



GO-VIKING

Research and Innovation Action (RIA)

This project has received funding from the Euratom research and innovation programme 2021-2025 under Grant Agreement No 101059603

Start date : 2022-06-01 Duration : 48 Months



Cantilever Rod (2-ph) simulation results

Authors : Federico BARAGLIA (EDF), William Benguigui (IMSIA, UMR 9219 EDF/ENSTA/CNRS, 7 boulevard Gaspard Monge 91120 Palaiseau, France, EDF R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatou, France), Federico Baraglia (EDF R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatou, France); Constantinos Katsamis, Ewan Harris, Ehimen Iyamabo, (EDF Energy R&D UK, 90 Whitfield Street, London W1T 4EZ, United Kingdom); Antti Timperi (VTT Technical Research Centre of Finland, Kivimiehentie 3, Espoo, 02044 VTT, Finland)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	Cantilever Rod (2-ph) simulation results
Author(s)	Federico BARAGLIA, William Benguigui (IMSIA, UMR 9219 EDF/ENSTA/CNRS, 7 boulevard Gaspard Monge 91120 Palaiseau, France, EDF R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatou, France), Federico Baraglia (EDF R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatou, France); Constantinos Katsamis, Ewan Harris, Ehimen Iyamabo, (EDF Energy R&D UK, 90 Whitfield Street, London W1T 4EZ, United Kingdom); Antti Timperi (VTT Technical Research Centre of Finland, Kivimiehentie 3, Espoo, 02044 VTT, Finland)
Number of pages	39
Document type	Deliverable
Work Package	WP4
Document number	D4.2
Issued by	EDF
Date of completion	2026-04-21 09:33:27
Dissemination level	Public

Summary

Multiphase flowinduced vibration of elongated structures is a common issue in many engineering systems and can affect structural integrity through deformation and wear. This report presents CFD simulations of a deformable rod subjected to axial airwater multiphase flow, aiming to improve understanding of the vibration mechanisms and support engineering design and assessment. Three numerical approaches were used, combining finite-element structural models with finite-volume multiphase flow solvers in Star-CCM+, ANSYS Fluent, and neptune_cfd. Multiphase flow was modeled using either the Volume of Fluid (VoF) method or a multi-Eulerian approach, while turbulence was treated with RANS or LES models. Two-way fluid?structure coupling was applied and results were compared with experimental data for specified inlet mass fluxes. The report describes the experiment, numerical methods, computational setup, and resulting comparisons.

Approval

Date	By
2026-04-21 10:56:43	William BENGUIGUI (EDF)
2026-04-21 15:35:29	Papukchiev ANGEL (GRS)



Task 4.1. Analysis of the two-phase Cantilever Rod experiment

Numerical simulation of an air-water flow along a clamped/free axial rod.

Version N°2.0
April/ 2026



William Benguigui^{1,2}, Federico Baraglia¹, Constantinos Katsamis³, Ewan Harris³, Ehimen Iyamabo³, Antti Timperi⁴

¹EDF R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatrou, France

²IMSIA – UMR 9219 EDF/ENSTA/CNRS – 7 boulevard Gaspard Monge 91120 Palaiseau, France

³EDF Energy R&D UK, 90 Whitfield Street, London W1T 4EZ, United Kingdom

⁴VTT Technical Research Centre of Finland, Kivimiehentie 3, Espoo, 02044 VTT, Finland



Funded by
the European Union

Disclaimer

The content of this report reflects only the author's view. The European Commission is not responsible for any use that may be made of the information it contains.

Document information

Grant Agreement / Proposal ID	101060826
Project Title	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
Project Acronym	GO-VIKING
Scientific Coordinator	Angel Papukchiev, mailto:Angel.Papukchiev@grs.de , GRS
Project starting date (duration)	1st June 2022 – 31st May 2026 (48 Months)
Related Work Package	WP4 – Multiphase flows
Related Task(s)	4.1
Lead Organisation	EDF
Contributing Partner(s)	EDF, EDF UK, VTT
Due Date	30 November 2025
Submission Date	21 April 2026
Dissemination level	PU

History

Date	Version	Submitted by	Reviewed by	Comments
16.02.2026	1.0	W. Benguigui	A. Papukchiev, K. Zwijssen	
21.04.2026	2.0	F. Baraglia	W. Benguigui	

Table of contents

1. Introduction	9
2. Experiment	10
2.1 Mock-up	10
3. Numerical modeling	13
3.1 VTT	13
3.2 EDF R&D	14
3.3 EDF UK	16
3.4 Sum-up of the numerical approaches	18
4. Calculation domain & Mesh	19
4.1 VTT	19
4.2 EDF R&D	20
4.3 EDF UK	21
4.4 Sum-up of the calculation domains & mesh	23
5. Numerical results	24
5.1 Qualitative analysis of the flow through numerical simulation	24
5.2 Comparison in terms of tip displacements	27
5.3 EDF R&D results - details	28
5.4 EDF UK results – details	32
5.5 VTT results - details	34

List of figures

Figure 1: Test rig for the two-phase flow experiments: schematics for a) flow loop of the clamped-free configuration; b) flow loop of the free-clamped configuration; c) mixing section design with inset showing the real components for the air injection section.....	10
Figure 2: a) Adapted Taitel map for two-phase flow regimes and conducted experimental cases; b) example flow images for measurement cases at $Re_{ls} = 24k$, void fraction increases from left to right.	12
Figure 3: (left) Dispersed regime highlighted in green, (centre) large interfaces highlighted in red and (right) multiple regimes highlighted.....	14
Figure 4. Representation of the water inlet to enter with resolved structures in the EDF R&D case with neptune_cfd.....	15
Figure 5. Fine mesh used in VTT simulations at the centre plane (top) and at the rod surface (bottom).	19
Figure 6. Fine mesh used in VTT simulations at the inlet (left) and at the annular section (right).	20
Figure 7. Coarse (left) and fine (centre) meshes used in the neptune_cfd simulations and immersed solid delimiting the beam (right).	21
Figure 8: Solid Mesh. Note the structured mesh for the cylinder part (left) and unstructured mesh for hemisphere (right). The two are fused at the boundary.....	22
Figure 9: Fluid mesh around rod tip with conical geometry.....	22
Figure 10: Mesh of the inlet on the left and fluid outlet (orange) including the solid rod (cyan) on the right.....	23
Figure 11. Bubble shapes (top) near the rod tip and air instantaneous volume fraction (bottom) in the EDF R&D simulations.	24
Figure 12. Average fields after 4 seconds in the EDF R&D simulations at $U_{gs} = 0.35$ m/s. (Top) Mixture velocity and (bottom) air volume fraction.	25
Figure 13. Bubble shapes and void fraction near the rod tip in VTT simulations with fixed geometry and inlet bubble diameter 2 mm ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).....	25
Figure 14. Time-averaged axial mixture velocity at the centre plane in VTT simulations with fixed geometry and inlet bubble diameter 2 mm ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).	25
Figure 15. Time-averaged void fraction at the centre plane in VTT simulations with fixed geometry and inlet bubble diameter 2 mm ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).	26
Figure 16: Bubble shapes and void fraction near the rod tip in the EDF UK simulations	26

Figure 17 Mixture velocity magnitude at the centre plane in EDF UK simulations ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).	26
Figure 18 Time-averaged mixture velocity magnitude at the centre plane in EDF UK simulations ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).	27
Figure 19 Time-averaged void fraction at the centre plane in EDF UK simulations ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).	27
Figure 20 (left) front tip displacement PSD comparison, and (right) side tip displacement PSD comparison in the $U_{gs} = 0.35$ m/s and $U_{ls} = 1.7$ m/s case.....	28
Figure 21 Tip displacement RMS as a function of inlet gas flux in the EDF R&D case.	29
Figure 22 Tip displacement in the neptune_cfd simulations (top) and experiments (bottom) in the case $U_{gs} = 0.16$ m/s.....	30
Figure 23 Tip displacement in the neptune_cfd simulations (top) and experiments (bottom) in the case $U_{gs} = 0.35$ m/s.....	30
Figure 24 Tip displacement in the neptune_cfd simulations (top) and experiments (bottom) in the case $U_{gs} = 0.79$ m/s.....	31
Figure 25 (left) front tip displacement PSD comparison, and (right) side tip displacement PSD comparison in the $U_{gs} = 0.79$ m/s and $U_{ls} = 1.7$ m/s case.....	31
Figure 26 Phase graphs for the three inlet fluxes simulated with neptune_cfd.	32
Figure 27: Time series graph of displacement in X	32
Figure 28: Time series graph of displacement in Y	33
Figure 29: Power Spectral Density graph of displacement in X	33
Figure 30: Power Spectral Density graph of displacement in Y	34
Figure 31: Tip trajectory - simulation vs experiment	34
Figure 32 Bubble shapes and centre plane void fraction with inlet bubble diameters of 2 mm, 4 mm and 8 mm (from top to bottom) when using the fine mesh ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).	35
Figure 33 Tip front (left) and side (right) displacement spectra obtained with different meshes and inlet bubble sizes ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).	35
Figure 34 Tip displacements with fine mesh and inlet bubble diameter 2 mm (top) and in experiment (bottom). Note that the total simulation sampling time is about 25 s.	36

List of tables

Table 1: All experimental flow setups. In this report, we focus on the first configuration.	11
Table 2. Properties of water and air used in the simulations.	14
Table 3. Rod properties applied in the simulations. The Rayleigh damping values α and β correspond to the experimental free vibration damping ratio in air $\zeta = 0.0009$	14
Table 4. Rod tip radial RMS displacement obtained in VTT FSI simulations with different meshes and inlet bubble sizes ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).	28

Abbreviations and Acronyms

Acronym	description
WP	Work package
GO-VIKING	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
FSI	Fluid Structure Interaction
RANS	Reynolds Averaged Navier Stokes
LES	Large Eddy Simulation
HRIC	High Resolution Interface Capturing
GLIM	Generalized Large Interface Model
FEM	Finite Element Method
CFD	Computational Fluid Dynamic
VOF	Volume Of Fluid
FIV	Flow-Induced Vibration

Summary

Multiphase flow–induced vibration of elongated structures is a common issue in many engineering systems and can affect structural integrity through deformation and wear. This report presents CFD simulations of a deformable rod subjected to axial air–water multiphase flow, aiming to improve understanding of the vibration mechanisms and support engineering design and assessment.

Three numerical approaches were used, combining finite-element structural models with finite-volume multiphase flow solvers in Star-CCM+, ANSYS Fluent, and neptune_cfd. Multiphase flow was modeled using either the Volume of Fluid (VoF) method or a multi-Eulerian approach, while turbulence was treated with RANS or LES models. Two-way fluid–structure coupling was applied and results were compared with experimental data for

specified inlet mass fluxes. The report describes the experiment, numerical methods, computational setup, and resulting comparisons.

Keywords

Key words: Two-phase flow, tube bundle, CFD, numerical simulation, flow-induced vibration

1. Introduction

Multiphase flow-induced vibration and deformation of elongated structures represent an ever-occurring configuration in engineering applications. The scales at which this issue is encountered can vary remarkably. Since the integrity of a structure can be compromised by the coupled behaviour of multiphase fluids and structure through deformation or wear, it is crucial to provide tools to accurately predict their evolution. This report presents the results of FSI simulations of an elongated deformable structure subjected to axial multiphase flow. Its main goal is to provide insights into the mechanisms of rod vibration in such configurations and support engineers in the design of new structures or the assessment of existing ones.

The participants (EDF UK, VTT and EDF R&D) involved in this task (*Task 4.1. Analysis of the two-phase Cantilever Rod experiment*) used three different methods combining finite-element models for the solid structures and a finite volume solver for the multiphase flow within StarCCM+, Ansys-Fluent and neptune_cfd. The multiphase aspect was treated with either the VoF model for simulations with StarCCM+ or Ansys-Fluent or the multi-Eulerian modelling for simulations with neptune_cfd. The obtained results are compared with experimental data for a specific water and air inlet mass flux. Turbulence is considered either with RANS modelling (k- ω SST or second order Reynolds stress model), or with Smagorinsky LES. The solid rod and the fluids are coupled with a strong two-way solver, one-way coupling was tested, but appeared to lack consistency for this scenario.

The experiment is described in Chapter 2, the numerical modelling for each participant is described in Chapter 3 and the mesh and domain setup are described in Chapter 4. The results are presented in Chapter 5.

2. Experiment

2.1 Mock-up

The Cantilever Rod experiments at the University of Manchester are conducted using a water tank with an open-flow loop driven by a centrifugal pump (10HP Leeson pump). The setup is an upgraded version of the WP2 design (Cioncolini et al 2023, Li et al. 2026) with additional flow paths to allow air injection for air-water two-phase flow experiments. Within the flow loop, a test section is installed which hosts a rod in the centre of a confining tube at an upright vertical position. The rod is clamped at the top end and free at the bottom. Flow is allowed to pass through the annulus between the rod and the confining tube. Schematics of the experimental rig and flow loop are provided in Fig. 1a-b.

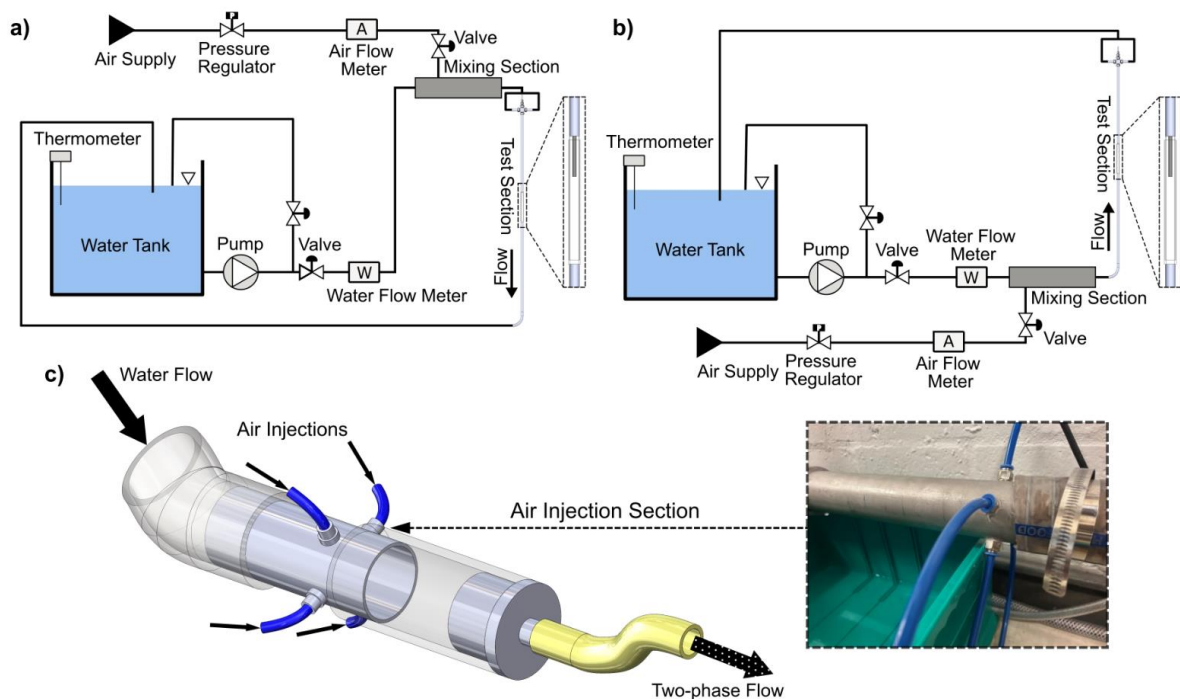


Figure 1: Test rig for the two-phase flow experiments: schematics for a) flow loop of the clamped-free configuration; b) flow loop of the free-clamped configuration; c) mixing section design with inset showing the real components for the air injection section.

A direct immersion digital thermometer (ETI-Electronic Temperature Measurements Ltd; temperature range: 248–343 K; accuracy ± 1 K) is used to measure the temperature of water allowing for estimation of its density and viscosity. The mass flow rate of water is measured using a turbine flow meter (Cole-Parmer; measurement range 37.8–378 L/min; accuracy ± 3.78 L/min) installed upstream of the test section. The air injection system is connected to the main flow loop via a mixing section, Fig. 1c. Within the mixing section, four injection ports are located around the air injection pipe, which are arranged in a circumferential manner. The injection ports are controlled by individual valves to allow better mixing of air and water upstream of the test section. For the experiments conducted in this study, air supply is provided via a compressed source at a constant supply pressure of 4 Bar. A flow meter (SMC PFM750S, measurement range 0–50 L/min; accuracy ± 0.5 L/min) is used to measure the total

flow rate for air injections. The hydraulic diameter, D_h is taken as the annular gap between the confining tube and the rod ($D_h = D_t - D_r$) and the velocity, U_{als} is taken as the annulus superficial velocity of the water phase. Experiments are conducted for five different Reynolds numbers, within a range between 24-73 K. For each Reynolds number, eight different air injection flow rates are considered, with the corresponding U_{gs} varying between 0-1.78 m/s. Note that, a single-phase water flow case with $U_{gs} = 0$ is also included. The experiment cases investigated in the current study are summarized in Table 1 including the range of void fraction, α , covered in each Reynolds number case.

Table 1: All experimental flow setups. In this report, we focus on the first configuration.

Re_{als}	U_{ls} (m/s)	U_{gs} (m/s)	α (%)
24 000	1.7	0.0 - 1.78 m/s	0.0 - 0.51
37 000	2.5	0.0 - 1.78 m/s	0.0 - 0.42
49 000	3.3	0.0 - 1.78 m/s	0.0 - 0.35
61 000	4.1	0.0 - 1.78 m/s	0.0 - 0.30
73 000	4.9	0.0 - 1.78 m/s	0.0 - 0.27

Additionally, Fig. 2a shows the experiment cases against the Taitel map [17] adapted for the current experimental conditions, illustrating the flow regimes for the different cases. It is evident that most of the experimentally investigated cases, over the higher Reynolds number range, fall into the bubbly flow regime, whereas slug/churn flow is only encountered for the lowest range of Reynolds numbers investigated. Representative two-phase flows for the current experiments are also illustrated in Fig. 2b, where an example hemispherical rod tip at a Reynolds number of 24 k is used for this demonstration. Details of the test section including dimensions of the cantilevered rod are shown in Figure 1. A union cross (Swagelok SS-25M0-4) is used to clamp the rod at the top end while at the same time leaving two horizontal ports for the flow loop connection. The confining tube is made of AISI 316 stainless steel which has an inner/outer diameter of 20.86/25.06 mm. The test section is rigidly fixed to a ground frame throughout the experiments. In the region near the rod tip, the confining tube is replaced with a transparent Perspex section with the same inner/outer diameter to allow optical access. The linear mass density of the rod, accounting for both the tube material and the lead shot filling, is 5.88 g/cm. An initial free vibration experiment was conducted to confirm the rod flexural rigidity, and the reader can refer to the previous work for detailed discussions on these verification experiments (Cioncolini et al, 2023). The rod tube is sealed with an aluminium cap with a mass of approximately 2.1 g. A cylindrical sensor mount-piece is attached to the flat end surface of the aluminium cap using secure adhesive. This component is used for the non-intrusive method developed for measuring the rod vibration dynamics. The sensor mount-piece has flat surfaces, which act as the end-piece for the blunt tip case investigated in the current study. Both the sensor mount-piece and the end-pieces are 3D printed using light-weight hard resin, hence the influence of their mass on rod vibrations are deemed negligible.

A simple non-intrusive method for the measurement of the rod vibration under two-phase axial flows has been developed in this study, based on the Hall effect sensors (Li et al 2026). Specifically, the methodology exploits Hall effect sensors to measure the position of the vibrating rod tip, and hence to track its motion during axial-FIVs.

Thus, for the present numerical work, the mass flow rates of the two phases at the inlet are the inlet conditions, and the rod tip vibration is the result.

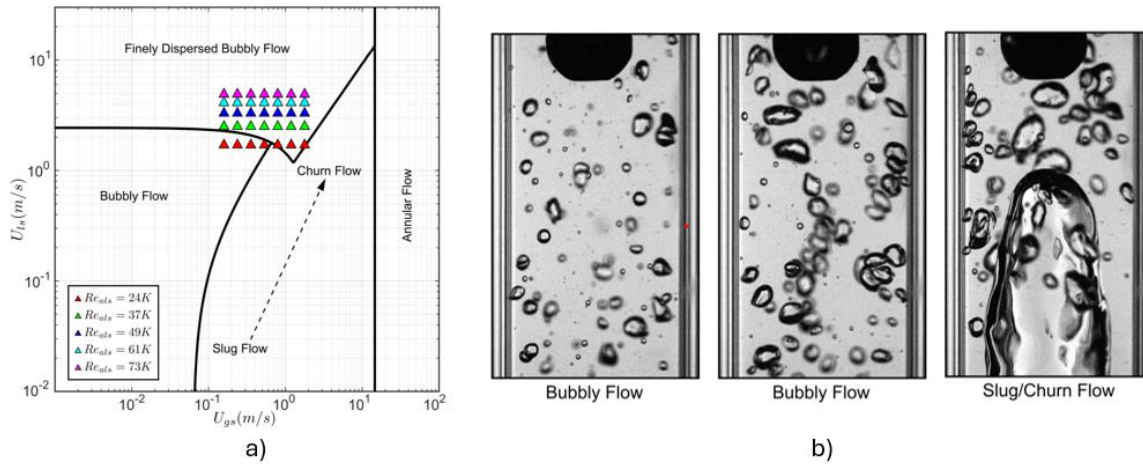


Figure 2: a) Adapted Taitel map for two-phase flow regimes and conducted experimental cases; b) example flow images for measurement cases at $Re_{als} = 24k$, void fraction increases from left to right.

For further details, the reader is invited to read the dedicated Go-Viking report ([https://go-viking.eu/images/resources/GO_VIKING_D4_1_Cantilever_Rod_experiment_\(2_ph\)__presentation_of_the_data.pdf](https://go-viking.eu/images/resources/GO_VIKING_D4_1_Cantilever_Rod_experiment_(2_ph)__presentation_of_the_data.pdf)) or its publication : Li et al, 2026.

3. Numerical modelling

Here are the three numerical approaches used for the simulations by the three participants involved. The summary of their models and the experimental configurations they simulated are summed up in Table 4 and Table 5.

3.1 VTT

The two-phase flow was simulated with the volume-of-fluid (VOF) model of Ansys-Fluent v2024. The geometric reconstruction scheme, which represents the interface as piecewise-linear, was used for tracking the air-water interface. This scheme resolves the interface sharply but is available only when using the explicit VOF formulation together with first-order time discretization. The compressive and high-resolution interface capturing (HRIC) schemes together with implicit VOF and second-order time discretization were also tested in a simplified one-bubble case, however, these resulted in significantly smeared interfaces compared to the geometric reconstruction using explicit VOF. Surface tension effects were modelled by the continuum surface force (CSF) model, which adds a volumetric source term in the momentum equation based on the local interface curvature (estimated from the void fraction gradient). Turbulence was modelled by using large-eddy simulation (LES), together with the wall-adapting local eddy-viscosity (WALE) sub-grid scale (SGS) model and wall functions. The default value $C_W = 0.325$ of Ansys-Fluent was used for the WALE SGS coefficient.

The bounded central differencing (BCD) with a boundedness factor of 0.5 was applied for the spatial discretization due to the LES modelling. The boundedness factor controls the balance between accuracy and stability (numerical diffusion) of the BCD scheme; a fairly low value of 0.25 is recommended for LES in general, in order to have relatively good accuracy. However, in LES-VOF modelling of the two-phase TREFLE experiment (Benguigui et al., 2026), the default factor of 0.25 resulted sometimes in numerical instabilities. Thus, the value 0.5 was chosen to guarantee the solution stability.

Also based on the TREFLE LES-VOF modelling, velocity limiting was applied to avoid local unphysical high velocities at the air-water interfaces, in order to stabilize the solution. The velocity was limited to 27 m/s, i.e. 10 times the mean homogenous mixture velocity in the annular region. As the limit is very high compared to the mean velocity, it is expected to have a minimal effect on the actual results. The velocity limiting was typically temporarily applied per time step only on few cells of the computational domain.

The SIMPLE pressure-correction algorithm was applied, together with the PRESTO scheme for pressure discretization. Time steps 66.7 μs and 50 μs were used with the coarse and fine mesh, respectively. These correspond to a CFL number of 0.58 based on the maximum mean velocity of the mixture (see Sec. 5) and the axial cell size at the rod tip. Due to the small time steps, only 10 iterations per time step were performed in the flow solution. The simulations were run for 25 s after settled conditions were reached for data sampling.

FSI was modelled by explicit coupling, where the surface load and displacement data was exchanged once in time step. The rod motion was solved by finite element method (FEM), by using the Euler-Bernoulli slender beam elements and the Newmark time integration (Cook et

al., 2002). Each element had two nodes involving the translational and rotational degrees-of-freedom (d.o.f.). The deflections of the rod in the two transverse directions were solved separately due to the relatively small displacements. At the coupling interface, the pressure and viscous forces were interpolated to the FEM model nodes. The CFD interface mesh nodes were displaced according to the interpolated FEM nodal translations and rotations, and the internal CFD mesh was deformed with the diffusion model based mesh smoother of Ansys-Fluent. The FEM solution and the explicit FSI coupling were implemented through user-defined functions (UDF). The water, air and solid properties are summed up in Table 2 and Table 3.

Table 2. Properties of water and air used in the simulations.

Fluid	ρ [kg/m ³]	μ [kg/(ms)]	σ [N/m]
water	998	0.001	0.072
Air	1.225	1.79×10^{-5}	

Table 3. Rod properties applied in the simulations. The Rayleigh damping values α and β correspond to the experimental free vibration damping ratio in air $\zeta = 0.0009$.

L [mm]	E [GPa]	I [mm ⁴]	m [kg/m]	α [1/s]	β [s]
1060	193	194.4	0.588	0.043	3.9×10^{-6}

3.2 EDF R&D

The simulations are carried out with neptune_cfd release 9.0. It is a multiphase fluid solver with 3D full unstructured finite volume discretization and co-located variables based on the multi-Eulerian approach. The generalized large interface model (GLIM) model is used to account for dispersed and resolved air structures. This model, embedded in neptune_cfd, enables the simulation of multiple air-water regimes from bubbly dispersed to droplet dispersed. It considers the interphase momentum transfer as a combination between a continuous-dispersed and continuous-continuous contribution. Thus, it can simulate dispersed bubbles with drag, lift and added mass at the same time it simulates large interfaces and the shear induced by surface friction (Merigoux 2022, Baraglia et al 2024).

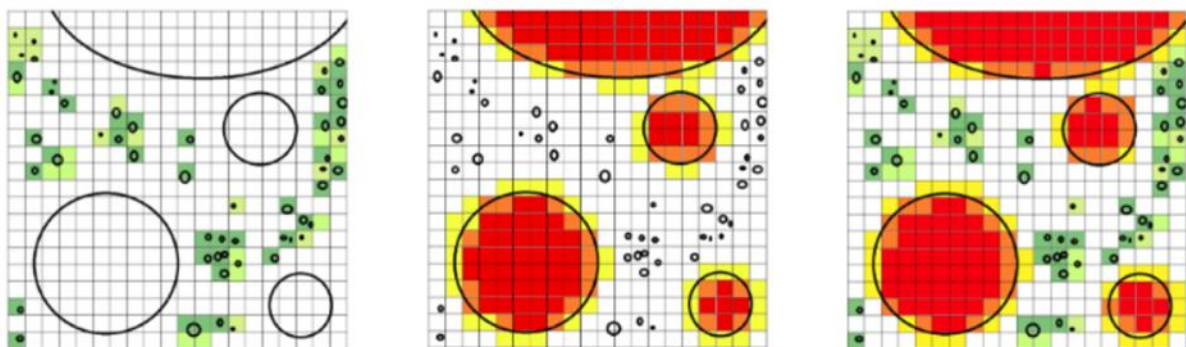


Figure 3: (left) Dispersed regime highlighted in green, (centre) large interfaces highlighted in red and (right) multiple regimes highlighted

Gravity is standard and directed along the channel axis. Second order $R_{ij} - \epsilon$ (EBRSM) turbulence model is selected for the water phase, and no turbulence model is selected for the gas phase. The fluctuations are accounted for in the turbulent dispersion model. Turbulence modulation due to large interfaces and dispersed structures is also activated. To keep air pockets stable, we also deactivate the added mass contribution to the bubble's turbulent dispersion. Only the drag contribution is selected. The interfacial area transport equation is activated to account for changes in disperse bubble diameter. The standard diameter for the model is set to $D_b = 4$ mm. Standard physical properties are set for water and air. A strong two-way coupling is used to simulate the fluid-structure interaction. It is based on a discrete forcing method (Benguigui et al 2018, Benguigui et al 2019) to represent the solid (Immersed boundary method) and a Euler-Bernoulli rod finite element solver for the solid deformation (Baraglia et al 2021).

Initially, the pressure is set to $P_0 = 0$ Pa and the channel is filled with water at rest.

The vertical boundary is a wall, and the two sides of the channel are an inlet and a pressure outlet respectively. The outlet is set to a mean outlet pressure with $P_{out} = 0$ Pa and the inlet is changed so that resolved structures enter the domain. 12 disks where the air volume fraction at the inlet equals 1 are defined. In the other part of the inlet, water volume fraction equals 1. The velocity of both fluids is set to maintain the correct superficial liquid and gas velocities. We simulate the cases $U_{ls} = 1.7$ m/s and $U_{gs} \in [0.16, 0.35, 0.79]$ m/s. Figure 4 shows the segmented inlet condition.

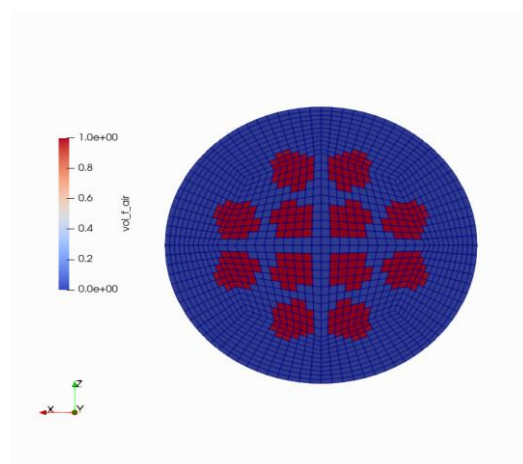


Figure 4. Representation of the water inlet to enter with resolved structures in the EDF R&D case with neptune_cfd

The beam Young's modulus is $E = 1.93 \times 10^{11}$ Pa, its density is $d = 7.49$ and its damping coefficient is $\xi = 0.013$ (The Rayleigh coefficients are $a = 1 \times 10^{-4}$ and $b = 7 \times 10^{-5}$). The Rayleigh coefficients are set by simulating the vibration of the rod in resting water. We consider between 1 and 6 FSI cycles and the convergence criteria is set to $\Delta = 0.01$. A wall-law type wall condition is set at its boundary.

The beam is initially fixed (for 0.5 seconds) to let the flow settle, at $t = 0.5$ seconds, the beam starts deforming and dos so for around 5 seconds. This means the mixture flows past the beam around 15 times.

3.3 EDF UK

Computations are conducted using StarCCM+ v2024.6 which relies on finite volume discretisation of the transport equations for the fluid domain, while finite element discretisation is used for the solid region.

The two-phase modelling is achieved with Volume of Fluid Method (VoF)-URANS framework to characterize the bubbly regime two-phase and the Solid Stress model was used to perform the structural analysis of a cantilever beam in StarCCM+. Closure of the transport equations is done using $k-\omega$ SST model. The inlet conditions are based on the mixture velocity, computed from the summation of the inlet superficial gas velocity with value of 0.35 m/s and from the superficial liquid velocity with value of 1.7 m/s. This approach is consistent with the way VOF CFD models mixture inlets using a shared velocity field for all phases. Once the flow is injected, inertia, gravity, shear and surface tension effects are modelled and result in flow segregation and bubbles development towards a steady state and gas volume fraction. This has been achieved by enabling the multi-phase submodels available in StarCCM+ to enable the bubble generation. These models are, the interface area density between the two fluids, the Interface Turbulence Damping which accounts for the turbulence fluctuations due to the large gradients in the boundary layer near the free surface and the surface tension force. A HRIC (High Resolution Interface capturing) convection scheme was selected for the Volume of fluid solver. HRIC sharpening factor was set to 0.9 to prevent smearing of sharp gradients at phase boundaries and mimic the convective transport of immiscible fluid components to track sharp interfaces.

The boundary condition for the cantilever rod at the base was specified as symmetry, while an FSI Interface was set for the boundary condition of the remainder of the rod surface. A Fixed Solid Stress Constraint was specified for the cantilever rod base. The motion specification for the cantilever rod was set to Solid Displacement. The outlet boundary condition was set to "Pressure Outlet".

For the solid stress solver, the Max Norm was selected as the type of norm used in the stress residual calculation. The option for strain energy was deactivated. For the test case, a stainless-steel material was selected for the rod. Note that the structure of the rod in the experiment consists of a shell filled with dense lead shots; however, we assumed a solid rod with homogeneous properties for the numerical modelling, with the properties taken from Salachna (2023): a density of 7490 kg m⁻³ and a Young's modulus of 76.4 GPa. The beam model, FSI and morphing set up, is based off that of previous work in Work Package 2 on single phase flow-induced vibration of the same cantilever rod geometry. The FSI was activated at the 2 second mark, after allowing a period for the fluid flow to develop. The FSI coupling was ramped over a 0.01 seconds time period to avoid any non-physical instabilities from sudden FSI coupling.

Two-way coupling is used to simulate fluid-structure interaction, done through a partitioned approach for both the fluid and solid domain calculations, where the fluid and solid domains are solved separately, but coupled strongly. At the interface between the fluid and solid domains, energy and momentum are conserved. To improve the stability of the calculation, the additional effect of a displaced mass of fluid acting on the solid domain is computed. This displaced fluid mass is added to the solid domain. Prior progress reports detail the calculation

of this “boundary added mass” from theoretical principles. StarCCM+ activates this additional effect through the specified value of the ratio of the displaced fluid volume to wetted surface area. The mesh needs to deform for the fluid-structure interaction calculation, done by activating the deformation option for the fluid and solid domains at every timestep. In StarCCM+, the deformation methods for the fluid and solid domains are activated in the "Tools Folder". The deformation parameter for fluid domain motion is called "Morphing" in StarCCM+, while the parameter for solid domain motion is called "Solid Displacement". After these deformation parameters have been activated, they must be specified in fluid and solid regions in StarCCM+ by labelling “Fixed” surfaces and morphing surfaces.



3.4 Sum-up of the numerical approaches

Table 4: Summary of the numerical approaches from the three participants involved.

Partner	Software	Two-phase modeling	Turbulence model	Type of Inlet	Type of Outlet	Space discretization
VTT	Ansys-Fluent	VOF	LES WALE	Velocity with SBM (Fluent UDF)	Pressure	Bounded central differencing with factor 0.5
EDF R&D	Neptune_cfd	Multi Eulerian	RIJ EBRSM	Velocity with full air inlet	Pressure	SOLU
EDF UK	StarCCM+	VOF	kw-SST	Velocity with full air inlet	Pressure	Second Order Upwind (SOU)

3.5 Summary of the experimental configuration simulated

Table 5: Summary of the experimental configurations simulated by the participants involved.

Partner	Liquid inlet flux	Air inlet flux
VTT	1.7 m/s	0.35 m/s
EDF R&D	1.7 m/s	0.16, 0.35, 0.79 m/s
EDF UK	1.7 m/s	0.35 m/s

4. Calculation domain & Mesh

4.1 VTT

The geometry used in the simulations covers the rod length of the experiment and an inlet section with a length of 5 diameters of the confining tube. The outlet of the flow domain is simplified, i.e. the exact geometry of the union cross in the experiment was not modelled and the outlet was set directly to the confining tube surface.

Two different hexahedral meshes were created, a coarse one with about 2.5 million cells and a fine one with about 6.2 million cells. The characteristic cell size in the inlet section is about 0.4 mm and 0.3 mm for the coarse and fine mesh, respectively. The fine mesh used in the simulations is shown in Figure 5 and Figure 6. The smallest air bubbles shown in the experiments (see Sec. 2) are not resolved accurately with the meshes used; resolving these with the VOF model would require a prohibitively dense mesh. However, presumably the rod motions are mainly due to the larger bubbles.

For the FSI calculation, the rod was modelled by FEM based on the Euler-Bernoulli beam theory. The element size is 5 mm, thus 212 elements were applied over the rod length, with the first element covering the rounded end of the rod.

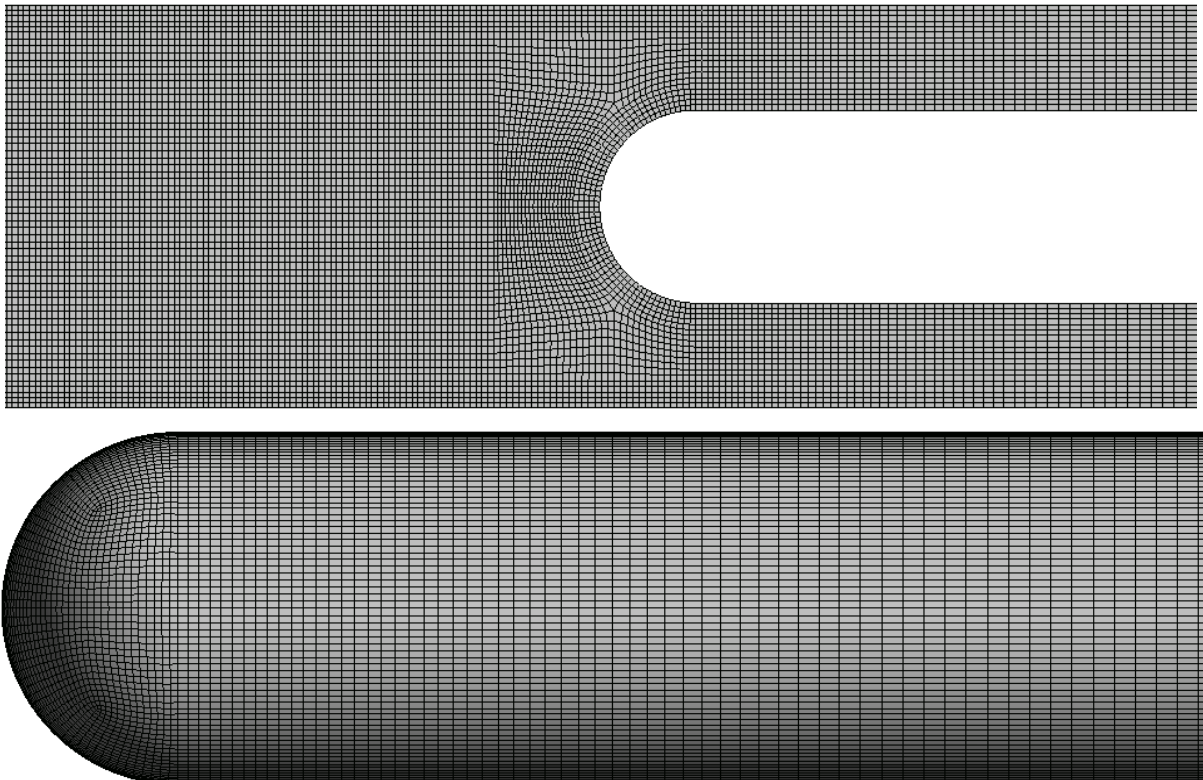


Figure 5. Fine mesh used in VTT simulations at the centre plane (top) and at the rod surface (bottom).

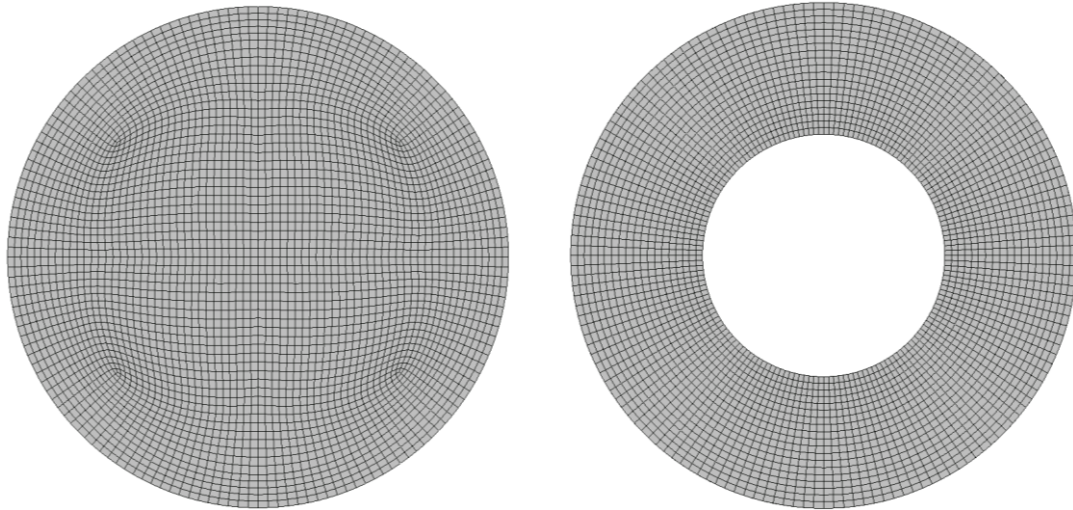


Figure 6. Fine mesh used in VTT simulations at the inlet (left) and at the annular section (right).

4.2 EDF R&D

In the simulations, we focus solely on the hemispherical tip and up flow configuration - the flow faces the round tip of the beam. The geometry is detailed in *Cioncolini et Al. 2023*. The simulated domain consists of a vertical cylindrical channel of diameter $D_t = 0.02$ m and height $H_c = 2$ m. The beam is fixed at the top end of the channel, its height is $H_b = 1$ m, its diameter is $D_r = 0.01$ m, and the round tip has a radius of $R_t = 5$ mm.

We used gmsh to generate a standard O-grid mesh of the cylinder. As mentioned above, only the two meters of the channel are represented in the simulation domain. Three different hexahedral meshes have been tested to assess the impact of mesh refinement on the fluid and solid fields. The Figure 4 shows the transverse section, the vertical segmentation of the mesh for the coarsest and finest meshes as well as the porosity field representing the beam on the finest mesh. The meshes tested are respectively 300k, 1M and 3M cells. Simulations with a fixed beam will determine which mesh is more appropriate considering accuracy and computing time.

For the FSI simulation, the beam is modelled with Euler-Bernoulli second order finite elements: 5 elements of size 20 cm are used. The rounded tip is included in the last element and impacts the rod behaviour only through the external forces. The elements characteristics are not modified.

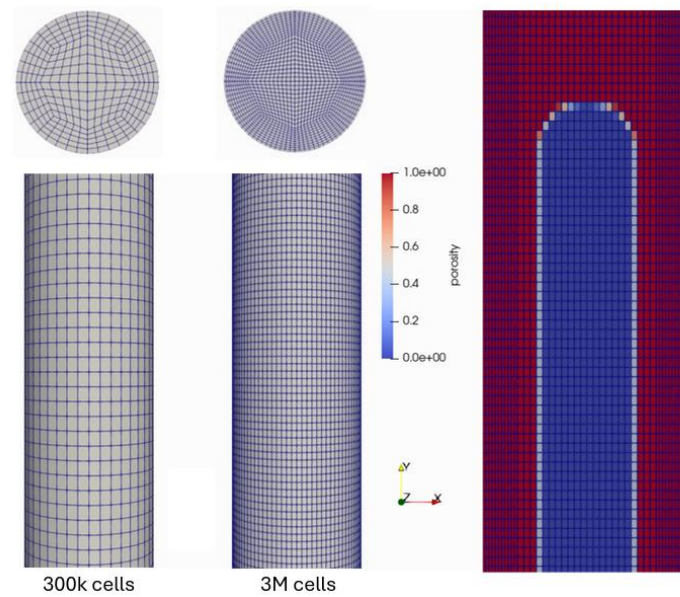


Figure 7. Coarse (left) and fine (centre) meshes used in the neptune_cfd simulations and immersed solid delimiting the beam (right).

4.3 EDF UK

The computational domain simulated is covering the rod length of the experimental configuration. The inlet section upstream of the cantilever rod was further extended from 5D to 15D compared to our previous single-phase calculations in Work Package 2, where D denotes the channel diameter. This was necessary to enable bubble generation and the two-fluid region to be established before reaching the rod surface.

Due to the strong coupling at the interface between the solid and fluid domains, a non-conformal mapping was necessary, to enable hexahedral elements to be used for most of the solid mesh except for the hemi-sphere, benefiting the FEM computation for the FSI part with reduced computational cost and a smaller mesh size for the fluid domain. Hence, a non – conformal interface was employed for the case, particularly as the fluid domain needed to be refined around the cantilever rod tip where flow acceleration occurred. Further details of the calculation domain are provided in Table 6 and Figure 8, Figure 9 and Figure 10.

Table 6: Summary table of fluid mesh properties for $Re = 21k$ case

	Value
Fluid Mesh size	2M
Solid Mesh size	62500
Maximum Wall y^+	39.4
Surface Average Wall y^+	31.6
Near-wall y^+ model / Wall Treatment Model	All y^+ wall treatment
Maximum Courant Number	0.5
Volume Averaged Courant Number	0.1

Number of inner iterations per time step	6
Time Step Size (s)	0.00015
Total Simulation Time (s)	8

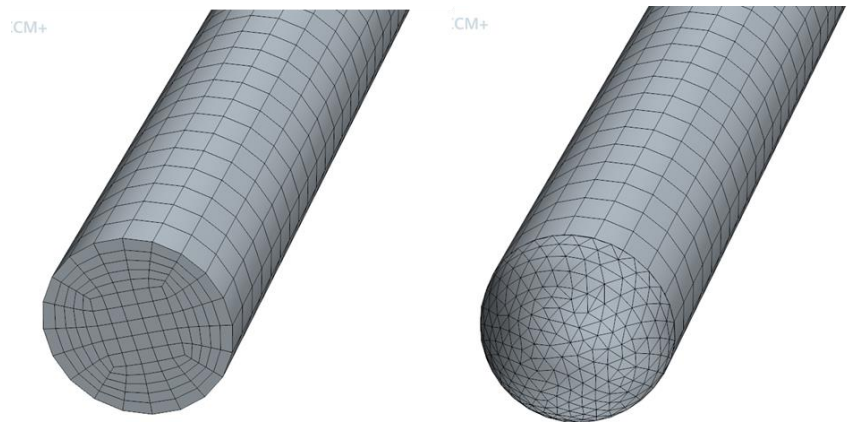


Figure 8: Solid Mesh. Note the structured mesh for the cylinder part (left) and unstructured mesh for hemisphere (right). The two are fused at the boundary.

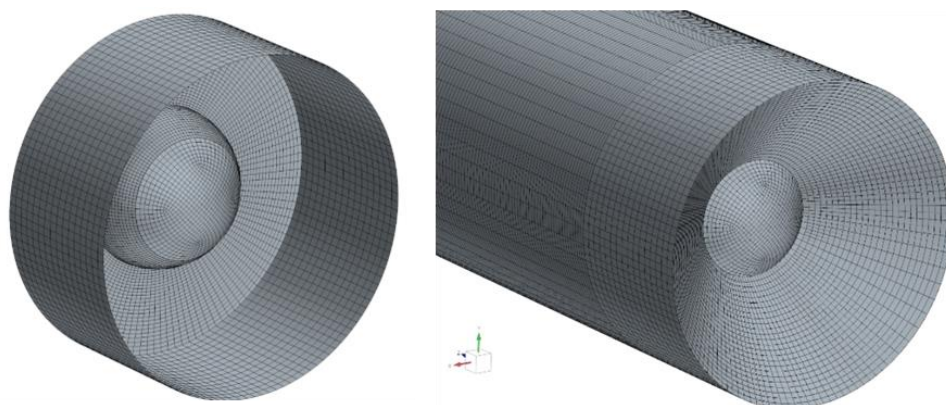


Figure 9: Fluid mesh around rod tip with conical geometry.

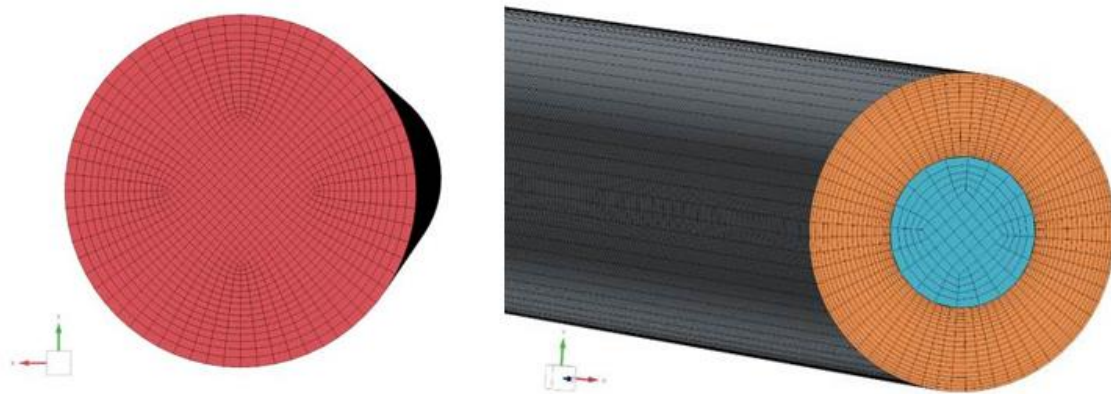


Figure 10: Mesh of the inlet on the left and fluid outlet (orange) including the solid rod (cyan) on the right.

4.4 Sum-up of the calculation domains & mesh

	2D or 3D	Tube Length	With upstream tube bundle	periodicity	Type of mesh	Total number of cells	Dr (mm) at cylinder wall	Dx _{mean} (mm)
EDF R&D	3D	2 m	No	no	Hexahedral	300K to 3M	1.5 mm to 0.6 mm	2 mm to 1 mm
VTT	3D	1.16 m	No	no	Hexahedral	2.5M, 6.2M	0.35 mm, 0.28 mm	0.6 mm, 0.45 mm
EDF UK	3D	2.0 m	No	No	Hexahedral	2M	1.5 mm	3 mm

5. Numerical results

5.1 Qualitative analysis of the flow through numerical simulation

Here are some qualitative comparisons between the different results obtained. For the $U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s case, the instantaneous and averaged air volume fractions and the axial air, water and mixture velocities are shown in the Figures 11 to 19 and differences are highlighted. Note that for the averaged velocities, the values for both phases are available in the EDF R&D case, whereas only the mixture velocity is available in the VTT case due to the one-field VOF modelling. In both the VTT and EDF R&D simulations, the velocity fields seem to be qualitatively similar. Upstream the beam tip, the velocity is at its maximum at the center and decreases near the wall. The width of the boundary layer is similar in both cases. After the rod tip, small differences can be highlighted. The overall velocity seems consistent in the two simulations, but the wall boundary layer is larger in the simulations with `neptune_cfd`. There are more differences in the average air volume field. In the `neptune_cfd` simulations (EDF R&D), the volume fraction profile seems more developed than in the VTT case; it is not changing in the vicinity of the beam tip. Downstream the rod tip, in the VTT case, the air is mainly found near the beam boundary whereas in the `neptune_cfd`, the air is stuck between the beam boundary and the channel wall. This change in behaviour could explain some of the differences observed in the beam deformation results. It also must be said that the air structures are slightly different. In the `neptune_cfd` simulations, the pockets are elongated compared to those observed in the VTT simulations. EDF UK qualitative findings of void fraction and mixture velocity prediction from the VOF computations are presented in Figures 16 to 19. The mean velocity prediction is consistent with the one from the VTT and EDF R&D simulations. The mean void fraction is larger at the centre of the tube which is consistent with the simulation results from VTT.

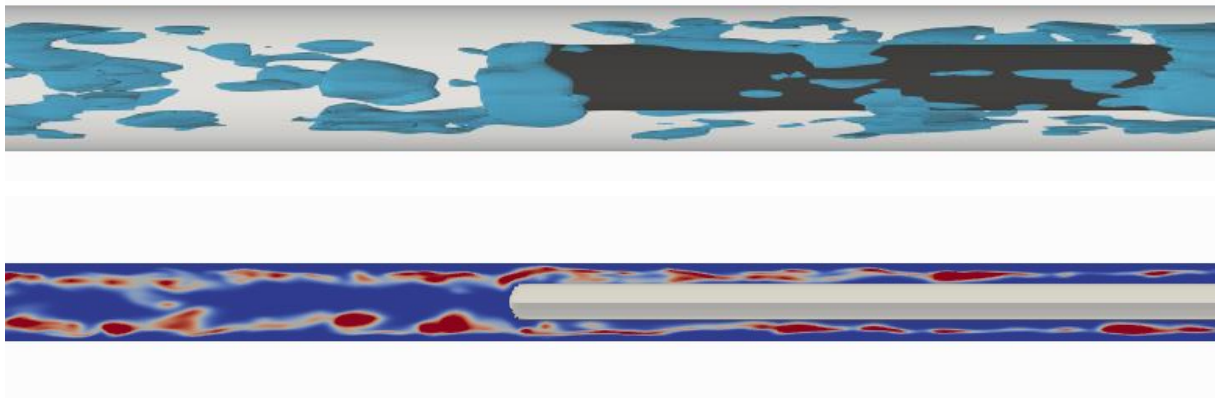


Figure 11. Bubble shapes (top) near the rod tip and air instantaneous volume fraction (bottom) in the EDF R&D simulations.

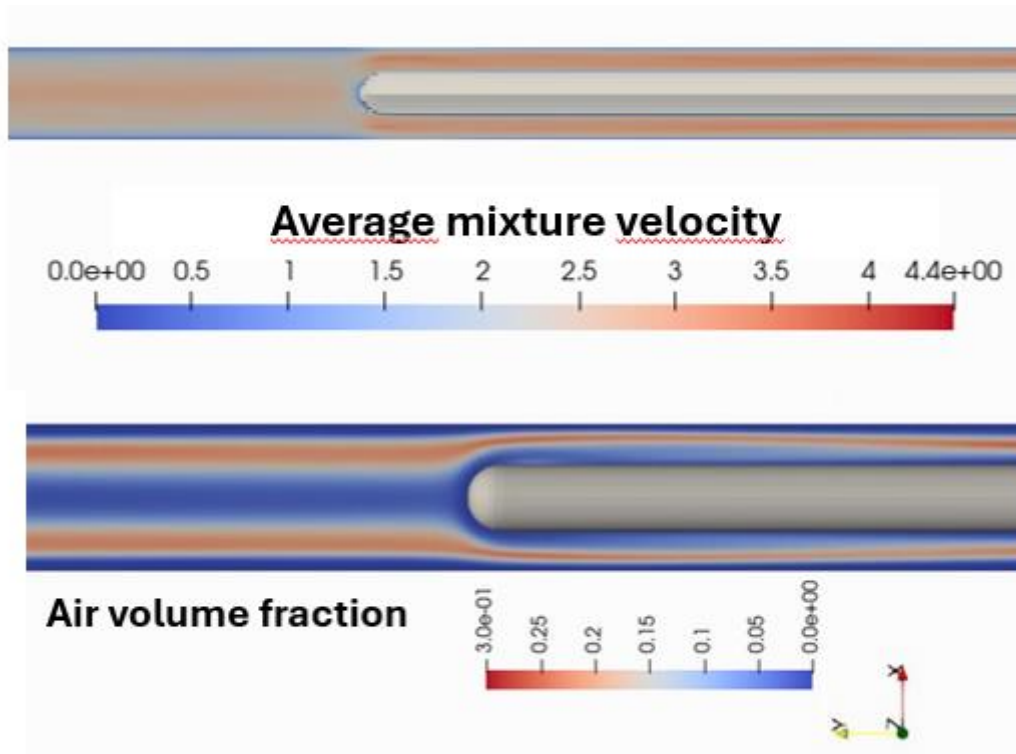


Figure 12. Average fields after 4 seconds in the EDF R&D simulations at $U_{gs} = 0.35$ m/s. (Top) Mixture velocity and (bottom) air volume fraction.

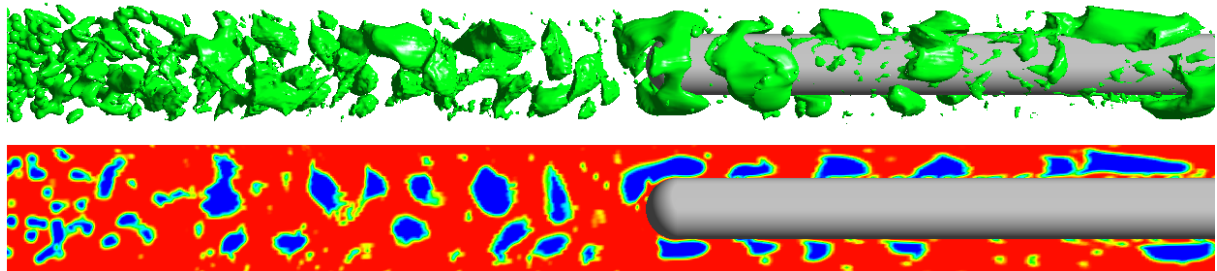


Figure 13. Bubble shapes and void fraction near the rod tip in VTT simulations with fixed geometry and inlet bubble diameter 2 mm ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).

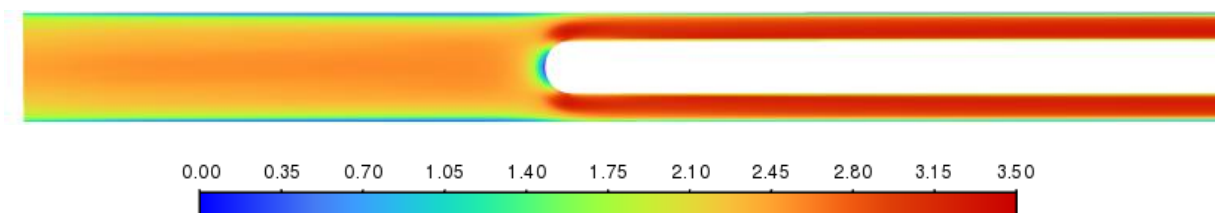


Figure 14. Time-averaged axial mixture velocity at the centre plane in VTT simulations with fixed geometry and inlet bubble diameter 2 mm ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).

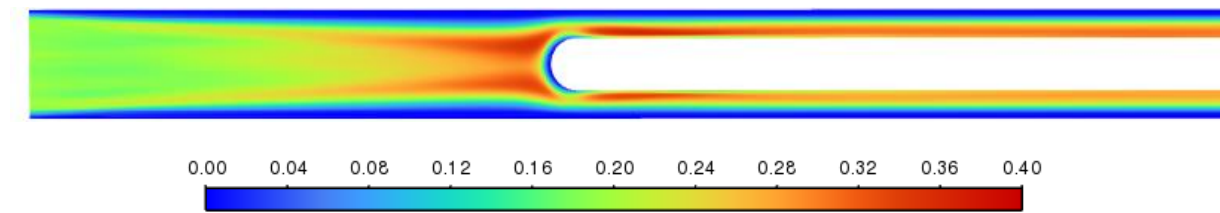


Figure 15. Time-averaged void fraction at the centre plane in VTT simulations with fixed geometry and inlet bubble diameter 2 mm ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).

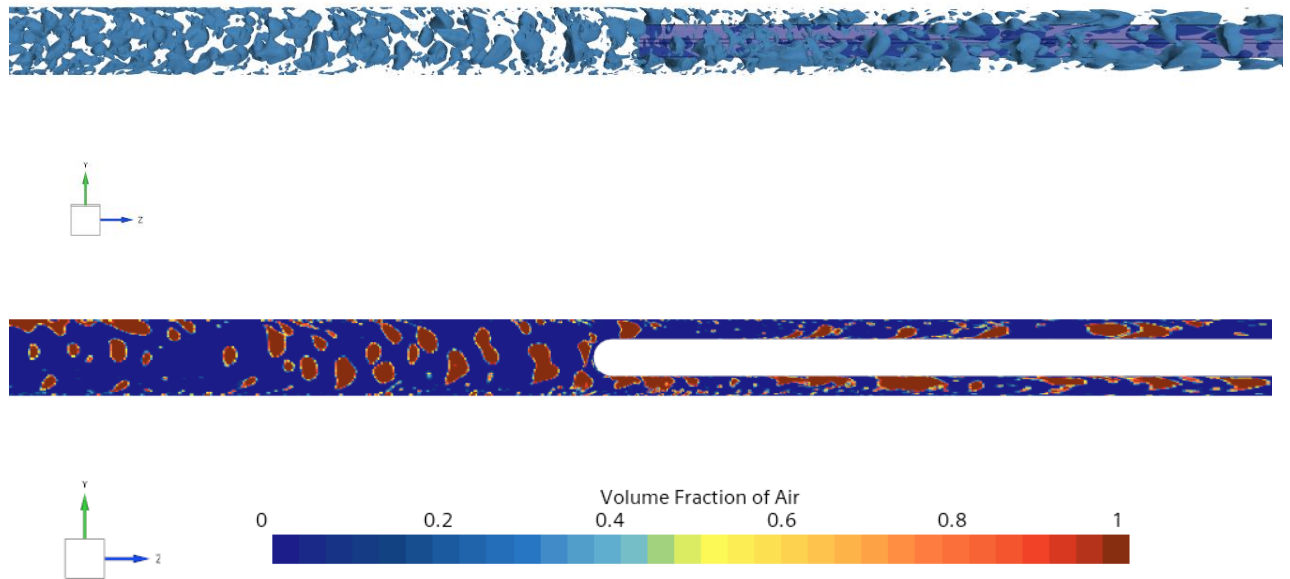


Figure 16: Bubble shapes and void fraction near the rod tip in the EDF UK simulations

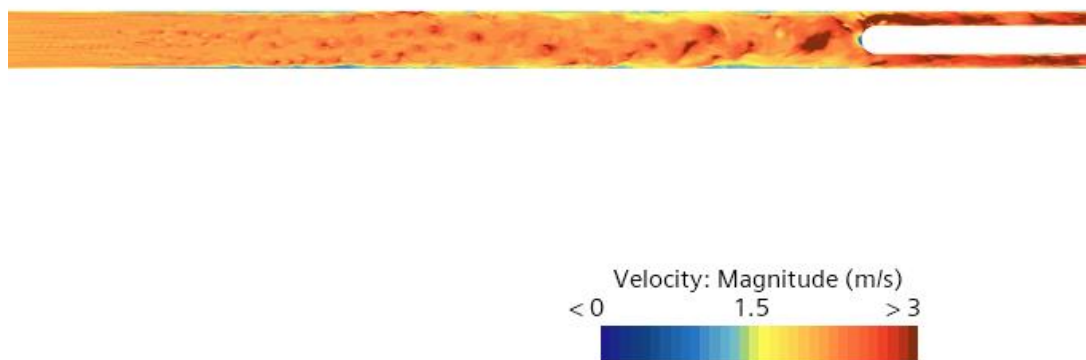


Figure 17 Mixture velocity magnitude at the centre plane in EDF UK simulations ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).

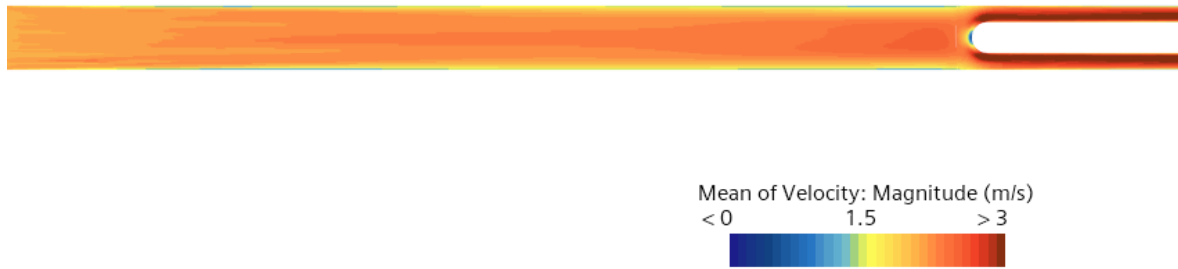


Figure 18 Time-averaged mixture velocity magnitude at the centre plane in EDF UK simulations ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).

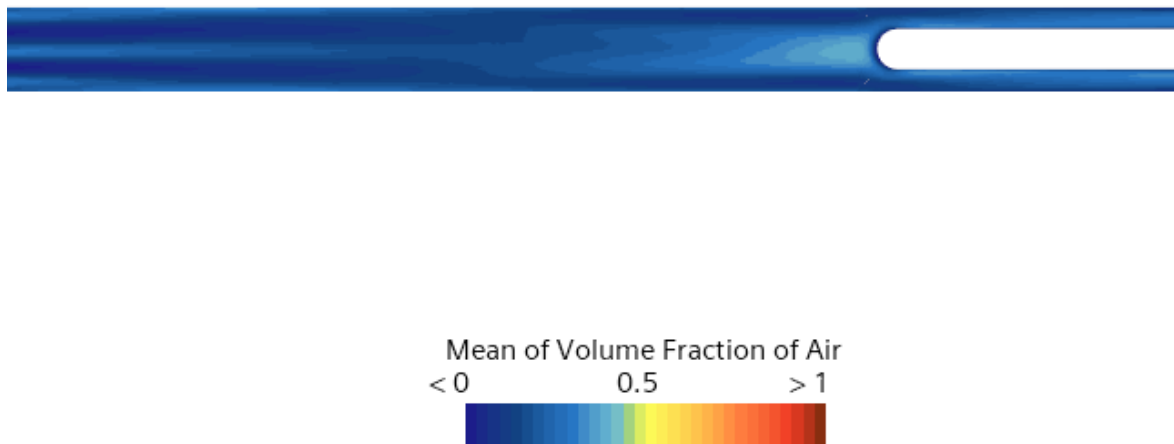


Figure 19 Time-averaged void fraction at the centre plane in EDF UK simulations ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).

5.2 Comparison in terms of tip displacements

For the case with $U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s, EDF R&D, EDF UK and VTT compared their tip displacement in terms of PSD. Numerical results appear different with:

- a better prediction of the amplitude around the main frequency for VTT and EDF UK simulations,
- A large peak at the main frequency for all participants with a slightly better predictions in terms of amplitude from EDF R&D.

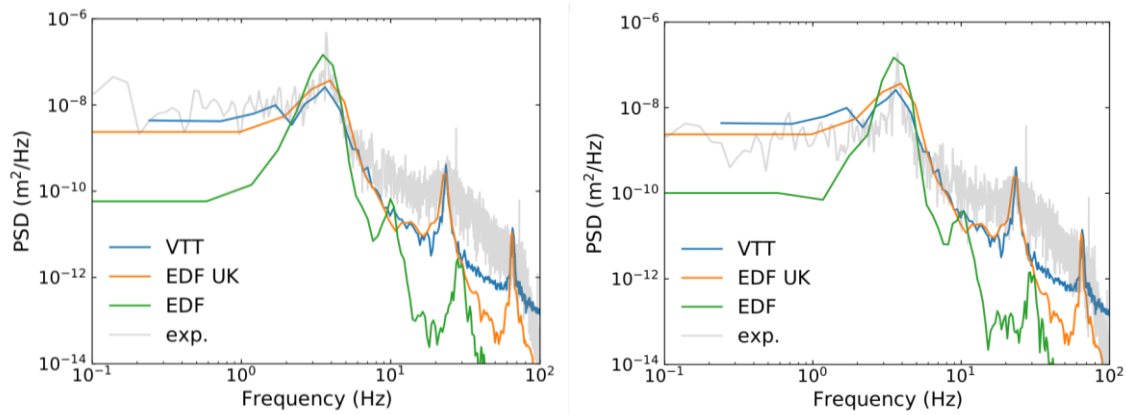


Figure 20 (left) front tip displacement PSD comparison, and (right) side tip displacement PSD comparison in the $U_{gs} = 0.35$ m/s and $U_{ls} = 1.7$ m/s case.

The difficulty with the present case is the absence of experimental bubble size characterization at the inlet. Depending on it, the simulation might predict a bubbly flow with small or large bubbles, even a slug flow. The present case, according to previous figures, appears to be a large bubbly flow. VTT did an interesting sensitivity analysis to highlight the significant role played by the inlet size distribution. In Table 7, we can see that for different inlet diameters, the RMS amplitude could change a lot.

Table 7. Rod tip radial RMS displacement obtained in VTT FSI simulations with different meshes and inlet bubble sizes ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).

Bubble diam. [mm]	RMS displacement [mm]	
	2.5 M mesh	6.2 M mesh
2	0.32	0.34
4	0.46	0.45
8	0.60	0.60
Experiment	0.41	

5.3 EDF R&D results - details

Simulations are carried out on a EDF R&D cluster for three different air superficial velocities $U_{gs} \in [0.16, 0.35, 0.79]$ m/s. In Figure 21, tip displacement RMS as a function of the inlet gas flux is given. In Figure 22, Figure 23 and Figure 24, the tip displacement along time from simulation (top) and experiment (bottom) are plotted for a gas superficial velocity of 0.16 m/s, 0.35 m/s and 0.79 m/s, respectively. These figures show the comparison between experimental (bottom graphs) and simulated (top graphs) displacement as a function of time. In the highest

gas flux case, the amplitude matches the experiment even though it seems that the variance is a bit lower in the simulation. For the two other gas fluxes, the amplitude is generally over-estimated, and the fluctuations are very small or non-existent. This was expected since simulations are carried out with a URANS model, effectively cancelling most of the finest velocity fluctuations. We also note that the phase delay between the two transverse directions is similar in the experiment and the simulation for the highest gas flux and it is not consistent in the two other cases. This may cause the difference in displacement amplitude that we observe at these gas fluxes.

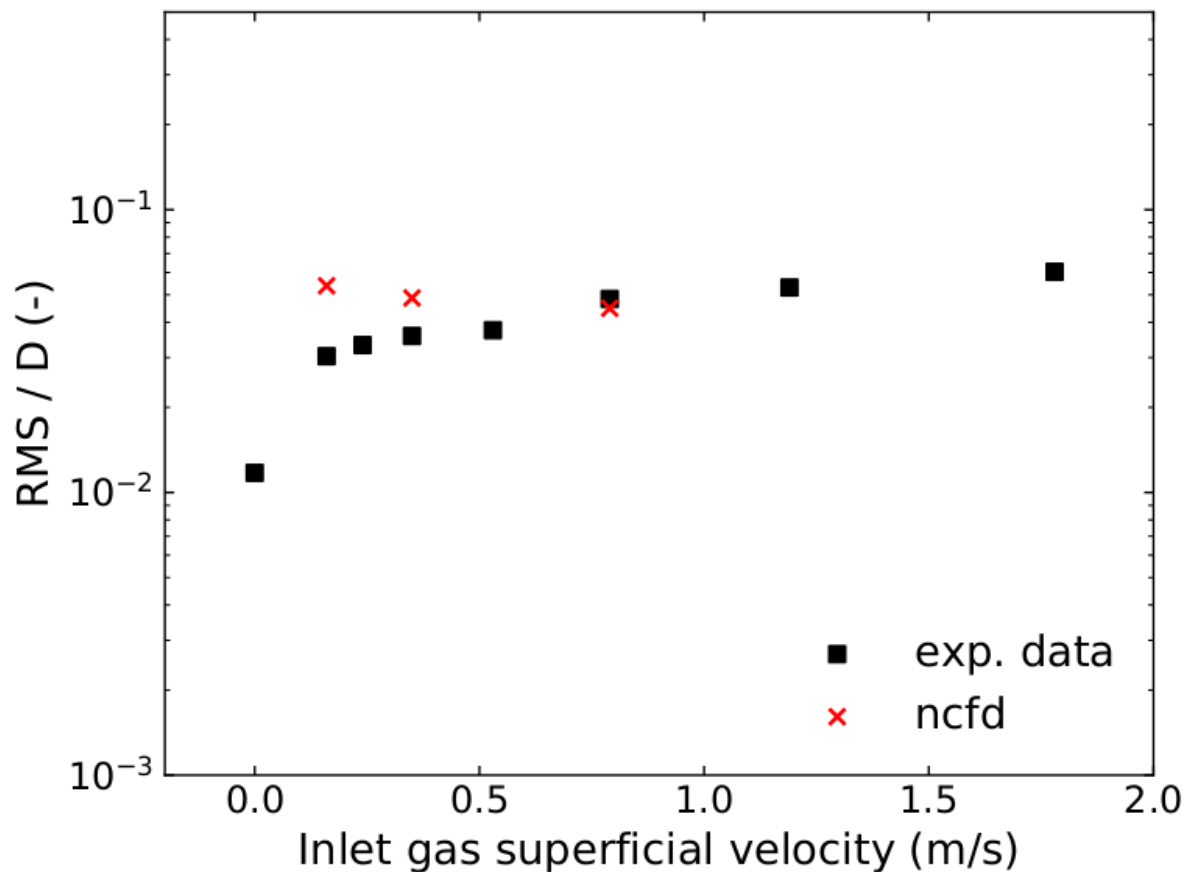


Figure 21 Tip displacement RMS as a function of inlet gas flux in the EDF R&D case.

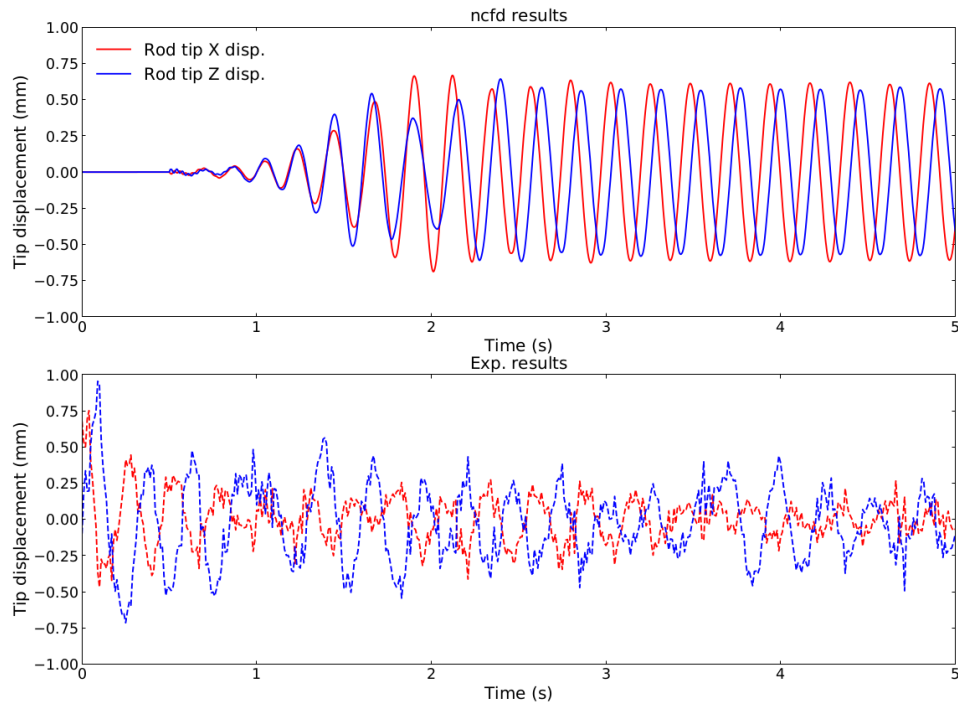


Figure 22 Tip displacement in the neptune_cfd simulations (top) and experiments (bottom) in the case $U_{gs} = 0.16$ m/s.

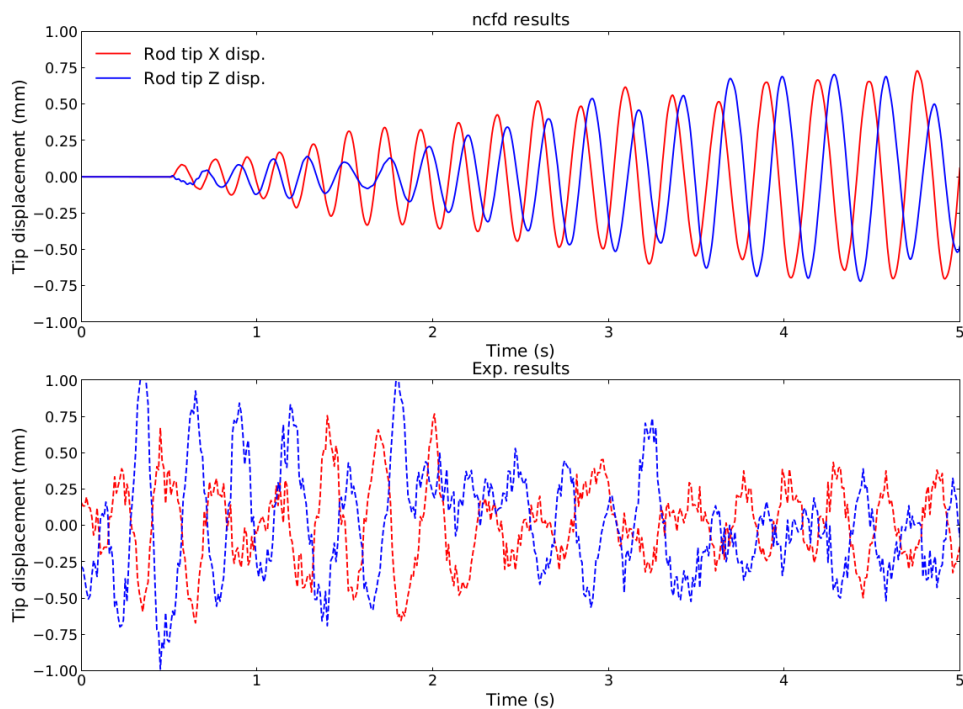


Figure 23 Tip displacement in the neptune_cfd simulations (top) and experiments (bottom) in the case $U_{gs} = 0.35$ m/s.

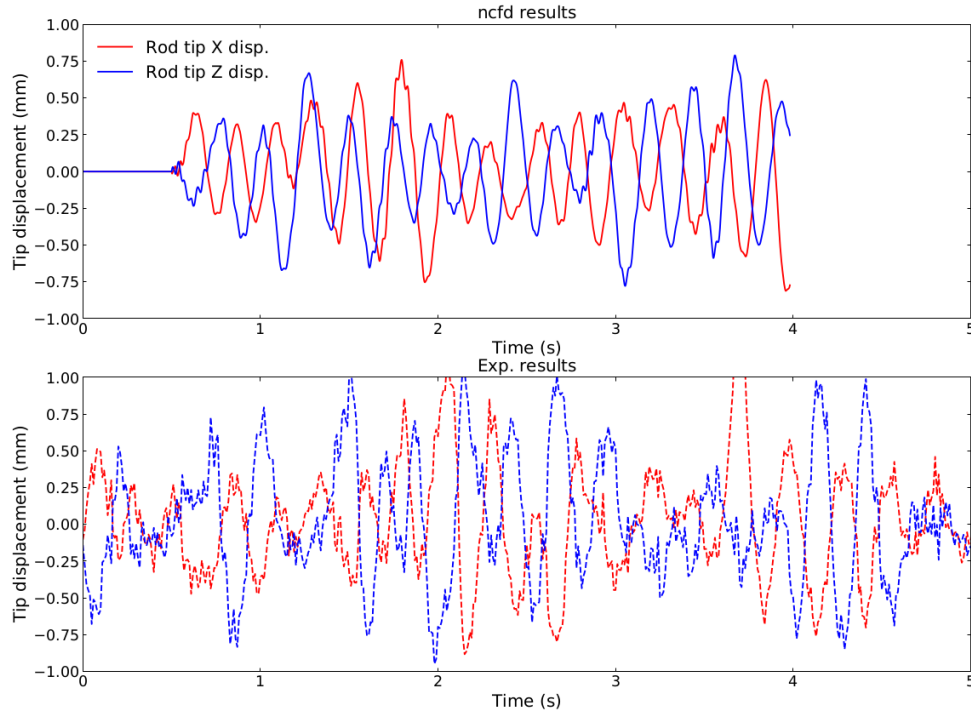


Figure 24 Tip displacement in the neptune_cfd simulations (top) and experiments (bottom) in the case $U_{gs} = 0.79$ m/s.

The comparison between the frequency spectrum is also mostly consistent even though the results are best at the highest gas flux, see Figure 25. We capture the first vibration harmonic of the cylinder and a bump in frequency amplitude around 27Hz. This bump's frequency seems a little overestimated compared to experiments. Even if the simulated frequency amplitudes are a little overestimated compared to the experiments, the results are satisfactory. In terms of RMS of amplitude as a function of gas flux, it seems that the smaller the gas flux, the larger the difference with the experimental results. This is probably because, as the volume fraction of gas decreases, the beam deformation becomes driven by liquid turbulent fluctuations, which are not captured correctly. In our simulations with neptune_cfd, we manage to better capture the beam deformation when it is mainly driven by impacting air pockets.

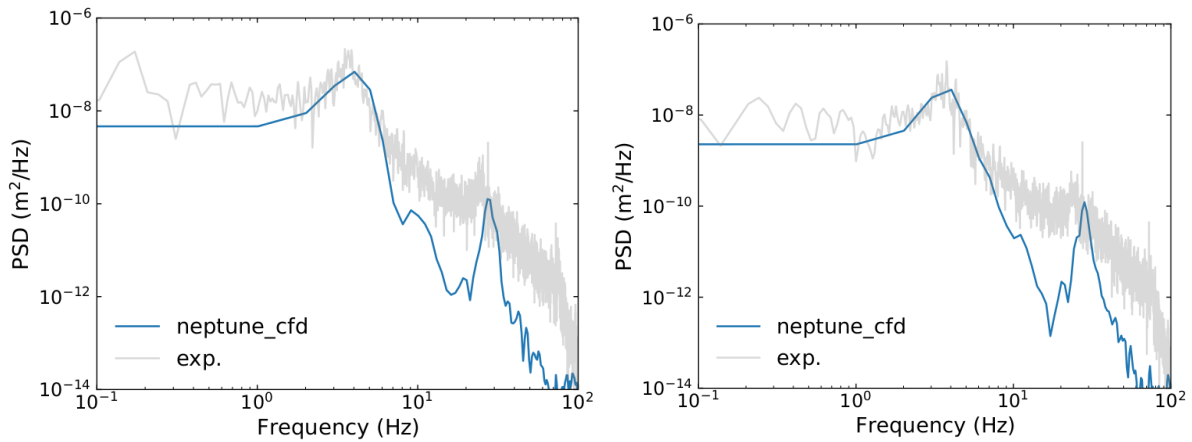


Figure 25 (left) front tip displacement PSD comparison, and (right) side tip displacement PSD comparison in the $U_{gs} = 0.79$ m/s and $U_{ls} = 1.7$ m/s case.

We also show in Figure 26 the displacement of the tip in the plane perpendicular to the beam. This allows us to see the difference in behaviour for the three fluxes simulated. In the smallest gas flux case, the beam turns around its equilibrium position rapidly at a fixed distance. In the case $U_{gs} = 0.35$ m/s, the beam also turns around its equilibrium, and the distance to the center slowly increases. In the highest gas flux case, the behavior is more erratic. This also confirms that velocity fluctuations are more pronounced in this last case.

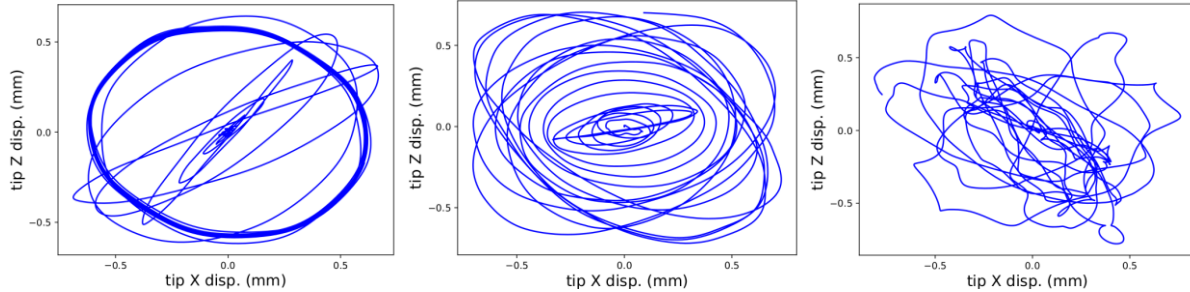


Figure 26 Phase graphs for the three inlet fluxes simulated with neptune_cfd.

At the end, it appears these results are considered as satisfactory considering the absence of experimental bubble size distribution or void fraction profile at the inlet to well define inlet conditions numerically.

5.4 EDF UK results – details

Visually, the amplitude of vibration in both front and side directions appear similar to the experimental results, see Figure 27 and Figure 28. This is confirmed by the RMS displacement values, which are 0.1968 mm and 0.2724 mm, for the front and side directions, respectively. This corresponds to errors of -13.6% and -24.9% relative to the experimental values of 0.2278mm and 0.3628mm. Note that the rms values were calculated after subtracting the mean, since the steady state displacement of the experiment would otherwise lead to an unfair comparison.

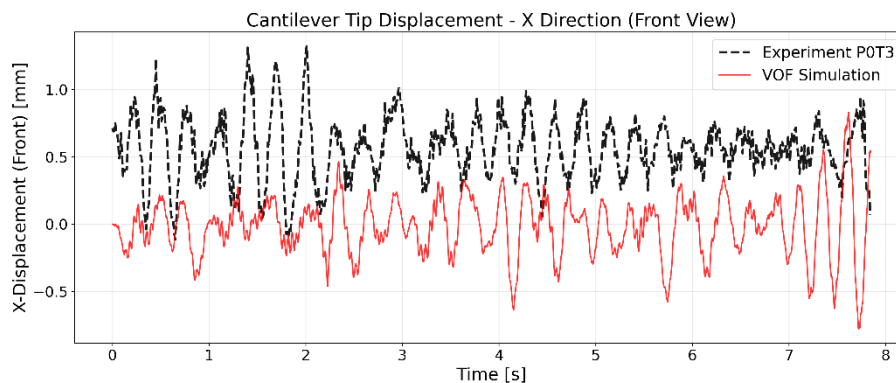


Figure 27: Time series graph of displacement in X

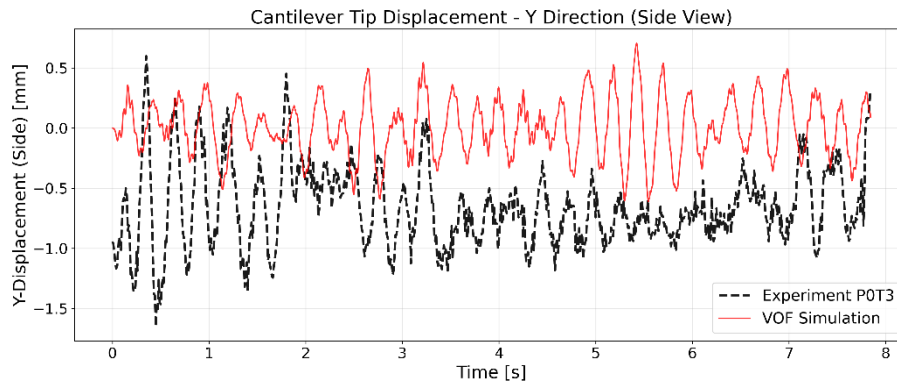


Figure 28: Time series graph of displacement in Y

Figure 29 and Figure 30 show peaks in the spectrum that match the natural frequencies of theory and experiment. The corresponding frequencies are 3.64 Hz for the first mode and around 23 Hz for the second mode. The peaks are broader and flatter than the experiment, which may be due to the shorter sequence length for the simulation, 8 seconds, compared to the experiment, 60 seconds. PSD magnitude is again similar at lower frequencies, but the error widens at higher frequencies, which is expected.

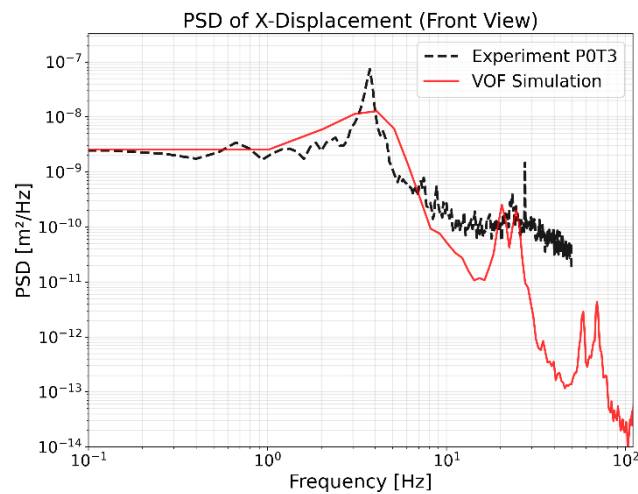


Figure 29: Power Spectral Density graph of displacement in X

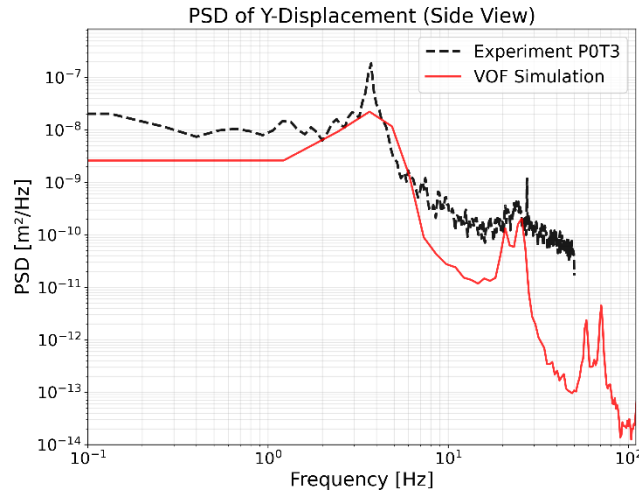


Figure 30: Power Spectral Density graph of displacement in Y

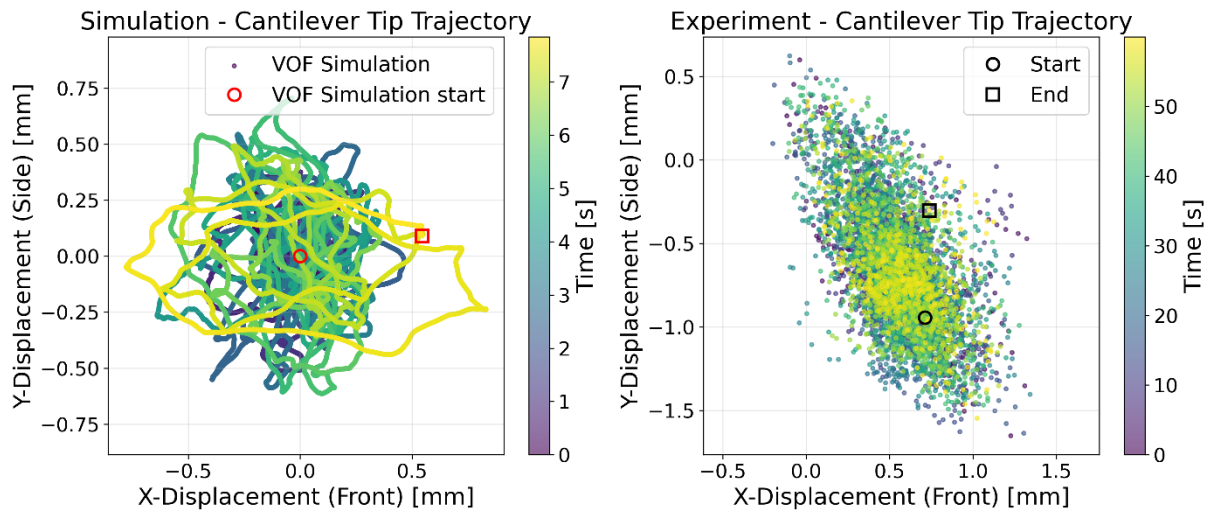


Figure 31: Tip trajectory - simulation vs experiment

The tip displacement trajectories, Figure 31, are compared for completeness. Overall, we see that the chosen method faithfully reproduces both the dominant frequencies and the amplitudes within 25%.

5.5 VTT results - details

Figure 32 shows development of the two-phase flow as iso-surfaces and as void fraction at the centre plane when using inlet bubble diameters of 2 mm, 4 mm and 8 mm. In all three cases, the bubbles start to merge after the inlet and larger bubbles are obtained already at the rod tip. At the same time, breakup of the bubbles leads also to small sizes which are not accurately resolved by the mesh. The rod tip displacement spectra obtained from different FSI simulations are compared with the experiment in Figure 33, whereas the displacement time signals for the fine mesh and inlet bubble diameter of 2 mm are compared with the experiment in Figure 34. The first peak at about 3.7 Hz due to the first vibration mode of the rod is approximately captured by the simulations, although the experimental peak is sharper. Also, the second peak at about 23 Hz is shown in the simulations, but at the higher frequencies

the simulated displacement amplitudes are clearly too low. This is shown also in the time histories of Figure 34, where the low-frequency oscillations are somewhat correctly represented while the experiment shows more high-frequency content. It is unclear whether the underprediction at higher frequencies is due to too high rod damping or too low flow loads at the high frequencies. It is seen that the underprediction of the spectral power worsens with increasing frequency. The spectral content of the forces at the rod tip with fixed geometry shows that the dominant frequency of the forces is around 100 Hz (determined by the flow velocity and size of the largest bubbles), which is high compared to the two lowest rod eigenmodes.

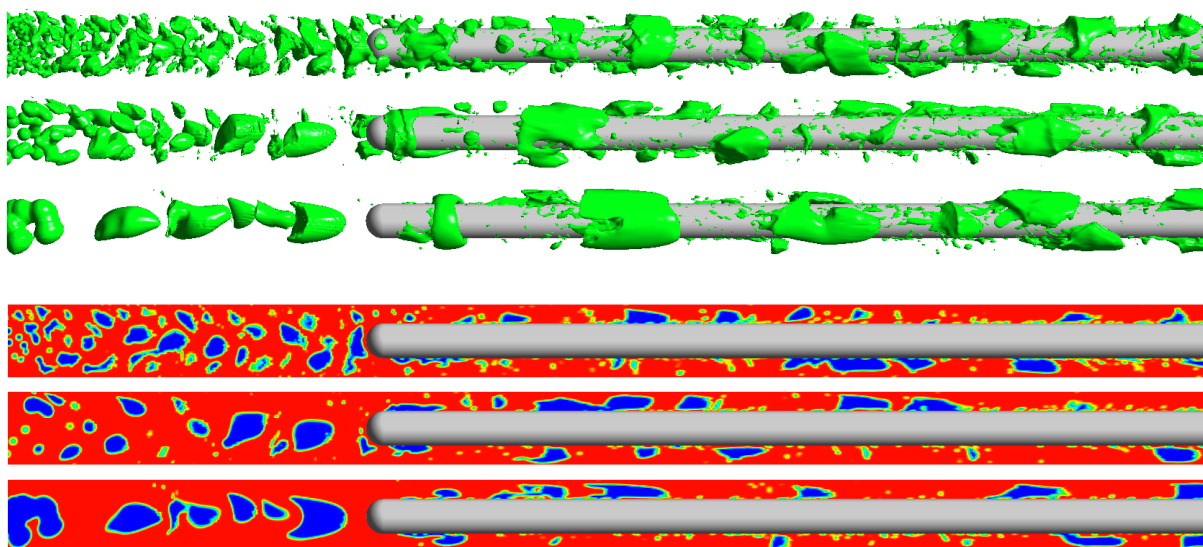


Figure 32 Bubble shapes and centre plane void fraction with inlet bubble diameters of 2 mm, 4 mm and 8 mm (from top to bottom) when using the fine mesh ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).

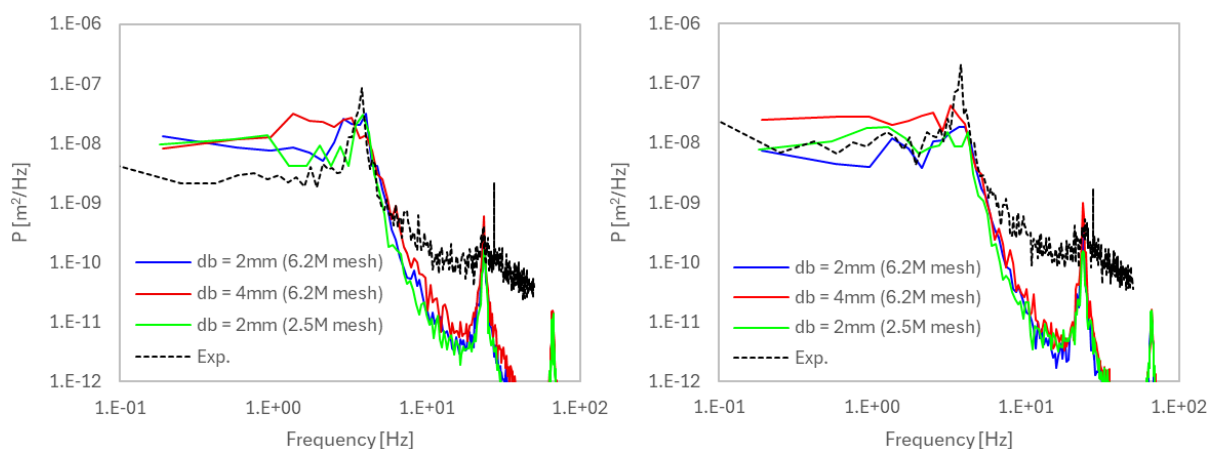


Figure 33 Tip front (left) and side (right) displacement spectra obtained with different meshes and inlet bubble sizes ($U_{ls} = 1.7$ m/s, $U_{gs} = 0.35$ m/s).

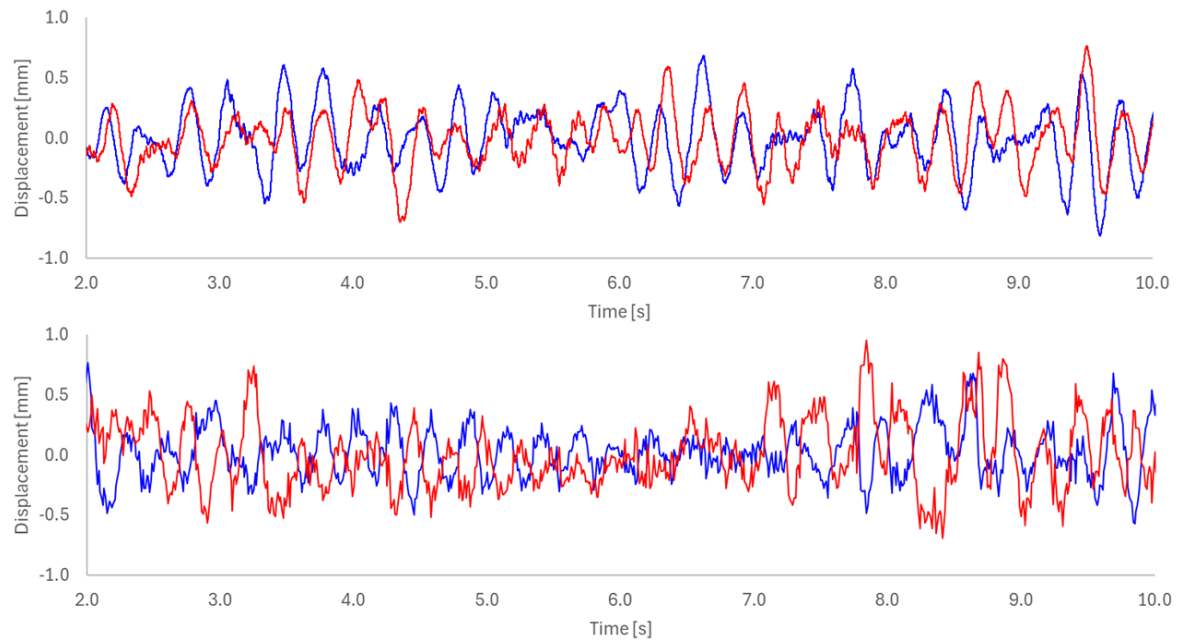


Figure 34 Tip displacements with fine mesh and inlet bubble diameter 2 mm (top) and in experiment (bottom). Note that the total simulation sampling time is about 25 s.

Conclusion

This report summarises the work carried out by EDF UK, EDF R&D and VTT on a multiphase axial fluid-structure deformation benchmark. The three partners provided simulation results for a specific air and water inlet mass flux and compared their flow and vibration results with the experimental data and the other participants. EDF UK and VTT used a VoF framework within StarCCM+ and Ansys-Fluent, respectively, while EDF R&D used neptune_cfd and its multi-Eulerian approach. VTT was the only participant to use LES, the other groups worked within the URANS framework, either with the k-omega SST model (EDF UK) or the second order Reynolds stress model (EDF R&D). All groups opted for a finite element modelling for the solid part and two-way coupling. One-way coupling only does not capture the instabilities and tends to yield incorrect rod vibration amplitudes. All in all, results obtained by the three partners are generally satisfactory. Exploratory simulations for different inlet gas fluxes have been conducted by EDF R&D showing the tendency of rod vibration prediction according to inlet gas flux. Some minor differences between the simulations of the three partners can be observed in the variable fields, these have been pointed out and linked to modelling choices each participant made. The three partners agree on the fact that the inlet boundary condition represented the key modelling choice of the simulation. Defining it without much experimental data has been the main challenge. To improve the simulation accuracy, having more details on the experimental inlet conditions would be necessary. In the present case, the displacement was mainly driven by larger gas structures, hence it was necessary to resolve them explicitly to catch their effect in terms of load. Thus, meshes were built in order to be able to track the gas structures in the VOF or multi-Eulerian approaches, consequently the inlet gas size distribution would have helped since it is directly related to displacement and the required mesh refinement. In terms of turbulence, URANS and LES simulations were performed, it appears that the main displacement is not driven by turbulence but by the instantaneous resolved liquid/gas interfaces. Multi-Eulerian simulations show, similarly to URANS single phase simulation, that the fluctuation when dispersed bubbles are modelled is very low. These conclusion in terms of two-phase models are similar in the other task of the WP4.

Bibliography

Baraglia, F., Benguigui, W. and Denèfle, R., 2021. "A corotational finite element approach coupled to a discrete forcing method to solve hyperelastic deformation induced by two-phase flow". *Journal of Fluids and Structures*, Vol. 107, p.103403.

Baraglia, F., Laviéville, J., Méricoux, N. and Simonin, O., 2024. "N-euler mathematical modelling and numerical simulation of liquid–gas–solid flows". *Nuclear Engineering and Design*, Vol. 418, p. 112886. ISSN 0029-5493.

Benguigui, W., Doradoux, A., Lavieville, J., Mimoumi, S. and Longatte, E., 2018. "A discrete forcing method dedicated to moving bodies in two-phase flow". *International journal of numerical methods in fluids*, pp. 1–19.

Benguigui, W., Laviéville, J. and Merigoux, N., 2019. "Fluid-structure interaction in two-phase flow using a discrete forcing method". *International journal of numerical methods in fluids*, Vol. 91, pp. 246–247

Cioncolini A., Zhang S., Nabawy M.R.A., Li H., Cooper D., Iacovides H., Experiments on Axial-Flow-Induced Vibration of a Cantilever Rod in Two-phase Flow, *Annals of Nuclear Energy*, 190, 2023

Cook R., Malkus D., Plesha M. and Witt R., 2002. *Concepts and applications of finite element analysis*. Wiley & Sons, Inc.

Li H., Cioncolini A., Iacovides H., Benguigui W., Nabawy M.R.A, Unveiling the fundamentals of two-phase axial-flow-induced vibrations of cantilever rods, *Sci Rep* 16, 5102 (2026).

Merigoux N., "Multiphase Eulerian-Eulerian CFD supporting the nuclear safety demonstration ". *Nuclear Engineering and Design*, 2022, Volume 397, , 111914.