Ref. Ares(2026)6212827 - 18/06/2026

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

Best practices for the analysis of FIV in twophase flow

Authors : William BENGUIGUI (EDF)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	Best practices for the analysis of FIV in twophase flow
Author(s)	William BENGUIGUI
Number of pages	16
Document type	Deliverable
Work Package	WP4
Document number	D4.7
Issued by	EDF
Date of completion	2026-05-06 22:50:50
Dissemination level	Public

Summary

Recommendations for best practices for simulations of FIV of tubes subjected to two-phase flows.

Approval

Date	By
2026-05-06 22:52:35	William BENGUIGUI (EDF)
2026-05-07 09:47:41	Papukchiev ANGEL (GRS)

GO VIKING

D4.7 Best practices for the analysis of FIV in two-phase flow

Version N°2
May/2026

William Benguigui[1,2], Federico Baraglia, Daniele Vivaldi[3], Lucas Rotily[3], Sina Tajfirooz[4], Kevin Zwijsen[4], Constantinos Katsamis[5], Ehimen Iyamabo[5] , Antti Timperi[6] , Clément Bazin[7]

[1]EDF R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatou, France
[2]IMSIA – UMR 9219 EDF/ENSTA/CNRS – 7 boulevard Gaspard Monge 91120 Palaiseau, France
[3]ASNR, Cadarache, St Paul-lez-Durance, BP3, 13115, France
[4]NRG, Westerduinweg 3, 1755 LE Petten, The Netherlands
[5]EDF Energy R&D UK, 90 Whitfield Street, London W1T 4EZ, United Kingdom
[6]VTT Technical Research Centre of Finland, Kivimiehentie 3, Espoo, 02044 VTT, Finland
[7]CEA, CEA Site de Saclay 91191 Gif-sur-Yvette, France, France

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; Kevin Zwijsen, kevin.zwijsen@nrgpallas.com
Project starting date (duration)	1st June 2022 – 31st May 2026 (48 Months)
Related Work Package	WP4 – Multiphase flow
Related Task(s)	D4.1, D4.2, D4.3, D4.4, D4.5, D4.6
Lead Organisation	EDF
Contributing Partner(s)	EDF, ASNR, NRG, VTT, CEA, EDF UK
Due Date	31.01.2026
Submission Date	06.05.2026
Dissemination level	PU

History

Date	Version	Submitted by	Reviewed by	Comments
21.04.2026	1	W. Benguigui	A. Papukchiev, K. Zwijsen	
06.05.2026	2	W.Benguigui		

Table of contents

1. Description of the experiments 6
1.1 Cantilever Rod in two-phase flow 6
1.2 TREFLE, tube bundle in square arrangement 7
1.3 TITAN, tube bundle in rotated triangular arrangement 8
2. Feedback from partners and Best Practice Guidelines 9
2.1 ASNR 9
2.2 CEA 10
2.2.1 Physical Modeling Strategy 10
2.2.2 Numerical Discretization 10
2.2.3 Structural Response and Spectral Analysis 10
2.3 VTT 11
2.4 EDF R&D 12
2.5 EDF UK 13
2.5.1 Two-phase modelling and Interface Capturing 13
2.5.2 2. Two-phase cantilever-rod case 14
2.5.3 TREFLE/TITAN Two-Phase FSI Cases 14
2.6 NRG PALLAS 15
3. Conclusions and Perspectives 16

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 (UoM reference for the pictures) 6

Figure 2 Sketch of the TREFLE experiment 7

Figure 3 Sketch of Titan Facility 8

Abbreviations and Acronyms

Acronym	description
BPG	Best Practice Guidelines
CFD	Computational Fluid Dynamic
FIV	Flow Induced Vibration
GLIM	Generalized Large Interface Model
GO-VIKING	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
HRIC	High-Resolution Interface Capturing
LES	Large Eddy Simulation
URANS	Unsteady Reynolds Averaged Navier Stokes
VOF	Volume Of Fluid
WP	Work package

Summary

The present document highlights the feedback and best practices from each partner of the WP4 on the 3 different simulated experiments.

Keywords

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

Introduction

To finalise the contributions of the WP4, each partner proposes its own feedback and best practice guidelines based on the work realized on the 3 different experiments – Cantilever Rod, TREFLE, TITAN. First, the experiments are summed up to highlight the important information available to simulate each of them. Then, each partner from WP4 highlights its lessons learned and BPGs. Finally, a conclusion is given highlighting:

- the important role played by two-phase characterization, especially at the domain inlet to simulate/validate the CFD tools.
- simulation approaches like Reynolds-Averaged Navier-Stokes (RANS) approaches for single phase flow, appear not to capture the amplitude and fluctuations of forces.
- the interest to use resolving approaches to predict accurately forces acting on structures. Most of the results with large gas structures were well predicted by the partners using resolved or hybrid approaches.

1. Description of the experiments

1.1 Cantilever Rod in two-phase flow

The experiments 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, 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.

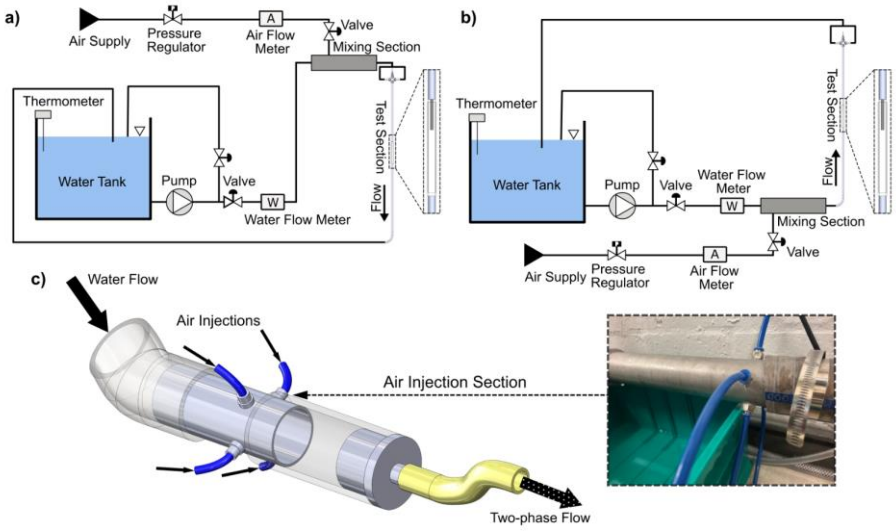

Figure 1: Test rig for the two-phase flow experiments: schematics for a) flow loop of the clamped-free configuration; b) flow loop of the free-clamped configuration; c) mixing

section design with inset showing the real components for the air injection section (UoM reference for the pictures)

The air injection system is connected to the main flow loop via a mixing section, Fig. 1c. 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 Ugs varying between 0-1.78 m/s. 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. 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 (UoM reference).

Thus, for the present numerical work, mass flow rates at the inlet are the inlet conditions, and the tip vibration is the result. Additionally, PIV measurements for low void fraction were performed at the end for a single Reynolds number, but different from the numerical benchmark.

1.2 TREFLE, tube bundle in square arrangement

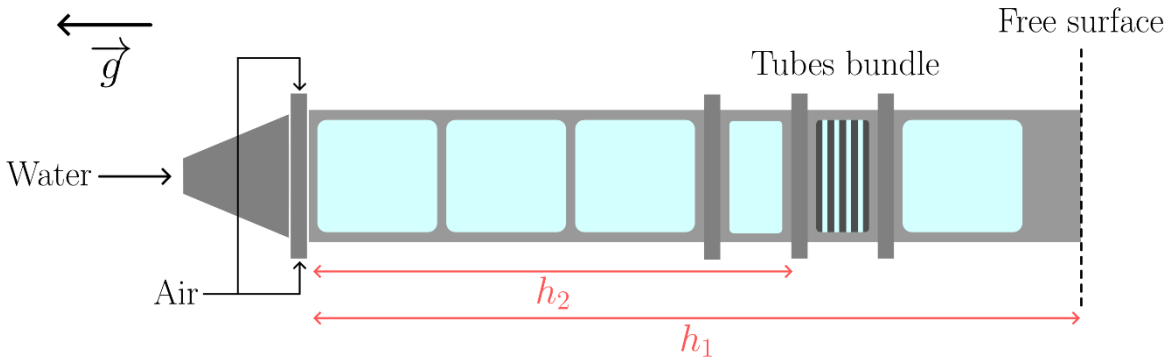

Figure 2 Sketch of the TREFLE experiment.

The TREFLE facility is designed to study the effect of a two-phase water/air flow on a 5x5 tube array, with the aim of investigating, at reduced scale, fluid-structure interaction phenomena similar to those present in a steam generator. The figure above schematizes the experiment (a more complete sketch may be found in deliverable D4.4 of Go-Viking project).

A water circulation is established from bottom to top with a hydraulic pump. The water is channeled at the base by a convergent designed to ensure a homogeneous flow in the test section. The water flow rate is kept constant for the different experiments at the facility maximum flow rate, namely 2500 g/s. The air is injected, with a flow rate starting from 0.4 g/s to about 50g/s, from the bottom of the section after the convergent through seven adjustable-size injectors. Two types of injectors were used: one type to produce millimeter-sized bubbles, simulating a bubbly flow regime; and the second type for flow regimes with large void fractions. The first type of injectors have an outer diameter (O.D.) of 15 mm, with each pipe containing 32 holes (1 mm diameter) placed diametrically opposed on opposite sides of the tubes, while the second type injectors have a pipe O.D. of 25 mm, with 60 holes per pipe (5 mm diameter) arranged in three rows of 20 holes. The total height of the water column measures approximately 3.5 meters (h_1 in the figure above), from the injection to the free

surface, maintained flat and at constant level by a gutter system around the upper outlet of the rectangular column. The facility has a large flow-homogenization zone of approximately 1.5 m between the air injection and the obstacle zone (h_2 in the figure above). Different windows allow for flow visualization at different heights. The test section of the facility contains an in-line, square pitch 5 × 5 tube bundle of fixed (i.e. non vibrating) tubes with pitch-to-diameter ratio (P/D) of 1.44. The tube length is 10 times the tube diameter, which measures 30 mm. Different experiments have previously enabled the quantification of the flow for different air flow rates, and different air injectors, through: a wire mesh sensor, a dual optical probe, and a high-speed camera. More recently, the forces experienced by the central tube are measured using two force sensors.

1.3 TITAN, tube bundle in rotated triangular arrangement

The experimental rig TITAN is located in the CEA/SEMT/DYN laboratory and is used to perform FIV tests on a rotated triangular tube bundle subjected to a two-phase air-water flow (Figure 3). TITAN is connected to the air-water loop of the RESEDA platform, which allows to reproduce flow conditions typically found in nuclear steam generators.

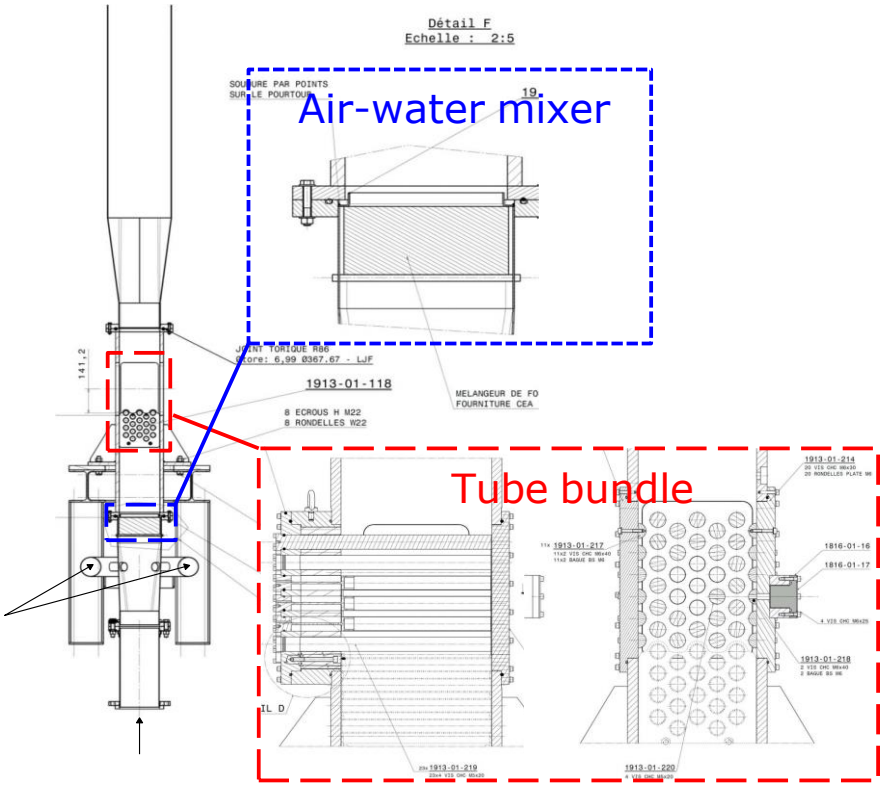

Figure 3 Sketch of Titan Facility

The two-phase flow loop includes a water centrifugal pump with maximum delivery capacity of 1100 m3/h and an air compressor which delivers a maximum flow rate of 3000 m3/h (at 273.15 K, 101325 Pa). The bundle consists of 30 stainless steel tubes, with two columns of half-tubes at both lateral plates. An additional bundle of 20 tubes is placed upstream the test section to regulate the inlet flow direction. The tubes have an outer diameter D =30.0 mm, thickness of 0.4 mm and are arranged in a rotated triangular pattern with a pitch ratio of 1.44 and an aspect ratio of 10.

For the fully rigid configuration, forces were measured on a single cylinder. To measure local quantities, a bi-optical probe was introduced, its tip is positioned in the middle of a canal with an uncertainty on the lateral position estimated to $\pm$ 0.5 mm.

In this experimental study, two flexible configurations were tested. The two configurations were subjected to increasing mass flow rates until the vibration amplitude was sufficiently high to indicate fluid-elastic instability. During a given test, the void fraction was kept constant.

For further details, please refer to the experimental deliverable of TITAN: https://go-viking.eu/images/resources/GO_VIKING_D4_5_titan_experiment_presentation_of_the_data_v2.pdf.

2. Feedback from partners and Best Practice Guidelines

2.1 ASNR

Starting from homogeneous void fractions higher than 50%, standard Eulerian two-fluid approaches based on dispersed flow closure laws fail to predict consistent results. This is expected, since increasing the void fraction and switching from bubbly to intermittent patterns the two-phase flow features a larger spectrum of gas-liquid interface sizes. So-called all-flow regime Eulerian approaches are required to correctly simulate such two-phase patterns.

The correct prediction of the force spectra of TITAN seems linked to the capability of the simulation to track and “resolve” large gas-liquid interfaces. Such large gas structures seem to rise inside the tube bundle region (across the tubes) without significantly breaking up. Numerical sets-up including compression methods for the void fraction seem to help the correct prediction of such intermittent flows. For some conditions, one may also wonder if the dispersed flow modeling has a significant impact on the results: results calculated by the AIAD do not seem to show an important sensitivity whether a constant bubble diameter or a population balance is used (provided that the constant value is chosen within consistent values).

Inlet boundary conditions play a major role in configurations with a limited number of tube rows, such as TREFLE. The flow pattern crossing the tube bundle region is mainly affected by the pattern found upstream. In this case, the simulation of the flow upstream of the bundle becomes fundamental: this means simulating as close as possible to the experimental configuration the gas-liquid injection and mixing, and the flow evolution between the injection/mixing position to the tube bundle. TREFLE results seem significantly influenced by these aspects. On the other hand, TITAN results seem overall significantly less affected by the numerical inlet boundary conditions employed: this because TITAN contains 10 rows of tubes, allowing sufficient flow development from the inlet to the outlet of the tube bundle.

The consideration that large gas-liquid interface tracking and resolution plays the primary role in terms of a consistent numerical prediction of the flow is also supported by the observation that one-fluid VOF simulations showed good results, in terms of both void fraction profiles

and force/vibration spectra, for both TITAN and TREFLE. The VOF simulations used mesh refinements unable to correctly track the smallest bubbles, but indeed allowed a precise simulation of the larger interfaces.

FSI simulations were run in the two-phase turbulence- (or buffeting-) induced force regime (meaning not in the fluid-elastic response regime). In this regime, the fluid-structure coupling was observed to converge after very few sub-iterations within each time step: at the second sub-iteration, the evolution of the tube displacement (compared to the first sub-iteration) was practically negligible. Of course, this behavior applies for a numerical time-step low enough compared to the cylinder natural period and the two-phase flow characteristic times. The time-step size employed ranged from 5e-5s to 1e-4s, depending on the simulating flow rates.

2.2 CEA

2.2.1 Physical Modeling Strategy

A precise identification of the flow regime (bubbly, slug, churn, annular) is essential, as itdetermines both the amplitude and the nature of excitation (single-phase turbulence vs. two phase flow-induced fluctuations). In low liquid superficial velocity regimes, fluctuations are often dominated by bubble dynamics rather than classical turbulence. In such cases, Euler-Euler averaged approaches may fail because bubble-induced agitation is not properly captured and two-phase turbulence models are not yet fully mature. If the liquid velocity is too low, avoid averaged models and favor interface-resolving instead. Monitoring only the void fraction is insufficient. The relevant conserved quantity is the phase flux (e.g. αU_g). A correct void fraction with inconsistent velocities is not physically reliable. Simple 1D calculations are highly recommended to check whether the experiment is close to equilibrium and provide order-of-magnitude estimates for the simulation.

2.2.2 Numerical Discretization

Mesh refinement is critical near the structure and in regions of strong void fraction gradients. Poor resolution leads to inaccurate force predictions and vibration amplitudes. Pressure being a non-local quantity, upstream mesh quality is also essential.

- Ensure a well-resolved mesh at least 5 characteristic lengths upstream
- Avoid abrupt mesh transitions and use progressive refinement

Without proper upstream resolution, structural loading predictions are unreliable. Grid convergence studies are mandatory. Without them, quantitative conclusions are not defensible.

2.2.3 Structural Response and Spectral Analysis

Spectral analysis is highly sensitive to time step, total simulation time, windowing method, sampling frequency.

Then a meaningful comparison between experimental and numerical spectra requires strict consistency: same sampling frequency, time window, windowing technique and cut-off frequency. If experimental data include a cut-off frequency, the numerical signal must be processed identically (including re-interpolation if needed). Spectra represent distributions of integrated quantities. Agreement can be artificially improved through post-processing choices. Always align postprocessing methodologies before drawing physical conclusions. Comparing spectra obtained with different processing is equivalent to comparing quantities expressed in different functional bases.

2.3 VTT

Explicit FSI coupling, where the CFD load and structural displacement data is exchanged only once in time step, can become numerically unstable. Parameters promoting the instability are e.g. low structure to fluid mass ratio and high structural flexibility. The numerical instability can be usually cured by using iterative implicit FSI coupling, however the implicit method may be computationally expensive and with some codes it might not be available. For explicit coupling, using smaller time steps typically improves stability. Adding realistic Rayleigh damping to the structure at the frequency range of interest may also prevent high-frequency numerical instabilities. The stiffness proportional contribution of the Rayleigh damping damps effectively very high frequencies. The frequency of the numerical instability corresponds often to the time step size, and is thus typically much higher than the structural eigen frequencies of interest. The stiffness proportional damping grows linearly with frequency, and thus a fairly low damping value may be sufficient for stabilization when small time steps are applied. Furthermore, for the flexible rod in axial flow simulated in GO-VIKING, the instability with explicit coupling showed later in the simulation when monitoring only the structural displacements. However, monitoring the structural acceleration signals can reveal the instability much earlier; this can be done also by numerically differentiating the displacement signal.

Generally, FSI calculations of vibration problems require longer sampling times for converged statistics (e.g. mean and RMS) compared to the flow field with fixed geometry. This is especially true if a reliable structural fatigue estimation is needed. This depends to a large extent on the structural eigen frequencies, and the computing time requirements can become large when the time scale of the structural motion is long compared to the relevant flow time scales. Furthermore, for ALE simulations smoothing the internal CFD mesh at each time step (sometimes called mesh morphing) can be expensive, so that the computing time may grow e.g. by a factor of 3-4 compared to fixed geometry (although this depends on the mesh smoothing method). It is thus advisable to deform the mesh only at regions where required and to optimize the smoothing algorithm settings for faster computations (e.g. number of iterations for diffusion-based methods). When taking into account all of the factors which add to the computational load, it can be said that two-way FSI simulations of vibrations issues may become very expensive compared to just obtaining flow field statistics on fixed geometry. Furthermore, the simulation setup becomes more demanding. Thus, it is also worthwhile to pursue one-way coupled FSI calculations with added fluid mass and flow-induced damping if the case allows it.

For two-phase modelling of FIV, the volume-of-fluid (VOF) method is a reasonable alternative to Eulerian multi-field approaches in cases with both bubbly and large interface flow regions.

Based on the simulations of the GO-VIKING two-phase benchmarks, the largest structural vibrations may generally be mainly due to the large-scale interfaces; this is also intuitively expected. With the VOF approach, the mesh cannot typically be fine enough to resolve the smallest bubbles, however this does not necessarily corrupt the solution severely and reasonable results for the two-phase fields and structural response may be nonetheless obtained. For having sharp bubble interfaces, the explicit geometric reconstruction VOF method can be recommended instead of the implicit algebraic method. Also, the VOF modelling of FIV might favor scale-resolving turbulence models (e.g. large-eddy simulation) compared to RANS, as the latter may yield too high turbulent viscosity and thus too large bubble sizes. Two-phase FIV remains still a challenging analysis case, mainly due to the error margins involved in predicting the flow field and structural loads, particularly if phase change is present.

The VOF modelling may require specific settings to stabilize the flow solution. Firstly, local unphysical high velocities sometimes appear at the air-water interfaces, causing divergence of the solution; this can be avoided at least by applying velocity limiting. It seems that fairly high values of the velocity limit are enough for stabilization, e.g. 10-20 times the maximum of mean mixture velocity. Possibly higher values than this could be used, although this was not tested. As the limit is high compared to the physical velocities, it is expected to have a minimal effect on the results as the limiting is typically temporarily applied only on a small group of cells. Secondly, with the bounded central differencing (BCD) scheme, the boundedness factor may require adjustment such that stability (numerical diffusion) is increased at the expense of accuracy. Thirdly, in conjunction with LES, the sub-grid scale (SGS) model parameter may need to be increased somewhat, to have higher eddy viscosity and thus a more stable solution.

2.4 EDF R&D

EDF R&D was involved in the different tasks of the WP4 where three experiments were proposed:

Cantilever Rod experiment, the measured variables were mainly the rod tip displacement and the inlet mass flow rates.

TITAN experiment, the measured variables were cylinder force, tube vibrations

TREFLE experiment, the measured variables were cylinder force, vibrations, and the void fraction at different locations (at the inlet, outlet and in the tube bundle).

The Cantilever Rod and the TITAN experiments were simulated. For these two-experiments, the inlet characterization (compared to TREFLE) was weak in terms of two-phase variables, liquid and gas mass flow rates were available, but no void fraction or gas size distribution was present. Thus, this was challenging since the size distribution appears to play a significant role in the forces acting on structures.

To do so, TITAN experiment was simulated in the rigid configuration with the multi-Eulerian approach from neptune_cfd, Generalized Large Interface Model (GLIM), where the large gas structures are resolved and the smaller ones are modeled (considered as dispersed). The strategy for this kind of case was to resolve the main gas structure of the flow to be able to

predict the forces. This condition drives the mesh refinement. Four cases in rigid configuration were simulated with sensitivity to the inlet condition. For high void fraction cases, there is close to no effect from the gas size distribution at the inlet, the tube bundle drives the gas size distribution. In comparison, for lower void fraction (below 50%), the inlet size distribution is of interest since bubbles are smaller than the gap, thus their evolution depends on the inlet. At the end, the presented numerical model is able to accurately predict forces and void fraction for different flow patterns. However, for low void fraction, a numerical sensitivity to gas size distribution was required. We did similarly for the cantilever rod case, where it was required to resolve gas structure in order to have a vibration amplitude similar to the experiment. However, like for TITAN, it is more difficult to simulate bubbly flows without having their size distribution which drives the force magnitude. Thus, results are in correct agreement with the experiment, but suffer from the lack of the gas size distribution to reproduce exactly each case. Discussion with other partners appeared to be in agreement on that point. Consequently, the necessity of resolved gas structures at the domain inlet resulted in testing different inlet conditions, in particular, the Synthetic Bubble Model from Ghent University and having air injectors at the inlet.. The first method was promising, but dependent on the bubble generation length and duration in the precursor. That appears to be observed in inlet mass flow rate fluctuations and therefore, influences the rod vibration. This kind of method is needed to reduce the size of the domain, but needs to be improved to overcome the present issue. Finally, air injectors were modeled at the inlet. These were able to generate resolved gas structures, however, no sensitivity has been performed to the injector diameter. In terms of vibration, the validated beam model from neptune_cfd coupled to the immersed boundary method is used to predict the deformation of the beam. This method appears to be interesting one more time for this kind of approach, also to avoid a strong coupling. For TITAN, an Arbitrary Lagrangian Eulerian (ALE) method was developed specifically for the present work in order to reduce the time of computation and the size of the mesh (since Adaptative Mesh Refinement is not available with our immersed bundary method in neptune_cfd), preliminary results are also promising.

2.5 EDF UK

This guide summarises the recommended practices and key learnings from medium-resolution, two-phase Fluid–Structure Interaction (FSI) 3D simulations performed using *StarCCM+* with Volume of Fluid (VOF) and Unsteady Reynolds-Averaged Navier–Stokes (URANS) methods.

2.5.1 Two-phase modelling and Interface Capturing

Chosen Approach: URANS combined with VOF was found to predict the high void fraction of the experimental configurations adequately, and capture the two-phase flow structures and bubbles developed. The Euler-Euler approach has also been assessed, but it was difficult to achieve convergence and requires further exploration of the sub-models available in the StarCCM+ solver.

Interface Sharpness: To maintain a well-defined interface between liquid and gas phases, the High-Resolution Interface Capturing (HRIC) scheme within StarCCM+ needs to be activated. HRIC coefficients showed that a value of **0.9** provided adequate interface sharpness and mitigated numerical diffusion linked with the coarse URANS grid.

Mesh refinement: Mesh refinement with sufficient prism layers is necessary around the cylinder walls for both the FIV modelling and bubble break-up effects. Adaptive mesh refinement significantly increases computational cost but can improve computational accuracy.

2.5.2 2. Two-phase cantilever-rod case

Non-Conformal Meshing: A conformal mesh between solid and fluid is not required. In fact, opposing gradient regions make a **mapped interface approach** preferable.

The **fluid mesh** should be refined near the **rod tip**, where flow gradients are strongest.

The **solid mesh** requires higher refinement at the **base**, where stress gradients peak.

Solid–Fluid Coupling: Assign motion for deformable solids to **"solid displacement"** and ensure **morphing** is activated in adjacent fluid zones.

All surfaces in the morphing region must be properly defined as **fixed** (stationary walls) or **free** (fluid–fluid and fluid–solid interfaces).

Solver Controls: Low inner iteration counts (around **6 per time step**) were sufficient to reproduce displacement amplitudes and peak frequencies accurately. Some damping in the power spectral density (PSD) was observed.

2.5.3 TREFLE/TITAN Two-Phase FSI Cases

Boundary Conditions: There is uncertainty in the mass flow rate of the air injected in the T5 experimental campaign via the air injector.

Synthetic Bubble Method (SBM) with a smaller upstream of the tube bank domain has been explored, modified to generate bubbles in a Gaussian distribution and limiting the bubble diameter to match the PDF of the experiment. Although the SBM approach predicted droplet breakup and coalescence, the solution is sensitive to the imposed SBM bubble distribution and is not representative of the experiment's expected intermittent behaviour. This suggests that SBM requires further refinement.

To overcome SBM issues, an elongated inlet was used with uniform velocity and void fraction conditions. This approach enabled natural bubble formation and coalescence governed by fluid physics, generating realistic upstream flow profiles without requiring prior bubble-size data. However, the engineer needs to ensure that sub-models, including the interaction area density between two fluids, drag force, tracking of interphase momentum transfer, interface turbulence damping, suppression of turbulence production in interfacial regions, and the surface tension force model, are enabled. This allows multiphase flow structures, such as bubble formation, coalescence, and breakup, to evolve naturally under the influence of inertia, shear, gravity, and surface tension, producing realistic velocity and void-fraction profiles upstream of the tube bundle. For the surface-tension force submodel and to improve numerical stability at large time steps, the "Semi-implicit Surface Tension" approach should be selected.

Turbulence Modelling: The **k–ω SST** turbulence model demonstrated stable behaviour.

Domain width: Short periodic domains captured key flow structures, though **3D domains** are recommended since the structures influence the FSI predictions.

Coupling Strategy: Among tested coupling methods, **sequential coupling** reproduced the tube acceleration reliably and offered an efficient alternative to full two-way coupling.

Temporal integration: The Adaptive Time-step was used with the maximum CFL number set to 0.95. The fluid domain used a 2nd order discretisation scheme, while the solid domain used a 2nd order Backward Differentiation scheme.

Spatial Discretisation: Second-order upwind convection scheme was used for all calculations.

Fluid Mesh Motion: The fluid domain was set to morph or move at every iteration within a time-step. This enabled a strongly coupled solution.

2.6 NRG PALLAS

NRG PALLAS studied two-phase flow-induced vibrations of cylindrical obstacles within triangular and square tube bundles using the HZDR hybrid two-fluid model. The focus was on a movable tube supported by a linear spring and damper, allowing motion only in the lift direction with a single degree of freedom. We employed a hybrid two-fluid approach, capable of resolving large-scale two-phase structures in a VOF-like manner when the grid resolution is sufficient. For smaller structures below the grid size, the model transitions to a traditional two-fluid method. The following paragraphs summarize the best practice guidelines and lessons learned.

Results indicate that the classical two-fluid model captures average flow quantities well in bubbly flows, but fails in slug/churn flows due to inaccurate sub-grid closures. Conversely, the hybrid model tends to overpredict bubble sizes in bubbly flows due to limited mesh resolution. However, only the hybrid model can reliably capture flow-induced vibrations, as the classical model fails to resolve temporal fluctuations.

For bubbly flows where structures cannot be explicitly resolved, sub-grid models are necessary to predict pressure fluctuations efficiently. The interface compression scheme in the hybrid model, which converts stochastic formulations into deterministic ones, requires improvement by accounting for local bubble size and mesh resolution.

When applying the hybrid model for flow-induced vibration studies, it is essential to include a sufficiently long entrance section (20–100 hydraulic diameters) with high enough resolution upstream of the vibrating structure to allow flow structures to develop. Proper placement of the object downstream, local mesh refinement, and monitoring of phase fraction, velocity, and forces help ensure accurate statistical characterization.

Force measurements in two-phase flows using the hybrid model often contain low-frequency content, necessitating long simulation times (100+ seconds) to achieve accurate averaging. Given that such low-frequency, slow trends are not present in the experimental signals, these transients likely stem from numerical artifacts, such as overestimated bubble sizes on coarse

meshes. Mesh refinement and sensitivity analyses are recommended to discern physical effects from numerical artifacts.

In experimental setups with flexible blades supporting tubes, a common approach models the tube as a rigid body undergoing translational motion with an attached spring-damper to represent blade flexibility. Accurate modelling using this approach requires determining the effective mass and stiffness from experiments or FEM modal analysis.

3. Conclusions and Perspectives

In conclusion, the three benchmarks performed within GO-VIKING WP4 led to satisfactory results, some partners were able to predict forces or vibration for each experiment. Different approaches were used in terms of two-phase and turbulent models in particular.The main lessons learned are

- it appears important to resolve large gas structures to predict accurately forces or vibration, with LES or URANS approaches

- meshes are defined similarly to single-phase flow cases and depends on gas size distribution to resolve the main gas structures, even if a mesh sensitivity study remains necessary.

- the numerical simulations highlight that it is easier to simulate high void fraction cases than low ones (where a dispersed approach is necessary). For low void fraction, the dependency to the inlet gas size distribution is significant, thus this information is required to perform accurate simulation of an experiment.

- Innovative injecting methods were used, like the SBM, with some encouraging results that need to be improved.

GO-VIKING WP4 tackled the difficult phenomenon of two-phase flow-induced vibration for NPP applications. Some progresses have been made to simulate these phenomena and show that the numerical models are now able to be closer to reality. It is now of interest to take the benefit of this feedback to continue experimentally and numerically to characterize the two-phase damping with analysis of local-scale variables.