respectively.

$$\rho \frac{\partial \bar{u}}{\partial t} + \rho \bar{u} \cdot \left(\nabla \bar{u}\right) + \nabla p - \mu_t \Delta \bar{u} + \rho \nabla \cdot \tau = f_f \ \text{ in } \Omega_f \ \text{ for } t > 0 \tag{1}$$

$$\nabla \cdot \bar{u} = 0 \text{ in } \Omega_f \text{ for } t > 0 \tag{2}$$

where $\bar{u}$ – Reynolds-averaged velocity, m·s$^{-1}$;
$p$–pressure of thefluid, Pa;
$\mu_t$ – fluid dynamicviscosity, N·s·m$^{-2}$;
$\Omega_f$ – bounded fluid domain;
$f_f$ – force, N;
$\tau$ – Reynold's stress tensor, N·m$^{-2}$.

Since fluid is incompressible, thermal effects are neglected. The density is constant in space and time due to the assumption of homogeneity and incompressibility of the fluid.

The problem must be supplemented with initial (3), Dirichlettype boundary ($\Gamma_{D,f}$) condition for velocity (4) and Neumann type of boundary ($\Gamma_{N,f}$) condition for normal stresses (5):

$$u = u_0 \text{ in } \Omega_f \text{ for } t = 0, \tag{3}$$

$$u = g_f \text{ on } \Gamma_{D,f} \text{ for } t > 0, \tag{4}$$

$$\sigma_f \cdot n_f = d_f \text{ on } \Gamma_{N,f} \text{ for } t > 0, \tag{5}$$

where $u_0$ – initial velocity, m·s$^{-1}$;
$n_f$ – normal vector;
$\sigma_f$ – stress tensor, N·m$^{-2}$;
$g_f$ – imposed fluid velocity on boundary, m·s$^{-1}$;
$d_f$ – imposed normal stresses on boundary, N·m$^{-2}$.

Dynamics of the structure is described in Lagrangian coordinates; equations are written in the initial configuration $\Omega s^0$. Moving structures are described with field equations(6) and Hook's law (7), which are valid for small displacements$\eta$. Boundary conditions have been defined by Dirichlet and Neumanntype conditions in solid domain with (8) and (9), respectively.

$$\rho_s \frac{\partial^2 \eta}{\partial t^2} \text{-} \nabla \cdot \sigma_s = f_s \text{ in } \Omega_s^{\ 0}, \tag{6}$$

$$\sigma_s = \lambda(\nabla \cdot \eta) I + 2\mu\varepsilon, \tag{7}$$

$$\eta = g_s \text{ on } \Gamma_{D,s}, \tag{8}$$

$$\sigma_s \cdot n_s = d_s \text{ on } \Gamma_{N,s}, \tag{9}$$

where $\lambda$, $\mu$ – Lamé constants;
$\varepsilon$ – strain tensor;
$I$ – second rank unity tensor;
$g_s$ – imposed solid displacement on boundary, m;
$\rho_s$ – density of solid,kg·m$^{-3}$.

On the interface ($I(t)$) boundary between fluid – ($F(t)$) and solid-domain ($S(t)$) kinematic (10), dynamic (11) and geometric (12) coupling conditions have been applied.

$$u_f = \frac{\partial \eta}{\partial t} \text{ on } I(t), \tag{10}$$

$$\sigma_f \cdot n_f = -\sigma_s \cdot n_s \text{ on } I(t), \tag{11}$$

$$\Omega(t) = F(t) \cup I(t) \cup S(t),\ F(t) \cap S(t) = 0. \tag{12}$$

Equation set of (1), (2), (6) and (10)-(12) formulate a coupled fluid-structure problem. The fluid equations are defined in a moving domain $\Omega_f^t$. One option to deal with the motion of the fluid-domain is to use the ALE framework, which is a well-established technique to model FSI.ALE introduces a fixed reference frame, which is mapped at every step to the desired physical domain and equations of motion are then recast in the reference frame.

**Numerical modelling technique**

Due to small displacements, the fixed geometry approach (the fluid domain is considered as fixed during the iteration) has been applied to couple Computational Fluid Dynamics (CFD) and Solid Mechanics interfaces. Segregated solver which sequentially solves for fluid flow, solid displacement, and moving mesh is used to reduce the computational costs. Backward Difference Formula has been chosen for the time stepping.

The fluid flow in this research is turbulent; corresponding Reynolds number $R_e$ = 10 000. For the approximation of the turbulent flow in the utilized software (Comsol Multiphysics) the RANS typea pproach has been used. Low Reynolds turbulence models have been investigated because of the importance of the flow near boundaries to model lift and drag forces and flow separation from surfaces with higher accuracy. Two different closure models have been tested to evaluate the most convenient one: low Reynolds number *k*-epsilon and SST (Shear-Stress Transport) *k*-omega [7]. The low Reynolds number *k*-epsilon is similar to the *k*-epsilon model alone without wall functions; the flow is solved near the walls, too. The SST *k*-omega model combines the best of two approaches: the *k*-epsilon in the free-stream and the *k*-omega models near the walls. The model does not use the wall functions. According to the comparison of convergence stability (see Fig. 1), the SST *k*-omega model has been chosen as the most suitable closure scheme for the studied cases.

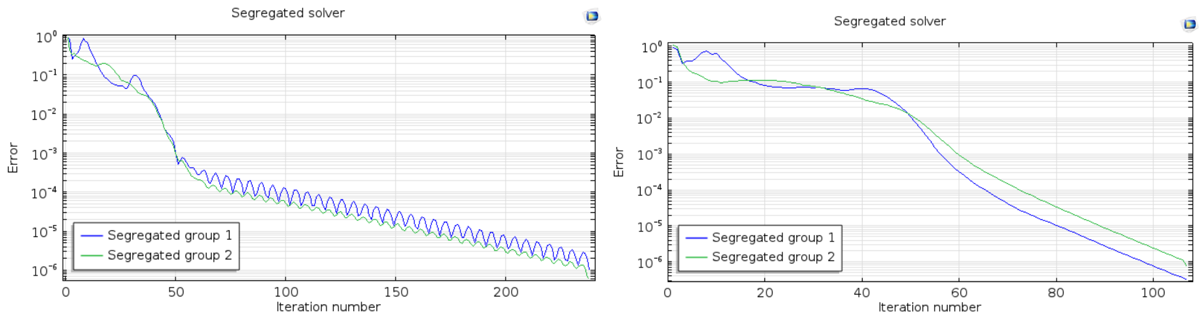


Fig. 1. **Comparison of convergence stability of two turbulence models:** left – low Reynolds number k-epsilon model; right – SST k-omega model

The oscillating flexibly-mounted rigid cylinder is modelled as a simple mass-damper-spring system. The equation of motion for the cylinder in the cross flow direction is:

$$m\ddot{\eta} + c\dot{\eta} + k\eta = f(t) \tag{13}$$

where $m$ – matrix of total oscillating mass of the system, kg;
$c$ – damping matrix, kg·s$^{-1}$;
$k$ – stiffness matrix, N·m$^{-1}$;
$f(t)$ –forces acting on the cylinder in the cross flow direction, N.

Firstly, there are calculated the lift and drag forces acting on the stationary cylinder in the rod bundle. After that, spring stiffness is defined using the previously calculated aerodynamics forces and determined maximum displacement. In order to compute high oscillation amplitude, the structural damping coefficients in streamline and transverse direction are set to zero.

The simulation has been performed in 2D space domain using the Finite Element method. The investigated geometry, domain dimensions and boundary conditions can be seen in Fig. 2, where $D$ is the diameter of the cylinder.

Constant velocity specified by an equation $u_\infty = u_0$ is defined as the inflow boundary condition of velocity. The outlet boundary condition is determined by the rule that the outflow of fluid must be perpendicular to the boundary. At the walls and the moving wall, no-slip condition is used: the tangential fluid velocity is equivalent with the velocity of rigid boundary and the normal component of velocity is zero.

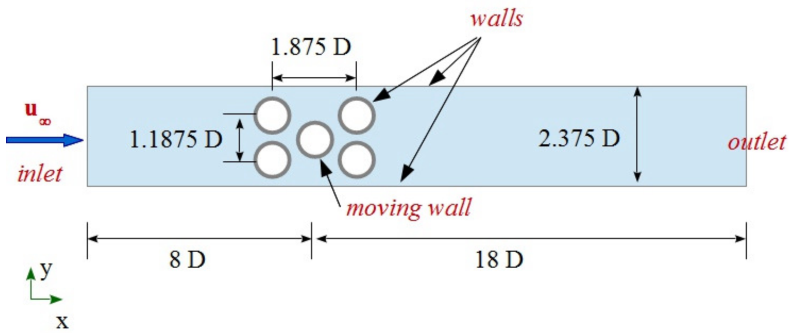


Fig. 2. **Computational domain, dimensions and boundary conditions:** *D* is a diameter of the cylinder and $u_\infty$ is the free-stream velocity

**Test cases**

Two test cases, one for CFD and one for FSI,were done to estimate the used approach, solvers, mesh size and other convergence criterions. The results of the test cases have been compared with the data from literature.

Classical CFD benchmark case with $R_e$ =10 000 has been chosen to evaluate turbulence models. Single stationary circular cylinder in cross-flow is simulated. Stable solution is achieved after approx. 0.5 seconds. Error of the calculated drag coefficient compared with the data in literature is 6.7 %.

To check the used FSI methodology, two cylinders in tandem have been analyzed. The first cylinder is stationary, while the second one has one degree of freedom to oscillate in the transverse direction. In this case $R_e$ = 1500. The sketch of the FSI computational domain and boundary conditions are shown in Fig. 3.

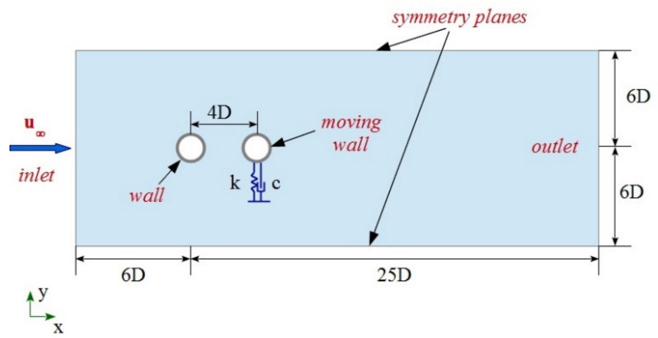


Fig. 3. **Computational domain and boundary conditions of FSI test case:** *D* is the diameter of the cylinder, *k* is the spring stiffness coefficient, *c* is the damping coefficient and $u_\infty$ is the free-stream velocity

The calculated results have been compared with the results from [8] and [9]. Comparison of the oscillation amplitude ratio ($y_{max}/D$) with the numerical and experimental data has been summarized in Table 1.

Table 1

**Comparison of the maximum amplitude of the oscillating cylinder**

| Case | Amplitude ratio ($y_{max}$/D) | Error ($e$) |
|---|---|---|
| Present study | 0.066 | 8.2 % |
| Derakhandeh, [8] | 0.065 | 6.6 % |
| Experimental value, [9] | 0.061 | |

From Table 1 it can be seen that the values of the numerical calculations are larger than the experimental value.

**Results and discussion**

Two cases have been investigatedto analyse the multi-cylinder system (see Fig. 4):

- moving cylinder has 1DOF, the movement is allowed in transverse direction (*y*-direction);
- moving cylinder has 2DOF (x and y direction).

The cylinder is modelled from steel and the used fluid is water. In order to get a faster convergence, a stationary case has been simulated first. The results of the stationary case have been applied as initial conditions in the dynamic case.

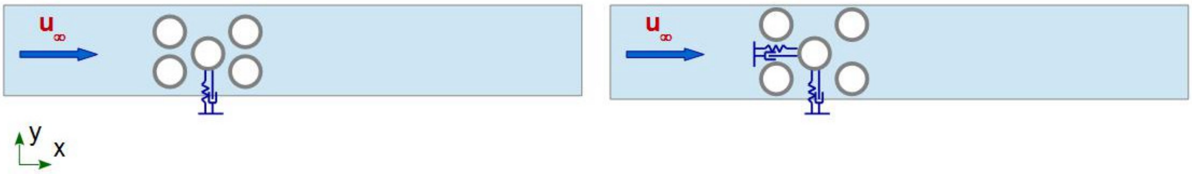


Fig. 4. **Computational domain and dimensions:** left – 1DOF model; right – 2DOF model

The cylinder does not oscillate in the case of 1DOF at the given initial and boundary conditions. After approximately 0.085 s, the moving cylinder reaches its maximum displacement at 0.015 D. Distribution of the flow field (velocity magnitude, $m \cdot s^{-1}$) around the rod bundle can be seen in Fig. 5.

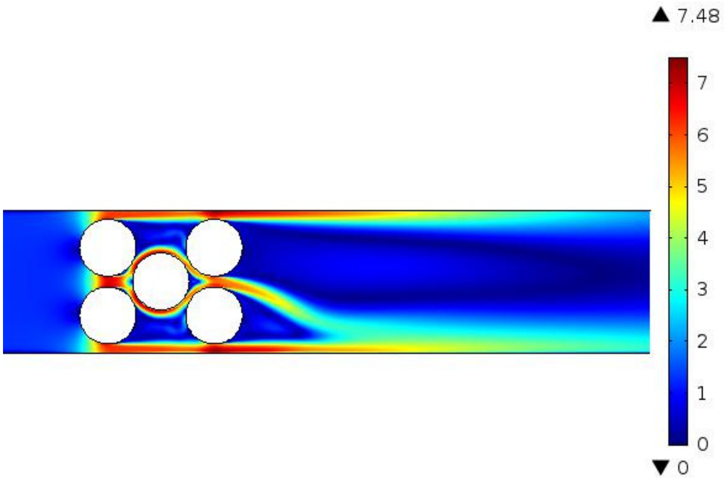


Fig. 5. **Distribution of the velocity magnitude ($m \cdot s^{-1}$) around the rod bundle of 1DOF case**

The flexibly-mounted cylinder oscillates, when it has 2DOF. A major impact on the total movement is y component (lift directions) of the displacement. Maximum displacement in *y* direction is 0.1 D, while in *x* direction it is 0.0125 D. The shedding frequency (Hz) of the oscillating cylinder has been displayedin Fig. 6.

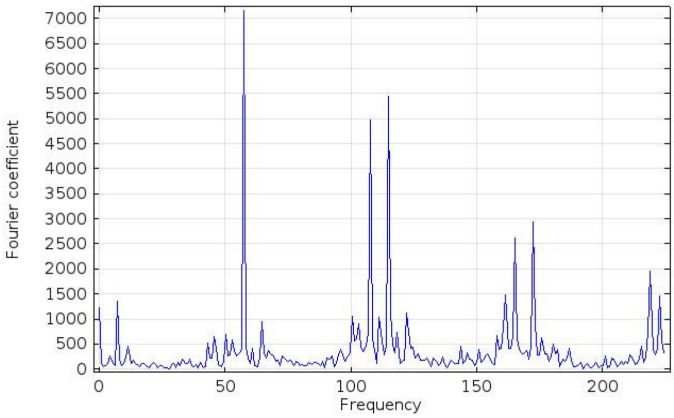


Fig. 6. **Lift force of the moving cylinder in the frequency domain using Fast Fourier Transform in 2DOF case**

At the given conditions the first expected frequency of the single cylinder is 31 Hz, moving the cylinder in the rod bundle the first frequency is higher – 57.5 Hz.

## Conclusions

1. Two-dimensional numerical simulations of flow-induced vibrations have been done to estimate the used FSI calculation methodology. Due to turbulent flow and structural element interaction it is a challenging task.
2. There is simulated a part of the cooling system with staggered arrangement rods. The middle circular cylinder of the rod bundle, which is mounted on the elastic system, can be oscillated in longitudinal and transverse directions. The cylinder motion has been modelled as a simple mass-damper-spring system.
3. Two Low-Re turbulence models have been analyzed. Based on the characteristics of convergence the SST k-omega turbulence model has been chosen as more suitable in the analysed case.
4. The numerical simulation results of CFD ($e$ = 6.7 %) and FSI ($e$ = 8.2 %) benchmark cases have a sufficient agreement with the experimental data from literature.
5. Further full rod bundle simulations will be required to compare the modelling results with the experimental measurements.

## References

1. Pettigrew M.J. et al. Vibration analysis and vibration damage assessment in nuclear and process equipment. Proceedings of “CORENDE: Regional congress on nondestructive and structural evaluation”, 1997, Buenos Aires, Argentina, pp. 127-138.
2. Lowdon A., Tonks N., Wilkinson T.S. Fluid-elastic instability of in-line tube arrays in cross-flow. Appl. Math. Modelling, vol 4, 1990, pp. 518-526.
3. Chakrabarti S.K. (Ed.) Numerical Models in Fluid Structure Interaction, Advances in Fluid Mechanics, vol. 42, WIT Press, 2005.
4. Yu K.R. et al. Flow-induced vibrations of in-line cylinder arrangements at low Reynolds numbers. J. of Fluids and Structures, vol. 60, 2016, pp. 37-61.
5. Nguyen V.-T., Nguyen H.H. Detached eddy simulations of flow induced vibrations of circular cylinders at high Reynolds numbers.J. of Fluids and Structures, vol. 63, 2016, pp. 103-119.
6. Arbitrary Lagrangian-Eulerian and Fluid-Structure Interaction. Numerical simulation. Edited by Souli M., Benson J. Eastbourne: John Wilew & Sons, 2010, 306 p.
7. Davidson L. An introduction to turbulence models. Goteborg: Chalmers University of Technology, 2015. 48 p.
8. Derakhshhandeh J.F. et. al. Numerical simulation of vortex-induced vibration of elastic cylinder. Proceedings of International conference “18th Australasian Fluid Mechanics Conference”, December 3-7, 2012, Launceston, Australia.
9. Assi G.R.S. Experimental Mechanisms for flow-induced vibration of interfering bluff bodies. London: Imperial College London, 2009. 228 p.