



## **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



---

### **TREFLE simulation results**

---

Authors : Kevin ZWIJSEN (NRG)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

|                     |                           |
|---------------------|---------------------------|
| Document title      | TREFLE simulation results |
| Author(s)           | Kevin ZWIJSEN             |
| Number of pages     | 55                        |
| Document type       | Deliverable               |
| Work Package        | WP4                       |
| Document number     | D4.4                      |
| Issued by           | NRG                       |
| Date of completion  | 2026-05-13 14:19:53       |
| Dissemination level | Public                    |

---

Summary

Model preparation and simulation results for the TREFLE experiment

---

Approval

|                     |                         |
|---------------------|-------------------------|
| Date                | By                      |
| 2026-05-13 15:07:46 | William BENGUIGUI (EDF) |
| 2026-05-13 15:09:10 | Papukchiev ANGEL (GRS)  |



## D4.4 TREFLE Simulation results

Numerical simulation of an air-water flow across a square tube bundle.

Version N°2  
May /2026

William Benguigui<sup>1,2</sup>, Daniele Vivaldi<sup>3</sup>, Lucas Rotily<sup>3</sup>, Sina Tajfirooz<sup>4</sup>, Kevin Zwijsen<sup>4</sup>, Constantinos Katsamis<sup>5</sup>, Ehimen Iyamabo<sup>5</sup>, Antti Timperi<sup>6</sup>, Clément Bazin<sup>7</sup>

<sup>1</sup>EDF R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatou, France

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

<sup>3</sup>ASNR, Cadarache, St Paul-lez-Durance, BP3, 13115, France

<sup>4</sup>NRG PALLAS, Westerduinweg 3, 1755 LE Petten, The Netherlands

<sup>5</sup>EDF Energy R&D UK, 90 Whitfield Street, London W1T 4EZ, United Kingdom

<sup>6</sup>VTT Technical Research Centre of Finland, Kivimiehentie 3, Espoo, 02044 VTT, Finland

<sup>7</sup>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

|                                         |                                                                                                     |
|-----------------------------------------|-----------------------------------------------------------------------------------------------------|
| <b>Grant Agreement / Proposal ID</b>    | 101060826                                                                                           |
| <b>Project Title</b>                    | Gathering expertise On Vibration ImpaKt In Nuclear power Generation                                 |
| <b>Project Acronym</b>                  | GO-VIKING                                                                                           |
| <b>Scientific Coordinator</b>           | Angel Papukchiev, <a href="mailto:Angel.Papukchiev@grs.de">mailto:Angel.Papukchiev@grs.de</a> , GRS |
| <b>Project starting date (duration)</b> | 1st June 2022 – 31st May 2026 (48 Months)                                                           |
| <b>Related Work Package</b>             | WP4                                                                                                 |
| <b>Related Task(s)</b>                  | T4.2                                                                                                |
| <b>Lead Organisation</b>                | EDF                                                                                                 |
| <b>Contributing Partner(s)</b>          | ASNR, NRG PALLAS, VTT, EDF UK, CEA                                                                  |
| <b>Due Date</b>                         | 30-11-2025                                                                                          |
| <b>Submission Date</b>                  | 12-05-2026                                                                                          |
| <b>Dissemination level</b>              | PU                                                                                                  |

## History

| Date            | Version | Submitted by | Reviewed by                | Comments |
|-----------------|---------|--------------|----------------------------|----------|
| 19 – Feb – 2026 | V1      | W. Benguigui | A. Papukchiev & K. Zwijsen |          |
| 12-May-2026     | V2      | W.Benguigui  | A. Papukchiev & K. Zwijsen |          |

## Table of contents

|         |                                                                    |    |
|---------|--------------------------------------------------------------------|----|
| 1.      | Experiment.....                                                    | 10 |
| 1.1     | Mock-up .....                                                      | 10 |
| 1.2     | Rigid configuration .....                                          | 11 |
| 1.3     | Flexible configurations .....                                      | 12 |
| 2.      | Numerical modeling.....                                            | 14 |
| 2.1     | VTT.....                                                           | 14 |
| 2.2     | NRG PALLAS.....                                                    | 15 |
| 2.3     | ASNR.....                                                          | 16 |
| 2.4     | EDF UK.....                                                        | 17 |
| 2.4.1   | Boundary conditions and numerical setup .....                      | 18 |
| 2.4.1.1 | Approaches to improve modelling and inlet conditions.....          | 18 |
| 2.4.2   | Numerical details, discretisation schemes and time stepping.....   | 20 |
| 2.5     | CEA.....                                                           | 21 |
| 2.6     | Sum-up of the numerical approaches.....                            | 22 |
| 3.      | Calculation domain & Mesh.....                                     | 23 |
| 3.1     | VTT.....                                                           | 23 |
| 3.2     | CEA.....                                                           | 24 |
| 3.3     | NRG PALLAS.....                                                    | 25 |
| 3.4     | ASNR.....                                                          | 25 |
| 3.5     | EDF UK.....                                                        | 27 |
| 3.5.1   | Geometry and computational domain for T5 configuration .....       | 27 |
| 3.6     | Sum-up of the calculation domains & Mesh.....                      | 29 |
| 4.      | Rigid configuration.....                                           | 30 |
| 4.1     | Qualitative analysis of the flow through numerical simulation..... | 30 |
| 4.1.1   | VTT.....                                                           | 30 |
| 4.1.2   | NRG PALLAS.....                                                    | 31 |
| 4.1.3   | CEA.....                                                           | 33 |
| 4.1.4   | ASNR.....                                                          | 36 |
| 4.1.5   | EDF UK.....                                                        | 37 |
| 4.2     | Void fraction: Numerical vs Experimental results.....              | 39 |
| 4.2.1   | CEA analysis on their quantitative results.....                    | 40 |
| 4.2.2   | EDF UK analysis on their quantitative results .....                | 44 |

|     |                              |    |
|-----|------------------------------|----|
| 5.  | Flexible configuration ..... | 47 |
| 5.1 | VTT.....                     | 47 |
| 5.2 | CEA.....                     | 48 |
| 5.3 | EDF UK.....                  | 49 |
| 5.4 | NRG PALLAS.....              | 51 |
| 6.  | Conclusion.....              | 53 |

## List of figures

|            |                                                                                                                                                                                                                                                                                                                   |    |
|------------|-------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| Figure 1 : | Detailed sketch of the TREFLE facility with dimensions reported in mm. ....                                                                                                                                                                                                                                       | 10 |
| Figure 2 : | Two side views of the tube bundle section.....                                                                                                                                                                                                                                                                    | 11 |
| Figure 3 : | Two type of air Injector, (a): Type 1 and (b) Type 2. ....                                                                                                                                                                                                                                                        | 11 |
| Figure 4 : | Position of the force sensors (left); direction of the forces (middle); photograph from the transparent side panel of the TREFLE, with the sensor at the centre of the 5x5 bundle (right).....                                                                                                                    | 12 |
| Figure 5 : | PSD of projected drag (left) and lift force (right) for various air flow rate. ....                                                                                                                                                                                                                               | 12 |
| Figure 6 : | Sketch of the vibrating tube inside TREFLE (top) and a picture of the tube bundle through the transparent panel (bottom), with the two vibrating tubes.....                                                                                                                                                       | 13 |
| Figure 7 : | PSD evolution of the displacement of one vibrating tube, after integration of the accelerometer signal, for different air flowrates (left) and comparison of the PSD of the displacement from the two accelerometers of the two vibrating tubes at an air flowrate of $q = 20\text{g/s}$ (right). ....            | 14 |
| Figure 8 : | Pierced tube injector used for high void fraction(left). SBM bubble generation based on Probability density functions (PDF) of bubble diameter [13]......                                                                                                                                                         | 19 |
| Figure 9:  | Bubble generated in a small domain using the SBM (left) and applying the SBM to a periodic segment of the domain, showing bubbles interacting with the tube bundle. The bubbles are illustrated using volume fraction = 0.5 iso surfaces. ....                                                                    | 19 |
| Figure 10  | Geometry and mesh used in the VTT simulations. ....                                                                                                                                                                                                                                                               | 23 |
| Figure 11  | Details of the VTT mesh at the tube bundle. ....                                                                                                                                                                                                                                                                  | 24 |
| Figure 12  | Cross sections of the computational domain in the NRG PALLAS simulations. (a) Full domain in the x-y plane. (b) Left: mesh in the vicinity of the tube bundle. Right: cross-section of the channel in the x-z plane. (c) The side view of the bundle region for flow-only (left) and FSI (right) simulations..... | 25 |

|                                                                                                                                                                                                                                                                                    |    |
|------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| Figure 13 Numerical domain, with the position of the “numerical” WMS. WMS1 and WMS2 correspond to the experimental WMS placed upstream and downstream of the tube bundle, respectively.....                                                                                        | 26 |
| Figure 14 Details of the numerical inlet boundary condition employed: (a) representation of the instantaneous bubble area ( $S_b$ ) injected with void fraction equal 1; (b) space-averaged void fraction imposed numerically and the experimental value, as a function of time... | 26 |
| Figure 15 Details of the ASNR mesh.....                                                                                                                                                                                                                                            | 27 |
| Figure 16 Schematic and geometry of computation domain with the long inlet. Zoom-in section of the grid around the tube bundle region for T5 experimental configuration. .                                                                                                         | 28 |
| Figure 17. Instantaneous void fraction (air shown in blue) at the centre planes in VTT simulation with fixed geometry (experiment T5). .....                                                                                                                                       | 30 |
| Figure 18. Time-averaged vertical mixture velocity in m/s (left) and void fraction (right) at the centre planes in VTT simulation with fixed geometry (experiment T5). .....                                                                                                       | 31 |
| Figure 19 Instantaneous (top) and mean (bottom) air void fraction at $t = 20$ s, case T5. ....                                                                                                                                                                                     | 32 |
| Figure 20 Mean water streamwise velocity at $t = 20$ s, case T5.....                                                                                                                                                                                                               | 32 |
| Figure 21 Qualitative comparison of the time-averaged void fraction (top-left) and instantaneous void fraction (top-right) in G1 simulations with T2 (bottom-left) and T3 photographs (bottom-right) [12]. .....                                                                   | 33 |
| Figure 22- Time-averaged void fraction for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in G1 simulations. ....                                                                                                                                 | 34 |
| Figure 23 Time-averaged gas velocity in G1 simulations. ....                                                                                                                                                                                                                       | 34 |
| Figure 24 Qualitative comparison of the time-averaged void fraction (top-left) and instantaneous void fraction (top-right) in T5 simulations with T5 (bottom-right) photographs (bottom-right). ....                                                                               | 35 |
| Figure 25 Time-averaged void fraction for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in T5 simulations.....                                                                                                                                   | 36 |
| Figure 26 Time-averaged gas velocity in T5 simulations.....                                                                                                                                                                                                                        | 36 |
| Figure 27 Instantaneous void along (left) the symmetry frontal plane, (middle) the symmetry lateral plane and (right) zoom on the tube bundle region of the frontal plane.....                                                                                                     | 37 |
| Figure 28 Instantaneous void fraction distribution at the centre planes from the T5 case in fixed configuration from the last time step of the EDF UK simulations. ....                                                                                                            | 38 |
| Figure 29 Time-averaged void fraction distribution at the centre planes from the T5 case in fixed configuration from the last time step of the EDF UK simulations. ....                                                                                                            | 38 |

|                                                                                                                                                                                                           |    |
|-----------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| Figure 30 Upstream/downstream void profile along x from each partner compared to the experiment.....                                                                                                      | 39 |
| Figure 31 Void fraction comparison inside the tube bundle. Point locations (left) and numerical results compared to experiment (with errorbars of +/- 15%) (right). ....                                  | 40 |
| Figure 32 Time-averaged void fraction within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in G1 simulations compared with experimental data. ....       | 40 |
| Figure 33 Time-averaged gas velocity within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in G1 simulations compared with experimental data .....        | 41 |
| Figure 34 Time-averaged Sauter mean diameter within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in G1 simulations compared with experimental data..... | 42 |
| Figure 35 Time-averaged void fraction within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in T5 simulations compared with experimental data. ....       | 43 |
| Figure 36 Time-averaged gas velocity within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in T5 simulations compared with experimental data. ....        | 43 |
| Figure 37 Time-averaged Sauter mean diameter within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in T5 simulations compared with experimental data..... | 44 |
| Figure 38 Distribution of void fraction in the lateral cross-flow x-direction for downstream and upstream WMS plane locations and average mixture velocity in the T5 configuration. .                     | 44 |
| Figure 39 Radial and cross-plane distribution of Void fraction.....                                                                                                                                       | 45 |
| Figure 40 PDF of bubble diameter distribution for T5 configuration .....                                                                                                                                  | 45 |
| Figure 41 Average void fraction at probe locations in the CFD and experimental data on the central tube in in the T5 configuration .....                                                                  | 46 |
| Figure 42 PSDs of transient monitors for average velocity and average void fraction at downstream and upstream WMS plane locations in the T5 configuration. ....                                          | 46 |
| Figure 43 PSDs of tube drag (left) and lift (right) force on the central tube in the T5 fixed configuration from EDF R&D UK simulations. ....                                                             | 47 |
| Figure 44. Tube acceleration spectrum in experiment T5 and in FSI simulations. Note that the air flow rate is 10 g/s and 11 g/s in the experiment and simulations, respectively.....                      | 48 |

|                                                                                                                                                         |    |
|---------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| Figure 45 PSD of the acceleration of the flexible central tube for G1 (top) and T5 (bottom) compared with experimental data. ....                       | 49 |
| Figure 46 PSD of the drag and lift forces of the flexible central tube for G1 (top) and T5 (bottom) compared with experimental data.....                | 49 |
| Figure 47 Power Spectra of stream-wise, $F_D$ (left), and cross-flow, $F_L$ (right) components of flow-induced forcing in two-way CFD-FEA coupling..... | 50 |
| Figure 48 Power Spectra of the acceleration magnitude of the rod end from the two-way CFD-FEA coupling simulations of the T5 EDF R&D UK simulation..... | 51 |
| Figure 49: Displacement signal of the moving tube in test T5 NRG PALLAS simulation.....                                                                 | 52 |
| Figure 50: Power Spectra of the moving tube acceleration in test T5 NRG PALLAS simulation. ....                                                         | 52 |

## List of tables

|                                                                                                                                                                                                               |    |
|---------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| Table 1 GLIM set-up .....                                                                                                                                                                                     | 17 |
| Table 2 Flow properties.....                                                                                                                                                                                  | 20 |
| Table 3 Sum-up of the numerical approaches from each partner.....                                                                                                                                             | 22 |
| Table 4 Computational domain details and computing expenses.....                                                                                                                                              | 28 |
| Table 5 Sum-up of the calculation domain and mesh.....                                                                                                                                                        | 29 |
| Table 6. Tube displacements in experiment T5 and in FSI simulations. The value in parenthesis corresponds to air flow 11 g/s, based on linear interpolation of the data with varying air flow from [12]. .... | 48 |

## Abbreviations and Acronyms

| Acronym   | description                                                                  |
|-----------|------------------------------------------------------------------------------|
| FIV       | Flow-Induced Vibration                                                       |
| GLIM      | Generalized Large Interface Model                                            |
| GO-VIKING | Gathering expertise On Vibration ImpaKt In Nuclear power Generation          |
| LES       | Large Eddy Simulation                                                        |
| RMS       | Root-Mean Square                                                             |
| SG        | Steam Generator                                                              |
| TREFLE    | Two-phase flow REgime and FLuid-structure interaction Experimental facility) |
| VOF       | Volume of Fluid                                                              |
| URANS     | Unsteady Reynolds Averaged Navier Stokes                                     |
| WP        | Work Package                                                                 |

## Summary

The present work is dedicated to the simulation of the TREFLE experiment, an air-water flow across a square tube bundle. Five partners present simulations with different codes, numerical setup or models on the rigid configuration for different two-phase flow pattern and for a flexible configuration to take into account the resulting vibration.

## Keywords

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



A homogenization section (about 1400 mm long) is used to make the two-phase flow develop and homogenize before reaching the tube bundle. The test section of the facility contains an in-line, square pitch  $5 \times 5$  tube bundle. The array, which features a pitch-to-diameter ratio of 1.44, is composed of cylinders with a diameter of 30 mm. This value is slightly higher than the one typically found in steam generators (SGs), but this design leads to wider gaps (13.2 mm) between the cylinders (see Figure 2), enabling detailed measurements of the two-phase flow dynamics. The tube length  $L$  is 10 times the tube diameter. The tubes can either be fixed or able to vibrate.

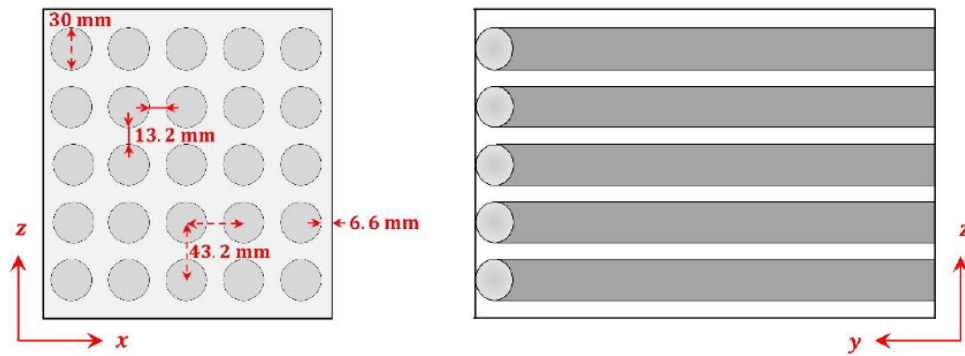


**Figure 2 : Two side views of the tube bundle section**

The gas injector is designed to be interchangeable (7 injectors), allowing for adapting the component based on the air flow rate. Two types of injectors were utilized in this study: Type 1 for bubbly flow and Type 2 for intermittent flow (see Figure 3).



**Figure 3 : Two type of air Injector, (a): Type 1 and (b) Type 2.**

The TREFLE facility has been operational since 2022, with various experiments already performed and some results presented to the GO VIKING partners (see Deliverable 4.3 for experimental results from Wire Mesh Sensor and Dual Optical Probe void fraction measurements). The experiments detailed in this document focus on both rigid and flexible tube configurations: an experimental campaign was conducted by varying the air flow rate from 0.5 g/s to 30 g/s (with the injector switched from type 1 to type 2 at 11 g/s) and a constant water flow rate equal to 2.5 l/s.

## 1.2 Rigid configuration

For the rigid configuration tests, the central cylinder of the tube bundle has been fitted with two piezoelectric force sensors. Each force sensor is placed at one extremity of the cylinder. One sensor is mounted to the TREFLE structure with a screw attachment, while the other is mounted on a tab that slots into a custom-made window panel.

Each sensor measures forces in two directions, which are then projected onto the drag and lift axes. The two Kistler 9317C triaxial piezoelectric sensors have a measuring range of  $\pm 1.5$

kN, a sensitivity of approximately 25 pC/N, and a natural frequency of about 5 kHz in fixed-free configuration. Due to sensor drift, the mean value must be removed from the signal, meaning only force fluctuations are experimentally reliable.

Results confirmed that both sensors measured consistent forces simultaneously, despite differences in the rod-to-structure mounting system that appear to dampen the response of the sensor located at the beam tip. Root-Mean Square (RMS) forces increase with flow rate, particularly in the bubbly flow regime, while this increase was less pronounced in the churn/intermittent regime. Drag forces were predominantly oriented in the flow direction and exhibited higher RMS intensity than lift forces. Spectral analysis revealed low-frequency vibrations consistent with bubble passage through the tube bundle cross-section (see Deliverable 4.3 for spectral analysis of void fraction measurements), with similar dominant frequencies observed for both force components, see Figure 5.

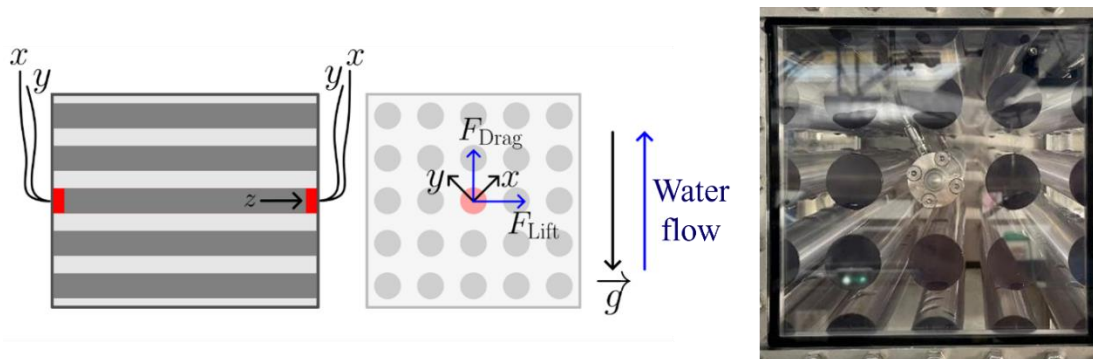


**Figure 4 : Position of the force sensors (left); direction of the forces (middle); photograph from the transparent side panel of the TREFLE, with the sensor at the centre of the 5x5 bundle (right).**

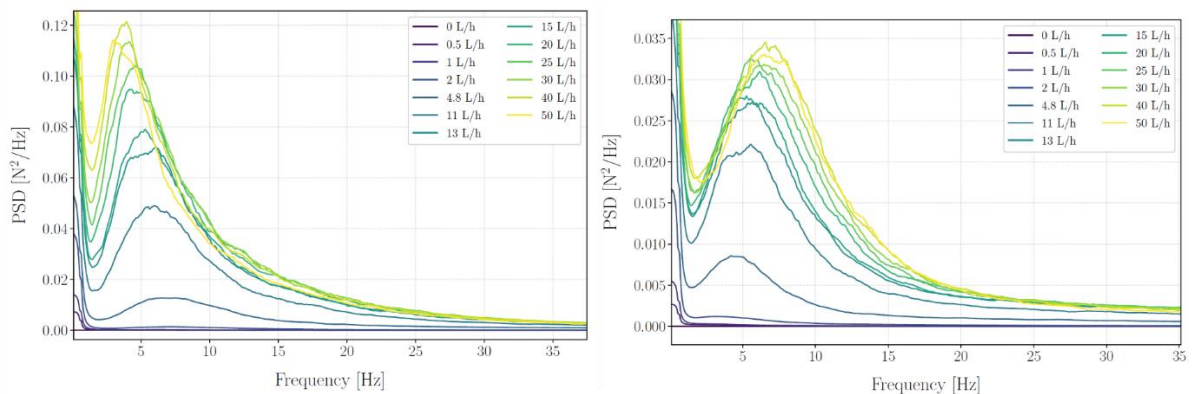


**Figure 5 : PSD of projected drag (left) and lift force (right) for various air flow rate.**

### 1.3 Flexible configurations

For the flexible configurations campaign, a system consisting of a flexible blade is used, aimed at measuring the vibration response of the tubes. A sketch of the vibrating tubes and an experimental picture are shown in Fig. 6. The vibrating tube is composed of a metal blade (Fig. 6 (e)) welded on a metallic support with a screw (Fig. 6 (a)) adapted to TREFLE fixation system. Around the blade, a system of two spacers (Fig. 6 (c)) fits over the blade at the top and bottom

to form a circular assembly, which is then covered with a hollow plastic tube (Fig. 6 (d)) to obtain the same diameter as the fixed tubes. For the present tests, the blade selected had a thickness of  $e = 2$  mm. The vibrating tubes are equipped with a strain sensor (Fig. 6 (b)) near the metallic support of the blade to measure the blade free end deformation, and with an accelerometer (Fig. 6 (f)). Two configurations were tested: one single vibrating tube (the one on top of the central one), and two vibrating tubes (the central one plus the one on top of it, shown in Fig. 6). Structural properties are the following : 0.39 kg, beam stiffness of 600 kN/m in the cross-flow direction, damping ratio of 1% in air and 4% in water; and the beam length is 300 mm. The natural frequency of the tube in air is about 13.4Hz.

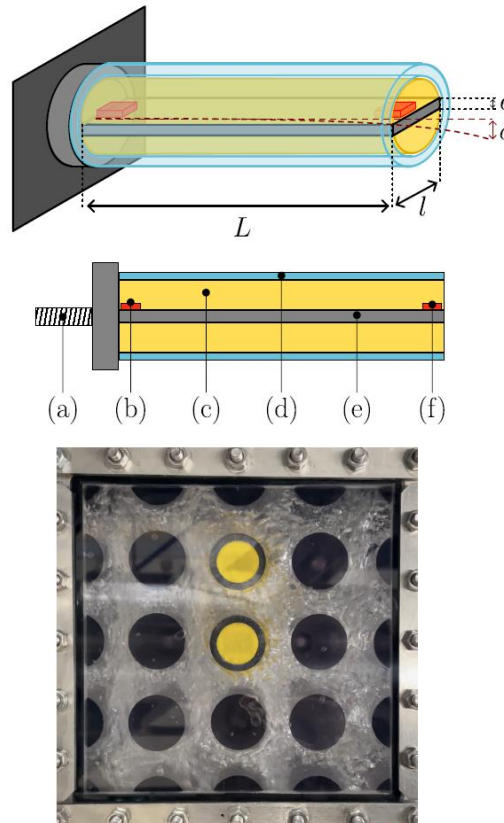


**Figure 6 : Sketch of the vibrating tube inside TREFLE (top) and a picture of the tube bundle through the transparent panel (bottom), with the two vibrating tubes.**

The vibrational response does not appear to change between the two configurations. Moreover, no trace of the intermittently rising gas structures could be found in the tube acceleration spectra; however, the PSD calculated through high-speed imaging presents harmonics that may be linked to the intermittent gas structures, see Figure 7.

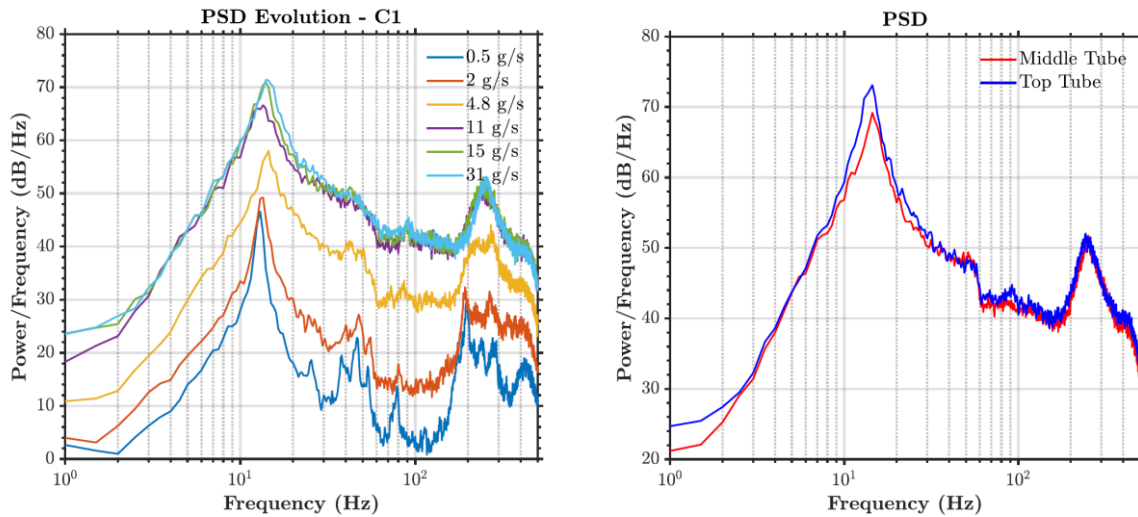


**Figure 7 : PSD evolution of the displacement of one vibrating tube, after integration of the accelerometer signal, for different air flowrates (left) and comparison of the PSD of the displacement from the two accelerometers of the two vibrating tubes at an air flowrate of  $q = 20\text{ g/s}$  (right).**

## 2. Numerical modeling

### 2.1 VTT

The two-phase flow was simulated with the volume-of-fluid (VOF) model of Ansys-Fluent v2024. The intermittent flow case T5 having relatively high air flow rate was focused on, which involves simultaneous bubbly flow and large-scale interface flow regimes. As horizontally the whole domain is considered, the mesh cannot be fine enough to resolve accurately the bubbly flow regimes (having a typical bubble size of  $\sim 5\text{--}9\text{ mm}$ ), however the largest tube motions are due to the large-scale bubbles intermittently passing the tube bundle.

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 HRIC interface 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 the large-eddy simulation (LES) approach, together with the Smagorinsky sub-grid scale (SGS) model and wall functions. Reynolds averaged Navier-Stokes (RANS) modelling by using the standard and re-normalization group (RNG)  $k\text{--}\epsilon$  models was also initially tested. LES gives much lower eddy viscosity and thus a smaller and more realistic bubble size compared to the RANS models, hence LES was chosen for the simulations. The value  $C_s = 0.1$  was initially used for the Smagorinsky SGS coefficient. However, this resulted in dispersed and clearly incorrect air-water interfaces, possibly due to unphysical

oscillations of the velocity field. Thus the coefficient was raised to  $C_S = 0.2$  which increased the eddy viscosity and stabilized the solution.

LES modelling employs usually second-order time discretisation since transient flow details need to be resolved accurately. However, in Fluent the geometric reconstruction with explicit VOF can be used only with first-order time discretization. Safarzadeh et al. (2021) compare first-order time discretization against two different second-order methods in single-phase LES. They find very small differences (mostly negligible) between these discretizations in the time-averaged mean and RMS values, especially when compared to the differences brought by the SGS model and spatial discretization. In addition, the present simulations are expected to have relatively large errors due to the under-resolved VOF modelling of the bubbly flow regions. Thus using LES with first-order time discretization is considered a minor issue and the effect of time step is tested separately.

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 low numerical diffusion. However, due to the numerical issues mentioned above in connection with the Smagorinsky coefficient when using the default factor 0.25, the value 0.5 was chosen as it was found to stabilize the solution, together with the increased Smagorinsky coefficient.

The SIMPLE pressure-correction algorithm was applied, together with the PRESTO scheme for pressure discretization. Time step 0.2 ms was used, corresponding to a CFL number 0.3 based on the maximum mean velocity of the mixture in the tube bundle (see Sec. 4.1). Due to the short time steps, only 10 iterations per time step were performed in the flow solution. The simulations were run for 50 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 tube motion was approximated as a rigid body, where a pivot point and a rotational spring were set at the fixed end of the tube. This corresponds to a single degree-of-freedom (d.o.f.) system, since in the experiment tube motion is allowed basically only in the lift direction (i.e. horizontally). At the coupling interface, the pressure and viscous loads were transferred to the single d.o.f. model. The CFD interface mesh nodes were displaced according to the tube rotation, and the internal CFD mesh was deformed with the diffusion model based mesh smoother of Ansys-Fluent. The tube motion and the explicit FSI coupling were implemented through user-defined functions (UDF).

## 2.2 NRG PALLAS

To model the two-phase flow, the single-pressure isothermal two-fluid model (TFM) is used. For this purpose, we use the multiphase Euler solver of OpenFOAM 12 [1][2]. For each phase, a set of balance equations is solved for averaged mass, momentum, and turbulence quantities. When it comes to studying two-phase fluid-structure Interaction (FSI), the classical TFM's fundamental limitation is that it averages out subgrid-scale two-phase structures, making it unsuitable for predicting dynamic stress fluctuations on solid surfaces in two-phase (FSI) problems. To overcome this, a hybrid approach is used to blend the classical TFM sub-grid closures (like drag and lift) with VOF-like 'tie-together' closures, using a blending function

based on the local volume fraction. This way, the solver can shift between treating a phase as dispersed and applying a fully resolved interface treatment (VOF-like) with surface tension, where necessary. For the classical TFM, the Ishii and Zuber drag [7], the Tomiyama lift [8], Antal's wall lubrication [9], and Burns' turbulent dispersion model [10] are used. The LogMoM model is used to describe the unresolved bubble (droplet) size distribution [4][5].

To model the FSI, we use OpenFOAM's SixDoFRigidBodyMotionSolver, which treats the obstacle's translational motion as a simple mass-spring-damper system, governed by Newton's second law. Using interpolation, a net force acting on the surface of the obstacle is calculated from the total stress tensor. The motion of the obstacle in streamwise and centerline direction, as well as any rotation, is constrained, meaning that the obstacle can only move in lift direction (i.e., orthogonal to both the obstacle's centerline and streamwise direction). The original 6-DoF FSI solver in OpenFOAM is limited to single-field solvers because hard-coded parameters restrict fluid force computations to single-phase flows. To overcome this limitation, a new solver has been developed that is compatible with the multiphaseEuler solver.

In OpenFOAM, the mesh motion method is based on an Arbitrary Lagrangian-Eulerian (ALE) formulation. The introduction of the PIMPLE algorithm in OpenFOAM enables the use of outer corrector loops to ensure the convergence of pressure and velocity. As a result, the mesh update function is incorporated within the PIMPLE outer corrector loop [10]. In OpenFOAM's FSI coupling approach, the following steps are executed at each time step. First, fluid variables (velocity and pressure, void fraction, etc.) are initialized using the solution from the previous step. Simultaneously, if the rigid body has moved, the mesh is updated accordingly. Next, through an inner PISO cycle, the flow fields are updated to ensure mass conservation. Finally, the hydrodynamic force and moment on the rigid body are calculated via surface integrals. For TREFLE simulations, we used one outer corrector, making the FSI coupling approach an explicit approach.

## 2.3 ASNR

The Generalized Large Interface Model (GLIM) was used. In the GLIM [22], the interfacial closure force between the gas and liquid phases depends on the flow regime in the cell, which is estimated based on the void fraction value and gradient. GLIM was implemented on top of the pre-existing LIM model, proposed by Coste [23]. The model tracks the presence of large interfaces based on a void fraction gradient criterion. If a large interface is detected, dedicated interfacial closure laws are used instead of the standard dispersed phase closure.

In the dispersed phase regime, the model includes all physical interfacial forces of bubbly/droplet flow: drag, lift, added mass, turbulent dispersion, wall lubrication forces. The flow is considered bubbly or droplet, depending on the void fraction of the cell.

In the large interface regime, only friction and a drag force are applied. No interface compression algorithms are used, and surface tension force is not taken into account.

The details of GLIM closure laws and set-up are provided in Table 1.

**Table 1 GLIM set-up**

| <b>GLIM set-up</b>                     |                                              |
|----------------------------------------|----------------------------------------------|
| <b>Closure laws (dispersed phase):</b> |                                              |
| • Bubbly flow diameter                 | IATE 1 group - Ruyer et Seiler               |
| • Drag coefficient bubbly flow         | Ishii and Zuber                              |
| • Drag coefficient droplet flow        | 0.44                                         |
| • Turbulent dispersion                 | Lavieville et al.                            |
| • Lift coefficient                     | Tomiyama                                     |
| • Virtual mass coefficient             | Ishii and Zuber                              |
| • Wall lubrication                     | Tomiyama                                     |
| <b>Closure laws (large interface)</b>  |                                              |
| • Drag coefficient                     | Mimouni                                      |
| • Friction force                       | Coste                                        |
| <b>Turbulence</b>                      |                                              |
| • model                                | RSM EBRSM (only applied to the liquid phase) |
| • turbulence interaction               | Bubble induced turb.                         |
| • turbulence interface damping         | None                                         |
| <b>Surface tension</b>                 | None                                         |
| <b>Space discretization</b>            | Second-order upwind (with slope test)        |
| <b>Time discretization</b>             | First order                                  |

## 2.4 EDF UK

EDF Energy R&D UK conducted 3D flow simulations for the two-phase (air/water) cross flow square in-line tube bundle of TREFFLE (Two-phase flow REgime and FLuid-structure interaction Experimental facility) at the Institute for Radiation Protection and Nuclear Safety (IRSN in Cadarache) [1]. Turbulent two-phase flow conditions and flow-induced vibration (FIV) could lead to high-cycle fatigue and fretting wear on the tubes of the Pressurised Water Reactor (PWR) steam generators. Comprehension of two-phase flow and fluid structure interaction mechanisms is required, along with robust numerical modelling practices to predict component failures and develop effective maintenance strategies.

Therefore, the CFD model focuses on the T5 5x5 tube array with a fixed and flexible central tube which experiments reported a relatively high-void fraction at the WMS location, generating bubbles and large slugs in the flow. The 3D transient computations were conducted

in the SIEMENS STARCCM+ v2024.6 finite volume CFD solver and Volume-of-Fluid (VoF) adopted for modelling the intermittent two-phase flow with URANS to close the turbulent transport equations. The comparisons against the experimental data focus on time series and comparisons of phase velocity, void fraction and dynamics of the vibrating tube.

### 2.4.1 Boundary conditions and numerical setup

In the experimental configuration for the T5 test, water is entering at a uniform mass flow rate and interacts with a gas injector, forming a two-phase intermittent flow (bubbly-slug flow) which continues to develop in the homogenization section, creating a non-uniform velocity and volume fraction profile. The water mass flow rate enters at  $\dot{m}_w = 2.5\text{kg/s}$  and the gas is injected with a flow rate of  $\dot{m}_g = 0.011\text{kg/s}$ , from the T7 injector tube shown in Figure 1. This leads to a distribution of bubbles, with the bubble diameter reported  $4\text{mm} < D_b < 15\text{mm}$  from the experiment. Due to the development of the flow and turbulence at such high void fraction, a mix of interface length scales from small bubbles ( $\sim 4\text{--}6\text{ mm}$ ) to large slugs/structures ( $\sim 25\text{--}100\text{ mm+}$ ) has been observed in the experiment. The largest structures are observed to coalesce with the inline tube bundle breaking up as the flow convects across the section [12]. The tube bank array contains a central vibrating rod that is free to vibrate, experimentally reproducing flow-induced vibration, with lift, drag and displacement data captured on this rod. The “rigid” rod is attached to a flexible beam with a total system mass of 0.39 kg, beam stiffness of 600 kN/m in the cross-flow direction, damping ratio of 4% and beam length of 300 mm (see [11],[12]). The beam was instrumented with an accelerometer to capture vibration response data.

#### 2.4.1.1 Approaches to improve modelling and inlet conditions

It is anticipated that due to the large density differences between the air and the water, the inertial effects of intermittent impingement of high- and low-density fluids with the tube bundle would be the dominant forcing effect compared to the background turbulent fluctuations within the water or the air phases. In addition, the larger scale bubbles are expected to be of primary importance, as very small bubbles are unlikely to generate significant forcing.

As a result, numerical modelling should ensure representative volume fraction, velocity and turbulence boundary conditions are defined and that a sufficient upstream length is provided to allow inertia, gravity and surface tension effects to interact and develop into the observed inlet void fraction and velocity profiles. Two approaches for modelling the inlet conditions have been assessed, namely a smaller domain with the Synthetic bubble generation method (SBM) of [13,14] combined with a short section upstream of the tube bank and a computational domain with a long upstream development section where the void fraction and velocity mixture are assigned based on the flow conditions measured in the experiment. In the latter, the mixture velocity and void fraction is defined at the inlet following:

- $j_l = 0.0386\text{ m/s}; j_g = 0.1386\text{ m/s}$
- Mixture velocity =  $j_l + j_g = 0.1772\frac{\text{m}}{\text{s}}$

- Gas volume fraction,  $\alpha_g = \frac{j_g}{j_g + j_l} = 0.7822$

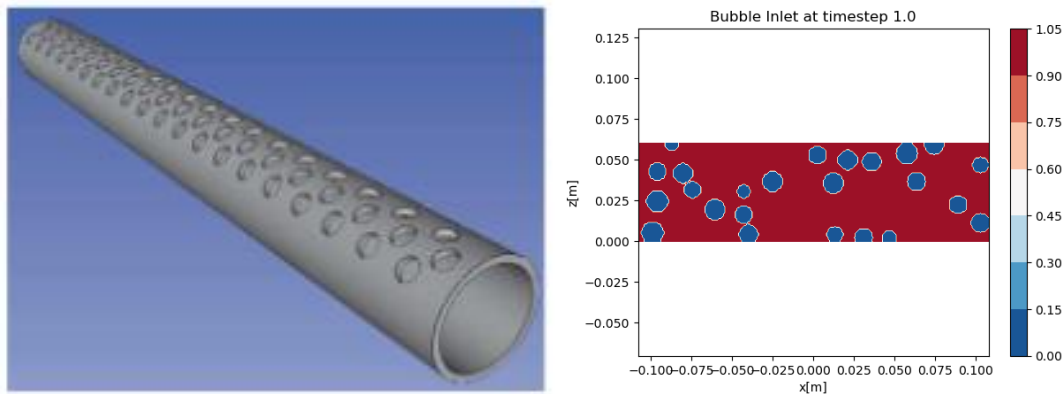


**Figure 8 : Pierced tube injector used for high void fraction(left). SBM bubble generation based on Probability density functions (PDF) of bubble diameter [13].**

The SBM has been modified to generate bubbles in a Gaussian distribution, limiting the bubble diameter to match those of the probability density function observed in the experiment (see Figure 8). The results for this initial investigation of the SBM on a small development section are shown in Figure 9.

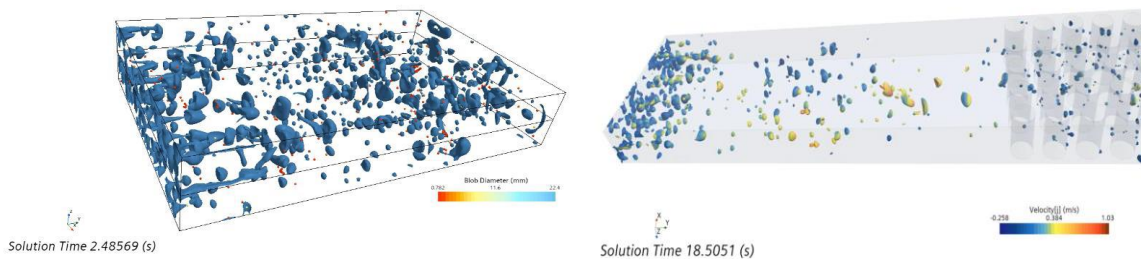


**Figure 9: Bubble generated in a small domain using the SBM (left) and applying the SBM to a periodic segment of the domain, showing bubbles interacting with the tube bundle. The bubbles are illustrated using volume fraction = 0.5 iso surfaces.**

An approach tested was to also parabolically scale the generated profile to match the void fraction and velocity of the experiment and run the computation long enough to enable the flow to reach fully developed profiles for velocity, VoF and turbulence. Although the SBM approach was able to predict droplet breakup and coalescence, offering significant computational savings compared to an elongated inlet domain, the overall solution was found to be strongly dependent on the imposed SBM bubble distribution and not to be representative of the expected intermittent behaviour. The SBM required further investigation, and it is highly dependent on the grid used and time interval for generating the bubbles, seen to lead to unphysical pulsating behaviour, which would introduce artificial fluctuation in the air flux PSD and potentially influence the vibration response.

To address the SBM issues, a simplified phase inlet derived from the sum of the superficial velocities of the different phases with a long inlet section for the present 3D simulations, assuming no-drift flux at the inlet boundary. Flow inertia, gravity, shear and surface tension

effects are modelled. Bubbles formation has been achieved by enabling the multi-phase sub-models available in StarCCM+. These included activating the interaction area density between the two fluids, which quantifies the interfacial surface area per unit volume between the two phases, computing the drag force and tracking the interphase momentum transfer. Interface Turbulence Damping has been enabled to account for the turbulence fluctuations due to the large gradients in the boundary layer near the free surface and suppressing turbulence production in the interfacial regions and near bubble surfaces. The surface tension force model was activated to calculate the surface tension force between each of the phases in the phase interaction. 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. The simulations used the semi-implicit surface tension approach to improve numerical stability in the computations by reducing the effect of spurious velocity. This achieved flow to develop with bubbles generated before reaching the tube bundle section, better representing the phenomena in the experiment and leading to the formation of large structures.

### 2.4.2 Numerical details, discretisation schemes and time stepping

Fluids are water and air, which are assumed to be isothermal, incompressible, with the properties defined in Table 1. Water is the primary phase, with air being the secondary phase.

**Table 2** Flow properties

|                              | Air    | Water     |
|------------------------------|--------|-----------|
| Density (kg/m <sup>3</sup> ) | 1.225  | 997.561   |
| Viscosity (Pa.s)             | 1.8E-5 | 8.8871e-4 |
| Surface Tension (N/m)        | 0.072  |           |

The Volume of Fluid (VOF) method [15] is used for the phase interface capturing model and relies on providing a definition for the volume fraction occupied by one of the phases as the phase-indicating scalar. This is obtained through the solution of a transport equation:

$$\frac{\partial}{\partial t} (\chi^k \rho^k) + \frac{\partial}{\partial x_j} (\chi^k \rho^k U_i^k) = 0 \quad (1)$$

where  $\chi^k$  is the phase-indicating scalar-volume fraction,  $\rho^k$  is the density and  $U_i^k$  the velocity for phase k.

The phase indicating function is constructed to take a value of 1 if the phase is present and zero otherwise. The numerical grid needs to be designed to provide sufficient resolution for the interface of the two phases. We have combined the VOF with the HRIC-blending scheme-1st /2nd order for interface density reconstruction and with specified interface sharpening set to 0.9 through slip velocity, with 1.0 leading to no numerical diffusion and a very sharp

interface for the simulations. When switching the interface sharpening in StarCCM+, it adds the following term in equation (1):

$$\nabla \cdot (U_i^k x^k (1 - x^k)) = 0 \quad (2)$$

The term in equation (2) enables the model to capture the interface and prevents numerical diffusion in the results as the flow passes through the tube bundle section.

Turbulence modelling is handled using the two-equation eddy viscosity model,  $k - \omega SST$  with all  $y+$  wall function for turbulence effects, with intermittency turned off and under-relaxation ramp to prevent numerical divergence. Pressure-velocity coupling is handled with SIMPLEC and 2nd order convective schemes for the turbulence model, phase interaction, and segregated solver. The time discretisation is constant with a 2<sup>nd</sup> order implicit scheme, but tests have been performed with the adaptive time scheme, with a 2nd order implicit time scheme, limiting the CFL = 0.25 for an acceptable VOF mass imbalance error (error <1% of total air mass flow, with higher CFL leads to mass imbalance error).

## 2.5 CEA

NEPTUNE\_CFD is a computational platform jointly developed by CEA, EDF, IRSN, and Framatome for the advanced simulation of two-phase flows. Its numerical framework relies on a finite-volume, collocated, cell-centered discretization. The code uses an “alpha–pressure–energy cycle” solver together with a first-order time-integration scheme based on a pressure-correction fractional-step method. This algorithm iteratively couples the mass and energy balance equations in order to ensure strict conservation of both quantities. Convective fluxes are discretized using a second-order upwind scheme. Boundary conditions are prescribed as follows: inlet sections are defined with uniform profiles of velocity, void fraction, turbulence intensity, and Sauter Mean Diameter (Dsm). Outlet boundaries impose a fixed pressure with a no-backflow condition.

The present six-equation model distinguishes three fields: a dispersed gas field (g1) representing quasi-spherical bubbles, a hybrid dispersed–continuous gas field (g2) for distorted bubbles and gas pockets, and a continuous liquid field (l). Each gas group is associated with its own transport equation for the equivalent bubble diameter (Dsm). The approach combines a two-group IATE model with an interface-detection method (LIM). Both gas fields are treated in an Euler–Euler framework, each with its own continuity and momentum equations, similar to the liquid phase [4]. The closure laws used in this model include drag [7], lift [16], added mass [17], turbulent dispersion [18], wall lubrication [19], and a liquid-phase turbulence model [20]. The full system therefore consists of three momentum equations, three mass-balance equations, one tensor equation for the Reynolds stresses  $R_{ij}$ , two scalar transport equations for  $\varepsilon$  and the elliptic-blending variable, and two interfacial area transport equations, yielding a total of 22 scalar equations. For the flexible-tube configuration, an Immersed Boundary Method (IBM) is employed to account for the motion of the central cylinder [21]. The acceleration obtained from the 2D fluid–structure interaction model is then projected onto a 3D flexible-beam model in order to predict the acceleration spectrum, which is subsequently compared with the experimental measurements. Drag and lift forces acting on the central cylinder are evaluated by integrating both viscous and pressure contributions over its surface.

## 2.6 Sum-up of the numerical approaches

Table 3 Sum-up of the numerical approaches from each partner.

| Partner | Software                          | Two-phase modeling                  | Turbulence model                                                | Type of Inlet                                                                                                         | Type of Outlet                                                                                          | Simulated case |
|---------|-----------------------------------|-------------------------------------|-----------------------------------------------------------------|-----------------------------------------------------------------------------------------------------------------------|---------------------------------------------------------------------------------------------------------|----------------|
| VTT     | Ansys-Fluent                      | VOF                                 | LES Smagorinsky                                                 | Velocity with SBM (Fluent UDF)                                                                                        | Pressure                                                                                                | T5             |
| CEA     | Neptune_CFD                       | Averaged Euler-Euler                | EBRSM (for liquid phase)                                        | Homogeneous                                                                                                           | Constant pressure                                                                                       | T5, G1         |
| NRG     | OpenFOAM 12 (Foundation branch)   | Hybrid two-fluid model (HZDR model) | K-Omega SST for liquid phase and K-Omega SST Sato for gas phase | Uniform velocities and volume fractions are set based on the experimental superficial velocities and volume fractions | Fixed pressure, zero-gradient for k and Omega, Void fraction, pressureInletOutletVelocity for velocity, | T5             |
| ASNR    | NeptuneCFD                        | Eulerian-EulerianGLIM               | EBRSM (only for the liquid phase)                               | Single large bubble synthesized                                                                                       | Constant pressure                                                                                       | T5             |
| EDF UK  | StarCCM+ v2024.6 Double Precision | Volume of Fluid (VOF) method        | $k - \omega$ SST                                                | Flat Inlet with mixture velocity and void fraction=0.7822                                                             | Pressure                                                                                                | T5             |

### 3. Calculation domain & Mesh

#### 3.1 VTT

The geometry and mesh used in the simulations are shown in Figure 10 and Figure 11. The calculation domain corresponds horizontally to the experimental setup with dimensions 216×300 mm, whereas vertically the total height of the domain was limited to 1100 mm. The height of the free-duct region before the tube bundle is 700 mm, which is about half of the free-duct in the experiment. A refined mesh is used in the tube bundle, with a characteristic cell size of 1 mm, i.e. half of that in the free duct. Cubic cells were used in the free duct to describe the bubbles accurately, together with close to cubic cells in the tube bundle. The total number of hexahedral cells is about 16 million. Note that non-conformal mesh interfaces exist between the tube bundle and free duct regions.

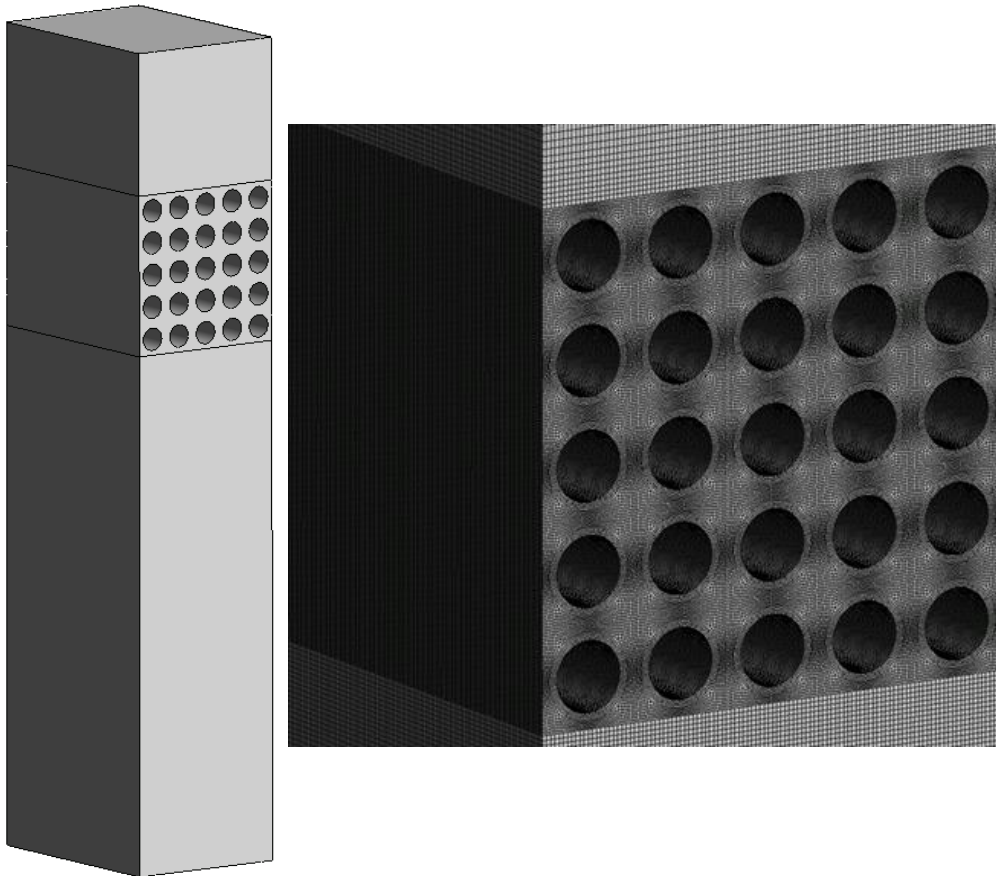


Figure 10 Geometry and mesh used in the VTT simulations.

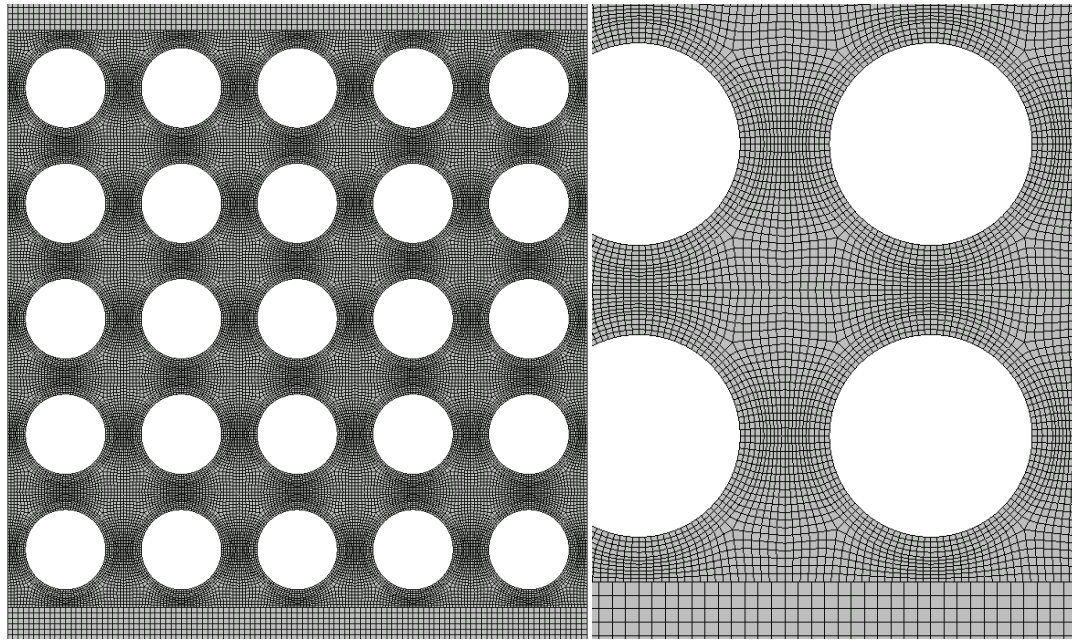


**Figure 11 Details of the VTT mesh at the tube bundle.**

### 3.2 CEA

The computational domain consists of a two-dimensional hexahedral mesh extruded in the third direction. A progressively refined boundary-layer meshing strategy is applied along the planar walls and the cylindrical rods. The domain accurately represents the full rod bundle, with upstream and downstream extensions corresponding to ten hydraulic diameters on both the inlet and outlet sides. Two operating conditions are considered, and therefore two meshes were generated. For case G1, the mesh contains 142,400 cells, with a typical cell size of 1.459 mm in the core region and 0.12 mm within the boundary-layer refinements. For case T5, the mesh consists of 320,100 cells, with an average size of 1.02 mm in the central region and 0.08 mm near the walls. The prescribed injection data do not directly provide the quantities to be injected for the two gas groups. To overcome this, we first analyse the distribution of void-fraction events at location P6. From this distribution, we infer an appropriate partition between the two groups. Using this partition, and by performing a manual balance in the inlet section based on the drag force, we determine both the injected void fraction and the required interphase slip velocity. To accelerate the convergence toward equilibrium without using a long inlet channel, a slip velocity between the liquid and the gas phases is imposed at the injection. The product of void fraction and superficial velocity is kept constant in order to preserve the total volumetric flow rate. For the bubble diameters, the probability density function (PDF) of the measured diameters is examined, and the two gas groups are separated using a critical diameter of approximately 11 mm. From this classification, a Sauter Mean Diameter is computed and then regularized so that the resulting total interfacial area matches the experimentally measured value.

For case G1, the liquid is injected with a velocity of 0.0542 m/s and a void fraction of 0.926. Gas group 1 is injected with a void fraction of 0.054, a velocity of 0.273 m/s, and a bubble diameter of 3.92 mm. Gas group 2 is injected with a void fraction of 0.020, a velocity of 0.272 m/s, and a bubble diameter of 11 mm.

For case T5, the liquid is injected with a velocity of 0.158 m/s and a void fraction of 0.670. Gas group 1 is injected with a void fraction of 0.160, a velocity of 0.347 m/s, and a bubble diameter of 3.28 mm. Gas group 2 is injected with a void fraction of 0.170, a velocity of 0.404 m/s, and a bubble diameter of 33.2 mm.

### 3.3 NRG PALLAS

The full bundle is modelled. The domain inlet boundary is located 1631 mm from the first row of the bundle (distance from the edge), upstream of the tube bundle. The outlet patch is located at 528 mm from the last row of the bundle (distance from the edge) downstream of the tube bundle. Total length of the domain is  $\delta x \times \delta y \times \delta z = 216 \text{ mm} \times 2159.2 \text{ mm} \times 300 \text{ mm}$ . The minimum mesh size is 2 mm. No prism layer is considered. The resulting mesh has a total of 3.5 M cells.

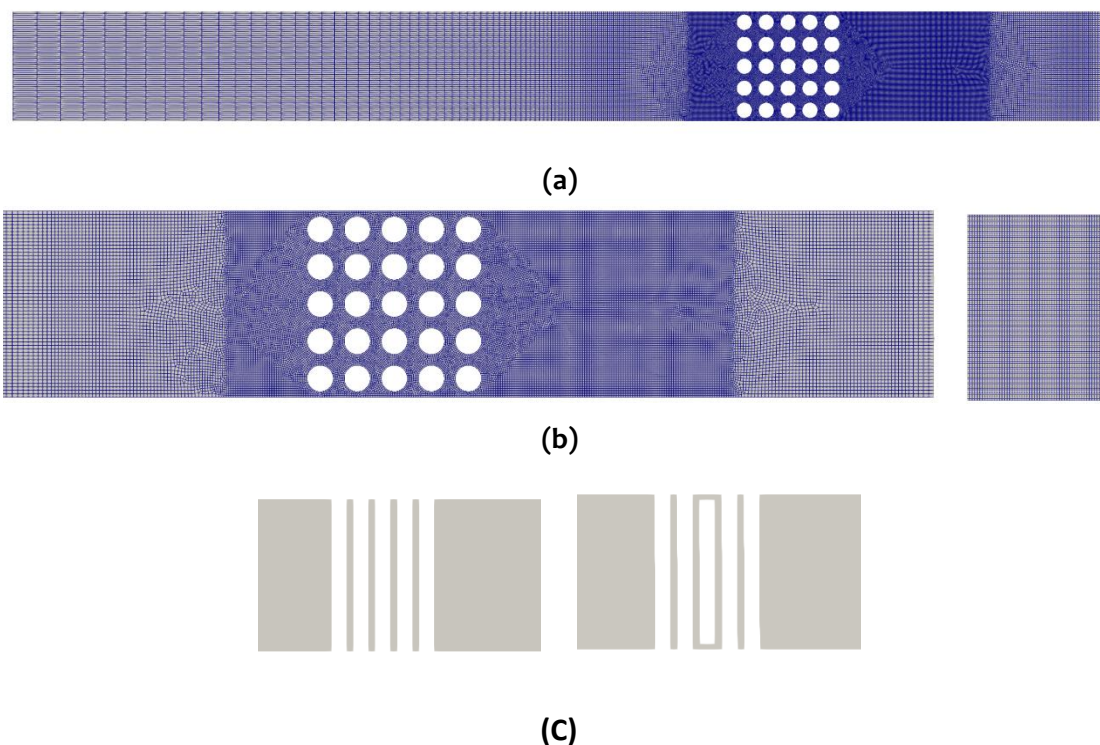


**Figure 12 Cross sections of the computational domain in the NRG PALLAS simulations. (a) Full domain in the x-y plane. (b) Left: mesh in the vicinity of the tube bundle. Right: cross-section of the channel in the x-z plane. (c) The side view of the bundle region for flow-only (left) and FSI (right) simulations**

Both cases are simulated for a total of 30 seconds. The last 10 seconds are used to extract the statistics. As shown in Figure 12, in the two-way coupled FSI simulations, the central tube is disconnected from the lateral walls by shortening the length of the tube by 2 cm (1 cm on each side) .

### 3.4 ASNR

The 3D numerical simulation of case T5 was realized, the entire tube length was simulated. The numerical domain is shown in Figure 13. The inlet of the numerical domain was placed at

a vertical position 0.5m upstream of the first row of the tube bundle, corresponding to about the middle of the tranquilization section.

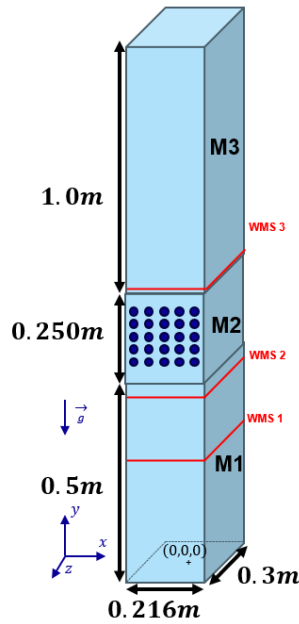


**Figure 13 Numerical domain, with the position of the “numerical” WMS. WMS1 and WMS2 correspond to the experimental WMS placed upstream and downstream of the tube bundle, respectively.**

The inlet boundary conditions were defined based on the WMS results measured upstream of the tube bundle. A large “bubble” with a diameter of 10cm was injected “explicitly” (void fraction equal to 1) in the middle of the inlet boundary face, with a frequency of 3.5Hz. A schematic of the boundary condition is given in Figure 14. The void fraction of the gas and liquid phases in the area outside of the bubble ( $S_o$  in Figure 14) was adapted at every time step as the large bubble area ( $S_b$  in Figure 14) gradually increases and decreases, in order to achieve the correct liquid and gas mass flow rates at every time step.

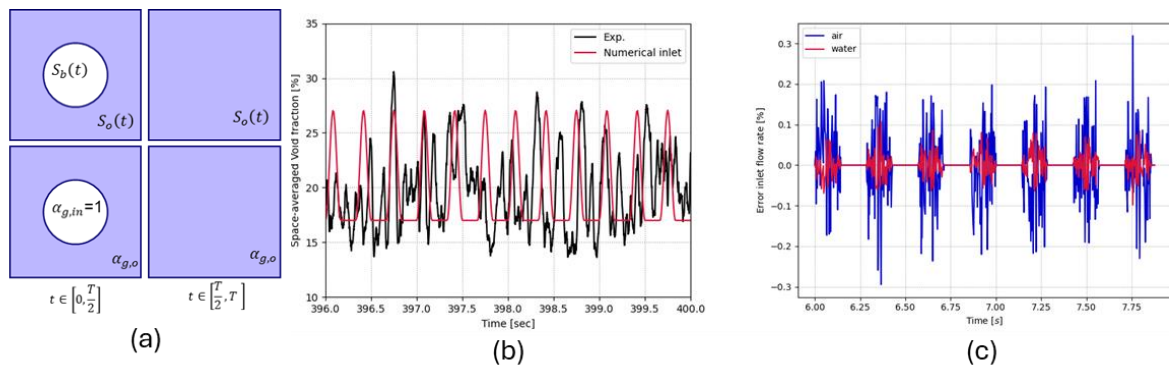


**Figure 14 Details of the numerical inlet boundary condition employed: (a) representation of the instantaneous bubble area ( $S_b$ ) injected with void fraction equal 1; (b) space-**

## averaged void fraction imposed numerically and the experimental value, as a function of time

The cell size in the bulk region is around 2mm. 150 layers of 2mm are used along the length of the tubes. Details of the mesh are provided in Figure 15.

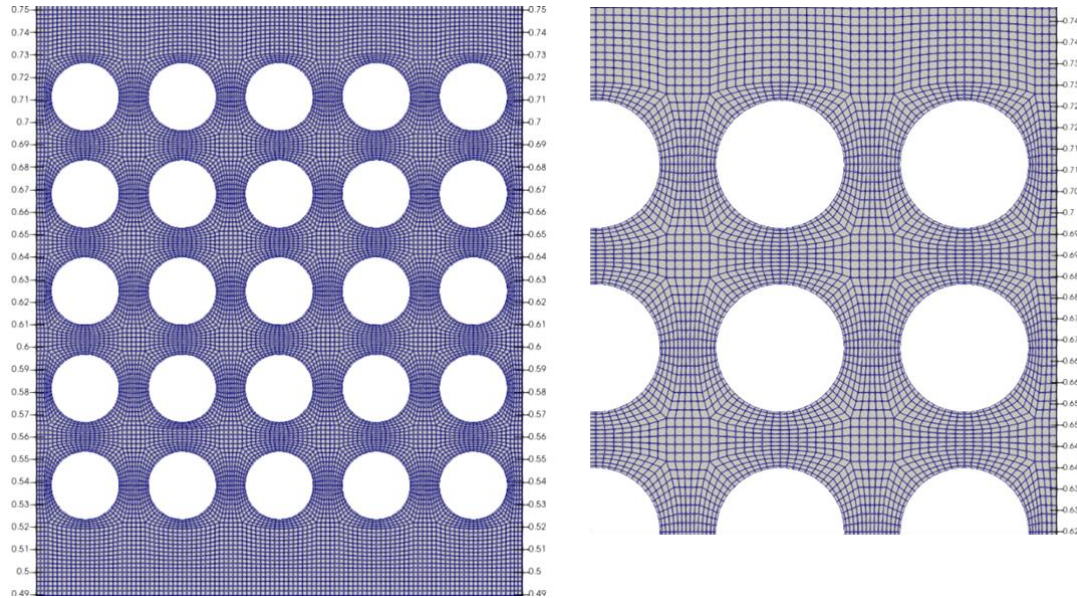


Figure 15 Details of the ASNR mesh

## 3.5 EDF UK

### 3.5.1 Geometry and computational domain for T5 configuration

The computational domain of the inline 5x5 tube bundle has been modelled as fully 3D with the same width of 10D in the spanwise direction as in the experimental configuration and sidewalls with no-slip condition for tube and side walls, shown in Figure 16. Initial computations and mesh sensitivity have been conducted with single-phase flow with water and a fully structured hexahedral grid, but the physics of the two-phase flow required further refinement in specific regions. Thus, the new mesh adopted was designed with a trimmed mesher of StarCCM+, which has been adopted with a 3.76 million cell baseline mesh with 1 level refinement, with  $y^+$  in the order of  $\sim 5$ , as provided in Table 1. Trimmed Cell mesh, which aligns a tetrahedral mesh on a Cartesian grid based on the geometry bounds, leading to a dominated hexahedral grid with resolution to capture expected scales, shown in Figure 16. Adaptive mesh refinement has been tested to reduce the cost of modelling the smallest scales and has also been assessed. However, once gas bubbles started interacting with the tube bank, adaptive mesh refinement became unstable and was switched off.

**Table 4 Computational domain details and computing expenses**

| Domain                          | 10D                             |
|---------------------------------|---------------------------------|
| Cell Count                      | 3.76 million cells (adaptation) |
| Base size[mm]                   | 3.0                             |
| Prism layers                    | 3                               |
| Prism layer total thickness[mm] | 0.75                            |
| Initial layer height            | 0.15                            |
| CPU time                        | 190 cores for 10500 CPU hrs     |

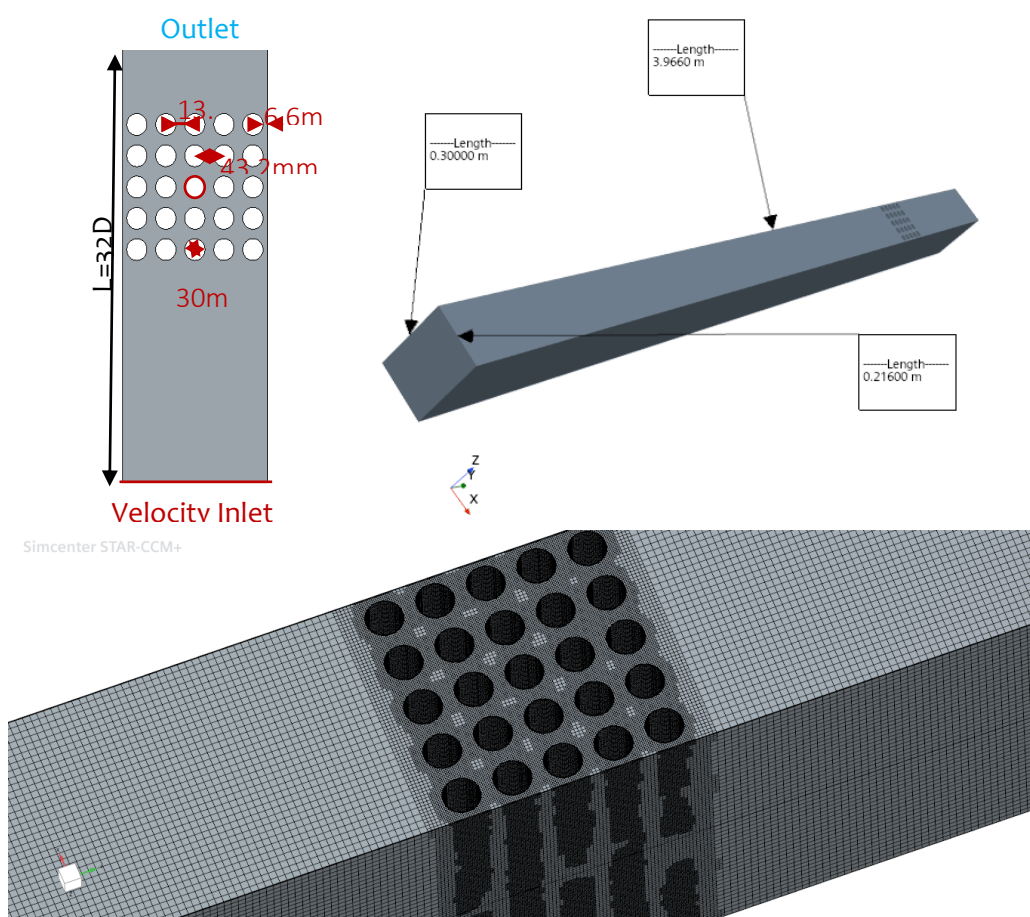


**Figure 16 Schematic and geometry of computation domain with the long inlet. Zoom-in section of the grid around the tube bundle region for T5 experimental configuration.**

### 3.6 Sum-up of the calculation domains & Mesh

Table 5 Sum-up of the calculation domain and mesh.

|               | 2D or 3D                   | Tube Length | Widthupstream tube bundle | periodicity              | Type mesh of                | Total number of cells     | Dr (mm) at cylinder wall | Dx_mean (mm)             |
|---------------|----------------------------|-------------|---------------------------|--------------------------|-----------------------------|---------------------------|--------------------------|--------------------------|
| <b>CEA</b>    | 2D (1 cell along cylinder) | 10D         | 0.3 m                     | No                       | Hexahedral                  | G1: 142400<br>T5 : 320100 | G1 : 0,12<br>T5 : 0,08   | G1 : 1,459<br>T5 : 1,022 |
| <b>VTT</b>    | 3D                         | 10D         | 700 m                     | none                     | Hexahedral                  | 16 M                      | 0.75                     | 1.0                      |
| <b>NRG</b>    | 3D                         | 280 mm      | 1631 mm                   | none                     | unstructured hexagonal mesh | 3.5 M                     | 2.0                      | 2.0                      |
| <b>ASNR</b>   | 3D                         | 10D         | 0.5m                      | -                        | Hexa                        | 2.5M                      | 0.5                      | 2.0                      |
| <b>EDF UK</b> | 3D                         | 10D         | 3966mm                    | In spanwise, z-direction | Hexahedral                  | 2.5M                      | 2.0                      | 2.0                      |

## 4. Rigid configuration

### 4.1 Qualitative analysis of the flow through numerical simulation

#### 4.1.1 VTT

The instantaneous and mean void fraction and the mean mixture velocity are presented in Figure 17 and Figure 18; these results are shown with the inlet bubble diameter of 20 mm. The bubbles start to breakup and merge almost immediately after the inlet, so that a bubble size distribution results. The largest bubbles are much larger than the initial size, and small bubbles are also obtained which the mesh cannot resolve accurately. The development of the time-averaged fields in the free duct region seems to be fairly rapid, and it is possible that the free duct length of 700 mm used in the simulation is sufficient. Negative mean velocity (i.e. downwards flow) and smaller bubbles are obtained near the free duct walls, which is qualitatively consistent with the experiments. At the tube bundle, the vertical mean mixture velocity is close to zero in the last gaps (between tube and duct wall), whereas fairly high velocities occur in the tube gaps. The void fraction increases significantly in the tube bundle compared to the free duct, consistently with the experiments.

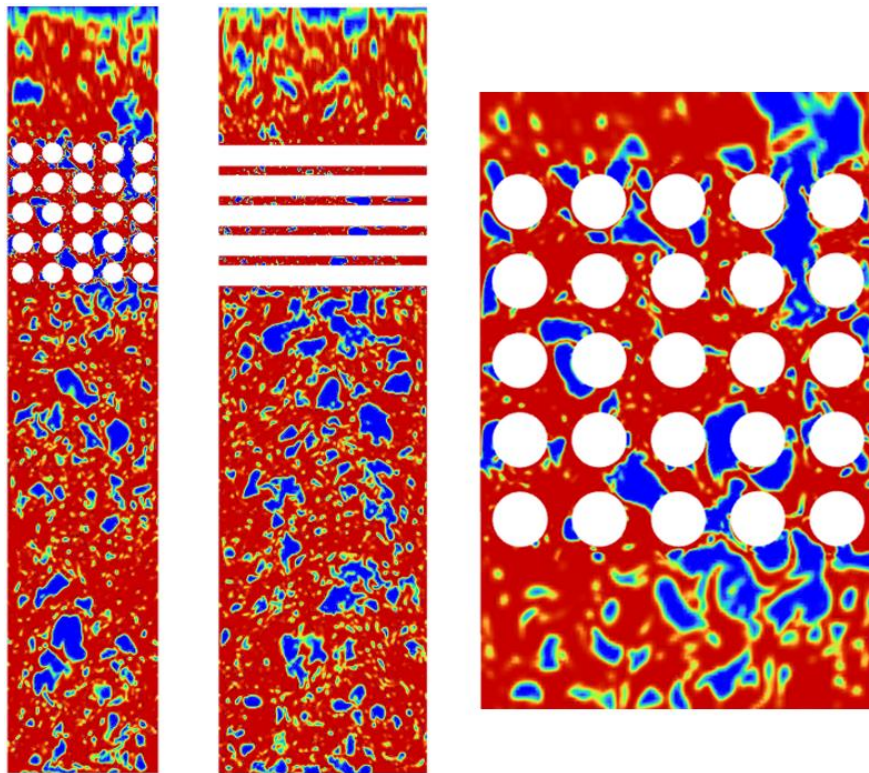


Figure 17. Instantaneous void fraction (air shown in blue) at the centre planes in VTT simulation with fixed geometry (experiment T5).

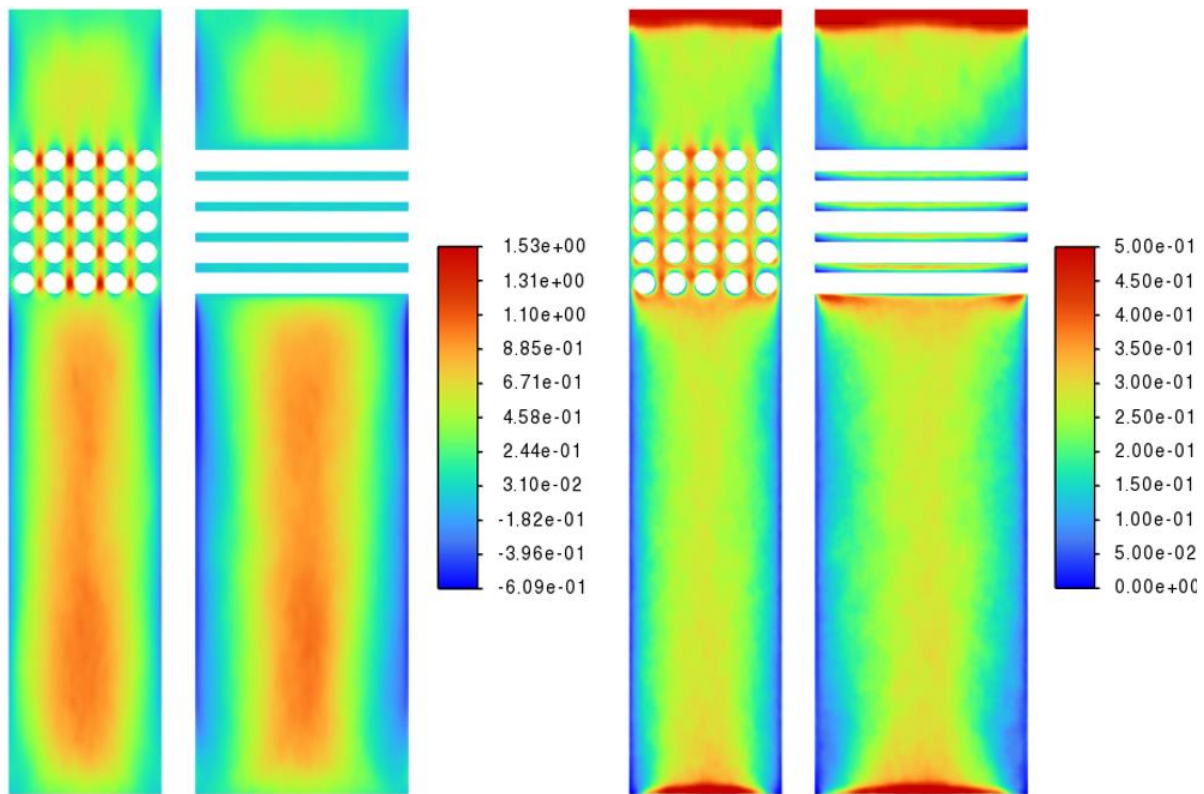


**Figure 18. Time-averaged vertical mixture velocity in m/s (left) and void fraction (right) at the centre planes in VTT simulation with fixed geometry (experiment T5).**

#### 4.1.2 NRG PALLAS

Inlet velocities for each phase are based on the experimentally provided superficial velocities and gas void fractions. Figure 20 shows the instantaneous and mean air void fraction and mean water streamwise velocity contours on the x-y and z-y planes, passing through the centre of the channel, for case T5.

As can be seen in the instantaneous void fraction plots in Figure 19, the simulation transitions to interface-resolving mode as the local mesh size decreases. Compared to experimental results, the resolved bubbles appear larger in volume. In other words, the hybrid model resolves two-phase structures too aggressively. The shift from unresolved/stochastic to resolved/deterministic morphology is driven by the interface-compression algorithm, which uses an arbitrary criterion not explicitly tied to sub-grid bubble size or local grid resolution. This limitation remains a key focus of ongoing research.

Once bubbles enter the high-resolution refined region, they undergo rapid breakup and coalescence, producing a broad bubble size distribution that includes bubbles significantly larger than the initial size as well as very small ones that the mesh cannot fully resolve. Moreover, examination of the time-averaged phase fraction fields reveals that the refined mesh region upstream of the tube bundle should be extended further upstream to allow a fully developed flow profile before the bubbles reach the bundle. Once again this development region is determined by the interface-compression algorithm and its length is also dependent on the two-phase flow regime. Further research is required to systematically investigate the effect of upstream refined length on bubble morphology at the bundle. The

mean void fraction shows a distinct local increase just before the first row of tubes. Within and downstream of the bundle, the mean water streamwise velocity field exhibits a relatively large asymmetry, most likely because the averaging time in the simulation was too short.

Near the free duct walls, the simulations consistently predict negative (downward) mean velocity accompanied by smaller bubbles, which qualitatively match the experimental findings. Overall, the simulations capture key flow features, such as wall-region downward flow, smaller bubbles near the walls, and the pronounced void fraction rise ahead of the tube bundle, in good agreement with experiments, though improvements in the upstream mesh refinement length and longer averaging times would help achieve more symmetric and fully developed mean profiles.

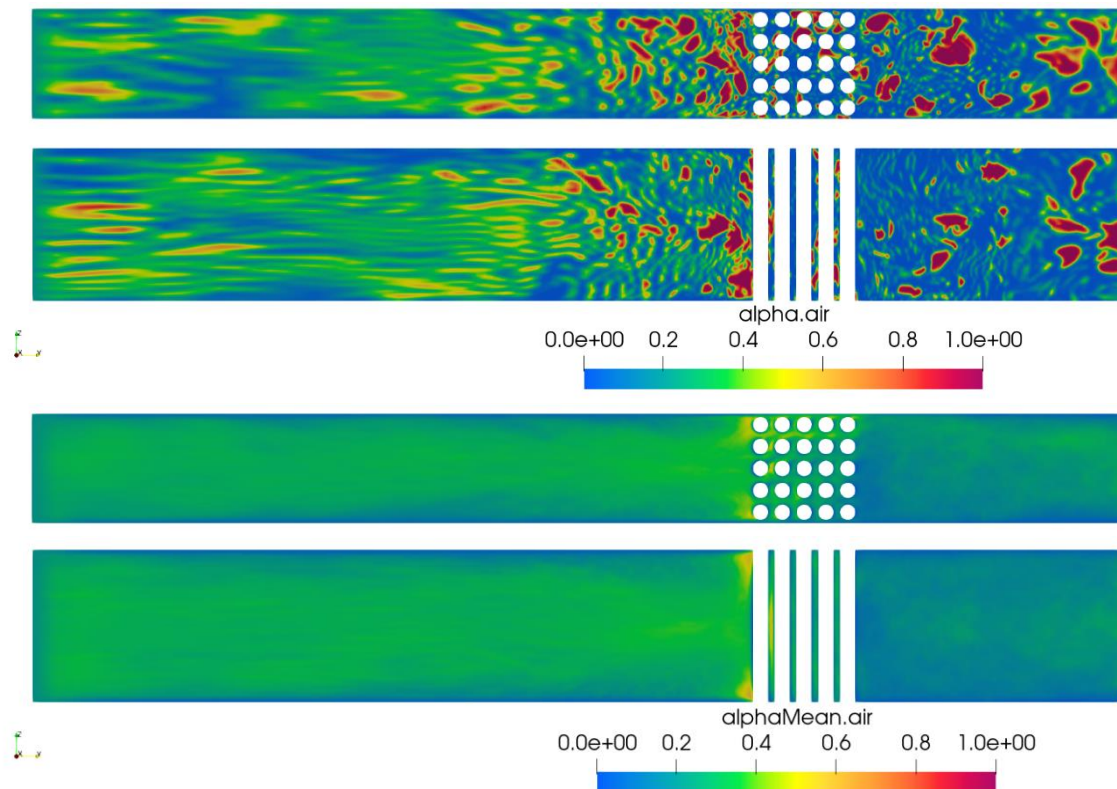


Figure 19 Instantaneous (top) and mean (bottom) void fraction at  $t = 20$  s, case T5.

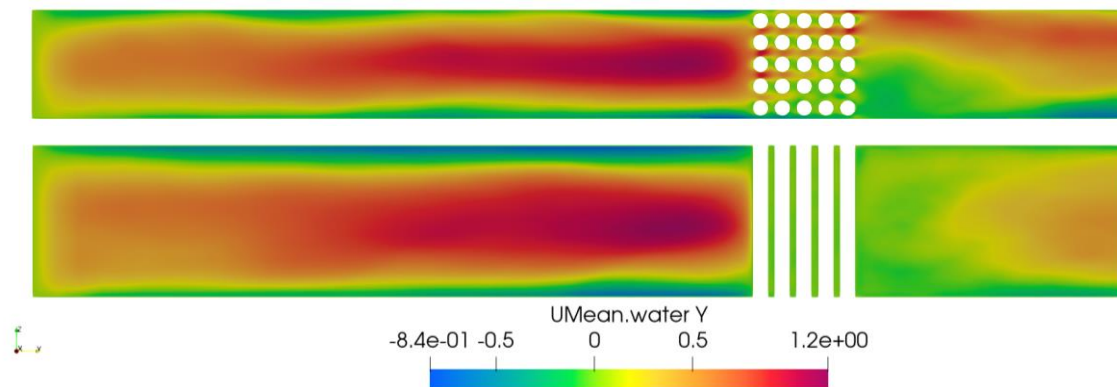


Figure 20 Mean water streamwise velocity at  $t = 20$  s, case T5.

### 4.1.3 CEA

We first focus on the G1 case [11]. Figure 1 displays the time-averaged total void fraction in our simulation in the top-left panel, along with the instantaneous void fraction in the top-right panel. These are to be compared with experimental photographs for cases T2 and T3 in the bottom panels (with G1 positioned between the two, closer to T3 in terms of injection) [12]. In our simulations, as shown in the top of Figure 21, we qualitatively observe that the void fraction is not homogeneous within the bundle, with higher values in the inter-tube regions and around the cylinders, and a zone of zero void fraction in the cylinder's wake. This description is consistent with previous simulations obtained in other configurations using our model [18]. When comparing our simulations with the experimental results (bottom of Figure 21), we note a qualitatively similar distribution, with most bubbles located in the inter-cylinder region or around the cylinders, and no-bubble zones behind the cylinders. We also observe preferential paths for the bubbles that are highly stable, as they are evident in the time-averaged visualization. Indeed, unlike T3, the group 2 (larger bubbles and gas pockets) is too weak to destabilize these preferential paths.

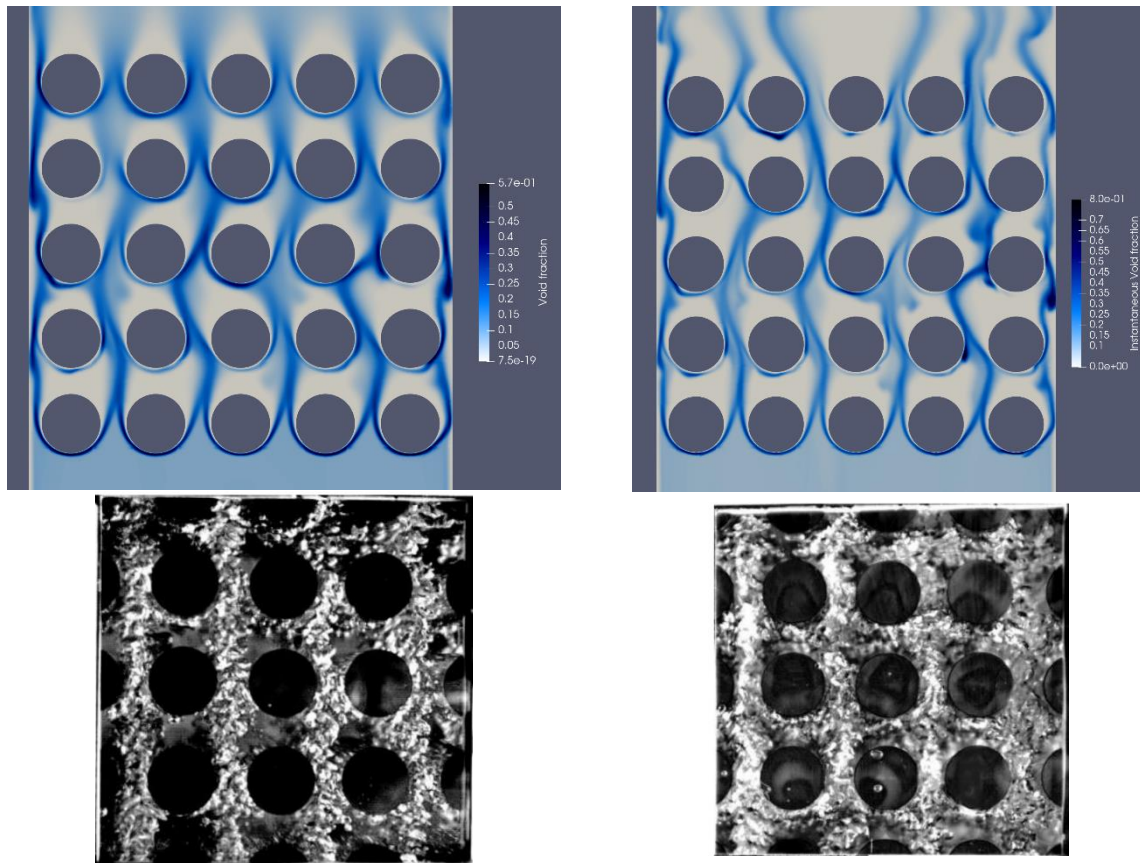


**Figure 21** Qualitative comparison of the time-averaged void fraction (top-left) and instantaneous void fraction (top-right) in G1 simulations with T2 (bottom-left) and T3 photographs (bottom-right) [12].

Figure 22 shows the time-averaged void fraction of nearly spherical bubbles (Group 1, left) and distorted bubbles (Group 2, right) in our G1 simulation. For Group 1 (left), the bubbles are primarily located in the inter-cylinder region with clearly defined preferential paths but are minimally diffused into the wake region. For Group 2 (right), the larger bubbles disperse more

into the wake regions but follow similar trajectories to those of the smaller bubbles. Given that their size is still relatively comparable to Group 1, they have a similar velocity and thus follow analogous paths. This explains why this configuration remains very stable in comparison with the experimental observations of T3. Figure 23 illustrates the time-averaged total gas velocity from the simulation. Accelerations are observed at the constrictions in the inter-cylinder regions, along with low-velocity zones in the wake of the cylinders, as expected. The paths taken by the two groups of bubbles are also discernible. Notably, the higher velocities observed behind the cylinders, where the overall void fraction is zero, are numerical artifacts resulting from the weighting of velocity by the respective void fraction of each group. This artifact arises because the velocity is averaged over regions where the void fraction is zero, leading to an apparent increase in velocity in those areas.

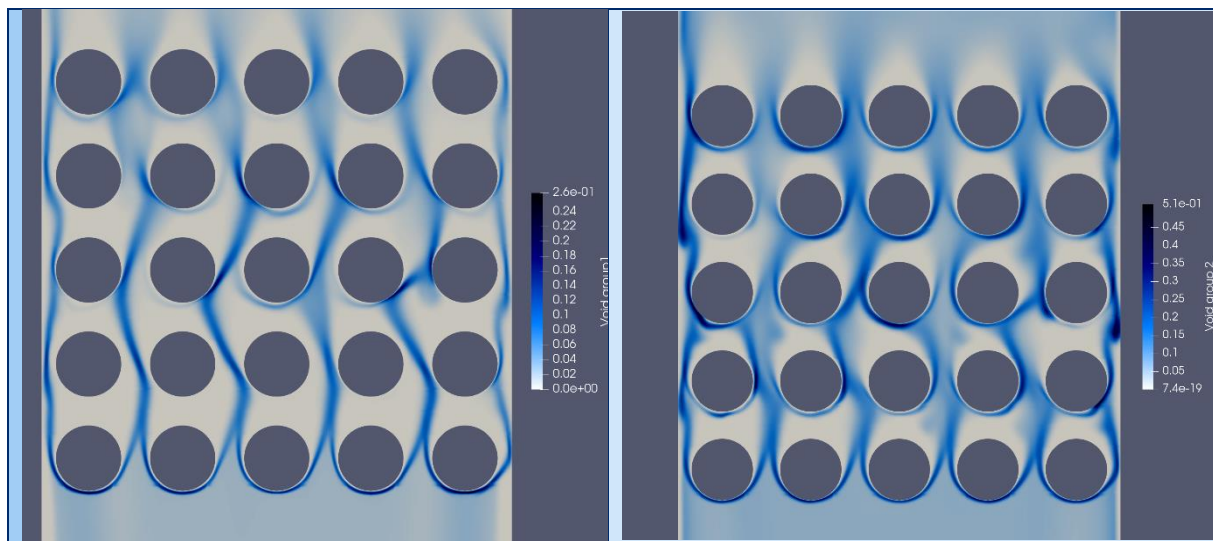


**Figure 22- Time-averaged void fraction for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in G1 simulations.**

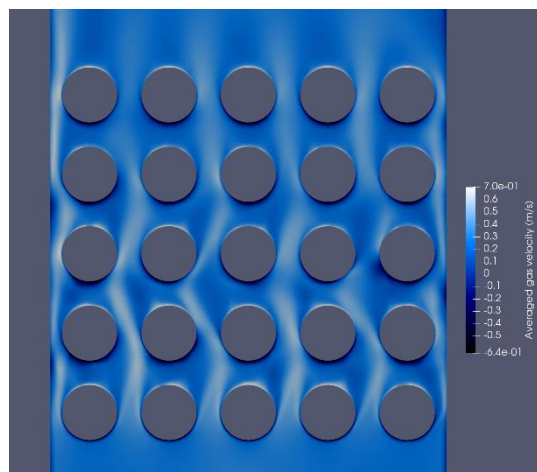


**Figure 23 Time-averaged gas velocity in G1 simulations.**

We now focus on the T5 case [11,12]. Figure 24 displays the time-averaged total void fraction in our simulation in the top-left panel, along with the instantaneous void fraction in the top-right panel. These are to be compared with experimental photographs of T5 in the bottom-right panel [12]. In our simulations, as shown in the top of Figure 24, we qualitatively observe that the void fraction is much more homogeneous within the bundle compared to G1, with a

slight concentration in the three central rows of tubes. We observe very small regions with no bubbles, which are mostly located in the near wake of the cylinders. This is due to the significantly increased cross-flow between the cylinders. This description is consistent with previous simulations obtained in this regime using our model. When comparing our simulations with the experimental results (bottom of Figure 24), we note a qualitatively similar distribution for the gas, which should be compared with the distribution of the two groups of bubbles. Unlike the G1 case, no preferential paths are distinguishable, which tends to homogenize the void fraction due to the presence of the second group of bubbles.

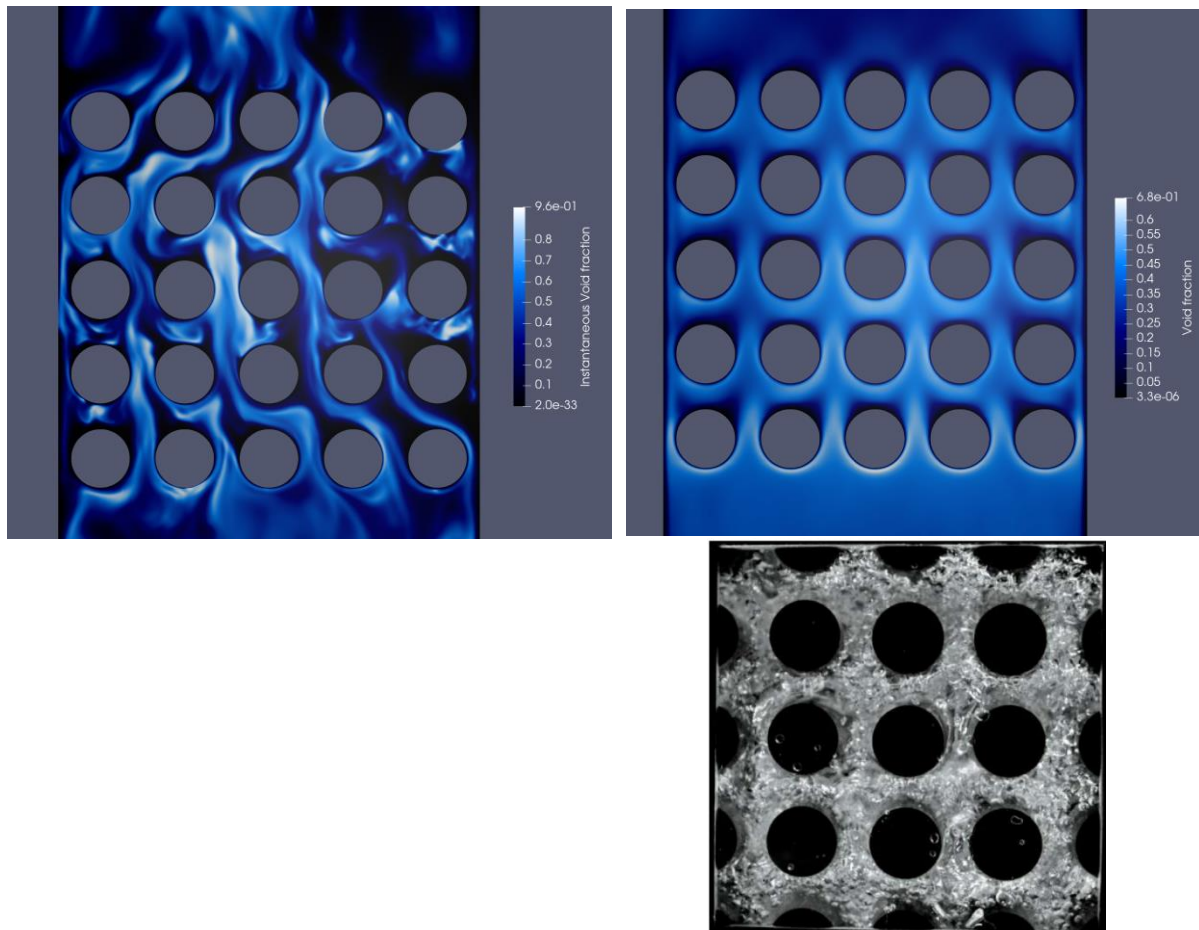


**Figure 24** Qualitative comparison of the time-averaged void fraction (top-left) and instantaneous void fraction (top-right) in T5 simulations with T5 (bottom-right) photographs (bottom-right).

Figure 25 shows the time-averaged void fraction of nearly spherical bubbles (Group 1, left) and distorted bubbles (Group 2, right) in our T5 simulation. For Group 1 (left), the bubbles are distributed homogeneously within the bundle, with preferential zones near the walls where their velocity is reduced due to the significant influence of the liquid velocity in these regions. Consequently, their presence is relatively higher in these near-wall zones compared to the central regions of the flow. For Group 2 (right), the highly deformed and larger bubbles are less affected by the near-wall regions and disperse more into the inter-cylinder zones. These structures concentrate in the three central rows, migrating toward the centre of the flow, and appear to occupy large spaces between the cylinders as well as in the wake behind the cylinders. These observations seem compatible with the experimental description reported in Giuseppe Spina's thesis [12]. Figure 23 illustrates the time-averaged total gas velocity from

the simulation. Similar to the G1 case, accelerations are observed between the cylinders. However, this time, the bubble velocities are more conventional, distributed in a more homogeneous and symmetric manner within the bundle. Compared to G1, the velocities at the end of the bundle disperse more rapidly, primarily due to increased turbulence and homogenization caused by the large gas structures.

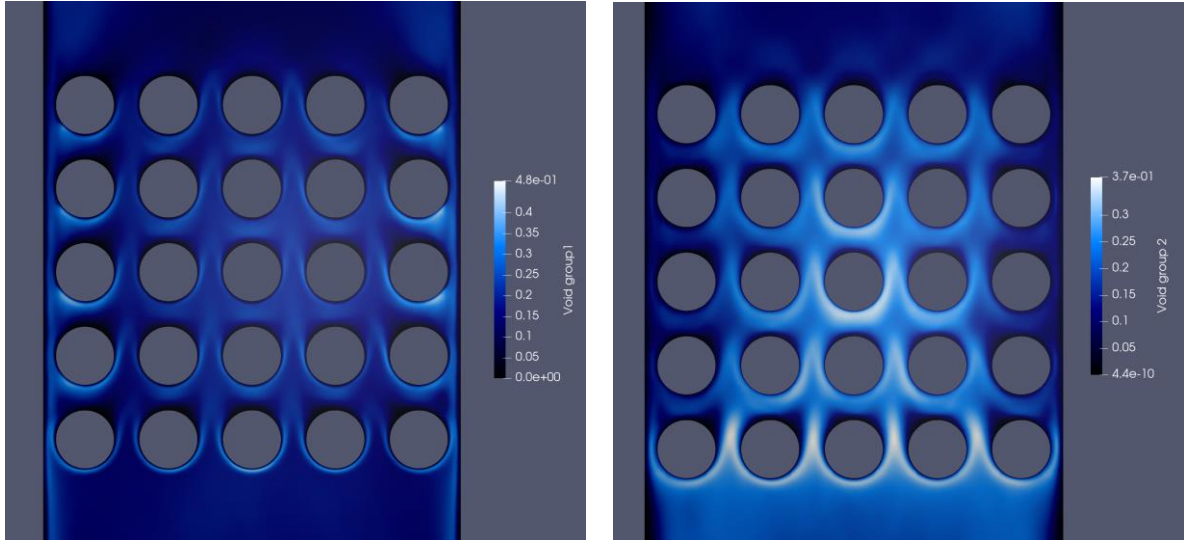


**Figure 25** Time-averaged void fraction for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in T5 simulations.

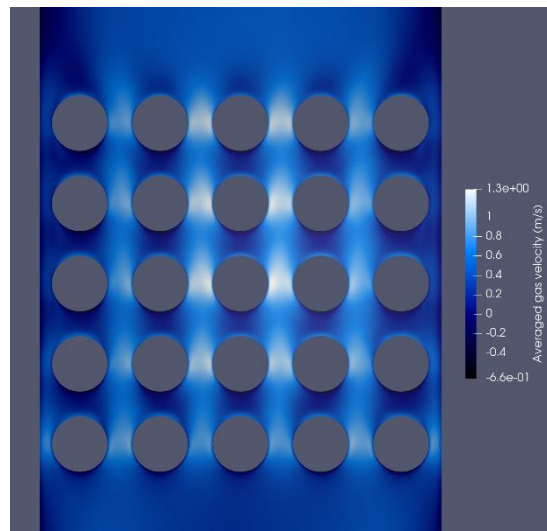


**Figure 26** Time-averaged gas velocity in T5 simulations.

#### 4.1.4 ASNR

Simulations were run along 20 seconds of physical time, and statistics were recorded starting from 6 seconds. In the central region of the numerical domain, one can observe the presence of some void fraction clusters, similar to large gas structures. Their size ranges from about 3 to 6cm, in both the frontal plane (Figure 27 (left)) and lateral plane (Figure 27 (middle)). These clusters are surrounded by large regions featuring a dispersed phase at a void fraction equal to about 20-50%.

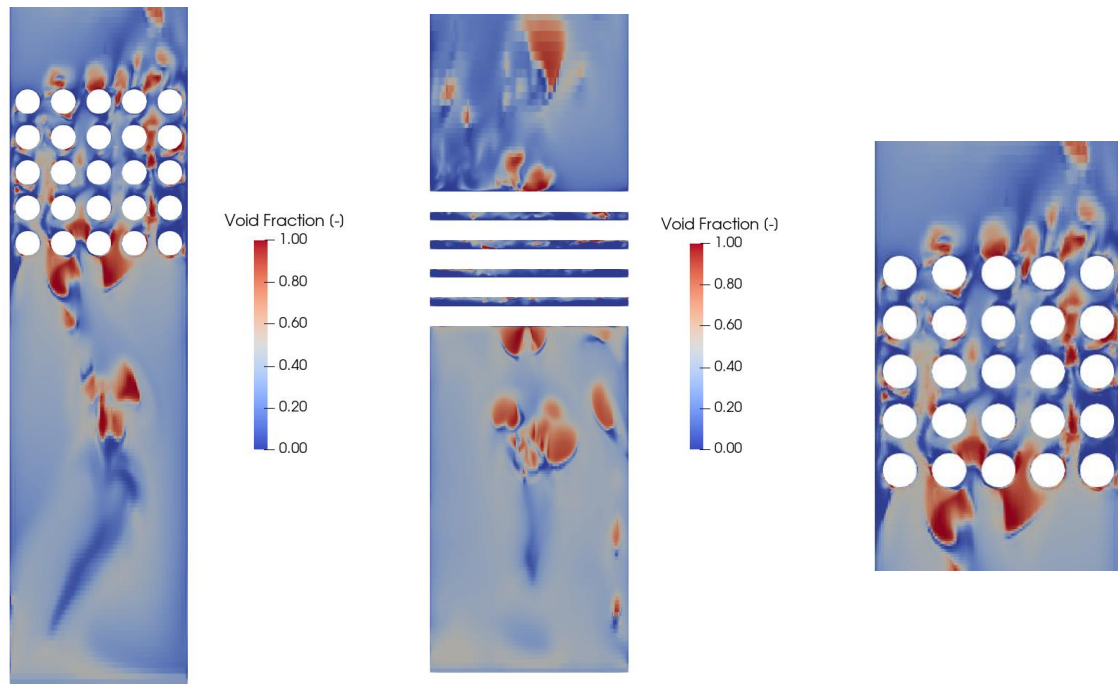


**Figure 27** Instantaneous void along (left) the symmetry frontal plane, (middle) the symmetry lateral plane and (right) zoom on the tube bundle region of the frontal plane.

#### 4.1.5 EDF UK

The flow was simulated for a total of  $\sim 42.65$ s and averaged beyond  $\sim 12.5$ s to remove the transient data once the flow had been established. Contours of the instantaneous and time-averaged void fraction distribution are presented in Figure 28-29, where bubbles are seen to develop, break-up and coalesce into a stable droplet size distribution, forming large air structures that are convected across until they reach the tube bundle section. This unsteady two-phase air-water cross-flow interaction with the tube bank generates dynamic forcing due to a combination of turbulence fluctuations and dynamic inertia effects from the impingement of intermittent two-phase flow on tube walls. It is also worth noting that from the flow animation, backflow is observed close to the side walls of the channel, a feature that is questioned whether it is a deficit of the CFD model or is actually present in the experiment. This was also verified from the vertical velocity profiles, which show negative velocity flow-reversal at side-walls (unclear whether this is representative of the test observations). Bubbles were seen generated from the inlet condition as a result of shear, inertia, buoyancy and surface tension effects. Blob detection was also enabled for the break-up and coalescence of resolved/unresolved blobs when the flow reached the tube bundle section.

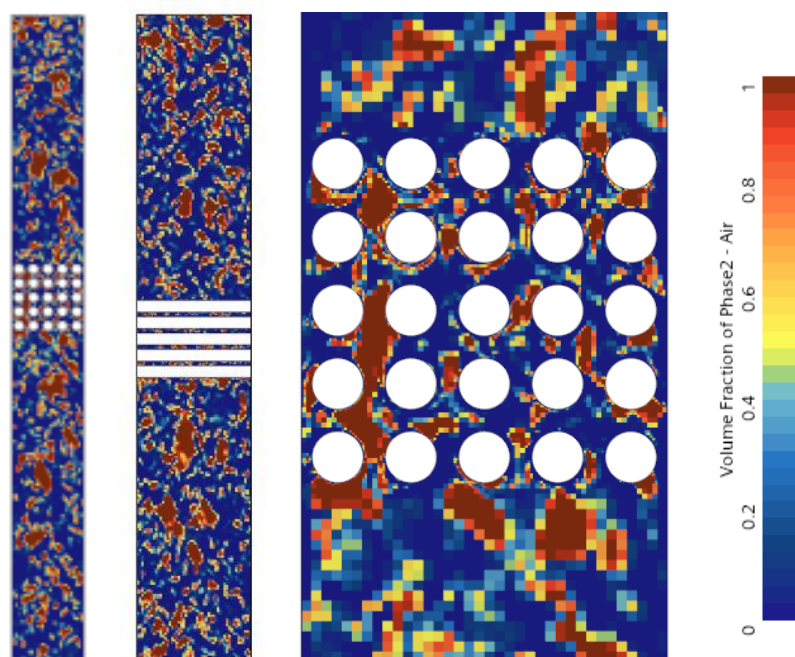


Figure 28 Instantaneous void fraction distribution at the centre planes from the T5 case in fixed configuration from the last time step of the EDF UK simulations.

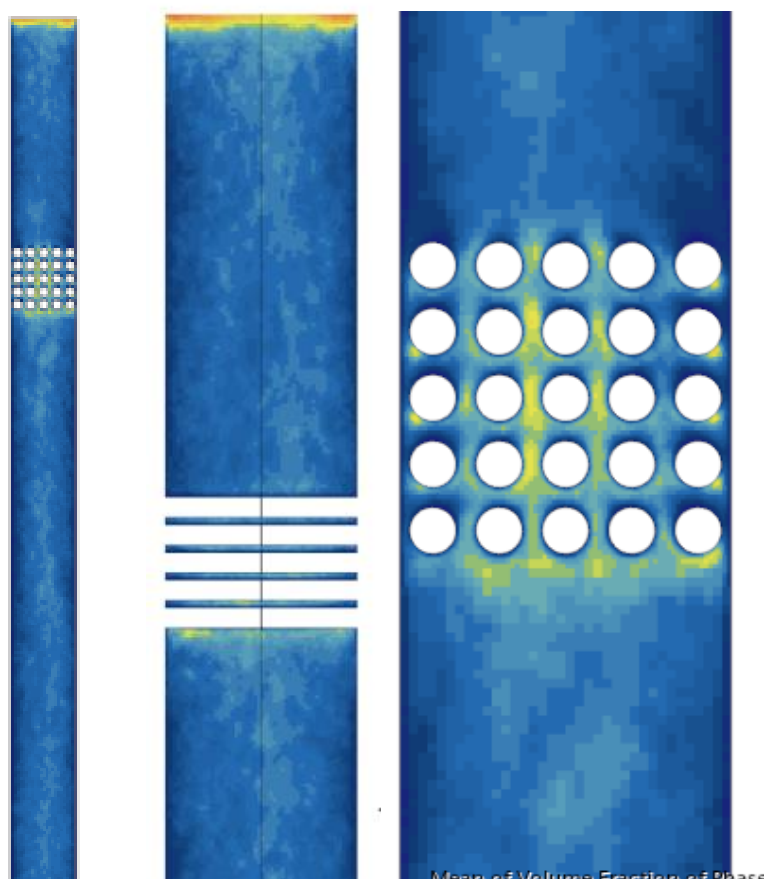


Figure 29 Time-averaged void fraction distribution at the centre planes from the T5 case in fixed configuration from the last time step of the EDF UK simulations.

## 4.2 Void fraction: Numerical vs Experimental results

For the case T5, partners did a cross comparison at the WMS upstream/downstream locations for void fraction and at the different bi-optical probe measurements in the tube bundle. T5 is a challenging case since the pattern is intermittent in the tube bundle, due to the upstream churn flow. This is due to the low liquid mass flow rate resulting in a high slip between phases. This kind of flow, coupled to the high density ratio of the air-water mixture, is very challenging to simulate.

In Figure 30, upstream (left) and downstream (right) void fraction profile are presented. First, it is important to note that the upstream experimental profile is not fully symmetric and that it is core peaked; illustrating the role played by larger gas structures at the center of the section. In terms of numerical simulations, predictions coming from the three partners using hybrid approaches appear to be different from the experiment in terms of amplitude and profile, whereas two partners using a VOF approach obtain numerical results in a satisfactory agreement with the experiment. If we look at the qualitative analysis previously described, it is possible to see that the discrepancy might come from the calculation domain. With a reduced inlet domain, with a 2D domain (1 cell along tubes), gas structures are not as resolved as with simulations using a full domain, leading probably to these kinds of discrepancies in terms of resulting void profile. In fact, hybrid models have already shown their ability to predict this kind of flow in literature, thus it highlights here the role played by the length of establishment in two-phase flow. Downstream, thanks to the mixing effect coming from the tube bundle, void fraction profiles are closer to the experiment, especially at the center of the duct. However at the wall, we still have discrepancies. An interesting fact to note here is that the influence of the tube bundle geometry is observed in different numerical simulations, whereas in the experimental profile it is not, illustrating a discrepancy in terms of dispersion downstream the tube bundle.

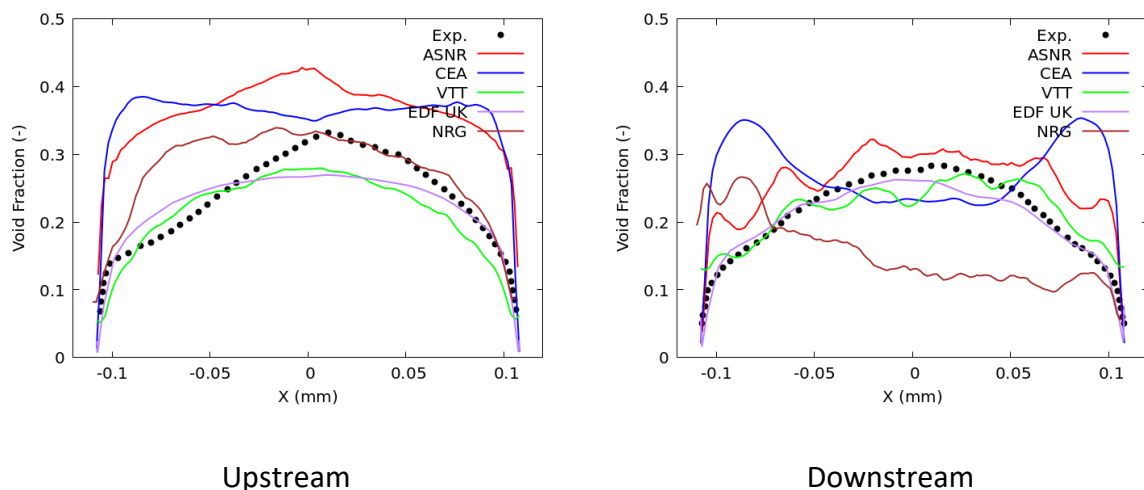


**Figure 30 Upstream/downstream void profile along x from each partner compared to the experiment.**

In Figure 31, void fraction measured inside the tube bundle by the bi-optical probe is compared with numerical prediction from each partner. Experimental measurements are shown with error bars of  $\pm 15\%$ . For most points, numerical predictions are around 15% of

error or less, this is related to the upstream prediction of the flow, where if the void fraction upstream is too high, predictions are also over estimated in the tube bundle. An interesting effect to note is that the void fraction is different depending if the location of the probe is in the gap or not, this effect is well reproduced by most of the simulations. The main discrepancies appear downstream at location 0, which is confirmed by the downstream profile previously discussed. Thus, the two-phase flow at the outlet of the tube bundle is not well predicted. For now, this is not well understood and needs further investigation.

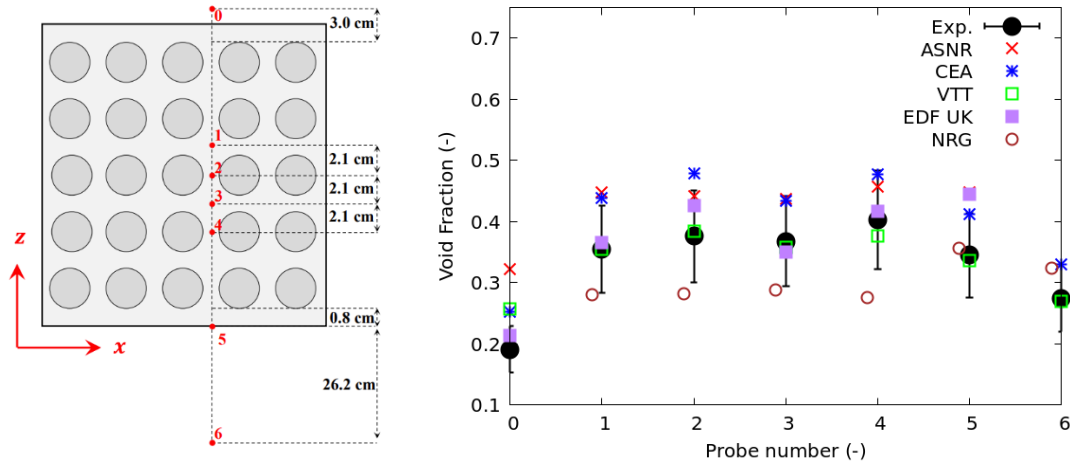


Figure 31 Void fraction comparison inside the tube bundle. Point locations (left) and numerical results compared to experiment (with error bars of  $\pm 15\%$ ) (right).

#### 4.2.1 CEA analysis on their quantitative results

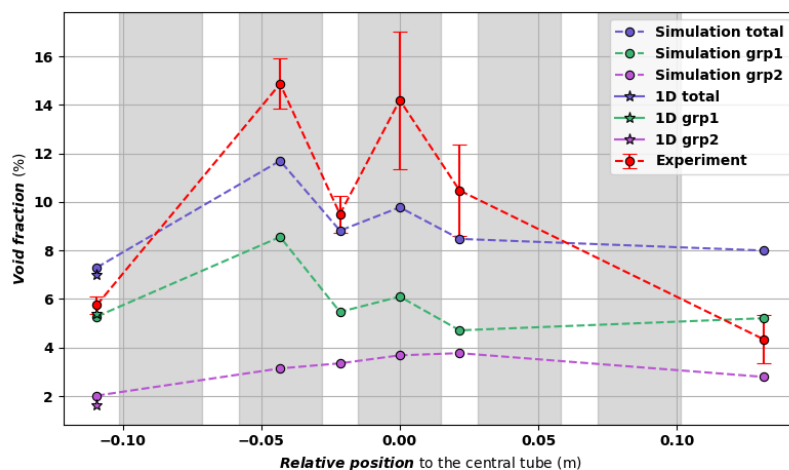


Figure 32 Time-averaged void fraction within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in G1 simulations compared with experimental data.

We first focus on the G1 case. Figure 32 shows the time-averaged void fraction within the bundle for Group 1 (nearly spherical bubbles, left) and Group 2 (distorted bubbles, right) in the G1 simulations compared with experimental data. In the experiment, depicted in red, the void fraction tends to increase the constriction zones between the tubes and decrease outside these zones. In our simulations, we have two groups of bubbles. We observe that only Group 1 shows a similar trend to the experiment, as the void fraction of the second group tends to

increase and then stabilise at the exit of the bundle. This is due to the size of these bubbles relative to the inter-tube spacing, causing them to concentrate between the tubes due to near-wall effects. We note that the sum of the two groups tends to follow the overall behaviour of the experiment, despite some discrepancies. However, the differences between the total void fraction and the void fraction of Group 1 compared to the experiment suggest that the experiment does not always accurately capture the bubbles of the second group, particularly in regions with larger diameters (upstream and downstream of the bundle). Nevertheless, it seems to correspond to the separation between the two groups within the interior. This would explain the differences between the experimental and simulation results. It is noteworthy that the first measurement point, P5, in our simulations and 1D calculations corresponds, indicating that our solution remained very homogeneous before entering the bundle.

Figure 33 shows the time-averaged gas velocity within the bundle for Group 1 (nearly spherical bubbles, left) and Group 2 (distorted bubbles, right) in G1 simulations, compared with experimental data. Experimentally, a high velocity is observed upstream of the bundle, which is greater than the velocity at the constriction. This appears surprising given the average bubble size. By calculating the equilibrium using experimental data and correcting the first velocity point by the experimental void fraction to maintain the flow rate, we obtain an experimental velocity of 0.3546 m/s. This results in a 30% discrepancy compared to the measured value, reducing the error between simulation and experiment from 40% to 15%. This initial discrepancy may be due to a highly inhomogeneous upstream flow or an anomalous value. Subsequently, the velocity errors for the first three rows range between 1.7% and 9%. In the simulation, the velocities then homogenize to reach the average velocity, unlike in the experiment. Finally, the exit velocity of the bundle exhibits a 46% error due to an acceleration at the bundle exit that is not numerically reproduced. This difference can be explained by the variation in the position of the oscillation at the bundle exit, which can induce significant acceleration. In our simulations, this oscillation is closer to the bundle than the measurement point.

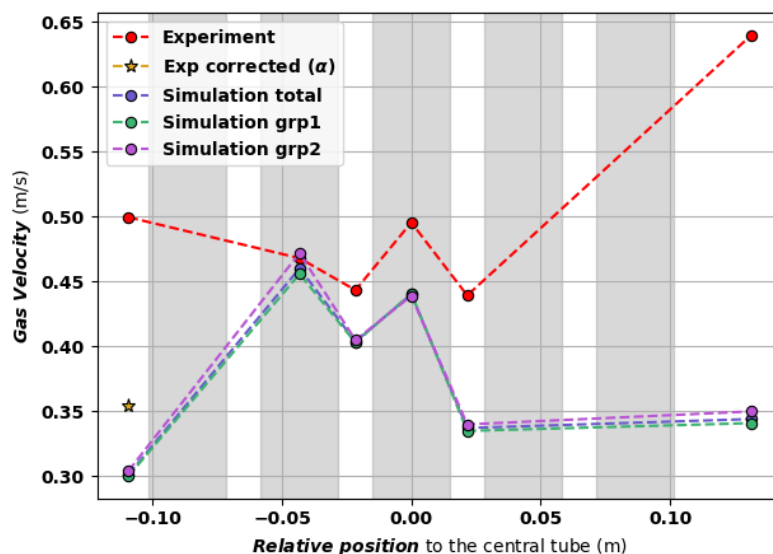


**Figure 33** Time-averaged gas velocity within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in G1 simulations compared with experimental data

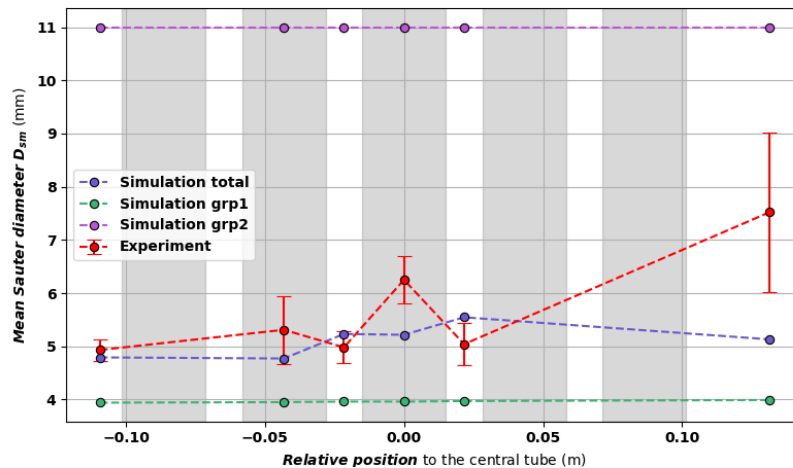


**Figure 34 Time-averaged Sauter mean diameter within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in G1 simulations compared with experimental data.**

Figure 34 shows the time-averaged Sauter mean diameter within the bundle for Group 1 (nearly spherical bubbles, left) and Group 2 (distorted bubbles, right) in G1 simulations, compared with experimental data. Experimentally, we observe that the diameter follows a trend similar to the experimental void fraction, except at the bundle exit point. This discrepancy at the exit can be attributed to either the high velocity at this point, which reduces the measured interfacial area and thus artificially increases the Sauter mean diameter, or to coalescence. For the remaining points, the experiment and simulation are in good agreement. The simulation data indicate minimal coalescence, and the void fraction distribution in our simulations corresponds well with the experimentally obtained trends. This suggests that the acceleration is the more likely cause of the observed differences.

We now focus on the T5 case. Figure 35 shows the time-averaged void fraction within the bundle for Group 1 (nearly spherical bubbles, left) and Group 2 (distorted bubbles, right) in the T5 simulations, compared with experimental data. In the experiment, depicted in red, the void fraction tends to increase in the constriction zones between the tubes and decrease outside these zones, but to a lesser extent than in G1. Additionally, a significant reduction in the void fraction is observed at the exit of the bundle. In our simulations, we still have two groups of bubbles. We observe that only Group 2 shows a similar trend to the experiment, as the void fraction of Group 1 tends to be relatively constant with a slight drop at the exit of the bundle. Group 2 exhibits a 28% difference between the void fraction measured in our simulations and that obtained through 1D calculations. This behaviour indicates a non-homogeneous redistribution upstream of the bundle, which was previously observed in Figure 5. This explains the discrepancy between the expected 1D value, corresponding to the experimentally measured void fraction, and that measured in our simulations, with an error of 24% but close to the experimental uncertainty. The overall behaviour of the total void fraction in our simulations is generally well-predicted, with a maximum error of 28%, and remains compatible with the experimental measurements. This suggests that there may be fewer Group 2 bubbles relative to Group 1 at injection or in their distribution toward the centre of the channel.

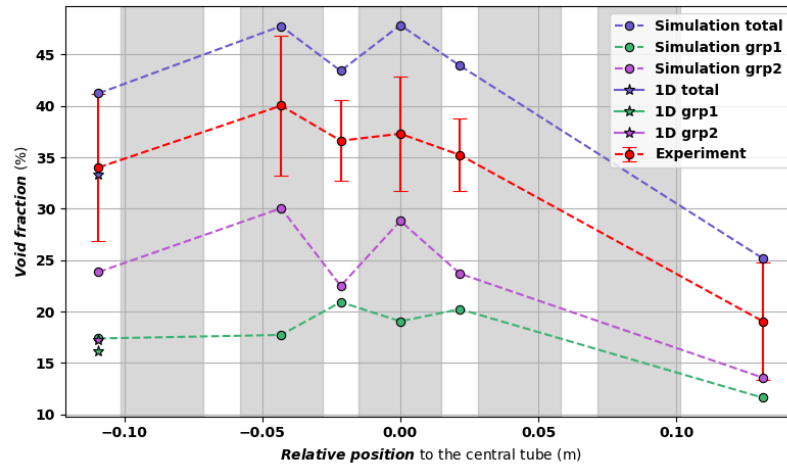


**Figure 35** Time-averaged void fraction within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in T5 simulations compared with experimental data.

Figure 36 shows the time-averaged gas velocity within the bundle for Group 1 (nearly spherical bubbles, left) and Group 2 (distorted bubbles, right) in T5 simulations, compared with experimental data. As for G1, a high velocity is observed experimentally upstream of the bundle, which is greater than the velocity at the constriction. This is also surprising given the higher mean velocity but could be attributed to large structure velocities. We also computed the equilibrium using experimental data and correcting the first velocity point by the experimental void fraction to maintain the flow rate, we obtain an experimental velocity of 0.3774 m/s. This results in a 63% discrepancy compared to the measured value, making the simulation results more compatible with the experimental data. However, this could be due to large structure not well reproduced in our simulations. Subsequently, the velocity errors for the first four rows range between 1.0% and 23%. Finally, the exit velocity of the bundle exhibits a 64% error due to an acceleration at the bundle exit that is not numerically reproduced. As for G1 case, this difference can be explained by the variation in the position of the oscillation at the bundle exit, which can induce significant acceleration.

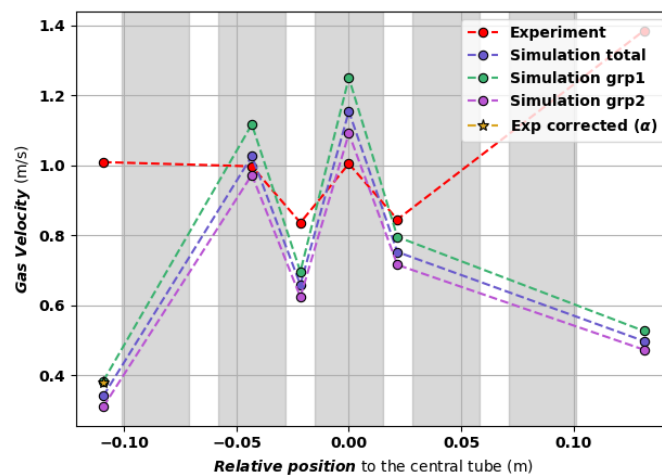


**Figure 36** Time-averaged gas velocity within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in T5 simulations compared with experimental data.

Figure 37 shows the time-averaged Sauter mean diameter within the bundle for Group 1 (nearly spherical bubbles, left) and Group 2 (distorted bubbles, right) in T5 simulations, compared with experimental data. Experimentally, we observe that the diameter remains almost constant within the bundle, except at the exit. As in the G1 case, this discrepancy at the exit - a 45% error - could be attributed to either the high velocity at this point, which reduces the measured interfacial area and thus artificially increases the Sauter mean diameter, or to coalescence. For the remaining points within the bundle, the experiment and simulation are in good agreement - between 3% and 35%.

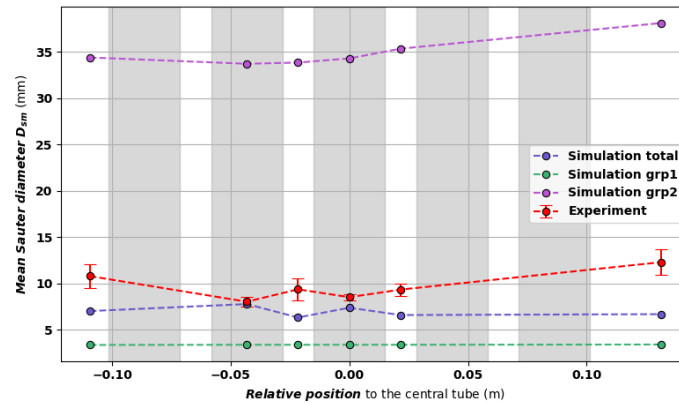


**Figure 37** Time-averaged Sauter mean diameter within the bundle for group 1 (almost spherical bubbles, left) and group 2 (distorted bubbles, right) in T5 simulations compared with experimental data.

#### 4.2.2 EDF UK analysis on their quantitative results

This section presents quantitative comparisons between the VoF EDF R&D UK computations against experimental or digitised data where available. In Figure 38, the time-averaged void fraction distribution predicted from CFD downstream of the WMS follows the trend of the experimental data, showing a near symmetric profile. Though the CFD computations return a parabolic upstream profile which does not capture the asymmetric behaviour and the peak measured from the WMS in the experiment, suggesting that at the upstream position, the VOF simulation shows more bubbly flow rather than intermittent.

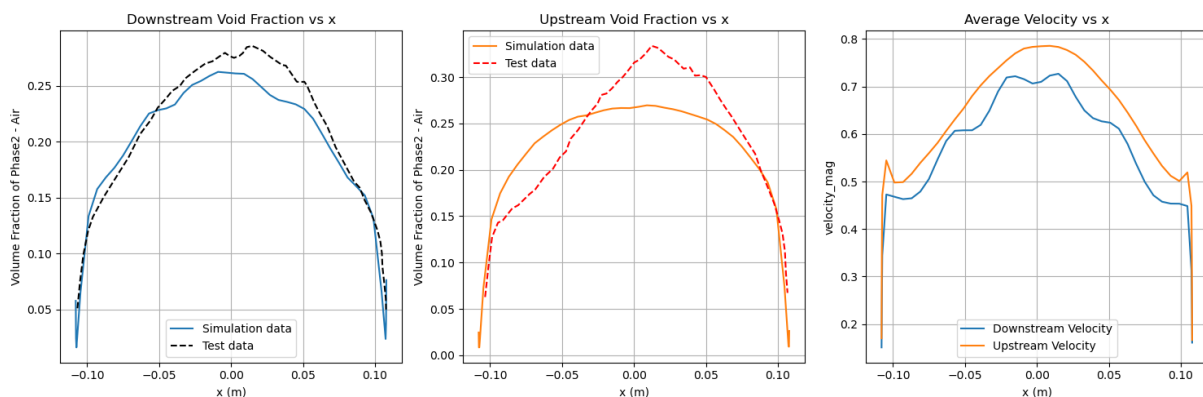


**Figure 38** Distribution of void fraction in the lateral cross-flow x-direction for downstream and upstream WMS plane locations and average mixture velocity in the T5 configuration.

Examining the radial void fraction distribution shown in Figure 39 the model follows closely the experimental data both in the upstream and downstream radial location of the WMS upstream/downstream. The deep seen in the radial distribution profile across both upstream and downstream profiles is due to the side walls.

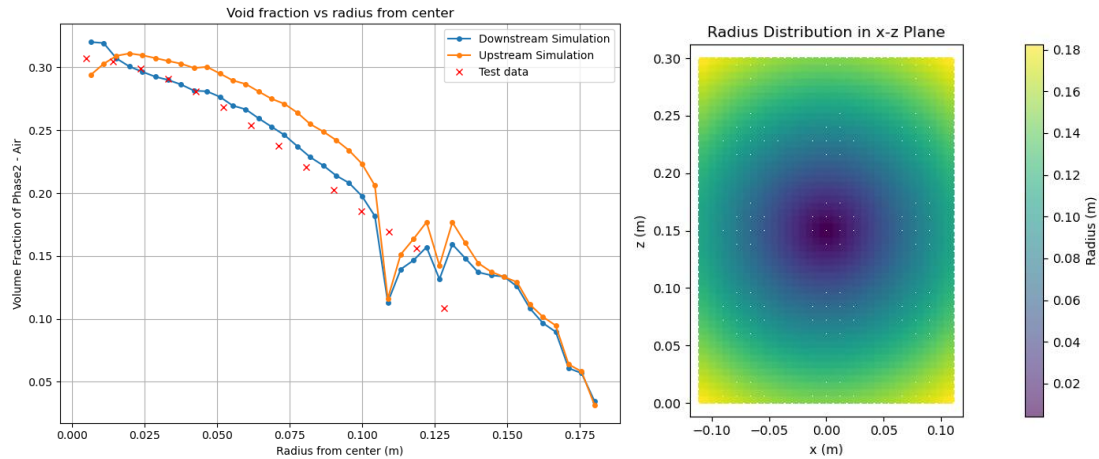


**Figure 39 Radial and cross-plane distribution of Void fraction**

The probability density function of the bubble size distribution in the tube bank region is shown in is shown in Figure 39. As seen, the computation predictions return high PDF for bubble diameters in the range of 1-3mm compared to the experimental data. One reason is that the numerical computations suggest the presence of greater proportion of small bubbles in the flow than seen in the probe test data. It's worth noting here that a single probe location was used in the experiment suggesting higher uncertainty whereas the simulation data is taken across a whole slice of the model.

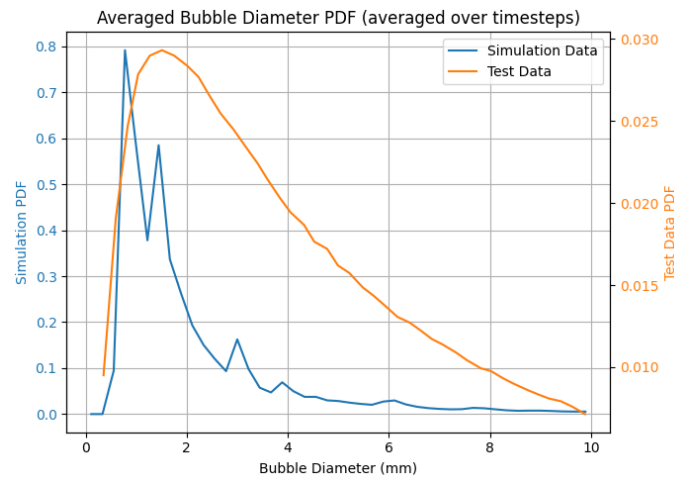


**Figure 40 PDF of bubble diameter distribution for T5 configuration**

Figure 40 shows the average void fraction in the tube bundle region compared with the digitised data from the location of the measurements from the dual optical probes and those the VoF simulations. As seen the simulation predicted void fraction close to the experimental data, though a higher discrepancy is shown at P5 and P6 location where the VOF over/under predicted the void fraction due to the gas accumulation close to the wall. Figure 42-43 present the PSD of void faction at WMS upstream, downstream and at various probes locations along

with the forces acting on the cylinder. As seen large gas structure are present at frequencies 0.5-4Hz at upstream and downstream WMS position.

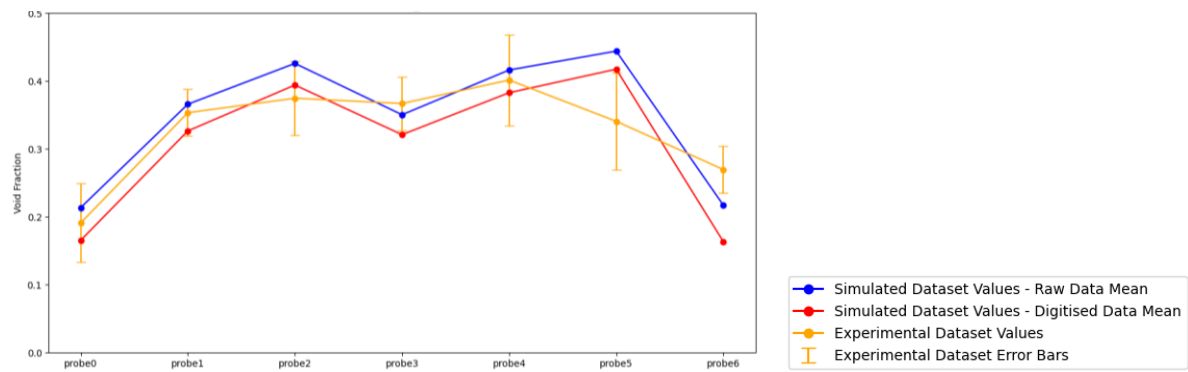


**Figure 41 Average void fraction at probe locations in the CFD and experimental data on the central tube in in the T5 configuration**

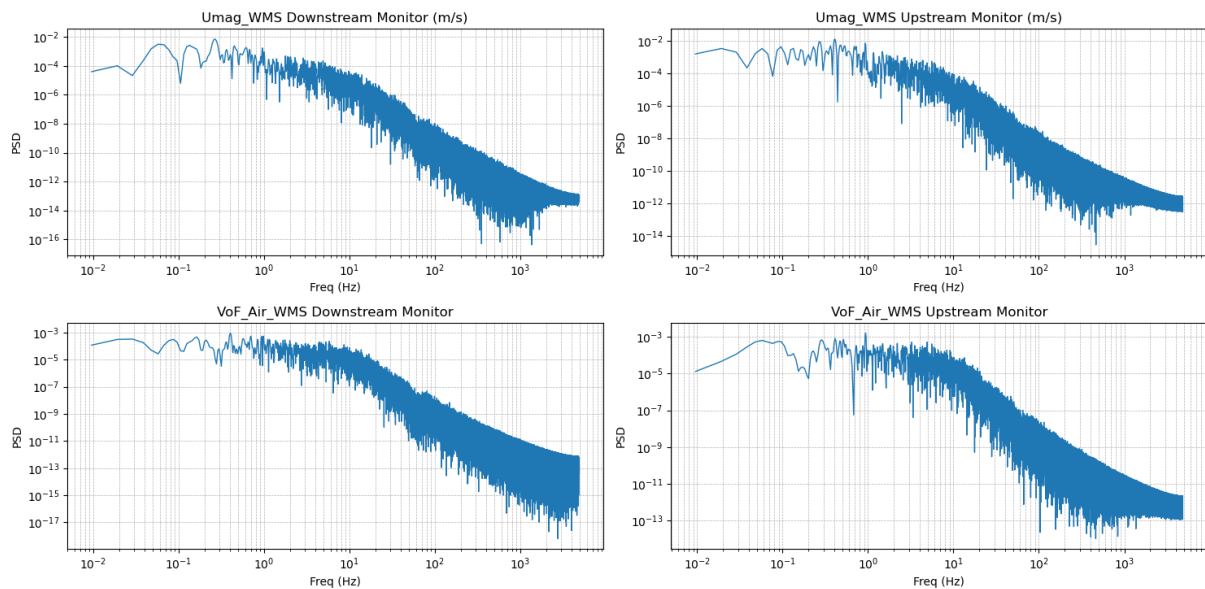


**Figure 42 PSDs of transient monitors for average velocity and average void fraction at downstream and upstream WMS plane locations in the T5 configuration.**

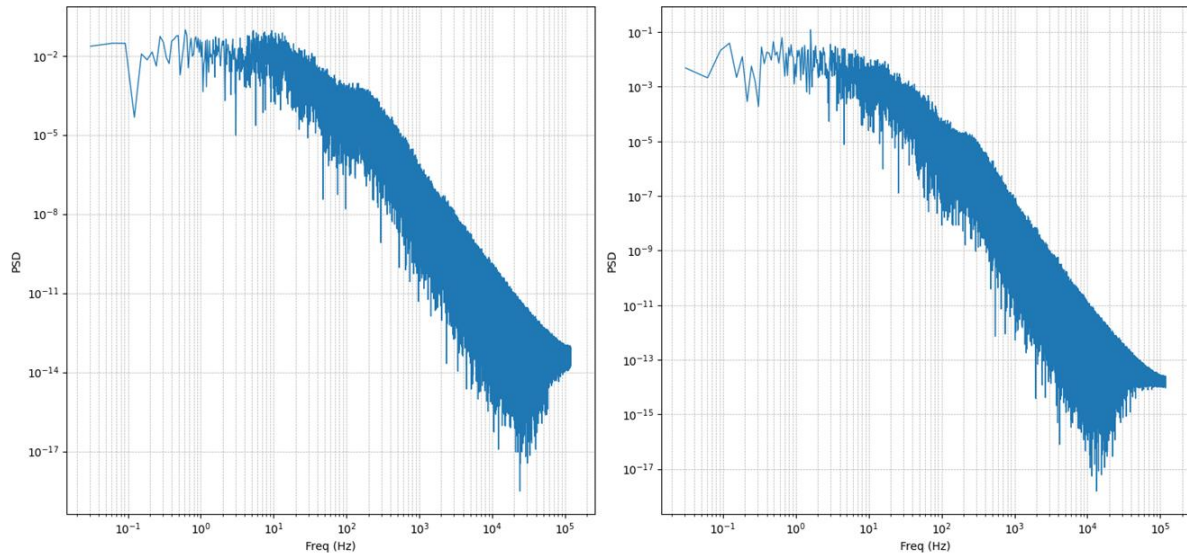


**Figure 43 PSDs of tube drag (left) and lift (right) force on the central tube in the T5 fixed configuration from EDF R&D UK simulations.**

## 5. Flexible configuration

### 5.1 VTT

The tube RMS displacement from the experiment and from the one- and two-way FSI simulations are compared in Table 6, whereas PSDs of the tube vibration are shown in Figure 44. Note that the experimental data is from the case T5, where the air mass flow was 10 g/s. The simulations correspond for the flow rate to the later T5 experiment with air mass flow of 11 g/s. For the one-way FSI, the tube moment signal from the fixed geometry simulation (see Sec. 4.1.1) was transferred to the single d.o.f. tube structural model described in Sec. 2.1. The added mass and damping from the two-phase flow surrounding the tube were included to the tube model in the one-way case. The additional damping was estimated from a simplified 2D FSI calculation by using the average mixture properties and single-phase flow, resulting in a damping ratio of 2.5%. The structural damping of the tube measured in air is 1% [12], hence a total damping of 3.5% was applied for the one-way simulation. From Figure 44 it is seen that the main vibration mode (i.e. basically rigid-body motion in the horizontal lift direction) is well captured by the simulations. The vibration amplitude is somewhat underpredicted in both cases, with the one-way coupling yielding closer agreement. The RMS value of the one-way simulation is affected by the added flow-induced damping, which was here only crudely estimated. The first mode peak in the experiment is at 10.6 Hz, which is almost the same for the free vibration test in still water, i.e. 10.4 Hz [12]. With the two-way coupling, this frequency shifts to 11.6 Hz, although the value in still water agrees with the experiment. The frequency shift is expected due to the reduced added mass of the two-phase flow and is shown in the newer experimental data in 44. The relatively small peaks in the experiment at higher frequencies are likely due to higher vibration modes of the tube and facility [12]; these are obviously not captured by the one d.o.f. tube model. Note that the added mass is constant in the one-way FSI calculation, whereas fairly high variations are expected in reality due to the intermittent large-scale bubbles.

**Table 6. Tube displacements in experiment T5 and in FSI simulations. The value in parenthesis corresponds to air flow 11 g/s, based on linear interpolation of the data with varying air flow from [12].**

| Value     | Experiment  | FSI one-way | FSI two-way |
|-----------|-------------|-------------|-------------|
| RMS [mm]  | 0.94 (0.97) | 0.90        | 0.79        |
| Max. [mm] | -           | 3.5         | 3.2         |

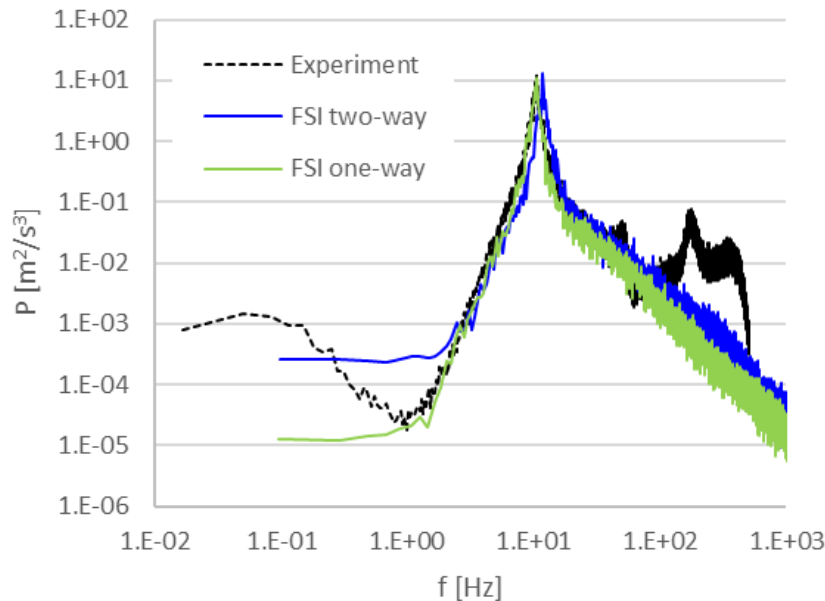


**Figure 44. Tube acceleration spectrum in experiment T5 and in FSI simulations. Note that the air flow rate is 10 g/s and 11 g/s in the experiment and simulations, respectively.**

## 5.2 CEA

Figure 45 shows the Power Spectral Density (PSD) of the acceleration of the flexible central tube for G1 (top) and T5 (bottom), compared with experimental data. We observe that the trends are similar for the G1 and T5 graphs. Notably, there is a significant difference in the integral under the curve between the experiment and the simulation, indicating a difference in the standard deviation of the vibrations. This discrepancy may be due to the 2D nature of our simulation and the assumed homogeneous distribution of loading, which can vary significantly depending on the location of the charge centres. Regarding frequencies, the fundamental frequency is recovered with a 4% error, but the third mode is not observed. However, a frequency around 63 Hz is present, which may be related to the experimental 50 Hz frequency (26% error). Additionally, a frequency of approximately 255 Hz is observed, which is also present in T1 and T2 but not in G1, with a 2.4% error. Finally, an excitation around 90 Hz is noted, which is not reproduced in the experiment and may be an artefact of our method. Actually, this frequency seems to be related to the drag according to T5 PSD of drag and lift forces.

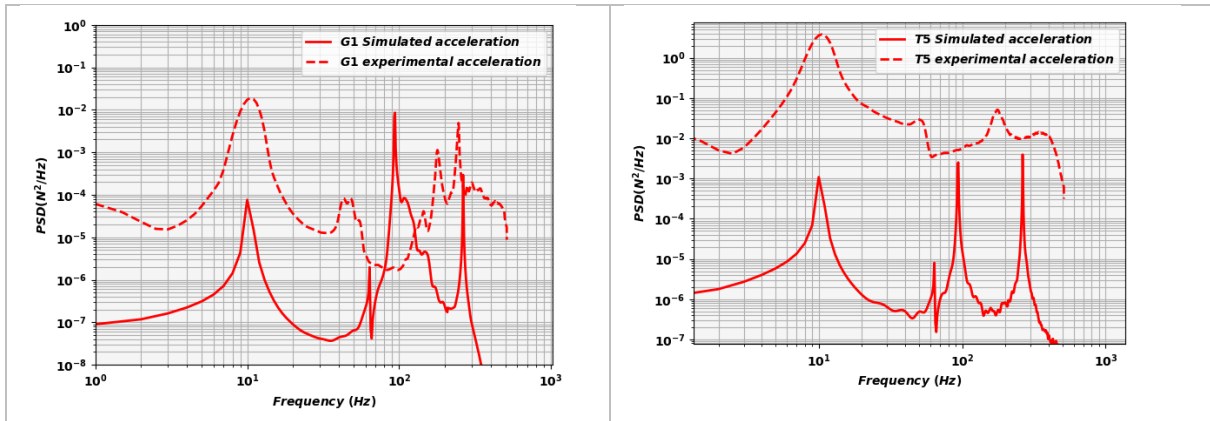


**Figure 45 PSD of the acceleration of the flexible central tube for G1 (top) and T5 (bottom) compared with experimental data.**

Figure 46 shows the Power Spectral Density (PSD) of the drag and lift forces of the flexible central tube for G1 (top) and T5 (bottom). For the G1 case, we observe a similar trend for both drag and lift forces, with a decay around 5 Hz followed by a change in slope around 15 Hz. For the T5 case, we also note a similar behaviour between the drag and lift forces, with a descent around 6 Hz, a plateau between 78 Hz and 155 Hz, and then a subsequent decay.

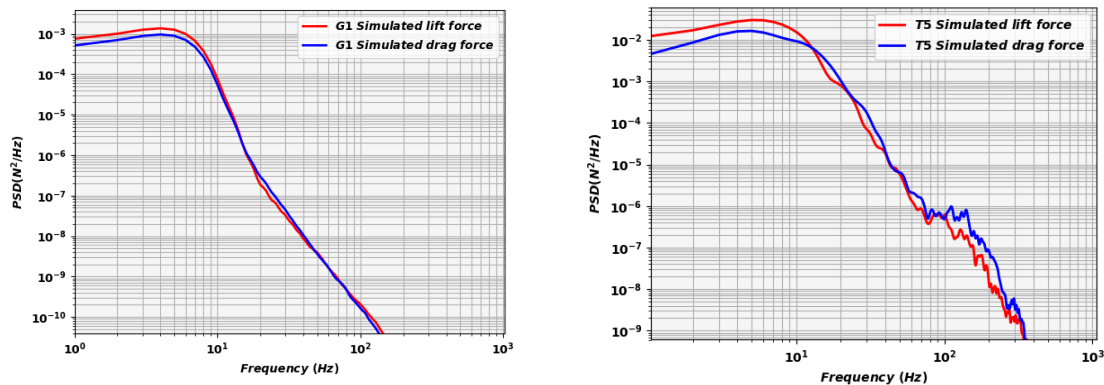


**Figure 46 PSD of the drag and lift forces of the flexible central tube for G1 (top) and T5 (bottom) compared with experimental data**

### 5.3 EDF UK

Simulations were conducted to resolve the fluid–structure interaction in full detail using a two-way CFD–FEA coupling approach in STARCCM+ for the 300 mm rod. To facilitate motion of the rod without distortion of the mesh where the rod meets the end walls, the rod was offset from the end walls by incorporating a gap of 5 mm at either end to give an effective 290 mm rod. The solid domain employed an isotropic, linear-elastic FE model of the rod–beam assembly with stainless steel properties (Poisson’s ratio 0.285, Young’s modulus 193 GPa). The rod density was scaled to 1835.3 kg/m<sup>3</sup> to achieve the correct total mass (0.39 kg), and hydrodynamic added mass ( $Ca = 1$ ) with 0.01 damping ratio was applied in line with subsea piping FIV practice. Fluid forces from the fluid domain were mapped to the solid structural domain, and the resulting structural displacements were returned to the CFD boundary via mesh morphing, closing the bidirectional loop. Models with the full coupling implemented are provided.

Instabilities were observed when morphing elements at the contact between the moving rod and static walls produced negative volumes; this was mitigated by offsetting the rod by 5 mm from the top and bottom walls and removing the inflation layer on the rod's top surface. Additional safeguards included bounding wall shear stress and pressure (while verifying values remained well within bounds so as not to artificially limit the solution).

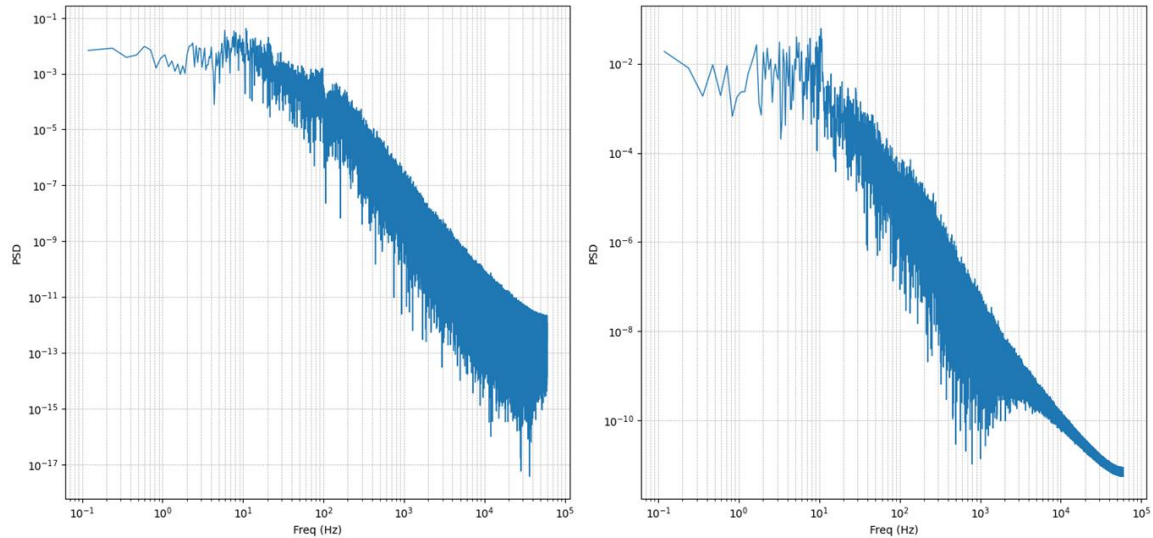


**Figure 47 Power Spectra of stream-wise,  $F_D$  (left), and cross-flow,  $F_L$  (right) components of flow-induced forcing in two-way CFD-FEA coupling**

CFD predicted forcing was used to drive the FEA transient forced response model, while the rod mesh was morphed using nodal displacements mapped from the structural model. The predicted forcing spectra from the CFD force monitors are shown in Figure 47, while the acceleration response spectra are shown in Figure 48. In the current configuration, the higher stiffness of the supporting beam in the stream-wise direction (relative to the cross-flow direction) leads to the dominant response being in the cross-flow direction. Alongside this dominant cross-flow response, a weak (orders of magnitude lower in amplitude) stream-wise response is also predicted with dominant peaks at the first ( $\sim 10\text{Hz}$ ) and second ( $\sim 100\text{Hz}$ ) structural modes. The dominant cross-flow response only shows a peak at  $10\text{Hz}$ , where the broadband high-amplitude dynamic forcing function excites the first structural mode.

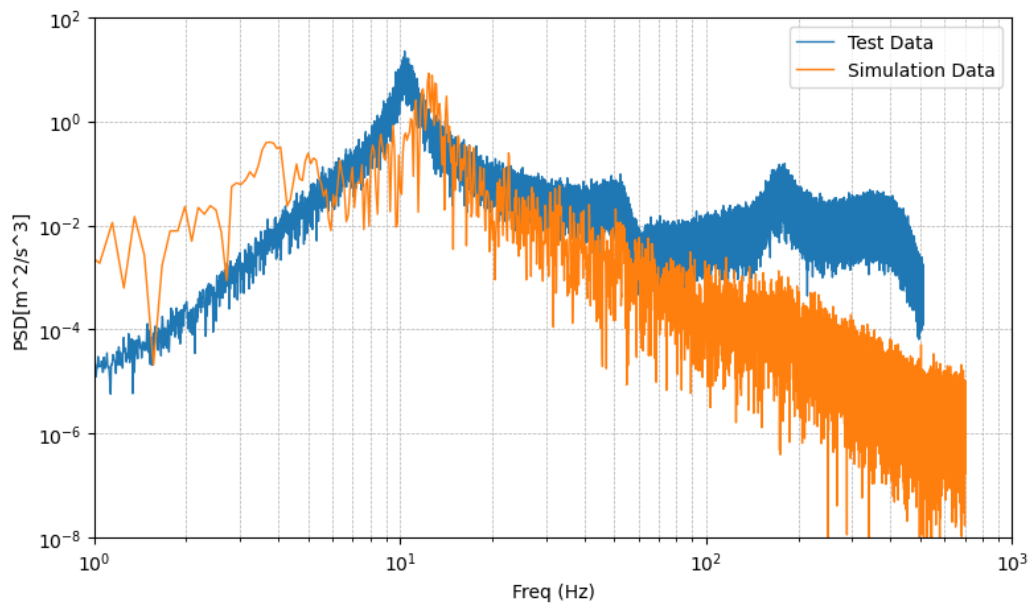


**Figure 48 Power Spectra of the acceleration magnitude of the rod end from the two-way CFD-FEA coupling simulations of the T5 EDF R&D UK simulation.**

## 5.4 NRG PALLAS

For the flexible configuration, the central tube's motion is modeled as a simple mass-spring-damper system governed by Newton's second law, allowing only translational motion in the lift direction (perpendicular to both the streamwise flow and the tube centerline). Motion in the streamwise direction, along the centerline, and any rotational degrees of freedom are fully constrained. The net force acting on the moving tube is computed from the total stress tensor (liquid and gas) using interpolation over its surface. The model incorporates a spring with a stiffness of 600 N/m and a damper with a damping coefficient of 0.3 Ns/m. The effective mass of the tube is set to 0.0846 kg. This effective mass was chosen to reproduce the measured natural frequency of the tube in air (13.4 Hz) using the experimentally measured stiffness.

We note that at the time of writing this report, the presented results remain preliminary, as only 9 physical seconds have been simulated. The initial phase of the vibration response is likely influenced by transient effects arising from the development of the flow field. Moreover, obtaining an accurate PSD analysis requires a significantly longer displacement time history to ensure statistical reliability and sufficient frequency resolution. A reliable assessment of FIV behaviour generally requires simulation of a much longer physical time span to capture the full dynamics beyond the initial transients

The displacement signal of the tube during the first 9 seconds of the simulated time is shown in Figure 49. We observe that the maximum vibration amplitude during this period is on the order of 2 mm.

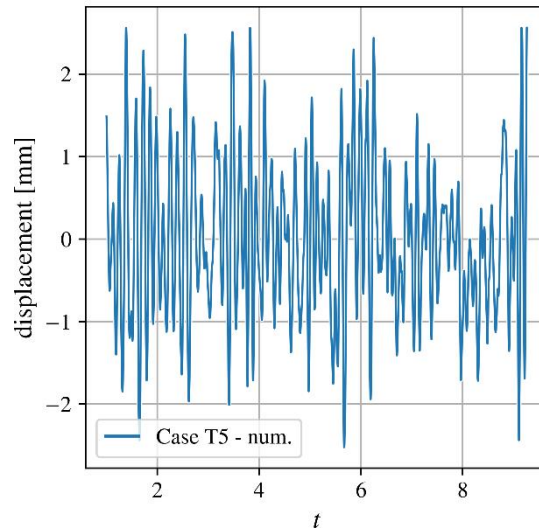


**Figure 49: Displacement signal of the moving tube in test T5 NRG PALLAS simulation.**

Figure 50 compares the power spectral density (PSD) of the tube's displacement computed from the hybrid two-fluid FSI simulation with the experiments. A clear peak is observed in the frequency range of approximately 810 Hz. This peak appears at a slightly lower frequency and with a smaller amplitude compared to the experimental PSD. This under-prediction of both the dominant frequency and the PSD magnitude is most likely attributable to the limited physical simulation time of the FSI cases (currently only 9 s) and the relatively course computational grid.

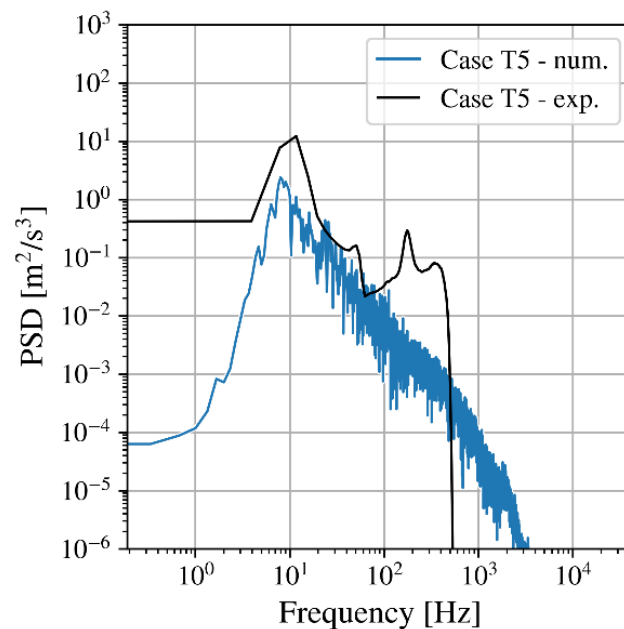


**Figure 50: Power Spectra of the moving tube acceleration in test T5 NRG PALLAS simulation.**

## 6. Conclusion

Considering the range of numerical models and calculation domains, having such quantitative and qualitative similarities between simulations strengthens confidence in the numerical predictions. However, it appears in the present case that there is less agreement with the experiment than for the triangular arrangement (see D4.6). Moreover, the high slip in the present cases also pose a challenge for the two-phase flow prediction. Thus, having a different void distribution than the experiment appears to have a significant role in the tube bundle. Most of the partners simulated two cases, with a reasonable agreement found upstream and downstream the cylinder by at least two of the partners. For those having the correct flow prediction, it can be seen that vibrations are in a correct agreement with experiment. Once again, similarly to the triangular case (D4.6), the force fluctuation in two phase flow numerical simulation appears to be driven by the resolved gas structure, highlighting correct prediction from VOF approaches without any treatment for the dispersed phase (oppositely to hybrid models). This is true since the present cross comparison was performed on an intermittent flow where the force is driven by large gas structures.

## Bibliography

- [1] R. Meller, F. Schlegel, and D. Lucas. Basic verification of a numerical framework applied to a morphology adaptive multifield two-fluid model considering bubble motions. *International Journal for Numerical Methods in Fluids*, 93.3: 748-773, 2021.
- [2] OpenFOAM Foundation (2025). OpenFOAM Version 12. Available at: <https://www.openfoam.org>.
- [3] V. Habiyaemye, E.M.J. Komen, J.G.M. Kuerten, and E.M.A. Frederix. Modeling of bubble coalescence and break-up using the log-normal method of moments. *Chemical Engineering Science*, page 117577, 2022.
- [4] E.M.A. Frederix, T.L.W. Cox, J.G.M. Kuerten, and E.M.J. Komen. Poly-dispersed modeling of bubbly flow using the log-normal size distribution. *Chemical Engineering Science*, 201:237–246, 2019.
- [5] E.M.A. Frederix, V. Habiyaemye, S. Tajfirooz, & J.G.M. Kuerten, Extension of the two-fluid model to bubble size distribution moment velocities. *Experimental and Computational Multiphase Flow*, 7(2), 151-166, 2025.
- [6] J. H. Chow and E.Y.K. Ng. Strongly coupled partitioned six degree-of-freedom rigid body motion solver with Aitken’s dynamic under-relaxation. *International Journal of Naval Architecture and Ocean Engineering*,
- [7] M. Ishii and N. Zuber. Drag coefficient and relative velocity in bubbly, droplet or particulate flows. *AIChE Journal*, 25(5):843–855, 1979.

- [8] A. Tomiyama, H. Tamai, I. Zun, and S. Hosokawa. Transverse migration of single bubbles in simple shear flows. *Chemical Engineering Science*, 57(11):1849–1858, 2002.
- [9] S.P. Antal, R.T. Lahey, and J.E. Flaherty. Analysis of phase distribution in fully developed laminar bubbly two-phase flow. *International journal of multiphase flow*, 17(5):635–652, 1991.
- [10] A.D. Burns, T. Frank, I. Hamill, J.-M. Shi, et al. The Favre averaged drag model for turbulent dispersion in Eulerian multi-phase flows. In 5th international conference on multiphase flow, ICMF, volume 4, pages 1–17, 2004.
- [11] A. Chahine, D. Vivaldi and G. Brillant, New experimental insights into two-phase air–water bubbly and intermittent flow in large duct and cross-flow configurations. *International Journal of Multiphase Flow* (2025), doi: <https://doi.org/10.1016/j.ijmultiphaseflow.2025.105266>.
- [12] Spina, G., (2023). Experimental and numerical study of two-phase flows and of fluid structure interactions inside a cross-flow tube bundle. Ph.D. thesis, INP Toulouse, Toulouse, France.
- [13] L. Moerlose, M. Paepe and J. Degroote, A Novel Inlet Model for Two-Phase Bubbly Axial Flow Through a Tube Bundle, in: WCCM-ECCOMAS2020. URL <https://www.scipedia.com/public/Moerlose et al 2021a>
- [14] H. Dolfen and J. Degroote, A Synthetic Bubble Model as Inlet for Fluid-Structure Interaction Simulations with Two-Phase Flow, in: ECCOMAS 2024. URL <https://www.scipedia.com/public/Dolfen Degroote 2024a>
- [15] Hirt, C.W., Nichols, B.D., 1981. Volume of fluid (VOF) method for the dynamics of free boundaries. *J. Comput. Phys.* 39, 201–225. [https://doi.org/10.1016/00219991\(81\)90145-5](https://doi.org/10.1016/00219991(81)90145-5)
- [16] K. Hayashi, H. Hessenkemper, D. Lucas, D. Legendre, A. Tomiyama, Scaling of lift reversal of deformed bubbles in air–water systems, *International Journal of Multiphase Flow* 142 (2021) 103653.
- [17] X. Cai, G. B. Wallis, A more general cell model for added mass in two-phase flow, *Chemical Engineering Science* 49 (10) (1994) 1631–1638
- [18] C. Bazin, Numerical and experimental studies of two-phase flows interacting with a bundle of tubes, Ph.D. thesis, Institut Polytechnique de Paris (2023).
- [19] A. Tomiyama, Struggle with computational bubble dynamics, *Multiphase Science and Technology* (1998) 369–405.
- [20] R. Manceau, Recent progress in the development of the elliptic-blending Reynolds-stress model, *International Journal of Heat and Fluid Flow* 51 (2015) 195–222.
- [21] W. Benguigui, J. Lavieville, N. Merigoux (2019). Fluid-structure interaction in two-phase flow using a discrete forcing method . *Internation Journal of Numerical Methods in Fluids*

- [22] N. Merigoux et al. A generalized large interface to dispersed bubbly flow approach to model two-phase flows in nuclear power plant. CFD4NRS-6, September 13-15 2016, Cambridge, MA – USA.
- [23] P. Coste. A Large Interface Model for two-phase CFD. Nuclear Engineering and Design 255 (2013) 38– 50.