Numerical investigation of flow-induced forces in the rods bundle

Sabine Upnere
Riga Technical University, Viskalu Street 36, Riga, Latvia
Ventspils University of Applied Sciences, Inzenieru Street 101, Ventspils, Latvia
E-mail: upnere@gmail.com

Abstract. The numerical modelling of cross-flow through the rods bundle with triangular arrangement has been done to analyse flow-induced forces on the rod located in the middle of the bundle. Significant problems of rods in the bundle during the operational time of the system can be caused by the cross-flow. At the same time, it is known that the behaviour of the system is strongly related to many parameters such as bundle geometry, flow, rods support and others. Therefore, there is needed to investigate characteristics of each type of typical bundles. In this paper is analysed flow-induced hydrodynamic forces in closely-packed rods bundle using Computational Fluid Dynamics. Unsteady Reynolds Averaged Navier-Stokes equations were solved using Finite Volume discretization. The impact of the size of the computational domain and the number of rows in it was investigated to find the optimal case for the numerical modelling. Obtained results were compared with references from literature and experimental data. Reynolds number effect on the test cylinder depending on the domain size and the bundle asymmetry impact on flow-induced force were investigated as well.

1. Introduction
Cross-flow around tubes or rods in the bundles can be often found in different engineering systems and structures. The interaction between flow and tubes is of great interest due to flow-induced vibration in the bundle associated with cross-flow and it can lead to long-term damages as well as serious problems in a very short time. Geometrical or mechanical variation of the array layout, necessitates a new set of experiments for each typical bundle [1]. Usage of techniques of numerical experiments usually allow decrease cost and time for the impact evaluation of such parameters as array configuration, pitch-to-diameter ratio, tube supports, etc. Widely used approach is computational fluid dynamics (CFD). However CFD has several limitations, for example, high computational cost for 3-dimensional real problems, selection of a proper turbulent model etc. [2]. A more convenient way would be to use low-fidelity CFD approaches such as Reynolds–Averaged Navier–Stokes (RANS) or unsteady RANS (URANS) instead of more complex techniques as Large Eddy Simulations or Direct Numerical Simulations, but for that, it would be important to understand where (U)RANS methods may be successful and where it becomes too inaccurate. In this research, a feasibility study of turbulent flow around a single circular cylinder was applied and obtained results were compared with data from the literature [2], [4]-[8] to evaluate the chosen methodology.

Additional studies were done analysing the effect of the computational domain size on calculated force to maximum reduce computational costs. The decoupling analysis measured the
differences in the flow-induced forces on the test cylinder of the simplified and original cylinder arrays, suggesting that 4 cylinder columns and 2 cylinder rows was suitable for the study of the cylinder array behaviours at an analysed range of Reynolds numbers. This evaluation is in contrast with other published results, for example, Weaver and El-Kashlan [9] recommended that 6 tube rows be used to simulate typical array behavior. Tang et al. [10] suggested 8 x 11 tubes array for a triangular bundle of tubes with pitch-to-diameter ratio 1.633. Mostly, the difference in estimation should be related to the flow rate in the array, i.e. the value of the Reynolds number. It is also important that this study looks at a bundle of closely-spaced rods. To check it out, the impact of small perturbations of the test cylinder position in the bundle was investigated introducing asymmetry moving the cylinder away from the centre by 0.5% of the cylinder diameter. This study showed that the gap size between neighbour cylinders has significant effect on flow-induced forces on the cylinder.

2. Numerical implementation and feasibility study

CFD method is applied to numerically investigate flow-induced forces in a cylinders bundle with triangle arrangement. The modelling was conducted via OpenFOAM 2.4.x (OF). The pressure and velocity coupling was realised by PIMPLE algorithm which is combination of PISO (Pressure Implicit with Splitting of Operator) and SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithms. This algorithm is an iterative solver for modelling time–dependent problems. The second order schemes for pressure and momentum discretization and first-order scheme for turbulence equations discretization were applied.

To decrease the computational time for simulations of the transient flow through the cylinders bundle the built-in OF solver potentialFoam was used thus obtaining the initial conditions for time–dependent simulations.

2.1. Governing equations and turbulence modelling

The flow is governed by incompressible Navier-Stokes equations. Based on the numerical complexity, RANS approach is applied to predict hydrodynamic forces on the test cylinder (TC) placed in the bundle in the two-dimensional space. As the dynamics of the system also is of interest the time–dependent simulations have been done solving URANS equations (1) and (2):

$$\frac{\partial \bar{U}_j}{\partial x_j} = 0 \text{ for } t > 0,$$

(1)

$$\left[\frac{\partial \bar{U}_i}{\partial t} + \bar{U}_j \frac{\partial \bar{U}_i}{\partial x_j}\right] = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 \bar{U}_i}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j}\{u_i u_j\} \text{ for } t > 0,$$

(2)

where \(\bar{\text{bar}}\) denotes averaged values, \(U\) is the velocity, \(P\) is the pressure, \(\rho\) is the fluid density, and \(t\) is the time. The density of the liquid is constant in the space and time due to the assumption that the liquid is non-compressible and homogeneous. Since the liquid is non-compressible, thermal effects are ignored as well.

The low-Reynolds turbulence model is applied to capture small scale effects in the bundle. The system closure is achieved using SST (Shear Stress Transport) \(k\)-\(\omega\) turbulence model introduced by Menter (1997).

2.2. Flow over single cylinder

To validate the computational methodology and chosen parameters, a benchmark study of water cross-flow passing a circular cylinder at \(Re = 11\ 000\) was undertaken. In the literature, both experimental and numerical data on integral parameters such as the lift coefficient \(C_L\), drag coefficient \(C_D\) and Strouhal number \(St\) at \(Re = 10\ 000\) and \(Re = 8000\) are available. Thus,
this model can be used to determine basic computing parameters (turbulence model, turbulence parameters, cell size of the mesh, boundary conditions) and to validate the model.

A rectangle computational domain of $52d \times 10d$ was created, where $d$ denotes the TC diameter. Based on a comparison of the results with other computational domains, it can be concluded that the selected size is appropriate so that it does not affect obtained results.

The domain mesh cells are designed so that close to the cylinder wall the cell size is small, and it gradually increases upstream and downstream from the cylinder. Considering that the low–Re turbulence model was used, the dimensionless wall distance $y^+$, which depends on the distance between the first node point and the cylinder wall in the radial direction, should be equal to or less than one. M1 mesh with an average $y^+$ value of 0.565 and M2 mesh with a finer breakdown with the average $y^+ = 0.455$ were created to analyse the effect of the mesh cell size. In Table 1 is shown a comparison of the drag coefficient $C_D$ and the Strouhal number $S_t$ using two meshes of the computational domain, as well as the corresponding computing time required for one computational iteration.

| Number of cells | $y^+$ | $C_D$ | $S_t$ | One iteration (seconds) |
|-----------------|-------|-------|-------|-------------------------|
| 1342077, M1     | 0.565 | 1.19  | 0.21  | 4.77                    |
| 2286170, M2     | 0.455 | 1.168 | 0.21  | 68.63                   |

From Table 1 it can be concluded that the mesh M1 is optimal for computing if both the integrated parameters and computing time are taken into account. Thus, M1 was used for future calculations. The numerical simulations are done using AMD Epyc 7251 8-core 2.1 GHz processor with 128 GB RAM.

The domain inlet is on the left side of the computing domain $14.5d$ from the origin of the coordinate system (i.e., the centre of the cylinder). In the right side has a domain outlet; $37.5d$ from the centre of the cylinder. A steady flow rate of 1.418 m/s was defined at the domain inlet, it corresponds to $Re = 11000$. A constant pressure of 0 Pa was applied to the domain output. Symmetry conditions were applied to the top and bottom of the domain located at $5d$ from the cylinder center. The no-slip condition was applied to the cylinder wall, which means that the flow rate increases from zero on the wall to the freestream velocity in the far region.

Using built-in OF solver *pimpleFoam*, a series of computational experiments were performed and the results obtained were compared with literature data.

It is known from earlier studies, for example, reported in Spalart [3] or Nguyen et al. [6] that URANS over-predicts the lift coefficient $C_L$. It is because in URANS case, the three-dimensionality is partially suppressed to a 2D geometrical set-up [6], but the length of the cylinder $L$ has a significant effect on the numerical prediction of lift force fluctuations. The cylinder with lower aspect ratio $L/d$ has a higher lift. In this study predicted lift coefficient (based on r.m.s. lift) is about 1.02 comparing to 0.2621–0.3629 obtained using DES (detached eddy simulations) reported by [6] or 0.25–0.46 measured experimentally by Norberg [8]. By contrast, the drag force and the Strouhal number can be predicted with sufficient accuracy. In Table 2 is summarized data from the literature and obtained results of this study using URANS approach with the SST k-omega turbulence model.

From Table 2 follows that the developed model can be used to predict drag coefficient or Strouhal number.
Table 2. Comparison of numerical and experimental results of the drag coefficient $C_D$ and Strouhal number $S_t$

|                | $C_D$     | $S_t$         |
|----------------|-----------|---------------|
| Khan et al. [2], RANS, 2D, Re = 10000 | 1.150     | 0.201         |
| Khan et al. [2], RANS, 3D, Re = 10000 | 1.210     | 0.203         |
| Dong et al. [4], DNS, Re = 10000     | 1.110 – 1.208 | 0.195 – 0.209 |
| Wornom et al. [5], LES, Re = 10000   | 1.22       | 0.200         |
| Nguyen et al. [6], DES, Re = 10000   | 1.133      | 0.200         |
| Gopalkrishnan [7], Exp, Re = 10000   | 1.186      | 0.193         |
| Norberg [8], Exp, Re = 8100         | –          | 0.203         |
| URANS, 2D (present study), Re = 11000 | **1.19**  | **0.21**      |

2.3. Computational domain of the bundle and its decoupling

The effect of the domain decoupling has been analysed to optimize computations decreasing the time for one simulation but keeping the acceptable accuracy of the result. Fig. 1 shows the domain configurations. The total length of the domain in the streamwise direction (x-axis) varied from five to three rows. The domain height in the spanwise direction (y-axis) was from five cylinders to two cylinders. Symmetry boundary conditions are applied to the top and bottom boundary of the domain.

![Figure 1](image-url)

**Figure 1.** Arrangements of the computational domain. The origin of the coordinate system is denoted by the red cross. It is the centre of the test cylinder as well.

The test cylinder (TC) is placed in the origin of the coordinate system (denoted by the red cross in Fig. 1). Six arrangement configurations are denominated by letters from (A) to (F).
The flow is from the left side to the right. Uniform inflow velocity is used for the inlet boundary and constant pressure is applied at outlet boundary.

The computational mesh is built from triangular cells in the space between cylinders. Structured quadrilateral cells are used in the boundary layer around cylinders, see Fig. 2. Each cylinder is discretized uniformly with 1800 nodes. In the radial direction, the size of the first layer is set that the dimensionless wall distance \( y^+ \) is less than 1.

![Figure 2. Structured mesh around circular cylinders and unstructured between them.](image)

### 3. Results and discussion

The impact of the size of the domain on the hydrodynamic forces was first evaluated. The forces on the TC were used for analysis. The position of the TC in the bundle can see in Fig. 1. In the first step, the flow velocity corresponds to \( Re = \frac{U_g d}{\nu} = 11.3 \cdot 10^3 \), where \( U_g \) is the velocity in the middle of the inlet gap.

The comparison of the forces depending on the domain size is done calculating the difference between the forces in the larger domain (A) and forces in the reduced domains (from (B) to (F)). Results of force difference in the percentage are shown in Fig. 3.

![Figure 3. The difference of the force in flow direction \( F_x \) between larger domain (A) and decoupled domains (B)-(F), see Fig. 1 for the notation of domains.](image)
Comparing forces $F_x$ on the TC in a domain with five cylinders per row and in a domain with one whole cylinder and two half-cylinders, the difference is less than 0.5%. The effect of the domain reduction in the flow direction can be evaluated comparing domains (C) versus (D) and (E) versus (F). There is no significant impact if the domain is reduced removing the row downstream the TC, see (C) and (D). The total number of rows is decreased from five to four. If it is removed the row before the TC (in the upstream direction, see (E) and (F)), then the difference between forces on the TC changes approximately by 4%. From Fig. 3 can conclude that to decrease time and costs for calculations can use reduced domains (E) or (F) at the given flow velocity. That allows decrease the size of the computational mesh from $2.2 \cdot 10^6$ for the domain (A) to $0.73 \cdot 10^6$ and $0.58 \cdot 10^6$ for (E) and (F) respectively.

Based on these results, in the next step was investigated velocity impact on forces depending on two domains (E) and (F). Numerical simulations were done at five different inflow velocities with the following maximum Reynolds numbers: $11.3 \cdot 10^3$, $24 \cdot 10^3$, $32 \cdot 10^3$, $41.6 \cdot 10^3$, and $53.3 \cdot 10^3$. Obtained results of force $F_x$ on the TC depending on the domain and velocity are illustrated in Fig. 4.

![Figure 4](image-url)

**Figure 4.** Comparison of force on the test cylinder in geometries (E), (F) and experimental measurements depending on the maximum inlet Reynolds number.

Fig. 4 shows that the domain (E) gives more stable results comparing to the geometry (F). As the input flow rate increases, the impact of the domain size on the calculated force on the TC also increases. For example, the difference at $Re = 11.3 \cdot 10^3$ is 4.4% but at $Re = 41.6 \cdot 10^3$ it is 11.71%. If there is compared calculated results with experimentally measured values then difference is $\approx 2.4\%$ and $2.9 - 14.4\%$ for the domain (E) and (F) respectively. The experiment is described in [11]. It can be concluded that the inflow velocity has a significant impact on the calculated forces depending on the domain size ((E) or (F)) if Reynolds number is larger than $32 \cdot 10^3$ in the triangular bundle with closely-packed rods ($P/d = 1.1$).

The effect of the TC position on the flow-induced forces in the unit cell (see Fig. 5) was analysed moving the TC by 0.5% of $d$ in flow direction and rotating it around the origin of the coordinate system. Due to the symmetry of the bundle, the rotational angle is from 0 to 180 degrees with increasing step of 45 degrees.

The schematic representation of the TC modified positions comparing to initial one is
presented in Fig. 6. The initial position of the TC is represented as a blue dotted circle in all cases. The position P0 corresponds to the case when the TC is in the centre of the unit cell. In the P1 case (orange circle), the TC is moved in a flow direction by 0.5% of d. The P2 case can be obtained rotating the P1 by 45 degrees around the centre of the unit cell. In a similar way can create other cases increasing the rotational angle by 45 degrees in each step, i.e. P3 to P5. It is assumed that the flow creates the unit force in the P0 case. Therefore flow-induced forces are normalized by force at the P0 position for all analysed cases.

As it was expected due to the symmetry of the bundle, the flow induced-force in the transverse direction $F_y$ is zero in the P0 position. The total normalized force $\hat{F}$ only contains the force component in the flow direction $F_x$. Similarly, in positions P1 and P5, the total flow-induced force is equal to $F_x$. The comparison of the P1 and P5 positions allows evaluating the gap size effect on the force. From Fig. 6 can conclude that the reduction of the gap by 0.5% of d leads to decreasing of the force from unit to 0.75. In contrast, the increase of the gap creates the force increase from unit to 2.43.
The $F_y$ component appears when the unit cell becomes non-symmetric moving the TC in the transverse direction, see positions P2, P3 and P4. A small downward component appears in position P2 but the dominant component is in the flow direction however it is smaller comparing to P2 ($\hat{F}=1.85$). A small upward component can be detect in position P3 and a magnitude of the total normalized force ($\hat{F}=1.1$) is a little larger than in position P0. The pronounced force component in the transverse direction $F_y$ is observed in the case of P4. The magnitude of the total normalized force is similar to position P5 ($\hat{F}=0.76$). Thus, P3 and P4 are positions when the TC tries to move away from the central flow axis comparing to P2 when the flow tries to return the TC to the x-axis.

4. Conclusions

The flow-induced forces on the cylinder in the centre of the triangular cylinder bundle with a maximum of 23 whole cylinders have been studied in a water cross-flow. A domain decoupling analysis was proposed by reducing the bundle in both the row and column directions to decrease the computational time. A series of numerical experiments of closely-packed cylinder arrays with a pitch ratio of 1.1 have been undertaken to determine the minimum number of rows and columns to study hydrodynamic forces in the rods bank. From calculation results follows that at low Reynolds number ($1.1 \cdot 10^3$ at the inlet gap) the domain size can be reduced from 5 x 5 cylinders to 4 x 2 cylinders and flow-induced force changes less than 0.5 %. The study of different inlet velocities showed that in cases if the maximum Reynolds number is less than $32 \cdot 10^3$ a difference between the force in computational domains with 2 x 2 cylinders ($7.3 \cdot 10^5$ cells) and 4 x 2 cylinders ($5.8 \cdot 10^5$ cells) is less than 5.6 %. The difference rapidly increase for higher Reynolds numbers. Furthermore, the fluid induced force of a cylinder were investigated to find how the position of the cylinder in the unit cell impacts the magnitude and direction of the force. The increase of the gap before the test cylinder leads to a significant increase in the force and vice versa - a reduction of the gap results in a smaller force on the cylinder. Introducing the asymmetry in a unit cell allows determine positions when the test cylinder tries to move away from the central flow axis and when it tries to return back to the central axis as a result of flow.

Future work will focus on creating implementations of URANS models for flow-induced vibration analysis describing the oscillating cylinder as a mass-spring system as well as on combining obtained models with metamodeling approach to develop a fast simulation tool.

References

[1] Andjelić, M., Austermann, R., Popp, K. 1992 *J Pressure Vessel Techn* 114 336
[2] Khan N B et al. 2017 *PLOS ONE* 12 101
[3] Spalart P R 2009 *Annu Rev Fluid Mech* 41 181
[4] Dong S and Karniadakis G E 2005 *J Fluid and Struct* 20 4 519
[5] Wornom S et al. 2011 *Comp and Fluids* 47 1 44
[6] Nguyen V-T and Nguyen H H 2016 *J Fluid and Struct* 63 103
[7] Gopalkrishnan R 1993 *Vortex-induced forces on oscillating bluff cylinders* Ph. D. Thesis (MIT Cambridge)
[8] Norberg C 2003 *J Fluid and Struct* 17 57
[9] Weaver D S, El-Kashlan M 1981 *J Sound and Vibration* 75 265
[10] Tang D et al. 2019 *Annals of Nuclear Energy* 124 198
[11] Upnere S et al. 2019: submitted to Journal of Vibroengineering