# NUMERICAL PREDICTION OF FLOW INDUCED VIBRATIONS IN NUCLEAR REACTOR APPLICATIONS

**E. ter Hofstede[1,2], A. Shams[1], A.H. van Zuijlen[2]**
[1]: Nuclear Research and Consultancy Group (NRG), Petten, the Netherlands
[2]: Delft University of Technology (TU Delft), Delft, the Netherlands
Shams@nrg.eu; A.H.vanZuijlen@tudelft.nl

## ABSTRACT

Flow induced vibration (FIV) plays an important role in nuclear industry. In nuclear power plants (NPP), FIV may cause fatigue problems, stress corrosion cracking, possible failure modes and fretting wear. In return, this can lead to nuclear safety issues and substantial stand-still costs due to unplanned outage. The demand for an increase in power density of future designs for nuclear reactors often results in the increase of coolant flow velocities or a change of cooling liquid. These changes may alter the flow behavior, which can lead to fluid elastic instability. It is therefore important to asses this phenomenon early in the design process.

Most of the experimental studies that have been performed are often simplified or only cover a single operation condition. Numerical methods can play an important role in analyzing complex industrial applications. To gain confidence in the available numerical methods, their validation is an important step that needs to be taken. In the present paper, such a validation study is performed on the used computational methods in Fluid Structure Interaction (FSI) for nuclear plant applications. A partitioned approach is used, in which the exchanges between the fluid and structure solver take place through the use of the Interface Quasi Newton with Inverse Jacobian from a Least Squares (IQN-ILS) coupling scheme.

Two different reference databases are used for this validation procedure. The first one is the numerical benchmark case of Turek. In this reference case, the deformation of an elastic flap, attached to a solid cylinder is studied. This case is well suited to validate the used coupling methods for applications with large structural deformations, and strong interaction. The second one is an experiment, performed by Vattenfall, where the damping of the flow along an excited (vibrating) slender tube is studied. The results of the simulation of the Vattenfall experiment are preliminary and do not include the coupled simulations.

**KEYWORDS**
Fluid Structure Interaction, Nuclear Reactor, Computational Fluid Dynamics, Strong Coupling.

## 1. INTRODUCTION

Flow Induced Vibration plays an important role in nuclear industry. In nuclear power plants, FIV may cause fatigue problems, stress corrosion cracking, possible failure modes and fretting wear [1]. This in return can lead to nuclear safety issues and substantial standstill costs due to unplanned outage. Reports of flow-excited failures of heat exchanger tubes began appearing in the 1950s [2]. The current demand for an increase in power density of nuclear plants often results in the increase of coolant flow, a change of cooling liquid or a change of component material or dimensions. These changes may alter the flow and structural behavior, and cause flow-induced vibration problems to become more prominent [2]. It is

therefore important to asses this phenomenon early in the design process. Because of the need of accurate prediction, FIV is an important area of research.

Many analytical models for slender bodies in axial flow do not use the full Navier-Stokes equations but split the fluid forces into an inviscid and viscous part. The contributions of the inviscid forces were first derived by Lighthill in 1960 [3], while the viscous forces were based on empirical relations, obtained from experiments on specific cases. The downside is therefore the lack in accuracy when applied to other or less simplified cases. Because of the increase of computational power it becomes more attractive to calculate the flow induced vibration by using a numerical simulation.

The mutual interaction between the fluid flow and the solid is often referred to as Fluid Structure Interaction (FSI). Solving an FSI problem numerically can be done in multiple ways. One can use a monolithic approach, this means that the equations of the fluid and the solid are written as one set of equations and solved all together. In this way their mutual influence can be taken into account during the solution progress. Using a monolithic approach therefore requires the implementation of a completely new code.

Another way to solve the FSI problem is the partitioned approach. In a partitioned FSI simulation the flow equations and the structural equations are solved with different numerical methods. This gives more freedom in selecting suitable methods for fluid and structural solvers than the monolithic approaches. The partitioned approach requires a coupling algorithm to model the interaction between the fluid and solid at the interface. The partitioned approach can be further divided according to the coupling method used, which makes a difference between loose-coupling and strong-coupling algorithms. If the interaction between the fluid and the structure is weak, a loosely coupled method, i.e. one coupling iteration per time step, can be used. When the interaction is strongly coupled, the solvers are called multiple times during a time step until both the kinematic and the dynamic equilibrium conditions are satisfied.

Using FSI to predict the occurrence of FIV for the development of $4^{th}$ generation reactors is challenging. Some of these reactors are using a cooling medium with a high density, like lead-bismuth eutectic. In this case the mass ratio of the solid (for example the fuel bundles) and the fluid is low ($\rho_s / \rho_f < 10$) and the added mass, i.e. the mass of fluid which is accelerated by the structure, is large. It is shown that in this case loosely coupled solvers suffer from poor convergence or even instability, therefore a strongly coupled method is necessary [4] [5] [6].

This paper therefore validates the use of a strongly coupled partitioned FSI method on a system with a low mass ratio and a large added mass. Two different reference databases are used for the validation procedure. The first is a numerical benchmark case of Turek. In this reference case, the deformation of an elastic flap, attached to a solid cylinder is studied. This case is well suited to validate the used coupling methods for applications with large structural deformations. The second one is an experiment, performed by Vattenfall, where the damping of the flow along an excited (vibrating) slender tube is studied. This is used to validate a case closer to the realistic application. This paper is organized as follows; section two gives a brief overview on the numerical methods, whereas section three contains the results of the two different cases. This is followed by the conclusions and the outlook.

## 2. NUMERICAL METHODS

When using a partitioned approach, the fluid and solid equations are solved with different numerical methods. In this case the fluid domain is solved using a computational fluid dynamic (CFD) method and the solid using a Computational Solid Mechanics (CSM) method. To be able to model the interaction of these two models, the two methods are coupled at the fluid-solid interface. A stable and efficient numerical technique is essential for the study of fluid structure interaction (FSI). When the interaction is strongly coupled, the solvers are called multiple times during a time step until both the kinematic and the dynamic equilibrium conditions are satisfied.

### 2.1. Governing Equations Fluid

The governing equations for a flow, i.e. the Navier-Stokes equations, are the conservation of mass and momentum. For an incompressible fluid these relations are given by:

$$\nabla \cdot \mathbf{v} = 0, \tag{1}$$

$$\rho\left(\frac{\partial \mathbf{v}}{\partial t} + \nabla \cdot (\mathbf{vv})\right) = \nabla \cdot \boldsymbol{\sigma} + \mathbf{f}, \tag{2}$$

with $\mathbf{v}$ being the fluid velocity, $\rho$ the fluid density, t the time, $\boldsymbol{\sigma}$ the fluid stress tensor and $\mathbf{f}$ are the body forces, e.g. due to gravity. For Newtonian fluids, the stress tensor can be written as:

$$\boldsymbol{\sigma} = -\mathbf{pI} + \mu\left[\nabla \mathbf{v} + (\nabla \mathbf{v})^T\right], \tag{3}$$

with $\mathbf{p}$ the pressure and $\mu$ the fluid dynamic viscosity.

### 2.2. Governing Equations Solid

The deformation of an elastic incompressible structure is governed by the conservation of momentum:

$$\rho \frac{\partial^2 \mathbf{d}}{\partial t^2} = \nabla \cdot \boldsymbol{\sigma} + \mathbf{f}, \tag{4}$$

with $\mathbf{d}$ being the structural displacement, $\rho$ the solid density, $\boldsymbol{\sigma}$ the solid stress tensor and $\mathbf{f}$ the body forces.

### 2.3. Coupling of the Fluid and Solid Equations

At the interface between the solid and the fluid, the kinematic condition requires the velocity of the fluid to be equal to the time derivative of the displacement of the solid interface:

$$\frac{\partial \mathbf{d}_s}{\partial t} = \mathbf{v}_f. \tag{5}$$

The dynamic condition requires the stress on the interface due to the fluid and solid surface normal to be equal, also called as equality of traction:

$$\boldsymbol{\sigma}_s \cdot \mathbf{n}_s = -\boldsymbol{\sigma}_f \cdot \mathbf{n}_f . \tag{6}$$

The code that solves the structure equations is called the structural solver *S* and the code that solves the fluid equations is called the flow solver *F*. When using a Dirchlet-Neumann decomposition, the flow equations can be solved for a given velocity (or displacement) of the fluid-structure interface (Dirchlet boundary condition), and the solid equations are solved for a given traction distribution on the interface (Neumann boundary condition). Taking $\mathbf{x} = \mathbf{d}_s$ and $\mathbf{y} = \boldsymbol{\sigma}_f \cdot \mathbf{n}_f$, the response of the structure solver can be therefore written as:

$$\mathbf{x} = S(\mathbf{y}). \tag{7}$$

The response of the fluid solver can now be written as:

$$\mathbf{y} = F(\mathbf{x}). \tag{8}$$

For every time step, the fixed point equation must be satisfied

$$\mathbf{x} = S \circ F(\mathbf{x}), \tag{9}$$

where $S \circ F$ is the result of function *F* is given as argument to the function *S*. Since the flow equations and the structural equations are solved separately, a residual at the interface can be found:

$$R(\mathbf{x}) = S \circ F(\mathbf{x}) - \mathbf{x}, \tag{10}$$

To reach convergence, this residual should be minimized, which can be done using an optimization algorithm. Well known approaches in FSI are the Gauss-Seidel method, fixed under-relaxation, Aitken under-relaxation, and the IQN-ILS method [7].

The Interface Quasi Newton with Inverse Jacobian from a Least Squares (IQN-ILS) is an efficient algorithm for strongly coupled problems [8] [9]. The IQN-ILS method is an Interface Quasi-Newton with an approximation of the Inverse of the Jacobian for a Least-Squares approximation. The IQN-ILS is a coupling technique that can be also used when both the flow solver and the structural solver are treated as a black box. In the current study the IQN-ILS method [7] [10] is applied.

### 2.4. Numerical Code

This study is performed using the software package OpenFOAM Extended [11]. OpenFOAM Extended is a fork of OpenFOAM [12] and is a free, open source CFD software package with also some capabilities to solve CSM. Both the fluid and structure equations are discretized using the Finite Volume Method.

## 3. RESULTS AND DISCUSSION

To validate the used numerical methods, two different reference cases are studied. The first case is a numerical benchmark case, here after referred to as the Turek benchmark case, and the second case is an experiment, hereafter referred to as the Vattenfall experiment. The Turek benchmark case is used to get confidence in the use of the IQN-ILS method for strongly coupled methods. The Vattenfall experiment is used to get a step closer to the realistic application for the nuclear field, since it is similar to most in-reactor applications that relate to FSI, which have a high aspect ratio in confined flow conditions.

### 3.1. Turek Benchmark Case

The Turek benchmark case [13] is a well-known FSI benchmark case. It is mostly used for the validation purpose and is therefore considered for the present study. This Turek case consists of a two dimensional incompressible laminar flow around a fixed cylinder with an attached elastic flap. In this paper it is considered as a two-dimensional configuration with a parabolic inlet velocity with a no slip condition at the wall. The considered computational domain is shown in Figure 1.

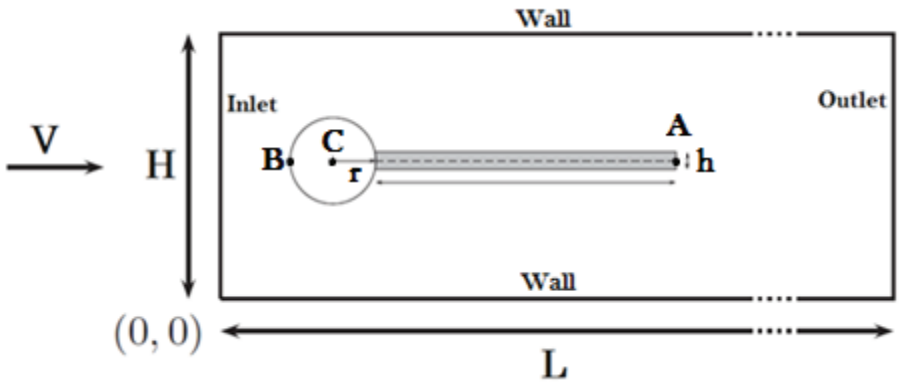


**Figure 1: Geometry of the cylinder with attached flap of the Turek benchmark case. With r=0.05m, l=0.35m, h=0.02m, H=0.41m, L=2.5m and A is the reference point for the measured displacement. The solid domain is shown in gray and the fluid domain in white. Amended from [13]**

Several test cases are described in [13] , however, for the present validation purpose only the case FSI3 is used, since here the mass ratio is small ( $\rho_s / \rho_f = 1$ ) and the velocity is relatively high (V=2m/s), such that the flap starts to oscillate. This case is therefore used to test the capability of the algorithm to solve a strongly coupled system.

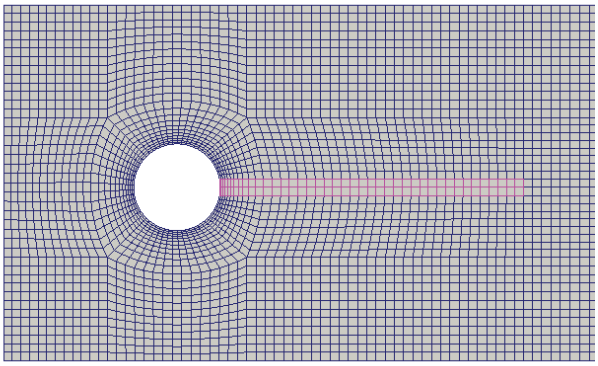

**Figure 2: Small section of the coarse mesh, 5981 cells in the fluid domain (blue) and 82 in the structure domain (pink), used for the Turek benchmark case.**

Two different mesh sizes are considered, a coarse grid and a finer grid. The coarse grid, as partly shown in Figure 2 has a mesh size of 5981 cells in the fluid domain and 82 cells in the structure domain. The finer grid has a mesh size of 23924 cells in the solid domain and 328 cells in the structure domain. With all domains having a hexahedral mesh. The time step used in both the solid and the fluid part is 0.001 seconds. A second order backward scheme is used as the temporal discretization scheme.

The resulting displacement of point A (see Figure 1), and the drag and lift forces after 19.5 up to 20 seconds are shown in Figure 3 and Figure 4, respectively. As can be seen here, the frequency of the oscillation is different from the oscillation frequency of the benchmark case. This difference is small but can be for example observed in the x-displacement plot of Figure 3. The frequency in the simulation here is approximately 11.1 Hz and the benchmark case has a frequency of approximately 10.9 Hz. This difference in frequency causes a phase shift after 19.5 seconds. However, in order to make the comparison of the difference between simulation and the Turek benchmark data easier this phase shift is removed in Figure 3 and Figure 4.

Figure 3 and Table 1 show that the displacement of point A is under estimated for both the fine and the coarse grid. The obtained results for the lift and the drag forces are shown in Figure 4 and Table 1. From this it can be found that the lift and the drag forces are slightly over estimated. When refining the mesh, the amplitude of the lift and drag forces are decreasing, as a result the amplitude of the displacement is slightly decreased.

As was also found in section 3.2, the decreased amplitude is caused by artificial damping introduced by the temporal discretization method. As with the phase shift, the artificial damping is also summing up after 19.5 seconds, introducing an underestimation in amplitude.

**Table 1: Amplitude and mean of the x-displacement, y-displacement, drag and lift for the simulation and the Turek benchmark case [12]**

| | **x-displacement [m][$10^{-3}$]** | | **y-displacement [m][$10^{-3}$]** | | **Drag [N]** | | **Lift [N]** | |
|---|---|---|---|---|---|---|---|---|
| | *Mean* | *Amplitude* | *Mean* | *Amplitude* | *Mean* | *Amp.* | *Mean* | *Amp.* |
| **Coarse** | -1.96 | 1.84 | 3.97 | 26.01 | 444.62 | 20.40 | -11.74 | 213.37 |
| **Fine** | -1.62 | 1.72 | 1.93 | 26.7 | 457.26 | 26.32 | -7.91 | 200.61 |
| **Turek** | -2.69 | 2.53 | 1.48 | 34.38 | 457.3 | 22.66 | 2.22 | 179.78 |

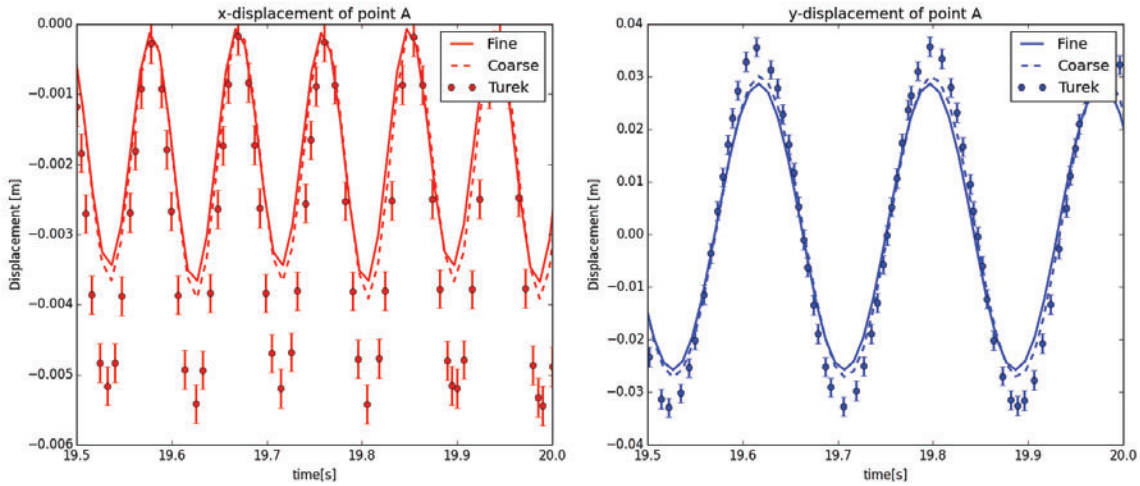


**Figure 3: Displacement over time of point A for a coarse grid, a finer grid and the numerical results presented in the Turek benchmark case [13]**

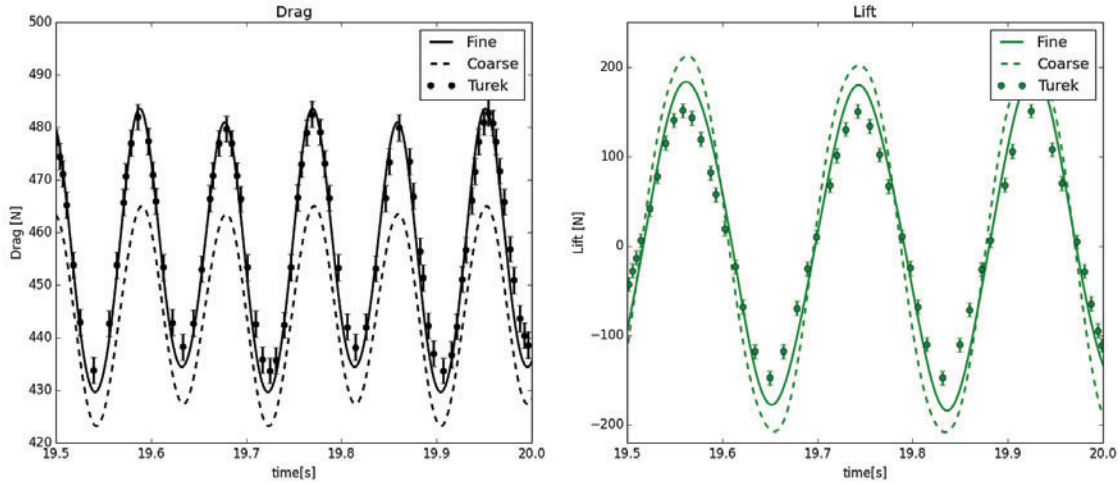


**Figure 4: Lift and drag forces on the whole submerged body (cylinder and flap) over time of point A. for a coarse grid, a finer grid and the numerical results presented in the Turek benchmark case [13]**

### 3.2. Vattenfall Experiment: Preliminary results

Vattenfall Research and Development in Sweden has performed an experiment [14] where a vertical slender rod with a pinned and a clamped side is displaced and then released. After which its vibrations and damping under the influence of the surrounding fluid are investigated. The considered computational domain is shown in Figure 5.

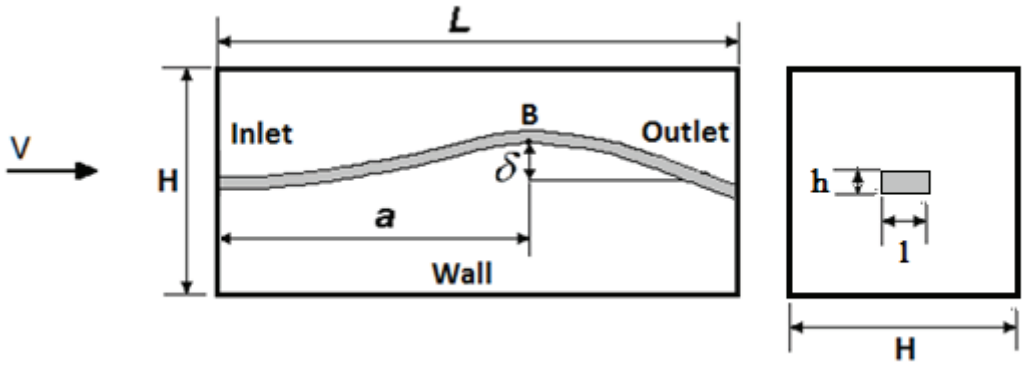


**Figure 5: A schematic drawing of the Vattenfall experiment where L=1.5m, a=0.75m, H=0.8m, h=0.008m, l=0.02m and the initial displacement $\delta$ =0.01 m. The gray represents the solid domain and the white the fluid domain.**

Measurements in air and water with no flow rate were performed initially and subsequently tests in water with 1.0 m/s and 3.0 m/s in the axial direction were performed. This case can therefore be used to test the influence of the added mass. Also, the mass ratio of the experimental case is $\rho_s / \rho_f = 8$ (using water as the fluid medium), which is larger compared to the Turek benchmark case. Therefore, it is expected that the IQN-ILS coupling algorithm can be used for this specific case. The rod is made of stainless steel with a density of $\rho = 8.00 \text{g/cm}^3$, a modulus of elasticity of $E = 193\text{GPa}$ and the moment of inertia is $I = 8.53 \cdot 10^{-10} m^4$. The left side of the rod is a fixed-fixed boundary condition and the right side is fixed in the all directions except for the beam longitudinal direction.

At this point in the study, only the structure solver of OpenFOAM is used to calculate the response of the beam after an initial displacement. The displacement of point B obtained with the three different simulations and the Vattenfall experiment are shown in Figure 6 and Table 2, along with their assigned nomenclatures. All simulations are 2D and the mesh size of the solid domain of simulations *sim_0.0001* and *sim_0.00001a* is 2912 cells with a base-mesh size of 2mm and in *sim_0.00001b* the solid domain contains 11968 cells with a base-mesh size of 1mm. For this relatively simple geometry a hexahedral mesh is chosen, this is optimal since it will gives the smallest amount of cells for a given base-mesh size. The time step of *sim_0.00001a* and *sim_0.00001b* is 0.00001 seconds and for *sim_0.0001* the time step is 0.0001 seconds. The results of the coupled simulation are still under review.

The resulting displacement of point B (see Figure 5) of three different simulations of the solid domain and the Vattenfall experiment in air at V=0m/s are shown in Figure 6 and Table 2. The three different simulations clearly show the influence of time and space discretization on the result. When comparing *sim_0.0001* and *sim_0.00001a* it can be clearly seen that the amplitude of the oscillation of *sim_0.00001a* is decreasing with time, the oscillation frequency of the two simulations are however almost equal (14.46 Hz and 14.58 Hz). This shows that using a time step of 0.0001 seconds is too big, and artificial damping caused by the numerical discretization will be introduced. A damping is however observed for the

experiment. When taking a sufficiently small time step this damping is not observed, this is because the used solid solver of OpenFOAM does not include a damping factor. The structural damping in this experiment is small compared to the aerodynamic damping, or in the other cases the hydrodynamic damping. It is therefore expected that when solving the coupled case, so when introducing aerodynamic damping in the simulation, a damping comparable to that found in the experiment will be introduced.

**Table 2: Oscillation frequencies of point B**

| | Frequency [Hz] |
|---|---|
| sim_0.0001 | 14.46 |
| sim_0.00001a | 14.58 |
| sim_0.00001b | 15.51 |
| Experiment | 12.30 |

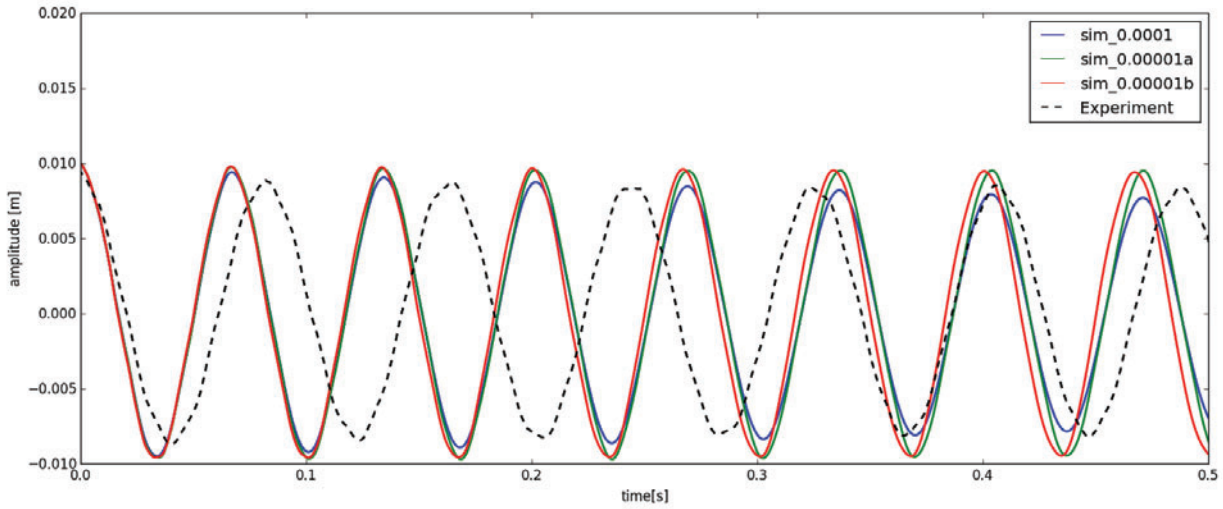


**Figure 6: Resulting displacement of point B of three different simulations and the Vattenfall experiment in air at V=0 m/s. sim_0.00001a and sim_0.00001b are both using a time step of 0.00001 seconds, and sim_0.0001 a time step of 0.0001 seconds. sim_0.0001 and sim_0.00001a have a grid size of 2912 cells and sim_0.00001b has a grid size of 11968 cells.**

When comparing *sim_0.00001a* to *sim_0.00001b* it can be observed that the frequency of the oscillation increases when the mesh is refined. The frequency changes from 14.46 Hz to 15.51 Hz, the effect of this difference in Figure 6 becomes more prominent over time. The frequency of the simulation is lower (12.30 Hz) compared to *sim_0.00001b* (15.51 Hz). Another influence on the frequency is the boundary condition.

When solving the case analytically, the frequency of a clamped-roller supported beam [15] can be calculated with:

$$\omega = \frac{15.418}{L^2}\sqrt{\frac{EI}{\rho A}}, \tag{11}$$

with $\omega$ being the natural frequency and, L the beams length, E the modulus of elasticity, I the moment of inertia, $\rho$ the density of the material and A the area. The analytic approximation of the beams frequency results in 12.37 Hz, this is very close to the value of 12.3 Hz measured in the experiment. As can be seen

from Figure 6, the influence of the time step and the mesh are not causing a big change in frequency. The influence of the added mass in quiescent air will be also minimal, and will not change the frequency so much. It is therefore possible that the boundary conditions applied in OpenFOAM simulation are not representing the experimental situation, and therefore causes the difference in frequency.

## 4. CONCLUSIONS

In the present paper, a validation study is performed on the used computational methods in Fluid Structure Interaction for nuclear plant applications. Two test cases are considered, i.e. the Turek benchmark case and the Vattenfall experiment. In both cases the mass ratio of the solid and the fluid is low ( $\rho_s / \rho_f < 10$ ) and the added mass, i.e. the mass of fluid which is accelerated by the structure, is large. For such cases it has been shown that a strongly coupled method is necessary. In this study a partitioned approach is used, in which the exchanges between the fluid and structure solver take place through the use of the IQN-ILS coupling scheme. The results of the Turek benchmark case show that the IQN-ILS method can be used to solve a strongly coupled problem.

In case of the Turek benchmark case and the Vattenfall experiment, the effect of the mesh and the time step can be clearly observed. If the chosen time step is too large, artificial damping will be introduced by the numerical time discretization scheme, whereas a mesh refinement will have an influence on the resulting oscillation frequency. Also, when considering a coupled simulation, a coarse mesh can cause an over prediction of the forces on the structure. By taking a small time step and a fine grid these effects can be eliminated. It is therefore suggested that a sensitivity analysis should be performed.

However, it can be concluded the IQN-ILS solver implemented in OpenFOAM can be used to solve a strongly coupled case. Therefore the coupled Vattenfall experiment and more complex nuclear applications can also be simulated using this method.

## 5. OUTLOOK

Since it is proven that the available IQN-ILS method is suitable for a strongly coupled case, the next step is to use the FSI simulation with the IQN-ILS coupling algorithm to simulate the other cases of the Vattenfall experiment. So using water as the fluid medium with a flow velocity of V=0m/s, V=1m/s and V=3m/s. The next goal will be to apply this method on a single rod with the same dimensions and confinement as a rod in a fuel bundle of a nuclear reactor.

## ACKNOWLEGEMENTS

We would like to thank Vattenfall Research and Development in Sweden for supplying us with their experimental data and David Blom from Delft University of Technology for supplying us with his numerical implementation of the FSI method in OpenFOAM.

## REFERENCES

[1] K. Luk, “Pressurized-Water Reactor Internals Aging Degradation Study,” Oak Ridge National Laboratory, 1993.

[2] D. Weaver, “Flow-Induced Vibrations in Power and Process Plant Components Progress and Prospects,” *Journal of Pressure Vessel Technology*, 2000.

[3] M. Lighthill, "Note on the swimming of slender fish," *Journal of Fluid Mechanics,* 1960.

[4] J. Banksa, "An analysis of a new stable partitioned algorithm for FSI problems. Part I: Incompressible flow and elastic solids," *Journal of Computational Physics,* 2014.

[5] Causin, "Added-mass effect in the design of partitioned algorithms for fluid-structure problems," *Computer methods in applied mechanics and engineering,* 2005.

[6] J. Yang, "A strongly coupled, embedded-boundary method for fluid structure interactions of elastically mounted rigid bodies," *Journal of Fluids and Structures,* 2008.

[7] J. Degroote, "Partitioned Simulation of Fluid-Structure Interaction," *Arch Comput Methods Eng,* vol. 20, pp. 185-238, 2013.

[8] J. D. Ridder, "Modal characteristics of a flexible cylinder in turbulent axial flow," *Journal of Fluids and Structures,* 2013.

[9] J. Degroote, "Partitioned simulation of the interaction between an elastic structure and free surface flow," *Computer Methods in Applied Mechanics and Engineering,* 2010.

[10] K.-J. B. a. J. V. Joris Degroote, "Performance of a new partitioned procedure versus a monolithic procedure in fluid–structure interaction," *Computers and Structures,* no. 87, p. 793–801, 2009.

[11] "The OpenFOAM® Extend Project," [Online]. Available: http://www.extend-project.de/.

[12] OpenFOAM, *User guide, Version 2.2.0,* OpenFOAM Foundation, 2013.

[13] S. Turek, "Proposal for numerical benchmarking of fluid-structure interaction between an elastic object and laminar incompressible flow," *Springer Links,* 2006.

[14] T. T. T. Vu, "Testing and optimization of Unicorn Fluid-Structure Interaction solver for simulating an industrial problem," Elforsk, 2014.

[15] S. Rajasekaran, Structural Dynamics of Earthquake Engineering: Theory and Application Using MATHEMATICA and MATLAB, Woodhead Publishing, 2009.