Numerical simulation of fluid-structure interaction by the example of a flow past a cylinder with a flexible thin structure

Irina Lantsova
Moscow State University of Civil Engineering, Yaroslavskoe shosse, 26, Moscow, 129337, Russia

E-mail: LantsovaIYu@mgsu.ru

Abstract. This article is devoted to the study of a numerical method of modelling fluid-structure interaction (FSI) problems, namely, optimization of computational resources. We consider a flexible thin-walled structure with clamped behind a fixed rigid non-rotating cylinder installed in a water channel. To solve the problem the "2-way FSI" mode of modelling in software ANSYS was used. The results are presented in tables and graphs in comparison with the data of the experiment and other authors.

1. Introduction
Problems of interaction between flexible structure and flow (Fluid-Structure Interaction, FSI) present a great and continuing interest in in the field of construction of unique buildings and structures. There are works [1-9] devoted to aero- and hydroelastic interaction, which study the applicability of certain methods (FEM, FVM, SPH, etc). They solve various test problems and benchmarks from different fields of science. This type of problem is very difficult, since turbulence modeling requires a fine computational grid, which, together with the non-stationary calculation, requires large computing power. In this paper, using the benchmark [11] as an example, we investigate the possibility of using the DES model and a coarser mesh in FSI problems in comparison with the authors of the benchmark itself, using LES and a very fine mesh. This benchmark is notable for the presence of a qualitatively described experiment, thus, it is possible to analyze in detail the results obtained and find out the necessary conditions (mesh size in the wall and distant regions, solver parameter, time step, etc.) for solving already practical problems of construction, such as the interaction of a bridge with wind flow. The first stage of this study, namely the solution of a two-dimensional laminar problem, is described in [10].

The purpose of this article is to verify the method of numerical modeling of the three-dimensional problem of aeroelasticity. The purpose of this article is to study the possibility of using a less expensive DES turbulence model (in comparison with LES) in FSI problems and a corresponding reduction in the size of the computational grid to speed up the solution of already real three-dimensional problems.
2. Methods

2.1. Description of the validation test case
As the verification task for the numerical technique of modeling fluid-structure interaction (FSI) civil engineering problems the benchmark [11] announced by German scientists (G. De Nayer, A. Kalmbach, M. Breuer - Helmut-Schmidt Universität Hamburg) was chosen. In [11] the results of experiment and numerical calculations are presented. The object of interest is a flexible thin structure clamped behind a fixed rigid non-rotating cylinder installed in a water channel (Fig. 1). The cylinder has a diameter \( D = 0.022 \) m. It is positioned in the middle of the water channel. The channel has a length of \( L = 0.338 \) m, a height \( H = 0.24 \) m and a width \( W = 0.18 \) m. The deformable structure has a length \( l = 0.06 \) m and a width \( w = 0.18 \) m. The thickness of the plate \( h = 0.0021 \) m. All parameters of the geometrical configuration of the benchmark are shown in Table 1.

The flow parameters are shown in the Table 2. Also, Table 2 shows the material parameters that correspond to the rubber material.

For comparison with the reference [11], dimensionless y-displacements \( U_y^*(t) \) was calculated:

\[
U_y^*(t) = \frac{U_y(t)}{D}
\]

where:

- \( U_y(t) \) is the vertical displacement of point \( A(t) \) (Fig.2);
- \( D \) is cylinder diameter.

The location of the point \( A(t) \) changes in time due to the plate dynamic reaction under flow excitation.

![Figure 1. Geometrical configuration of the Benchmark.](image)

| Table 1. Geometrical configuration |
|-----------------------------------|
| Cylinder diameter \( D \)         | 0.022 m |
| Cylinder center x-position \( L_c \) | 0.077 m |
| Cylinder center y-position \( H_c \) | 0.12 m |
| Channel length \( L \)            | 0.338 m |
| Channel height \( H \)            | 0.24 m |
| Channel width \( W \)             | 0.18 m |
| Deformable structure length \( l \) | 0.06 m |
| Deformable structure height \( h \) | 0.0021 m |
| Deformable structure width \( w \) | 0.18 m |

| Table 2. Physical and mechanical characteristics of materials |
|---------------------------------------------------------------|
Flow parameters

| Parameter                        | Value         |
|----------------------------------|---------------|
| Inflow velocity $u_{inflow}$     | 1.385 m/s     |
| Flow density $\rho_f$            | 1000 kg/m$^3$ |
| Flow dynamic viscosity $\mu_f$   | 0.001 Pa s    |

Structure parameters

| Parameter                        | Value         |
|----------------------------------|---------------|
| Density $\rho_{plate}$           | 1360 kg/m$^3$ |
| Young’s modulus $E$              | 16 MPa        |
| Poisson’s ratio $\nu$            | 0.48          |

2.2. Numerical simulation methodology

To solve the problem, multiphysics engineering simulation software ANSYS was used. To study the fluid-structure interaction (FSI), the “2-way FSI” mode of modelling was used. The 2-way FSI setup procedure is shown in Fig. 3.

The modeling procedure is divided into steps:
1. Geometry and mesh creation;
2. Setting up a Computational Structure Dynamics model, CSD-model (material model, loads and constraints, identification of the Fluid-Structure Interface);
3. Setting up a Computational Fluid Dynamics model, CFD-model (flow properties and flow boundary conditions, identification of the Fluid-Structure Interface and indication of mesh motion model);
4. Setting up System coupling (time duration and time steps, coupling sequence, number of coupling iterations per time step, interface exchange under-relaxation and convergence criteria)

2.2.1. Numerical CSD Setup. For the CSD model, a structured finite element model was created as follows: in the $oX$ direction, the flexible structure (plate) was divided into 10 elements, in the $oY$ direction - into 2 elements, in the $oZ$ direction - into 30 elements.
On the CSD side, the flexible plate is loaded on the top and bottom surfaces by fluid forces that are transferred from the fluid mesh to the structure mesh. These Neumann boundary conditions for the project reflect the constraint conditions. As for the Dirichlet boundary conditions, four edges need a corresponding support modeling: the fixed support is realized from the side of the entrance to the rigid cylinder, and all degrees of freedom are equal to zero. On the opposite exit side of the trailing edge, the rubber plate is free to move and all nodes have a full set of three degrees of freedom. The z-axis displacement of the structure nodes on the lateral boundaries is forced to be zero.

![Mesh for CSD model.](image)

**Figure 4.** Mesh for CSD model.

### 2.2.2. Numerical CSD Setup

The mesh for the CFD model with indication of the parameters is shown in Fig. 5.

![Mesh for CFD model.](image)

**Figure 5.** Mesh for CFD model.

On the CFD side, adhesion boundary conditions are applied to the rigid front cylinder and flexible structure. It is assumed that the upper and lower walls of the channel are sliding walls. At the inlet, a constant longitudinal velocity is specified as the inflow condition without any disturbances. The choice of the zero level of turbulence is based on the fact that such small disturbances imposed at the inlet usually do not reach the cylinder due to the roughness of the grid at the outer boundaries. Since a
turbulent flow regime was considered, the Detached Eddy Simulation (DES SST) turbulence model was used to close the Navier-Stokes equations.

2.2.3. Coupling conditions. The time-step size for CFD and CSD parts of $\Delta t=0.005$ s Furthermore, a constant underrelaxation factor of $\omega=0.1$ is considered for the displacements and the loads are transferred without underrelaxation. 5 FSI sub-iterations were assigned to achieve the convergence criterion.

3. Results
The obtained results of numerical simulations are shown below on Table 3 and Fig.6 compared to the reference [11]. Figure 7 shows the velocity fields and structure deformations at different instants of time

|                | $U_{y\text{max}}^*$ | $U_{y\text{min}}^*$ | $f_{FSI}$ |
|----------------|---------------------|---------------------|-----------|
| Numerical simulation | 0.423               | -0.447              | 7.47      |
| Numerical simulation [11] | 0.456               | -0.464              | 7.08      |
| Ref [11] | 0.418               | -0.42               | 7.10      |

Table 3. Comparison of the results with the reference [11]

![Table 3. Comparison of the results with the reference [11]](image)

Figure 6. Dimensionless $y$-displacements $U_y$ dependency on time calculated by numerical simulation (blue solid line). The red dotted lines are the maximum and minimum dimensionless $y$-displacements obtained in the experiment [11].
Figure 7. Velocity fields, m/s and structure deformations at various points in time $t$, s.
4. Conclusion
The results obtained are in very good agreement with the experimental data. The authors of the benchmark [11] used the LES turbulence model and finer mesh (13.5 million CVs), which is significantly more computationally expensive. The results obtained show that to solve such problems, it is sufficient to use a lighter DES model and a larger grid (1.5 million CVs). This allows us to approach even closer to solving real problems of aeroelasticity in a direct nonstationary three-dimensional formulation.

Acknowledgments
The reported study was funded by RFBR, project number 19-31-90128.

References
[1] Tu, J., Inthavong, K. & Wong, K. K. L. Computational Fluid Structure Interaction. in 95–154 (2015). doi:10.1007/978-94-017-9594-4_5.
[2] Turek, S. & Hron, J. Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow. Lect. Notes Comput. Sci. Eng. 53, 371–385 (2006).
[3] Yang, Q., Jones, V. & McCue, L. Free-surface flow interactions with deformable structures using an SPH-FEM model. Ocean Eng. 55, 136–147 (2012).
[4] Wang, L., Quant, R. & Kolios, A. Fluid structure interaction modelling of horizontal-axis wind turbine blades based on CFD and FEA. J. Wind Eng. Ind. Aerodyn. 158, 11–25 (2016).
[5] Lee, H., Song, M. C., Suh, J. C. & Chang, B. J. Hydro-elastic analysis of marine propellers based on a BEM-FEM coupled FSI algorithm. Int. J. Nav. Archit. Ocean Eng. 6, 562–577 (2014).
[6] Liu, H., Xu, H., Ellison, P. J. & Jin, Z. Application of computational fluid dynamics and fluid-structure interaction method to the lubrication study of a rotor-bearing system. Tribol. Lett. 38, 325–336 (2010).
[7] Calderer, A., Kang, S. & Sotiropoulos, F. Level set immersed boundary method for coupled simulation of air/water interaction with complex floating structures. J. Comput. Phys. 277, 201–227 (2014).
[8] Gilmanov, A., Le, T. B. & Sotiropoulos, F. A numerical approach for simulating fluid structure interaction of flexible thin shells undergoing arbitrarily large deformations in complex domains. J. Comput. Phys. 300, 814–843 (2015).
[9] Guivier-Curien, C., Deplano, V. & Bertrand, E. Validation of a numerical 3-D fluid-structure interaction model for a prosthetic valve based on experimental PIV measurements. Med. Eng. Phys. 31, 986–993 (2009).
[10] Afanasyeva I. N. and Lantsova, I. Y. Numerical simulation of an elastic structure behavior under transient fluid flow excitation. in AIP Conference Proceedings vol. 1800 (2017).
[11] De Nayer, G., Kalmbach, A., Breuer, M., Sicklinger, S. & Wüchner, R. Flow past a cylinder with a flexible splitter plate: A complementary experimental-numerical investigation and a new FSI test case (FSI-PfS-1a). Comput. Fluids 99, 18–43 (2014).