Ref. Ares(2026)5868043 - 09/06/2026

GO-VIKING

Research and Innovation Action (RIA)

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

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

BPGs on the use of FSI methods for stakeholders

Authors : Papukchiev ANGEL (GRS)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	BPGs on the use of FSI methods for stakeholders
Author(s)	Papukchiev ANGEL
Number of pages	69
Document type	Deliverable
Work Package	WP5
Document number	D5.5
Issued by	GRS
Date of completion	2026-06-05 09:50:15
Dissemination level	Public

Summary

Synthesis of best practices for FSI methods for FIV evaluation, based on the project results.

Approval

Date	By
2026-06-05 09:50:58	Papukchiev ANGEL (GRS)
2026-06-05 09:51:19	Papukchiev ANGEL (GRS)

GO VIKING

D5.5 Best Practice Guidelines on the use of FSI methods for FIV evaluation

Version N°1
May / 2026

Angel Papukchiev[1], Kevin Zwijsen[2], Daniele Vivaldi[3], Hidajet Hadžić [4], William Benguigui[5], Philippe Planquart[6]

[1]Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Garching, Germany
[2]NRG PALLAS, Petten, The Netherlands
[3]ASNR, Saint-Paul-Lez-Durance Cedex, France
[4]Framatome GmbH (FRA-G), 91052 Erlangen, Germany
[5]Electricité De France (EDF) R&D, Chatou, France
[6]Von Karman Institute for Fluid Dynamics, Sint-Genesius-Rode, Belguim

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.

This document may contain information that may be subject to European and/or national export authorization. Any transfer or disclosure of the information contained in this document which requires an export license will not be made without the authority of an export license demanded by the competent authorities. The exporter is responsible for obtaining the export license. No rights may be derived from the (absence of an) EU DuC.

Funded by
the European Union

Document information

Grant Agreement / Proposal ID	101060826
Project Title	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
Project Acronym	GO-VIKING
Scientific Coordinator	Angel Papukchiev, angel.papukchiev@grs.de , GRS Kevin Zwijsen, kevin.zwijsen@nrgpallas.com, NRG PALLAS
Project starting date (duration)	1st June 2022 – 31st May 2026 (48 Months)
Related Work Package	WP5
Related Task(s)	Task 5.3 Best practice guidance for FIV
Lead Organisation	GRS
Contributing Partner(s)	NRG PALLAS, EDF, ASNR, FRA-G, VKI
Due Date	31-03-2026
Submission Date	31-05-2026
Dissemination level	PU

History

Date	Version	Submitted by	Reviewed by	Comments
31-05-2026	1.0	A. Papukchiev	K. Zwijsen	

Table of contents

1. Introduction 11
3. Short introduction to fluid-structure interaction 13
3.1 Fluid-structure interaction types, methods and modelling 13
3.1.1 FSI types 13
3.1.2 Coupling approach and data transfer 13
3.1.3 Coupling approach and data transfer 14
3.1.4 Fluid-solid interface grid 14
3.1.1 Solution Sequence 15
4. Best practices for FIV analysis in single-phase axial-flow 16
4.1 CAD and mesh 16
4.2 Flow field modelling 18
4.2.1 Flow field modelling in single rod configurations 18
4.2.2 Flow field modelling in fuel bundle simulations 20
4.3 Structural modelling 22
4.3.1 Structural modelling in single rod configurations 22
4.3.2 Structural modelling in bundle configurations 22
4.4 Coupling and FSI simulations 23
4.5 Strategies to increase the efficiency of bundle simulations 24
4.6 Validation and experimental data 25
4.7 Conclusions on FIV analysis in single-phase axial-flows 25
5. Best practices for FIV analysis in single-phase cross-flow 28
5.1 CAD and mesh 28
5.1.1 Geometry representation 28
5.1.2 Mesh topology and refinement 29
5.1.3 Near-wall resolution and turbulence scale considerations 30
5.2 Flow field and turbulence modelling 32
5.3 Inlet boundary conditions 33
5.4 Temporal resolution and convergence 34
5.5 Structural modeling 36
5.5.1 Structural representation and degrees-of-freedom 36
5.5.2 Structural properties and dynamic response 38
5.6 Coupling and FSI simulations 39
5.7 Conclusions on FIV analysis in single-phase cross-flows 41
6. Best practices for FIV analysis in multiphase flows 43

6.1 Numerical simulation of multiphase flows 43
6.2 Mesh and resolution of the gas-liquid interface 45
6.3 Inlet boundary conditions 47
6.4 Coupling and FSI simulations 48
6.5 Structural modeling 50
6.6 Conclusions on FIV analysis in multiphase flows 50
7. Best practices for the utilization of fast-running and uncertainty quantification methods 52
7.1 Reduced-order model for the structural domain 52
7.2 Reduced-order model for the fluid domain 55
7.2.1 AniPFM 55
7.2.1 Synthetic Turbulence Method 56
7.3 Best practices on the implementation and utilization of uncertainty quantification methods 60
7.3.1 Polynomial Chaos Expansion 60
7.3.1 Uncertainty Quantification of Turbulence Models 63
7.4 Conclusions on fast-running and uncertainty quantification methods 64
8. Overall conclusions 66
9. Bibliography 69

List of figures

Figure 1: Numerical mesh of a cantilevered rod for a LES simulation (EDF). 17

Figure 2: Vorticity distribution behind the spacer grid in a single span of the ALAIN experiment calculated with an LES approach by CEA (Michel, 2025). 18

Figure 3: DNS performed by Penn State University (Bistany, 2024): Q-criterion revealing coherent structures in the vicinity of the rod tip. 19

Figure 4: Beam model of the ALAIN 5x5 assembly with torsion springs and dampers, developed by GRS (Blaesius, 2026). 23

Figure 5: Examples of meshes used by the various partners within WP3, with top left: mesh GRS for AMOVI, top right: mesh EDF for AMOVI, and bottom: mesh NRG PALLAS for GOKSTAD. 29

Figure 6: Example mesh independence studies performed for AMOVI, with left: for a single flexible tube (GRS) and right: for a flexible tube inside a rigid tube bundle (IPP). 30

Figure 7: GLL spacing of a spectral element method mesh used in Nek5000 for LES (Top) and Taylor microscale computed from a RANS simulation using Nek5000 (Bottom) (VCU). 31

Figure 8: Resolved structures by GRS for AMOVI for single rigid tube in cross flow. Right: k-ω SST, left: SAS. 32

Figure 9: The computational domain created by EDF for the single cylinder configuration of AMOVI, clearly showing the precursor used to generate proper inlet conditions. 34

Figure 10: Time evolution of the mean drag coefficient (left) and the standard deviation of the lift coefficient (right) as obtained by EDF for the single rigid cylinder configuration. 35

Figure 11: CAD models based on the used Rigid Body Motion approaches as used by GRS for AMOVI, with left: rotational motion model and right: translational motion model. 37

Figure 12: Tube displacements, calculated with linear and angular momentum algorithms (GRS). 37

Figure 13: Lift displacement time series (left) and PSD (right) as obtained by NRG PALLAS for GOKSTAD. 39

Figure 14: Test section with marked channel wall (left), Boundary condition for wall parallel to moving side of flexible tube 40

Figure 15: Volume rendered streamwise velocity contour plot of a reduced computational domain of the GOKSTAD tube bundle. 41

Figure 16: Void distribution in the TREFLE experiment, calculated with FLUENT using VOF in combination with LES (VTT). 44

Figure 17: High-quality numerical grid for the TITAN experiment, generated by EDF. 46

Figure 18: Difference between a single air jet and SBM as inlet for a tube bundle in multiphase cross flow (University of Ghent). 48

Figure 19: Measured and calculated PSDs with one- and two-way coupling by VTT for the TREFLE experiment. 49

Figure 20: PSDs of tube displacements calculated with FOM (red) and ROM (blue) (GRS). ... 52

Figure 21: Comparison of two simulations of that consider 6 and just the first tube vibration mode (GRS). 52

Figure 22: Tube displacements calculated with ZLES (red) and SST (blue) (GRS). 53

Figure 23: PSDs calculated with ZLES (red) and SST (blue) (GRS). 53

Figure 24: Distribution of the URANS (SST) and LES zones in the CFD domain that represents the AMOVI experiment (GRS). 54

Figure 25: 3D time history of non-dimensional rod tip displacements calculated with LES and AniPFM (NRG PALLAS)..... 56

Figure 26: Visualization of the main instantaneous fields calculated using ANSYS CFX + ST for the single flexible tube case with Q = 4.0 l/s (a – integration time step 2.5 ms, K10; b – integration time step 1.0 ms, K31) 58

Figure 27: Results of displacements calculation for the single flexible tube with Q = 4.0 l/s . 59

Figure 28: Effect of the begin of the fitting window on the damping for multiple free vibration signals after an initial excitation, each indicated with a different colour (Universtiy of Ghent). 61

Figure 29: Convergence of the RMS vibration amplitude for a cylinder as a function of the final time step included in its calculation. The legend indicates different diameters and velocities (Universtiy of Ghent). 61

Figure 30: UQ framework for turbulence model calibration in CFD (TUD)..... 64

Funded by
the European Union

Abbreviations and Acronyms

Acronym	Description
AIAD	Algebraic Interfacial Area Density
ALE	Arbitrary Lagrangian–Eulerian
CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
CFL	Courant–Friedrichs–Lewy number
CSM	Computational Structural Mechanics
DES	Detached Eddy Simulation
DNS	Direct Numerical Simulations
DOF	Degree-Of-Freedom
FA	Fuel Assembly
FEI	Fluid-Elastic Instability
FIV	Flow-Induced Vibrations
FP	Flow Passage
FSI	Fluid-Structure Interaction
GOKSTAD	GO-VIKING experimental SeTup to Assess FIV by cross-flow in tube bunDles)
GO-VIKING	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
IDDES	Improved Delayed Detached Eddy Simulation

LES	Large-Eddy Simulations
PCE	Polynomial Chaos Expansion
PSD	Power Spectral Density
PWR	Pressurized Water Reactor
(U)RANS	(Unsteady) Reynolds-Averaged Navier Stokes
Re	Reynolds (number)
RMS	Root-Mean Square
RSM	Reynolds Stress Model
SAS	Scale Adaptive Simulation
SBM	Synthetic Bubble Model
SFA	Stable Fluid Algorithm
STG	Synthetic Turbulence Generator
TIV	Turbulence-Induced Vibrations
TKE	Turbulent Kinetic Energy
UQ	Uncertainty Quantification
VIV	Vortex-Induced Vibrations
WM-LES	Wall-Modelled Large Eddy Simulation
WP	Work Package
WR-LES	Wall-Resolved Large Eddy Simulation

Funded by the European Union

Summary

This report presents a consolidated set of Best Practice Guidelines on the use of Fluid-Structure Interaction (FSI) methods for Flow-Induced Vibration (FIV) evaluation using CFD tools and also CSM methods. It contains dedicated chapters dealing with FIV in single-phase axial-flows, single-phase cross-flows, and multiphase flows. Further, it focuses on the development and application of fast-running and uncertainty quantification methods. By combining results from multiple partners on the application of different simulation approaches, including turbulence modelling, structural representation, coupling strategy, etc., the report provides a consistent framework that helps the selection of appropriate flow and structural modelling. Key aspects addressed include CAD modeling, mesh generation, flow field modeling, structural representation, and FSI coupling. The objective of the "Best Practice Guidelines on the use of fluid-structure Interaction methods for flow-induced vibration evaluation" is to synthesize the knowledge gathered in the four years of the GO-VIKING project duration, and thus, help the stakeholders dealing with similar problems.

Keywords

Flow-Induced Vibrations, benchmark, tube bundle, cross-flow, FIV, FSI, CFD, single-phase, high-resolution simulations, LES, medium-resolution simulations, URANS

1. Introduction

The prediction of flow-induced vibrations (FIV) in NPP components remains a critical and challenging problem in nuclear engineering, heat exchanger design, and fluid–structure interaction (FSI) modelling. These systems exhibit highly complex flow behaviour characterized by strong turbulence production, unsteady wake interactions, and coupling between fluid forces and structural motion. The accurate representation of such phenomena requires careful consideration of modelling assumptions, numerical methods, and coupling strategies.

This report presents a consolidated set of Best Practice Guidelines on the use of FSI methods for FIV evaluation using CFD tools and also CSM methods. It is based on the experience gathered in GO-VIKING WP2 "Flow-induced vibrations in fuel assemblies", WP3 "Flow-induced vibrations in steam generators", Wp4 "Flow-induced vibrations in multiphase flows" and WP5 "Fast-running methods, uncertainty quantification and best practice guidelines". This report contains dedicated chapters dealing with the lessons learned in these four work packages. By combining results from multiple partners on the application of different simulation approaches, including turbulence modelling, structural representation, coupling strategy, etc., the report provides a consistent framework that helps the selection of appropriate flow and structural modelling. Some of the key aspects addressed include:

- CAD modeling
- mesh generation for CFD
- flow field modeling
- representation of structures for CSM
- FSI coupling

The objective of the report is to synthesize the knowledge gathered in the four years of the GO-VIKING project duration, and to provide this as a guide to the stakeholders dealing with similar problems. Therefore, this document shall not be considered prescriptive, but rather as a synthesis of lessons learned that might help scientists or industrial engineers to make the right choices when dealing with a certain type of a FIV phenomenon.

These best practices are based on different codes and numerical approaches used for the analysis of the following experiments:

- Cantilever Rod experiment (rod exposed to single-phase axial-flow)
- ALAIN experiment (rod bundle exposed to single-phase axial-flow)
- AMOVI experiment (tube bundle exposed to single-phase cross-flow)
- GOKSTAD experiment (tube bundle exposed to single-phase cross-flow)

- Cantilever Rod experiment (rod exposed to two-phase axial-flow)
- TREFLE experiment (tube bundle exposed to two-phase cross-flow)
- TITAN experiment (tube bundle exposed to two-phase cross-flow)

All these experiments were dedicated to FIV in nuclear reactor cores and steam generators. Still, the derived lessons learned and the guidelines formulated here can be applied also for other nuclear power plant (NPP) components. Since Computational Fluid Dynamics (CFD) and Computational Structural Mechanics (CSM) are general purpose numerical methods that do not strongly rely on correlations for specific geometries and boundary conditions, this document may be useful also for practitioners from industries other than nuclear.

The detailed descriptions of all experiments as well as the results from the numerical activities can be found on the GO-VIKING webpage under https://go-viking.eu/results. The current document is a synthesis of the GO-VIKING deliverables D2.5 "Best practices for the analysis of FIV in single-phase axial-flow", D3.5 "Best practices for the analysis of FIV in single-phase cross-flow", D4.7 "Best practices for the analysis of FIV in two-phase flow" and contributions from partners that worked on fast-running and uncertainty quantification methods. The aforementioned documents can be also found on the GO-VIKING webpage together with all other project deliverables.

Funded by
the European Union

3. Short introduction to fluid-structure interaction

This chapter will give just a brief introduction to general FSI with the aim of easing the understanding of the next chapters. The text below is heavily based on the "OECD/NEA Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications – 2024 Update" (OECD/NEA, 2025), which was partly written by some of the GO-VIKING consortium partners.

3.1 Fluid-structure interaction types, methods and modelling

The interplay between fluid and the adjacent structure is called FSI, and it is considered in the design process and the analysis of technical systems in many engineering disciplines such as nuclear, automotive, aerospace, energy, healthcare, construction, oil, and gas, etc. The structure might be either immersed in the fluid or confine it.

In the past, the issues with FSI phenomena were investigated with the help of empirical correlations. Today, modern 3D CFD programs can be coupled with CSM tools to allow an FSI analysis. In each domain (solid and fluid), mathematical equations are solved to determine loads and mesh deformations. These equations are solved on discretised domains, represented with numerical meshes. In coupled FSI applications, the CFD program calculates the forces on the common boundary interface (usually a wall) and provides these to the CSM code. The CSM code uses these as boundary conditions and then calculates the mesh deformation. With the deformed mesh as a boundary condition, the CFD performs the next iteration or time step.

3.1.1 FSI types

A very simple example for FSI is a pipe, filled with water, and a heated wall. In this case, the energy is transferred to the water through the wall and the fluid temperature under transient conditions will depend also on the material properties of the pipe wall (thermal conductivity and inertia). In this simple case of interaction between and the pipe wall, there is no common moving boundary between the solid and the fluid.

When dealing with FIV, the FSI is significantly more complex, because displacements/deformations take place. In such cases, the boundary between the solid and fluid domains is moving. This happens when highly energised flow impinges on long, slender and flexible structures such as rods, tubes, etc., and thus deforms these structures, causing FIV.

3.1.2 Coupling approach and data transfer

Depending on the analysed FSI phenomena and of the expected results, different FSI methods may be applied. Thus, competences in both fluid and structural mechanics are needed to choose the appropriate CFD and CSM methods and models.

3.1.3 Coupling approach and data transfer

The level of coupling between the CFD and the CSM simulations required to obtain consistent results depends on the specific application. There are three levels with increasing complexity of data transfer between the fluid and structural domains:

- **Decoupled approach:** The whole analysis is first performed with one of the codes (CFD), and then the solution variables are provided as time-dependent boundary conditions to the second code (CSM). In this analysis two stand-alone simulations (CFD and CSM) are sequentially performed.

- **One-way coupling:** In this coupling scheme, both codes are executed sequentially or in parallel. The data transfer is unidirectional, which means the CFD code provides data (forces/momenta) to the CSM code after each time step or iteration. There is no feedback from the CSM result to the CFD program. Such approach is feasible for FSI cases with small structural displacements (such as turbulence-induced vibrations (TIV)), in which the flow is negligibly affected by the deforming structure.

- **Two-way coupling:** In this coupling scheme, the CFD provides the forces/momenta after each iteration or time step to the CSM program. It uses these as boundary conditions and performs the structural simulation. Its result is a new shape of the common boundary, which is then passed back to the CFD program. The CFD code uses the new geometry and again calculates forces/momenta acting on the common boundary. This process is repeated till the specified problem time is reached. The two-way coupling scheme is used when the structural displacements are large enough to alter the flow pattern inside the fluid domain.

3.1.4 Fluid-solid interface grid

In a two-way coupling approach, it is possible to consider two main options for the modelling of the fluid-structure interface:

- **Methods with moving fluid grid and explicit representation of the fluid-structure interface:** The structure is solved through a dedicated mesh, which deforms as a function of time, space, and the forces, provided by the CFD code. The result of the CSM simulation is the new, deformed common CFD-CSM boundary (wall/surface). This approach is referred to as Arbitrary Lagrangian-Eulerian (ALE).

- **Methods with fixed fluid grid:** Typical representative is the Immersed Boundary method, in which the Eulerian fluid grid is fixed, while Lagrangian structure nodes can move over it. The elastic forces, calculated by the CSM program, are interpolated from the structural nodes to the surrounding fluid. The calculated velocity is then interpolated from the CFD domain to the structural nodes.

There are further FSI methods like Chimera (moving and fixed fluid grid), particle methods (SPH), etc. A good overview on the different FSI methods can be found in (Degroote, 2020).

3.1.1 Solution Sequence

Concerning the execution of the coupled simulation, two approaches can be distinguished:

- **Partitioned approach:** The equations governing the flow and the displacement of the structure are solved with two separate (CFD and CSM) solvers. Both solvers exchange data during the simulation.
- **Monolithic approach:** The equations governing the flow and the equations describing the structural displacement are solved simultaneously in one single solver.

4. Best practices for FIV analysis in single-phase axial-flow

4.1 CAD and mesh

Reduce the geometrical details: From the simulation of the ALAIN benchmark, it turned out that the configuration is very complex and computationally demanding. To reduce the computational costs some simplifications of the geometry could be applied. The geometry simplification pays off with a lower overall number of elements and thus faster computations.

Special care is necessary for the mesh preparation: while today an automatically generated mesh (Figure 1) is standard, the analyst shall always check and further improve the mesh at the locations of interest for the simulation. Especially is important to prevent poor cell quality that usually appears in regions with sharp edges, like it is the case at the contact between fuel rods and spacer grids in a fuel assembly (FA). "Smoothening" of the geometry at such location may significantly improve the mesh quality and avoid divergence problems during CFD simulations. This is especially important in connection with the scale-resolving approaches when using the non-diffusive, but non-bounding discretization schemes that are more sensitive to mesh quality.

Mesh resolution for fuel assembly configuration: Application of scale-resolving methods, like Large-Eddy Simulation (LES), typically requires the use of a numerical mesh much finer compared to Reynolds-Averaged Navier-Stokes (RANS) methods. A very fine mesh is required to resolve fine turbulent eddies being generated particularly close to the solid wall boundaries. For fuel assembly (FA) configurations, the use of a fine numerical mesh is prohibitively expensive and from the perspective of industrial application, in most cases, not realizable at all. Consequently, the use of wall-modelling instead of wall-resolving high-resolution approach (e.g. LES) is the only reliable alternative. In addition, the use of a rather coarse than fine numerical mesh from the standpoint of the LES requirements (or other similar scale-resolving simulation approaches) is also inevitable. Generally, it is recommended to use different length scales of turbulent flow (e.g integral turbulent length scale, Kolmogorov length scale or Taylor microscale) as reference to estimate roughly the required mesh cell size. For cell expansion the recommendations from the literature can be used (Piomelli, 2022). Close to wall boundaries use multiple cell layers and adjust the y+ value in a suitable range, typically 20-30.

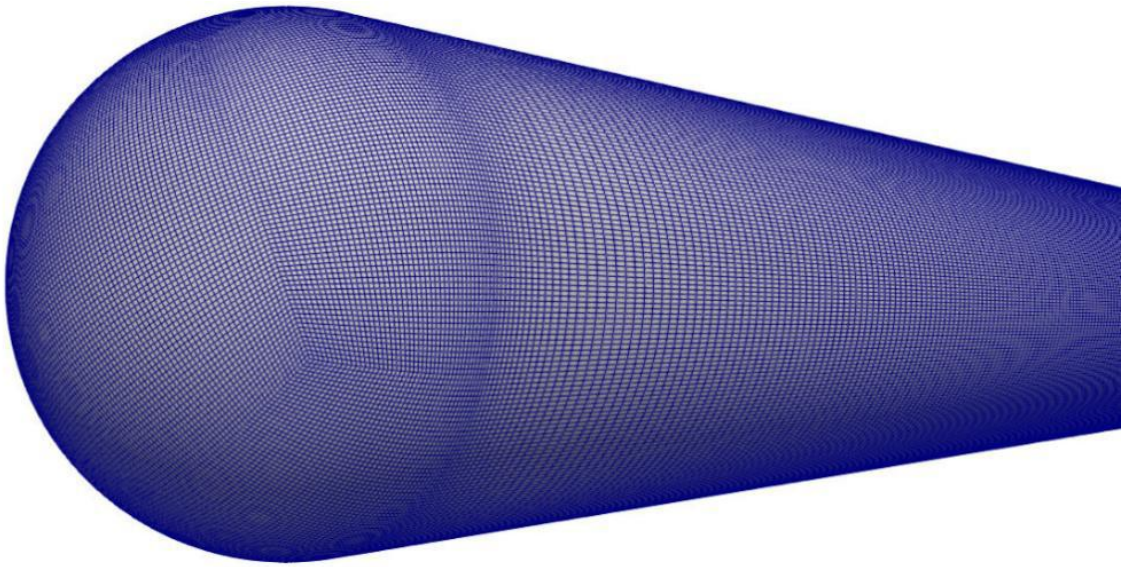

Figure 1: Numerical mesh of a cantilevered rod for a LES simulation (EDF).

Use geometry reduction to reduce the computational effort: Given the high geometrical complexity and the large CFD model size, a kind of reduced modelling approach on the CFD side would be very beneficial. This is especially true for the modelling of the full-scale FA, which is obviously not realistic with the models used so far.

An alternative may be the combination of a reduced model for the full-scale fuel assembly to describe the global (mean) flow field and a detailed LES model of a FA segment (e.g. one span (Figure 2)) to derive some parameters to fit the reduced full-scale model and to derive turbulence parameters. The span might be the FA segment between two spacer grids or comprise one spacer grid and half or the rod length up- and downstream of the spacer grid (like in Figure 2). By reducing the overall geometry to a single span, the efficiency of the analysis may be increased remarkably. The use of reduced-geometry models such as one span of the fuel assembly configuration is also beneficial for the purpose of testing of the modelling approaches or for mesh refinement studies. In similar way, a domain reduction may also be utilized in the case of cantilever rod simulation by limiting the computational domain to only a portion of the rod length in which the flow field changes.

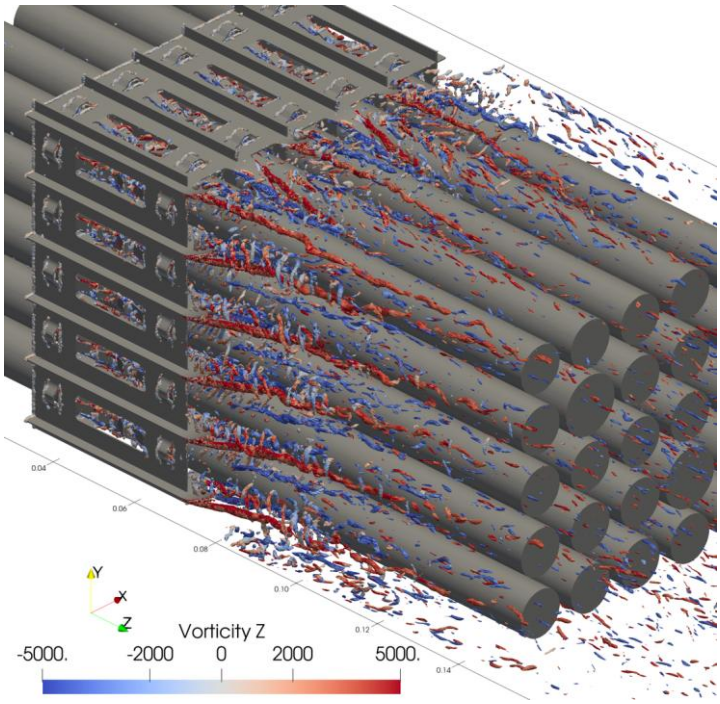

Figure 2: Vorticity distribution behind the spacer grid in a single span of the ALAIN experiment calculated with an LES approach by CEA (Michel, 2025).

4.2 Flow field modelling

4.2.1 Flow field modelling in single rod configurations

High-resolution approaches

The Direct Numerical Simulations (DNS) performed for the Cantilever Rod experiment revealed that the evolution of the statistics is remarkably different near the rod tip and for a considerable distance along the rod wall before returning to expected annular flow statistic behaviors.

Wall-resolved LES (WR-LES) was also used to investigate the effect of the rod vibrations by simulating the fluid flow only (without rod movement). FIV were analysed with WR-LES within coupled FSI simulations. FSI and fixed rod simulations yielded very close velocity and Reynolds stress profiles, which means that the rod motion, resulting from turbulence-induced vibrations (TIV), does not have a strong impact on the flow. Results also showed good predictions of vibration frequencies, but still underpredicted the vibration amplitude compared to the experiment.

High-resolution methods perform better than medium-resolution ones, but are expensive: LES and other scale-resolving techniques provide better results compared to medium-resolution methods (URANS), but are remarkably more expensive primarily due to the finer mesh and smaller time steps required. In case of turbulence-induced vibrations (TIV), scale resolving simulations are irreplaceable. A decision on the resolution level has to

be taken on a case-by-case basis, considering the main objectives of the FIV simulation and the added value provided by high-resolution methods towards these objectives.

Post-process further quantities: Post-processing first-order statistics is not always sufficient. We recommend to evaluate the Q-criterion (Figure 3) and turbulent kinetic energy (TKE) budget in order to carefully examine and analyse the flow phenomena that take place in the CFD domain. These statistics provide high-fidelity data that could also be used for benchmarking and validation of turbulence approaches with lower resolution.

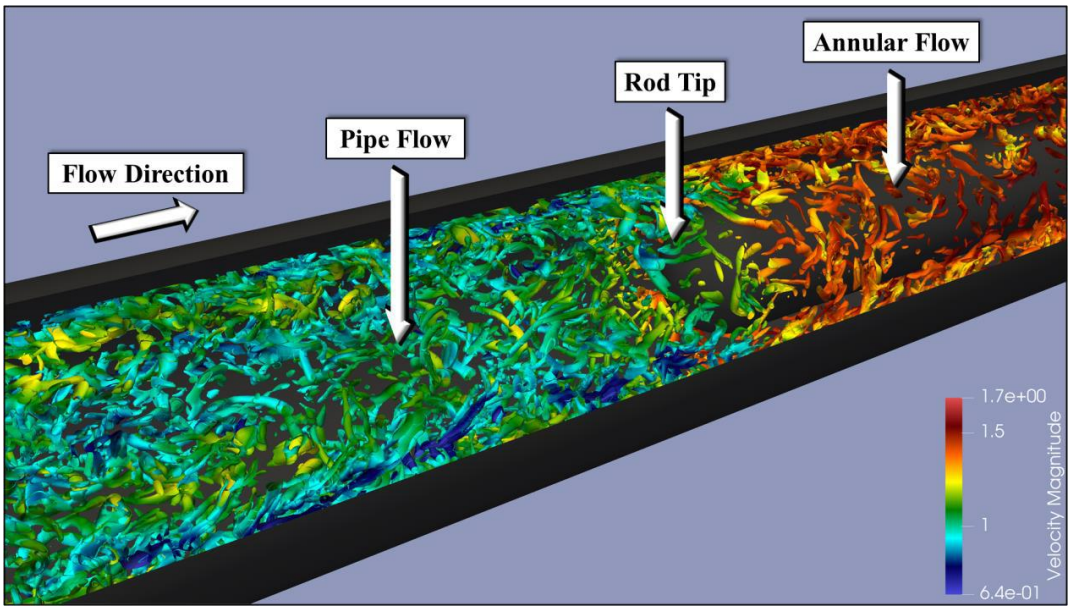

Figure 3: DNS performed by Penn State University (Bistany, 2024): Q-criterion revealing coherent structures in the vicinity of the rod tip.

Medium-resolution approaches

Medium-resolution simulations were performed for the Cantilever Rod experiment using several medium-resolution approaches based on different RANS turbulence models.

Medium-resolution FSI simulations were performed using the SSG RSM (Reynolds stress) model. The key vibrations results showed that the approach worked well in predicting vibrations of a rod with a flat (blunt) tip, although with some underprediction of the rod tip displacement. On the other hand, it completely failed to predict any significant vibration for a rod with a rounded end.

FSI simulations using the k–ε RANS model with cubic constitutive relations were also performed. The low Re-number formulation of the model was used, meaning that the fine mesh resolution is used close at the solid wall boundaries with targeted wall y+ values below 1. Good predictions of the flow field and the vibration frequencies were found. Again, the vibration amplitude was underpredicted. As seen with the SSG RSM model, for the blunt-ended rod the displacements were higher by an order of magnitude than in the cases with curved-ended rod.

FSI simulations using the k–ω SST model were performed as well. The flow field data showed good agreement with experimental data. However, the obtained dynamics of the rod tip displacements in both the front and side directions were peculiar. The vibration amplitude was found to be in the same order of magnitude as the LES prediction. On the other hand, the frequency of vibration was found to be very dissimilar in the two directions of measurement – with more than a factor of 2 between them.

URANS formulations are not sufficient for accurate predictions: Simplified URANS formulations and isotropic turbulence models struggle to accurately process the boundary layer and the generation of vortex structures at the rod tip. These structures have varying frequencies and phases that isotropic models cannot fully restore. The medium resolution methodology predicted flow separation and reattachment around the rod with a blunt tip, where the separation point is defined by the geometry, and this was missing in the cases with curved-ended rod tip. Consequently, the generation of unsteady flow structures around the cantilever free end, which is one of the instigating factors for vibration, is significantly lower for the round tip case than the blunt-ended one. Subsequently, the medium resolution methodology struggled to resolve the unsteady turbulent flow structures around the rod free end for the curved-ended rod case; this led to flow-induced vibration not being predicted correctly.

Pay attention to the choice of turbulence model: it has the largest impact on the accuracy of the results: We found that the choice of turbulence model had the strongest impact on the accuracy of the results. The impact of this choice overshadowed any potential gain in accuracy of using sophisticated CSM models instead of simple beam models.

LES seems to perform best and provide closest results to the experimental ones, but it comes with its price. The URANS approach could be used to some extent for the blunt-ended rod cases, since unsteady flow structures are generated there through the impingement of the flow on the blunt-ended rod. Turbulent pressure fluctuations could not be captured with URANS approaches.

Pay attention to wall treatment: One conclusion derived from the medium resolution simulations on cantilever rod is that the wall treatment plays a significant role not only for high-resolution approaches like LES but also for URANS simulations. The application of the $k-\varepsilon$ RANS model with cubic constitutive relations and in it low Re formulation showed better performance in prediction of FIV than the RANS models that were combined with a kind of the wall-models. This suggests that wall-resolved approach performs better also in combination with RANS models at least for the rounded rod end.

4.2.2 Flow field modelling in fuel bundle simulations

The main feedback from the ALAIN benchmark regarding flow modelling was that the essential role for the success of the CFD simulations plays the choice of the turbulence modeling approach. In this context, the scale-resolving simulation approaches, with LES as the main representative, showed a superior performance in prediction of flow fluctuations and FIV compared to URANS approaches. Furthermore, the wall-modelling turned out as the only feasible option for simulation of the ALAIN benchmark configuration. Among different SRS models used in the ALAIN benchmark, wall-modeled LES showed a better

performance than the Detached Eddy Simulation (DES) and Scale-Adaptive Simulation (SAS) models.

The main drawback from the performed simulations was the computational effort which is associated with the model size required for the simulation of the experimental setup. Therefore, any alternative approaches that can further reduce the computational effort, while still providing sufficient accuracy, are of great value.

The main BPGs for the flow modeling in fuel bundles derived from ALAIN benchmark are:
Use LES or other scale-resolving methods to resolve turbulent flow fluctuations: Since the turbulent pressure fluctuations are the main driver of the FIV, these fluctuations need to be captured properly by the CFD simulation. This requires in general the application of the methods that are capable of resolving turbulent flow structures. The recommended approach is the wall-modeled LES.

Use wall-modelling as the only practical option for larger structures like fuel assemblies: wall-resolving simulations of a single, complete fuel assembly is computationally extremely expensive, and therefore, not feasible for industrial applications. Therefore, wall-modelling is the only reliable alternative in combination with LES or any other scale-resolving simulation approach. Even a wall-modeled LES is still very expensive when applied for a full fuel assembly.
Use of alternative approaches: Alternatively, any other method can also be used that is capable to provide the turbulent pressure fluctuations over the time, which can be integrated over the structural surface to calculate turbulence-induced fluid forces acting on the structure. In GO-VIKING WP5, the AniPFM method and the Synthetic Turbulence approach were developed. These generate pressure/velocity fluctuations, based on information on the turbulence kinetic energy from an URANS approach. More information can be found in (van Zuijlen A. Z., 2026) and (van Zuijlen A. Z., 2024).
URANS might be inappropriate for flows in FA: If the unsteady forces/momenta on the structures result from turbulent fluctuations, URANS simulations are not appropriate. The turbulence model can have a significant influence on the fluid forces/momenta acting on the structures depending on the nature of the unsteady flow. Unsteady flows with vortex-shedding-like behaviour (vortex-shedding can occur also in axial flows) can in principle be simulated with URANS using typical two-equation turbulence transport models. If the unsteady forces/momenta on the structures are primarily a result of the turbulent fluctuations URANS, simulations are not appropriate for this purpose, because their theoretical derivation is based on applying ensemble averages that remove turbulent fluctuations for the "averaged" fields of pressure and velocity. Therefore, more detailed turbulence simulations like LES are necessary to produce the correct fluctuations needed to calculate the fluctuating forces/momenta for the structural mechanical simulations. Comparing pressure and velocity fluctuations between URANS simulations using traditional turbulence models and more detailed simulations like LES can be used to estimate which kind of flow behaviour to expect.
For the analyses of bundles, use a pragmatic approach with respect to numerical mesh resolution: The analyst should not necessarily stick to strict LES requirements (mesh density, y+, Taylor microscales), but rather violate some of the requirements (y+, Taylor microscale, max. CFL, etc.). This will lead to an affordable high-resolution simulation, but the analyst should first question the results and then compare these with available and reliable data in order to gain trust in the generated solution. Further, the investigated

multiple scale-resolving (hybrid) methods showed a reasonable level of agreement with the experimental results for the mean and fluctuating flow field components.

4.3 Structural modelling

4.3.1 Structural modelling in single rod configurations

Be aware that even if high-resolution methods provide good agreement in the flow field statistics with data, they still may underpredict the vibration amplitudes: It was found than even with a very detailed high-resolution CFD domain, which is also in a good agreement with the PIV data, the structural displacements can still be underestimated. In such cases, carefully check your structural model and the CFD-CSM coupling, if such is used.

Apply Euler-Bernoulli instead of Timoshenko beams: The comparison of two beam approaches, the Euler-Bernoulli and the Timoshenko, revealed that while Timoshenko theory is more comprehensive, the simpler Euler-Bernoulli model provided excellent results for the analysed slender rod, with differences only appearing at higher frequencies (mode > 7). The rod vibration frequencies are relatively low, and therefore, the Euler-Bernoulli approach shall be preferred whenever it works well, which is true in most cases.

Investigate the calculation of the forces: A critical finding was that the calculation of forces (qx, qy) on the rod was sensitive to domain decomposition in ANSYS CFX. Therefore, in case of simulation of the beam motion by applying the User Libraries in ANSYS CFX, it is recommended to ensure the entire rod surface resides within a single kernel during parallelization to maintain accuracy. This suggestion is code-specific and relevant to ANSYS CFX users.

Be aware that axial-flow (up-flow or down-flow) introduces compressive or tensile forces that can further shift vibration frequencies.

4.3.2 Structural modelling in bundle configurations

Use beam models also for the structural representation of a whole FA: The mechanical modelling of a FA also requires special attention. Similarly to the use of a fine wall-resolved mesh in the CFD simulation, a representation of the FA structure using 3D structural elements including explicit modelling of the contacts between spacer grids and fuel rods is also considered as not practical or even not feasible at all. Because of its long slender structure, the mechanical model can be well represented by beam elements. In this way, the whole fuel assembly may be modelled by just one or by multiple beams.

Fit your beam model on available data first: A beam model with additional springs to model the stiffness introduced by the spacer grids, which is fitted against experimental values, should be suitable for the simulation of FA general dynamic behaviour. Further effects like internal vibration modes, however, require a more detailed model. For the fitting of the model, a damped modal analysis is suitable, using vibration mode,

natural frequencies and logarithmic decrement for the fitting of the beam moment of inertia, stiffness of the springs and damping (Figure 4).

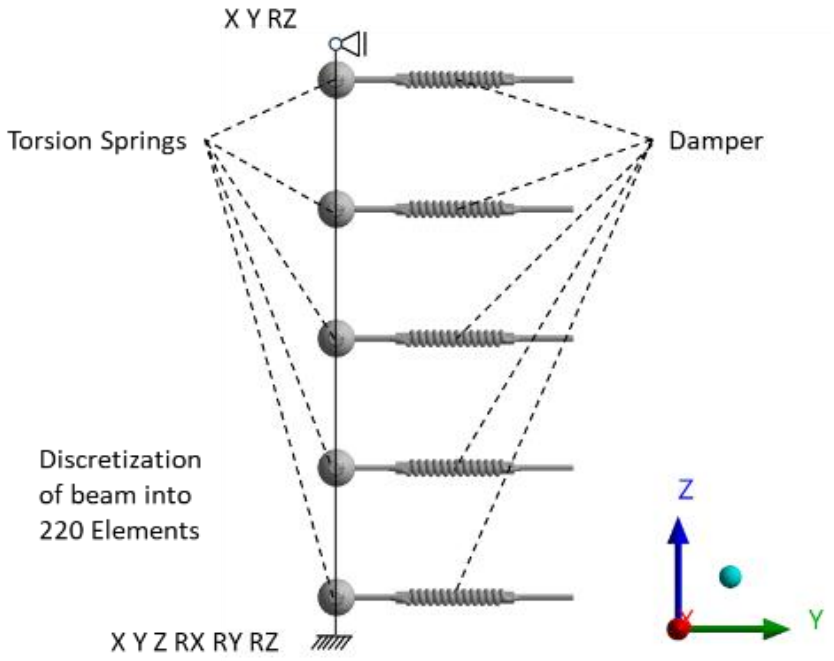

Figure 4: Beam model of the ALAIN 5x5 assembly with torsion springs and dampers, developed by GRS (Blaesius, 2026).

Use other types of reduced-order models to represent the structure: In the frame of ALAIN benchmark IPP used a multiple degree of freedom model to represent the fuel assembly structure. Although not typical, this approach could also be followed for the representation of the fuel assembly structure. For more details the reader is referred to the D2.4 deliverable document.

4.4 Coupling and FSI simulations

For the ALAIN benchmark, coupled FSI simulation turned out to be very challenging for FA and could not be realized in the present project. Although, based on the very small vibration amplitude in the simulated ALAIN case, the usefulness of a coupled FSI simulation may be questioned. At the same time, the decoupled approach showed the capability for reliable prediction of the vibration amplitude, given that all needed parameters are properly provided and evaluated. It should be noted here that the simulations performed with the imposed movement did not show any effect of this very small displacement on the flow field. The same conclusion was drawn also from the FSI simulation of the cantilever rod with wall-resolved LES. Also, in this case no effect of the rod motion on the flow field characteristic was observed.

Thus, the following BPGs can be derived from this case:

Analyse, if coupled FSI simulations are necessary and make sense: Are the expected changes in the geometry due to forces/momenta of the fluid large enough to significantly affect the fluid computational grid? If the cells of the fluid computational grid at the moving walls are much larger than the amplitudes of the wall movements, no effect is to be expected on the flow, and therefore, the simulations of the fluid and the structural domains of the model can be simulated separately. Then the forces/momenta calculated by a stand-alone fluid simulation can be taken as input for a stand-alone simulation of the structural behaviour.

Add the fluid terms in the decoupled approach: In the case of decoupled simulations, in which CFD and mechanical simulations are performed separately, it is very important to add the fluid terms (added mass and fluid damping) in the structural dynamic model. These effects are quite significant for rod bundles, contrary to some simple structures such as a single rod, where these effects are much smaller. Particularly the fluid damping may become significant with increasing axial flow velocity. One has to be careful when deriving these parameters, since their effect on the FIV can be significant.

For the Cantilever Rod case, all FIV simulations were performed as coupled FSI simulations. Different strategies were used. In some cases, the standard FSI procedures implemented directly in the CFD code (STAR-CCM+), while in other cases special self-implementations for FSI coupling were used.

Some remarks on the FSI coupling in context of structural modelling:

The strategy for structural modelling based on the beams as the basic elements affects the strategy for coupled FSI simulations. Most of the standard tools for FSI coupling are based on the common sharing boundary surface between fluid and structural domain. This boundary does not exist explicitly when the fuel assembly is modelled using beam elements. Hence, an alternative strategy is needed to establish the interface between fluid and structural domain in a coupled FSI simulation according to discretization of the structural dynamic model. This, in turn, requires special non-standard solutions for FSI coupling, typically based on some user coding solutions within the CFD codes. This requires some programming and testing effort, since certain non-trivial implementations by the user itself are necessary. However, on the other hand, it may bring a significant benefit in terms of computational efficiency compared to standard general-purpose solutions. This was demonstrated by EDF in the case of Cantilever Rod simulation using Euler-Bernoulli beam as structural model and self-implemented FSI coupling within the CFD code Code_Saturne. The computational cost of the FSI simulation was not appreciably higher compared to stand alone CFD simulation. Similar was done by IPP. In the FSI simulation of both benchmark cases they used user supported libraries for FSI coupling.

4.5 Strategies to increase the efficiency of bundle simulations

To allow the bundle analysis on currently available hardware, the computational effort may need to be reduced through model simplifications. This can be realized through a suitable combination of the following measures:

- Usage of element sizes larger than required (increases uncertainty)

- Reduction of the calculated CFD time interval and space with subsequent extension of the results: Several methods are described in the literature for the extension of CFD data, including staggered sequencing, cutting and reassembling, transfer to the frequency domain and resynthesis.

- Performance of 1-way coupling in combination with added values (added mass, fluid damping) from literature, experiments, experience.

- The settling-in period of the mechanical model has to be cut off when interpreting the results. Its length can be reduced by removing the static part from the data.

4.6 Validation and experimental data

Validate structural and fluid models with vibration tests in vacuum and still water before moving to the cases with non-stagnant fluid: This will provide information on the main fluid and structural mechanics phenomena.

Validate on multiple experimental runs: An extensive validation is necessary to decrease the uncertainty in the predictive capabilities of the codes: the same numerical approaches that give good predictions for certain Reynolds numbers tend to underestimate/overestimate displacements for other Reynolds numbers. Therefore, code validation shall be performed for a number of experimental runs with different boundary conditions to check the validity of the model.

Use high-quality experimental data: High-quality experimental data is crucial for the validation of high-fidelity simulation approaches. Having, for example, just displacements is not sufficient for the proper validation of an FSI code. Particle Image Velocimetry (PIV), Laser Doppler Velocimetry (LDV) or other data that shed light on the flow phenomena in the CFD domain is very helpful. Further, force measurements on rigid structures are very useful for CFD validation as well.

4.7 Conclusions on FIV analysis in single-phase axial-flows

The turbulence modelling approach used in the CFD simulation obviously plays a significant role for the predictability of the vibration response. Based on the very small vibration amplitudes measured in the FIV tests, turbulence is assumed to be the main driving vibration mechanism. According to this assumption, it is essential to use turbulence modelling approaches and turbulence models that are capable of resolving relevant turbulent flow structures that are responsible for excitation of vibratory modes of the fuel assembly structure.

The class of scale-resolving simulation approaches turned out to be the right choice. Thereof, LES was found to be the best performing one in combination with wall-modelling, which is the only feasible option in combination with LES for the given configuration. Wall-resolved LES is generally out of reach for industrial applications due to its disproportional costs. The SAS approach, used in combination with different CFD software and different meshes, showed underperformance in prediction of flow fluctuations compared to LES. It is, however, not quite clear whether this underperformance is just associated with an

insufficiently fine mesh or if it is due to the model itself. This issue is worthwhile to investigate in further research to assess better the performance of the SAS modelling approach for application to FIV simulation.

The modelling approaches based on RANS turbulence models show poor performance in prediction of turbulent flow fluctuations and thus also in prediction of FIV for given cases. Therefore, this class of turbulence models is not suitable for application to FIV, especially in the cases when the FIV is driven by turbulence. In some other cases, when other flow features such as vortex shedding are driving the vibration, the application of RANS models may be successful.

Computational effort turned out as to be a significant limitation in the analysis of the ALAIN benchmark. The requirement for more advanced turbulence modelling approaches such as LES makes it even more difficult to manage a CFD simulation of this very challenging configuration. This was also a significant limitation in terms of performing coupled FSI simulations, which proved to be unfeasible for most participants.

In the context of computational effort, a reduction of the computational domain can be a useful simplification. This is especially recommended for the purpose of assessment of different models or in general different simulation approaches. For a given configuration a single span model represents such a geometrical reduction. The assessment of the performance of different models can be done more efficiently on a single span configuration. When the assessment is completed the best performing model, i.e. approach, can then be applied to the full-scale model.

Given the fact that the vibration amplitudes measured in the FIV tests were very small, just a few micrometres RMS, a decoupled simulation approach seems to be sufficient for the simulation of FIV in such cases. However, it needs to be noted in the context of decoupled simulation approach that the effects of fluid in terms of added (hydrodynamic) mass, fluid damping and fluid stiffening play also an essential role for reliable prediction of the fuel assembly vibration response. The effect of these parameters is probably in the same order as the effect of turbulence-induced excitation forces. In this context, effort needs to be made to evaluate these parameters properly either by simulations or by other means, like e.g. from available test data.

With respect to mechanical modelling of the FA structure, the use of beams as the basic structural elements in combination with additional elements (e.g. springs) for the simulation of a fuel assembly was proven as to be a good choice. This modelling approach keeps the computational costs of the mechanical simulation at a very low level and is practically negligible compared to the computational costs of a CFD simulation, while providing sufficient accuracy for the results of interest. However, this approach does affect the strategy for coupled FSI simulations. Most of the standard tools for FSI coupling are based on the common shared boundary surface between fluid and structural domain. This boundary does not exist explicitly when the fuel assembly is modelled using beam elements. Hence, an alternative strategy is needed to establish the interface between fluid and structural domain in a coupled FSI simulation according to discretization of the structural dynamic model.

Finally, the uncertainties of the experimental measurements, which could not be evaluated exactly, may play a non-negligible role for the assessment of the simulation results. In future experimental investigations, more attention should be paid to the assessment of the measurement uncertainties to have a better basis for the assessment of the simulation results.

5. Best practices for FIV analysis in single-phase cross-flow

5.1 CAD and mesh

The definition of the computational geometry and corresponding mesh forms the foundation of any CFD or FSI simulation of tube bundles in cross-flow. Both D3.2 and D3.4 (see https://go-viking.eu/results) show that inaccuracies introduced at this stage—such as insufficient domain extent or inadequate local refinement—cannot be compensated for by more advanced turbulence models or coupling strategies later in the simulation. The CAD model must, therefore, strike a balance between physical representativeness and computational efficiency, while the mesh must be capable of resolving the key flow features driving turbulence generation and fluid–structure interaction.

In the context of the GO-VIKING simulations, different partners employed a range of modeling tools and approaches, yet converged on a number of consistent best practices regarding geometry extent, simplification strategies, and mesh refinement.

5.1.1 Geometry representation

The benchmark simulations demonstrated that the geometric extent of the computational domain strongly influences both turbulence development and wake organization within the tube bundle. Particular attention should therefore be paid to inlet development length, outlet extent, and preservation of large-scale bundle interactions:

Sufficiently long inlet and outlet regions to ensure realistic flow development: The inlet region should typically extend at least several tube diameters upstream of the first tube row to allow development of velocity and turbulence profiles before entering the bundle and to allow pressure blockage to take place. The outlet region should also be sufficiently long to prevent artificial confinement of wake structures and recirculation zones. Insufficient domain length can lead to incorrect prediction of both flow field and resulting fluid forces.

Full tube bundle geometry: The flow through tube bundles is inherently non-uniform and involves large-scale interaction between neighbouring tubes. Simplified domains, such as periodic subsets, suppress important phenomena like jet switching and asymmetric recirculation. Accurate prediction of fluid forces and vibration response, therefore, requires modelling the complete bundle configuration including the side walls.

Consistency between geometric simplification and inlet boundary conditions: When upstream geometry (e.g., bends or flow conditioning devices) is omitted, realistic inlet conditions must be prescribed to compensate for that simplification and its impact on inlet turbulence structure. Experimental profiles or precursor simulations should be used where possible. Failure to do so can result in incorrect turbulence levels and flow asymmetry.

- **Use 2D simulations for preliminary or engineering-level predictions, but switch to 3D models when high accuracy is needed**: Two-dimensional simulations offer clear

advantages in terms of computational cost and flexibility, especially when studying multiple tube elements or performing parametric analyses. Results from the AMOVI benchmark indicate that 2D approaches can provide reasonably accurate predictions of mean flow behaviour and are therefore suitable for engineering assessments. However, because tube bundle flows are inherently three-dimensional, 2D models are limited in capturing vortex dynamics, wake interactions, and the resulting force fluctuations that drive vibration. These limitations become particularly important in transitional and turbulent regimes. Therefore, while 2D simulations are valuable for preliminary studies, fully three-dimensional models are required for accurate prediction of unsteady forces and flow-induced vibration behaviour.

5.1.2 Mesh topology and refinement

The choice of mesh topology and its local refinement directly determines the ability of the simulation to resolve key flow features such as jet formation, vortex shedding, and wake interactions. Across both D3.2 "AMOVI model preparation and simulation results" and D3.4 "GOKSTAD model preparation and simulation results", all partners converged on the use of predominantly hexahedral meshes, with strong local refinement in physically important regions. This is illustrated in Figure 5, where mesh clustering is clearly visible around the tubes and in the inter-tube gaps.

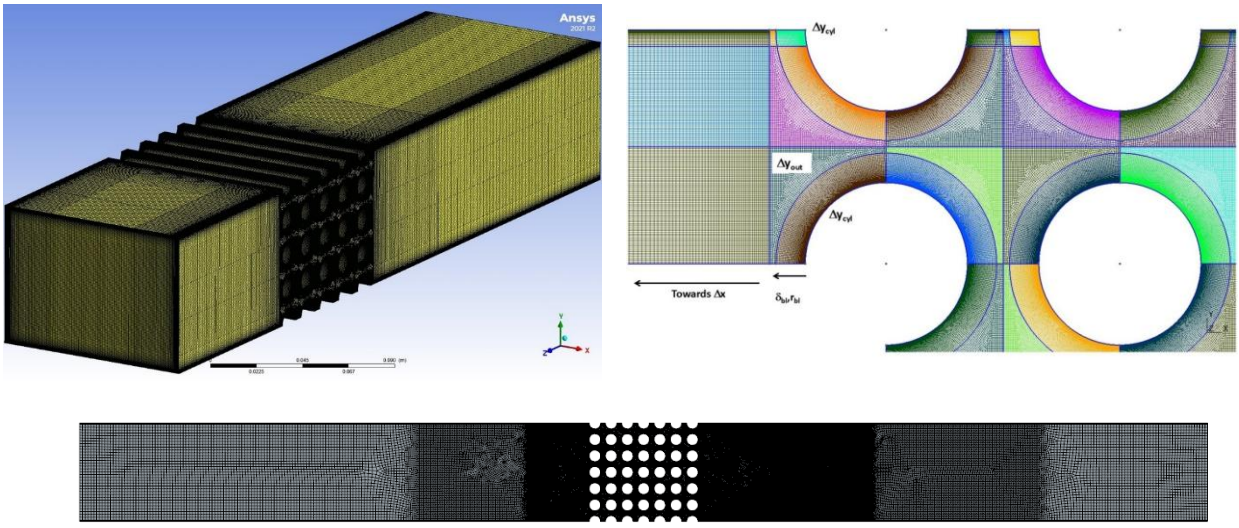

Figure 5: Examples of meshes used by the various partners within WP3, with top left: mesh GRS for AMOVI, top right: mesh EDF for AMOVI, and bottom: mesh NRG PALLAS for GOKSTAD.

Based on these observations, the following Best Practice Guidelines are identified:

- **Use hexahedral-dominant meshes to resolve flow structures efficiently**: Hexahedral meshes are preferred due to their alignment with the primary flow direction and their ability to capture narrow gap flows with reduced numerical diffusion. In tube bundle simulations, this is particularly important for resolving jet-like structures between cylinders and maintaining accuracy in shear layers.

- **Apply strong local refinement in inter-tube gaps and wake regions**: The flow acceleration between tubes leads to the formation of high-velocity jets, which subsequently interact and form recirculation zones downstream. These regions require fine spatial resolution to capture gradients in velocity and turbulence. As shown in Figure 5, mesh density is highest in these regions to accurately resolve the dominant flow features.
- **Ensure adequate mesh resolution around tube surfaces**: Boundary layers and separation points on the cylinder surfaces strongly influence wake development and force generation. A sufficiently refined mesh around each tube is therefore essential to capture vortex shedding and pressure distribution accurately.
- **Distribute mesh resolution according to physical importance rather than uniform refinement**: Uniform refinement of the entire domain is computationally inefficient. Instead, mesh density should be concentrated in regions where flow gradients are highest (e.g., near walls, gaps, and wakes), while coarser resolution can be used in less critical regions such as far upstream or fully developed outlet zones.
- **Perform mesh sensitivity studies to verify solution independence**: Mesh independence should be assessed by comparing quantities such as velocity profiles, TKE, and force coefficients across successively refined meshes. A mesh is considered adequate when further refinement leads to negligible changes in these quantities, see Figure 6.

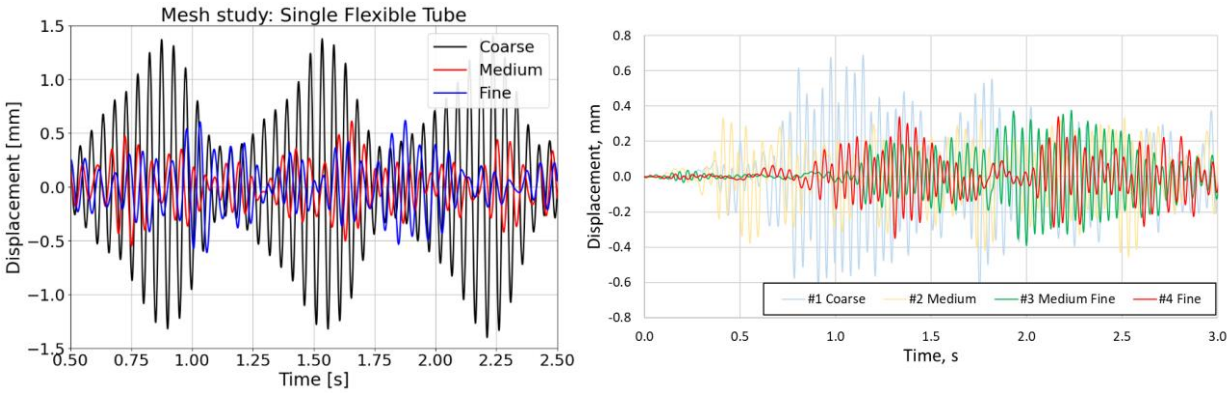

Figure 6: Example mesh independence studies performed for AMOVI, with left: for a single flexible tube (GRS) and right: for a flexible tube inside a rigid tube bundle (IPP).

5.1.3 Near-wall resolution and turbulence scale considerations

Accurate prediction of flow-induced forces requires proper resolution of near-wall behaviour and relevant turbulence scales. In high-resolution simulations, this is achieved through wall-resolved meshes and validation against characteristic turbulence length scales. Hence, before starting the FSI simulations, it is crucial the distribution of these parameters is properly checked and validated.

Based on these findings, the following core Best Practice Guidelines are identified:

- **Maintain low y^+ values for wall-resolved simulations**: For LES or wall-resolved approaches, y^+ values should be kept close to or below 1 across most of the geometry. This ensures that boundary layers are fully resolved without reliance on wall functions, enabling accurate prediction of separation and shear-layer development. The aspects ratio of the near-wall computational cells should remain of the order of few tens in LES.

- **Resolve turbulence scales relevant to the flow regime**: The mesh should be fine enough to capture the dominant turbulent structures in the flow. This means at least a couple of bulk cells should be present in between cylinders for URANS approaches. For LES, one useful metric to compute before running LES on the full bundle geometry is the Taylor microscale. The Taylor microscale acts as an average of the dissipative scales in the turbulent spectrum. Comparing the computed Taylor microscale to the local mesh spacing allows an assessment of whether the resolved turbulence is adequately captured when doing LES. Therefore, beginning the simulation campaign with a RANS simulation can be of great use. The Taylor microscale can be computed using turbulent variables from the RANS simulation, and can be compared against the mesh spacing in the LES mesh. Figure 7 shows an example of this comparison for the GOKSTAD tube bundle.

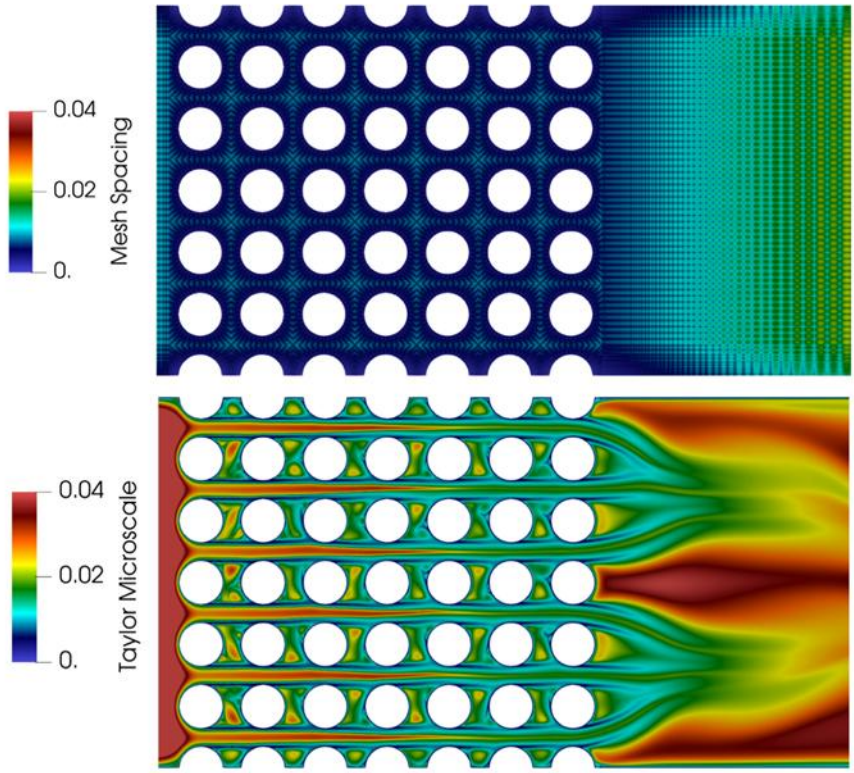

Figure 7: GLL spacing of a spectral element method mesh used in Nek5000 for LES (Top) and Taylor microscale computed from a RANS simulation using Nek5000 (Bottom) (VCU).

- **Avoid over-reliance on turbulence models to compensate for insufficient resolution**: Insufficient near-wall or turbulence resolution cannot be fully

compensated by advanced turbulence models. Even with LES or hybrid models, inadequate mesh resolution will result in inaccurate prediction of TKE, shear layers, and ultimately fluid forces.

5.2 Flow field and turbulence modelling

The accurate modelling of the flow field is central to predicting flow-induced vibrations in tube bundles, as the fluid forces driving structural motion originate from turbulent flow structures. Both D3.2 and D3.4 demonstrate that the choice of turbulence model, inlet conditions, and temporal resolution strongly influence the predicted velocity field, turbulence levels, and force spectra. In particular, resolving the dominant flow features within the bundle—such as jet formation, recirculation, and turbulence generation—is essential for reliable FSI predictions.

Accurate prediction of flow-induced vibrations requires a consistent and physically correct representation of the turbulent flow structures within the tube bundle. The flow is dominated by strong acceleration through inter-tube gaps, leading to jet formation resembling to Coanda effect, wake interactions, and progressive turbulence generation, which directly determine the fluid forces acting on the structure. Different turbulence models reproduce these features with varying degrees of accuracy, ranging from high-fidelity LES to more approximate URANS or hybrid approaches, see Figure 8. While the choice of turbulence model depends on the desired accuracy, and available computer power, it was found that certain models give better predictions for similar costs.

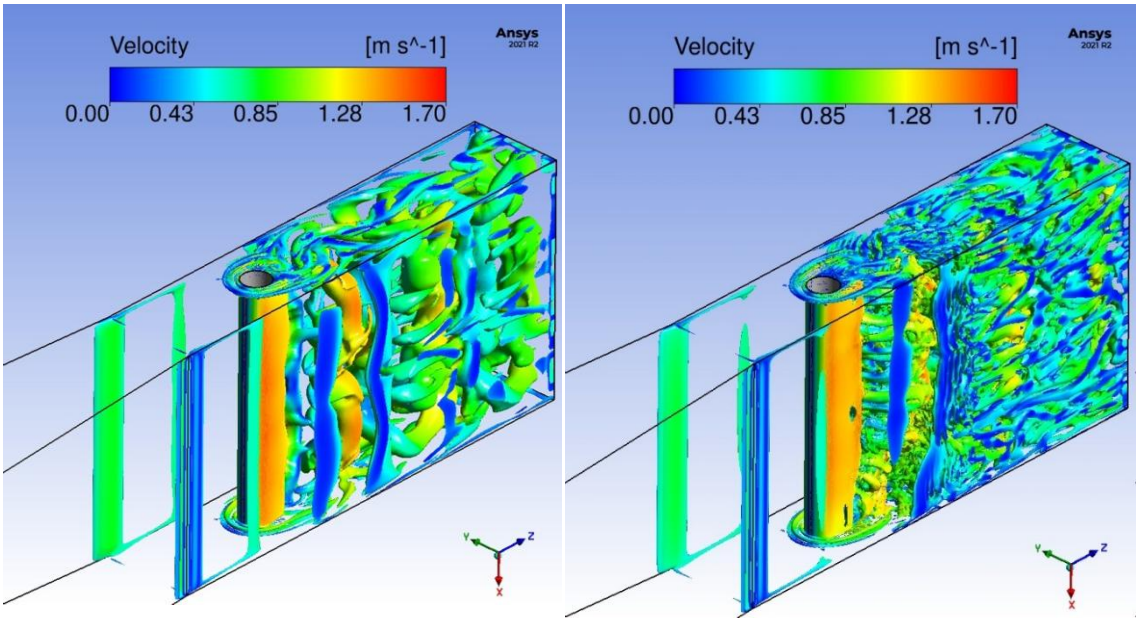

Figure 8: Resolved structures by GRS for AMOVI for single rigid tube in cross flow. Right: k-ω SST, left: SAS.

Based on these findings, the following core Best Practice Guidelines are identified:

- **Ensure that the selected turbulence model can reproduce the dominant flow physics inside the bundle**: The simulation must capture key mechanisms such as jet formation, jet switching, shear-layer instability, and wake interactions. These features directly determine the distribution of velocity, pressure, and turbulence, and therefore the forces acting on the tubes. A model that fails to reproduce these mechanisms will lead to inaccurate FIV predictions.

- **Prefer LES or hybrid turbulence models when accurate prediction of unsteady flow features is required**: LES provides the highest level of fidelity, resolving the temporal and spatial evolution of turbulent structures. Hybrid approaches such as IDDES, or other scale-resolving models offer a practical alternative, capturing large-scale turbulence while reducing computational cost compared to full LES.

- **Use URANS models cautiously and only for mean flow analysis**: URANS approaches using the two-equation eddy viscosity models or resolving Reynolds stresses can provide acceptable predictions of time-averaged quantities, but generally underpredict turbulence intensity and fail to accurately reproduce fluctuating forces, and the resulting tube displacements. Their use should therefore be limited to preliminary design studies or cases where detailed FSI behaviour is not of primary interest.

5.3 Inlet boundary conditions

The specification of inlet boundary conditions is a critical factor in determining the accuracy of CFD and FSI simulations of tube bundles in cross-flow. The inlet not only defines the initial velocity field, but also governs the development of turbulence structures entering the bundle, which in turn directly influence jet formation, wake interactions, and the resulting fluid forces. Inadequate or overly simplified inlet conditions—such as uniform velocity profiles without proper turbulence specification—may lead to significant deviations in the predicted flow field. Since these discrepancies propagate downstream, they can affect turbulence levels, force fluctuations, and ultimately the structural response. Ensuring physically realistic and properly developed inlet conditions is therefore essential for reliable simulation results.

Based on these findings, the following Best Practice Guidelines are identified:

- **Use physically realistic inlet conditions that include both velocity and turbulence characteristics**: Prescribing only a uniform velocity profile is insufficient for capturing the development of turbulence inside the bundle. Inlet conditions should include turbulence intensity and length scales representative of the physical system.

- **Prefer inlet generation techniques that produce developed turbulence for LES**: A common failure mode in LES is insufficient turbulence development in the inlet region of the domain. The inlet region must provide enough development length, or be supplemented by a precursor channel simulation, to allow turbulence to develop in the inlet before reaching the region of interest, see Figure 9. Embedded recycling methods are also an effective tool to ensure the inlet is properly

developed. A recommended methodology is to first run a precursor LES of only the inlet channel, using periodic boundary condition, on a channel length of 10 Dh. It is worth stating that Dh is the hydraulic diameter of the inlet channel, not the hydraulic diameter related to the tube bundle. Once the inlet channel is turbulent and fully developed, the solution can be initialized onto the full domain. The 10 Dh length is kept as the inlet for the full bundle, and a recycling boundary is used to impose the developed solution downstream on the inlet boundary. This ensures that a fully developed turbulent inlet is used when doing LES.

- **Align inlet conditions with available experimental data where possible**: Experimental velocity and TKE profiles provide the most reliable basis for defining inlet conditions. Using such data improves agreement between simulations and measurements and enhances predictive accuracy.

- **Recognize the impact of inlet conditions on downstream predictions**: Errors introduced at the inlet propagate throughout the domain and affect velocity fields, turbulence levels, and ultimately structural response. Proper initialization of the inlet is therefore essential for accurate FSI simulations.

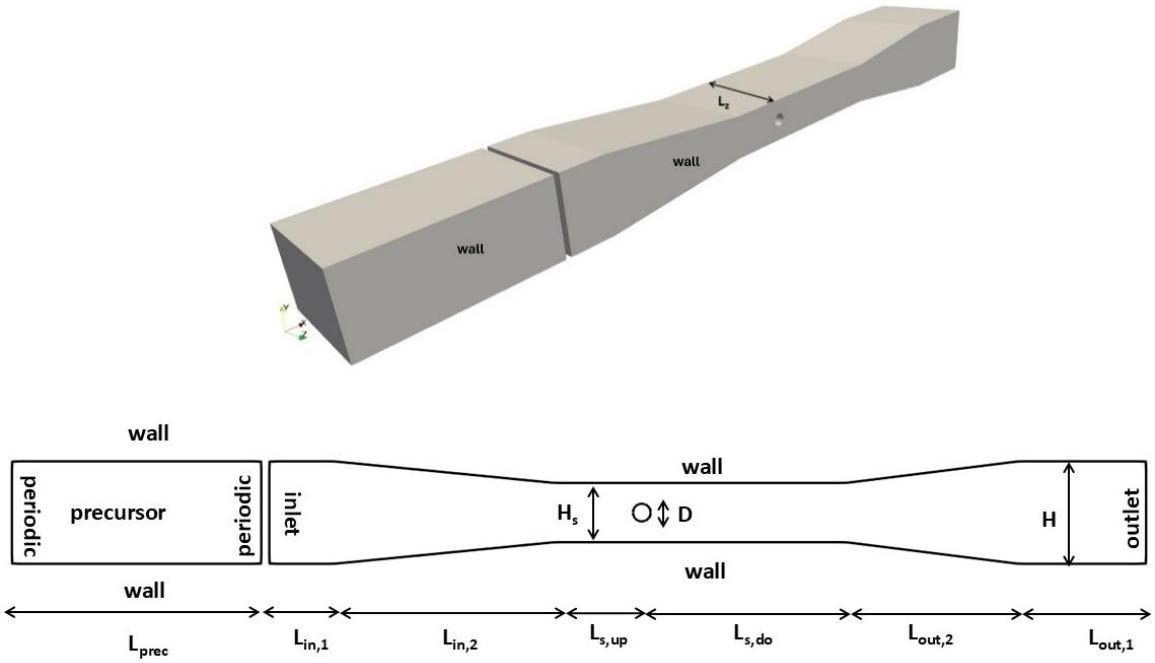

Figure 9: The computational domain created by EDF for the single cylinder configuration of AMOVI, clearly showing the precursor used to generate proper inlet conditions.

5.4 Temporal resolution and convergence

Accurate prediction of flow-induced vibrations requires not only proper spatial resolution, but also sufficient temporal resolution to capture the unsteady dynamics of the flow and its interaction with the structure. The time step directly affects the ability to resolve turbulence fluctuations, vortex shedding, and force oscillations, while the total simulation time determines whether statistically meaningful results can be obtained, see Figure 10. Inadequate temporal resolution or insufficient simulation duration can lead to

underpredicted fluctuations, distorted frequency content, or premature conclusions regarding convergence. Careful selection of time-stepping parameters, simulation length and monitoring of convergence is therefore essential for reliable CFD and FSI simulations.

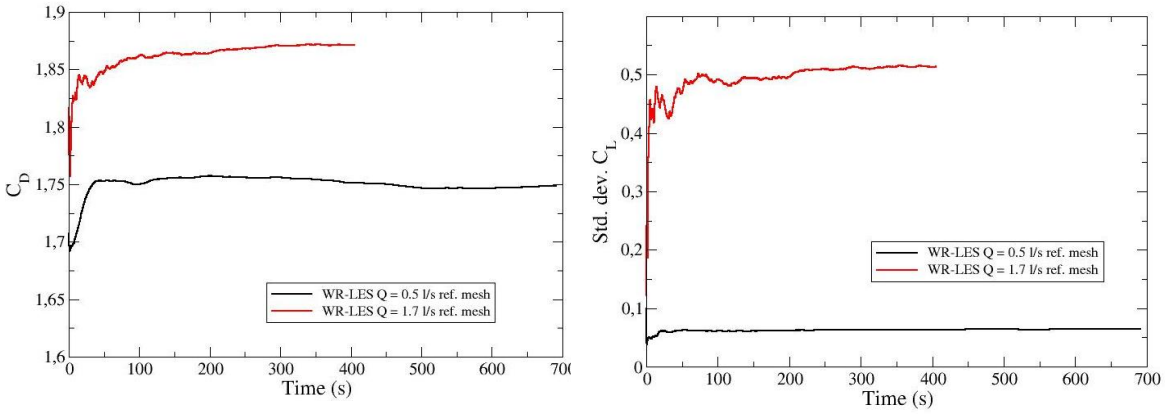

Figure 10: Time evolution of the mean drag coefficient (left) and the standard deviation of the lift coefficient (right) as obtained by EDF for the single rigid cylinder configuration.

Based on these observations, the following core Best Practice Guidelines are identified:

- **Select time steps that ensure both numerical stability and temporal resolution of turbulence**: Time-step size should be chosen such that the CFL number remains mostly below unity, especially in the zones of interest. This ensures that transient flow structures, such as vortex shedding and turbulence fluctuations, are resolved accurately.

- **Run simulations for sufficiently long durations to obtain statistically converged results**: Flow-induced vibration analyses rely on time-averaged quantities and fluctuating signals, such as RMS values, dominant frequencies, and force spectra. In tube bundles, the flow is not only turbulent, but can also exhibit large-scale, low-frequency phenomena, such as jet switching and flow bifurcation between parallel flow paths. These effects lead to long-time variability in the flow field, where the flow may alternate between different quasi-stable states. Capturing this behaviour requires simulations to be run over several flow-through times, often significantly longer than what is needed for simple steady or periodic flows. If the simulation time is too short, these slow dynamics may not be fully sampled, leading to biased averages, underpredicted fluctuations, or missing frequency content. Ensuring sufficiently long sampling times is, therefore, essential for obtaining statistically converged and physically representative results.

- **Use higher-order temporal discretization schemes**: Second-order schemes provide a good balance between accuracy and computational cost, and were consistently used across all partners. Lower-order schemes may introduce excessive numerical diffusion and reduce the accuracy of unsteady predictions

- **Verify convergence using both flow and force statistics**: Convergence should be evaluated by monitoring key physical outputs, such as velocity profiles, turbulent

kinetic energy, force time histories, and displacement signals. Residual reduction alone is insufficient to guarantee that the flow field and FSI response are statistically converged.

5.5 Structural modeling

The structural modelling of tubes in cross-flow conditions plays a crucial role in the accurate prediction of FIV. While the fluid domain governs the excitation mechanisms, the structural model determines how these forces are translated into motion, and therefore directly influences displacement amplitudes, frequencies, and coupling behaviour.

Both D3.2 and D3.4 demonstrate that, despite the complexity of the flow field, the structural response of tube bundles can often be represented using rather simplified models. In particular, the use of beam modelling or rigid-body approximations, possibly combined with mass–spring–damper formulations, was found to provide robust and computationally efficient predictions across a wide range of simulation approaches

5.5.1 Structural representation and degrees-of-freedom

The structural representation defines how fluid-induced forces are translated into motion and therefore directly influences the predicted vibration amplitudes, dominant frequencies, and resonance behaviour. In many tube bundle applications, the structural response is dominated by relatively simple kinematics, allowing the use of reduced-order models. However, the choice of representation also determines the ability to capture more complex dynamic effects, such as multi-modal behaviour or transition to resonance, which may become important depending on the flow regime and system configuration.

In the GOKSTAD benchmark, the experimental setup was designed to allow a cylinder to move in two orthogonal directions, which was consistently represented numerically using mass–spring–damper systems with two degrees of freedom (2-DOF). In contrast, the AMOVI configuration consists of a rigid cylinder mounted on a flexible blade, resulting in motion that is primarily constrained to a single degree of freedom in the lift direction. While simplified structural models can adequately capture this behaviour for small displacements, the AMOVI results indicate that the structural representation becomes more important as the dynamic response becomes more complex. Figure 11 shows two such possible representations for the moving cylinder in AMOVI.

Based on these findings, the following core Best Practice Guidelines are identified:

- **Use rigid-body mass–spring–damper models for systems with predominantly translational motion:** For configurations such as GOKSTAD, where the cylinder motion is intentionally constrained to pure translation (e.g. 2-DOF in lift and drag), a rigid-body mass–spring–damper representation is sufficient to capture the dominant vibration amplitudes and frequencies. This approach provides a robust and computationally efficient model that performs consistently across different solvers and turbulence modeling approaches.

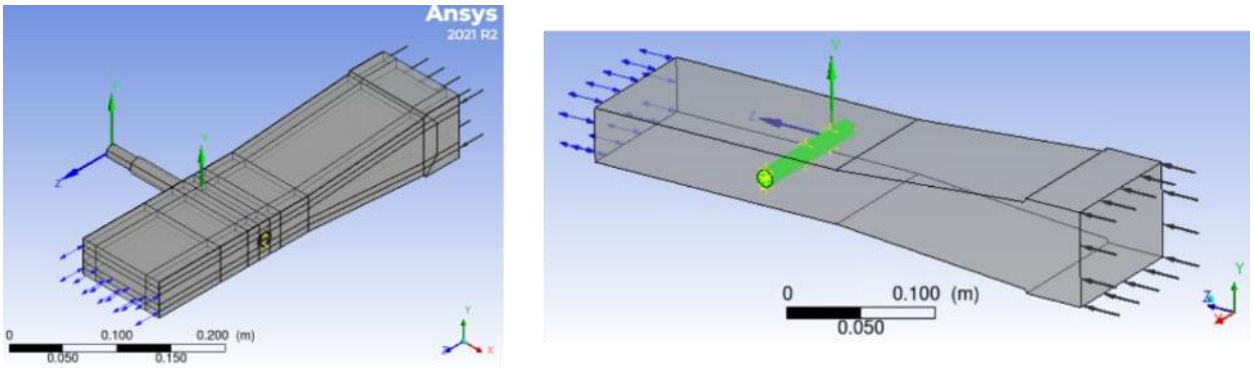

Figure 11: CAD models based on the used Rigid Body Motion approaches as used by GRS for AMOVI, with left: rotational motion model and right: translational motion model.

- **Select the appropriate degree of freedom and motion formulation based on the physical support structure**: The structural model should reflect how the tube is mounted and allowed to move. In AMOVI, where the cylinder is attached to a bending blade, the motion is governed by the deformation of the support and can be represented either as translational motion or as an equivalent rotational (pendulum-like) motion. Results indicate that explicitly accounting for this kinematic behaviour (e.g. through rotational formulations, as done by GRS) can improve the representation of the system dynamics compared to overly simplified assumptions, see Figure 12.

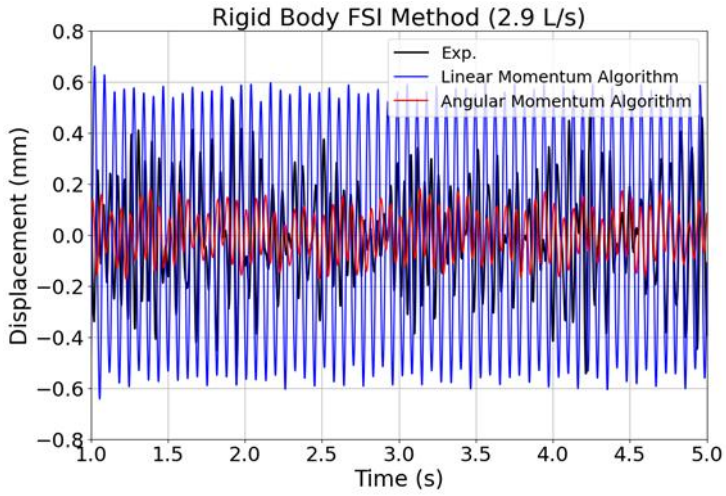

Figure 12: Tube displacements, calculated with linear and angular momentum algorithms (GRS).

- **Increase structural model fidelity when kinematics, deformation, or coupling complexity become important:** While simplified rigid-body models are sufficient for small-amplitude and purely translational motion, they may become inadequate when structural deformation, non-uniform motion, or resonance effects play a role. In such cases, more advanced representations—such as beam models or multi-degree-of-freedom systems (as explored by IPP)—allow inclusion of distributed structural properties and multiple vibration modes. This improves the prediction of coupled dynamics, particularly near resonance and in cases where the response is governed by more than a single frequency. The selection of model fidelity should

therefore follow the physical complexity of the problem and the required level of accuracy.

5.5.2 Structural properties and dynamic response

The dynamic response of the structure is governed by a small set of physical parameters—primarily mass, stiffness, and damping—which together define the system's natural frequencies and damping characteristics. These parameters control how the structure responds to fluid forcing, including the amplitude of oscillations, the onset of resonance, and the overall stability of the motion. As such, even when simplified structural models are used, the accurate specification and interpretation of these parameters is essential for reliable prediction of flow-induced vibrations.

Based on the findings in D3.2 and D3.4, the following Best Practice Guidelines are identified:

- **Use experimentally validated structural parameters and ensure consistency across simulations**: Mass, stiffness, and damping coefficients should be derived from experimental data or validated references whenever possible, as they directly determine the structural response. Using consistent and accurate input parameters ensures that differences between simulations reflect modelling choices rather than inconsistencies in structural definition.

- **Ensure that the structural model reproduces the correct dynamic characteristics, particularly natural frequency and damping behaviour**: The natural frequency is a key parameter governing resonance and must be verified against experimental or expected values. Damping should also be carefully specified, as it strongly influences the amplitude of oscillations: underestimating damping can lead to exaggerated responses, while overestimating it may suppress physically relevant vibration modes.

- **Assess structural response in both time and frequency domains to ensure physical realism and convergence**: Time-domain signals (e.g. displacement and force histories) provide insight into transient and steady-state behaviour, while frequency-domain analysis is essential for identifying dominant oscillation modes and resonance effects. Consistent interpretation of both domains is required to validate the dynamic response and ensure that key features of the coupled system are correctly captured, see e.g. Figure 13.

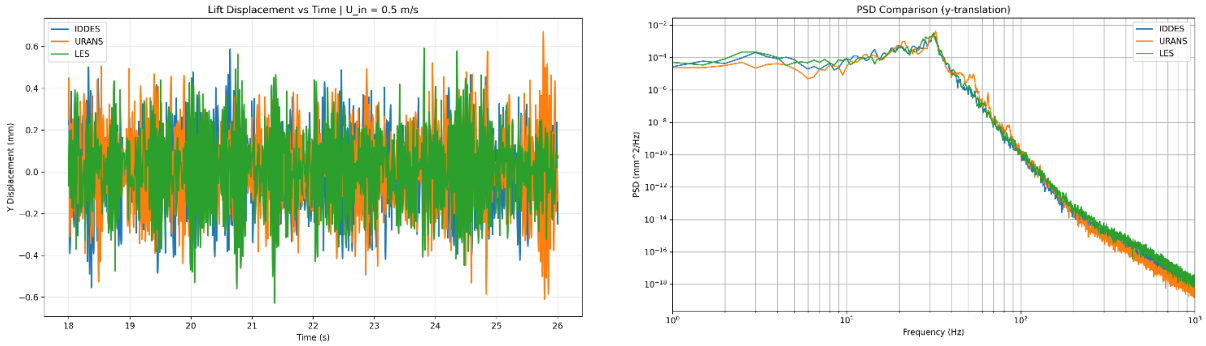

Figure 13: Lift displacement time series (left) and PSD (right) as obtained by NRG PALLAS for GOKSTAD.

5.6 Coupling and FSI simulations

The coupling between the fluid and structural domains determines how flow-induced forces drive structural motion and how this motion feeds back into the flow. The importance of this interaction depends on the vibration amplitude and flow regime, with weakly coupled cases requiring less complex treatment than strongly coupled systems. Results from D3.2 and D3.4 show that both one-way and two-way coupling approaches can be appropriate, provided that the selected method matches the physical level of interaction and the desired accuracy.

Based on the findings of the participants to WP3, the following core Best Practice Guidelines are identified:

- **Select the coupling strategy (one-way vs two-way) based on the strength of fluid–structure interaction**: One-way coupling is generally sufficient for small-amplitude oscillations where structural motion does not significantly influence the flow field, and provides an efficient approach for engineering studies. However, when structural motion alters the flow—such as near resonance or in higher-amplitude regimes—two-way coupling is required to capture feedback effects and accurately predict force fluctuations and spectral behaviour.

- **Ensure consistent and stable initialization of the coupled simulation**: FSI simulations should be initialized from a developed flow field, and sufficient time should be allowed for the coupled system to evolve before collecting FSI statistics. Structural motion or coupling should be introduced gradually to avoid non-physical transients and numerical instability. Sequential or phased approaches, as used in D3.4, are effective in achieving stable and physically representative coupling.

- **Use appropriate temporal resolution and coupling convergence settings**: The time step must resolve both fluid and structural dynamics, while in two-way coupling sufficient inner iterations per time step are required to ensure accurate force transfer. Inadequate temporal or coupling resolution can lead to damping or distortion of vibration signals or even to a divergence of the solution.

- **Apply robust mesh motion strategies to maintain mesh quality during structural deformation:** Mesh deformation, particularly in confined regions near moving boundaries, can lead to severe cell distortion and simulation failure. Results from GRS and EDF show that appropriate boundary conditions and mesh motion strategies—such as allowing tangential sliding along walls or applying rigid motion in near-wall regions, see Figure 14—are essential to preserve mesh quality and ensure numerical stability during FSI simulations.

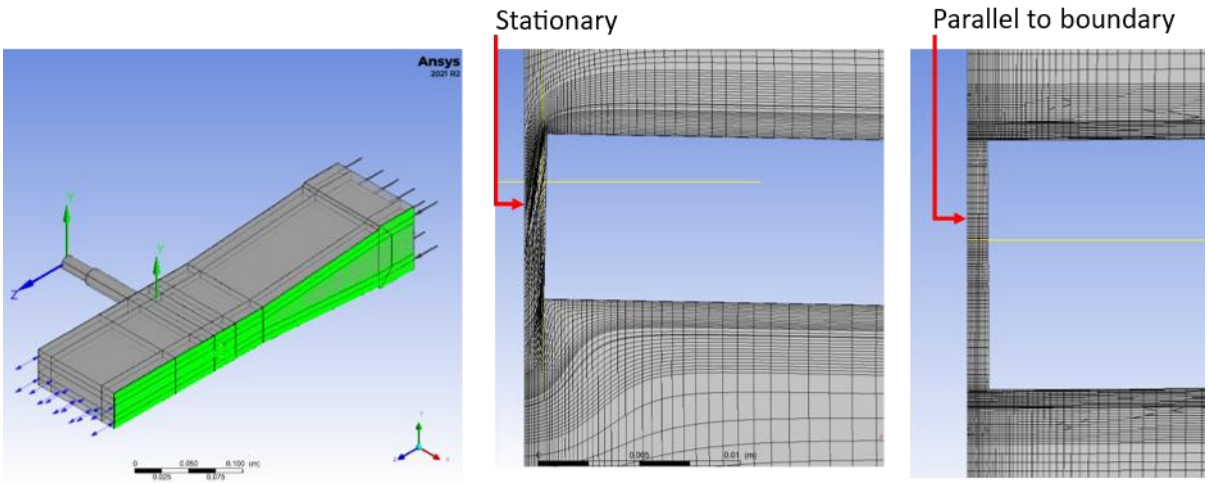

Figure 14: Test section with marked channel wall (left), Boundary condition for wall parallel to moving side of flexible tube

- **Perform reduced-scale tests before launching the full-scale simulations:** Before simulating the full tube bundle, a recommended strategy is to first perform a small periodic tube bundle precursor simulation. A reduced domain containing only a few repeating tube pitches and periodic boundary conditions allows rapid testing of mesh quality, numerical stability, and turbulence behaviour in the domain at a small fraction of the full computational cost. These preliminary calculations provide an efficient environment for tuning simulation parameters and identifying setup issues before scaling to the complete geometry. However, when employing this approach, appropriate domain sizing needs to be selected to ensure the periodic boundary conditions do not artificially constrain the flow. As a general guideline, a minimum spanwise domain extent of three tube diameters and a streamwise domain length of at least three tube pitches is recommended. Figure 15 shows an example of a reduced computational domain of the GOKSTAD tube bundle using periodic boundary conditions.

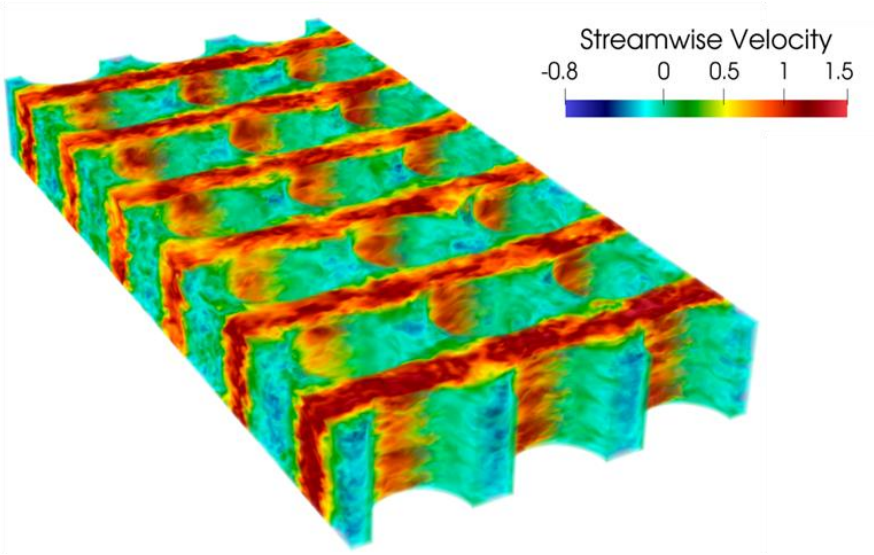

Figure 15: Volume rendered streamwise velocity contour plot of a reduced computational domain of the GOKSTAD tube bundle.

5.7 Conclusions on FIV analysis in single-phase cross-flows

The combined analysis of the GOKSTAD and AMOVI benchmarks demonstrates that reliable prediction of flow-induced vibrations requires a consistent treatment of both the fluid and structural domains, as well as their interaction. The studies confirm that relatively simple modelling approaches—such as rigid-body mass–spring–damper systems combined with appropriately chosen turbulence models—can provide satisfactory predictions for cases with limited structural deformation and weak coupling. This makes such approaches well-suited for engineering applications, where computational efficiency is often a key requirement.

At the same time, the results clearly show that increasing physical complexity—such as strong turbulence, flow asymmetry, structural deformation, or proximity to resonance—requires increases in model fidelity. In the fluid domain, this is reflected in the need for more advanced turbulence modelling and realistic inlet conditions, while in the structural domain, more detailed representations (e.g. beam models or multi-degree-of-freedom systems) may be required to capture the full dynamic response. Similarly, the selection of coupling strategy must reflect the strength of fluid–structure interaction, with two-way coupling becoming essential when structural motion significantly influences the flow.

A key overarching conclusion is that many of the largest discrepancies between simulations do not arise from the choice of the solver itself, but rather from modeling assumptions and setup choices. In particular, inlet boundary conditions, mesh resolution, and temporal convergence were identified as dominant sources of uncertainty. In addition, practical aspects such as mesh motion and deformation handling in FSI simulations were shown to play a critical role in ensuring numerical stability and solution reliability. These findings highlight the importance of careful setup and validation procedures in addition to selecting appropriate physical models.

Finally, the work demonstrates the value of a hierarchical modelling approach, in which model complexity is increased only when required by the physics of the problem. Simplified models can be used effectively for screening studies and initial design assessments, while higher-fidelity approaches should be applied when detailed understanding of flow dynamics, force fluctuations, or resonance behaviour is required. The Best Practice Guidelines presented in this report provide a structured framework to support such decisions, enabling users to select appropriate modelling strategies based on the specific requirements of their application.

Funded by
the European Union

6. Best practices for FIV analysis in multiphase flows

6.1 Numerical simulation of multiphase flows

Identify the simulated two-phase flow regime: A precise identification of the flow regime (bubbly, slug, churn, annular) is essential, as it determines both the amplitude and the nature of excitation (single-phase turbulence vs. two-phase flow-induced fluctuations). In low liquid superficial velocity regimes, fluctuations are often dominated by bubble or large gas structure dynamics rather than classical turbulence. In such cases, Euler-Euler dispersed approaches may fail to predict forces because bubble-induced agitation is modeled, and two-phase turbulence models are not yet fully mature.

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

In this sense, the Generalized Large Interface Model (GLIM) from neptune_cfd, where the large gas structures are resolved and the smaller ones are modelled (considered as dispersed) was applied for the multiphase simulations. The strategy here is to resolve the main gas structures in the flow in order to predict the forces. This condition drives the mesh refinement.

The Volume-of-Fluid (VOF) method is a good option for FIV analyses in two-phase flows: For two-phase modelling of FIV, the VOF method is a reasonable alternative to the Eulerian multi-field approaches in cases with both bubbly and large interface flow regions. Based on the simulations of the GO-VIKING two-phase benchmarks, the largest structural vibrations may generally be mainly due to the large-scale interfaces; this is also intuitively expected. With the VOF approach, the mesh cannot typically be fine enough to resolve the smallest bubbles (as at least 5-6 cells will be necessary for their resolution), , in the simulated cases, this does not necessarily corrupt the solution severely and reasonable results for the two-phase fields and structural response may be nonetheless obtained (Figure 16). However, for bubbly flow or flow pattern transition, the absence of bubble modelization might fail to predict the void distribution so the forces.

Stabilize the VOF solution: The VOF modelling may require specific settings to stabilize the flow solution. Firstly, local unphysical high velocities sometimes appear at the air-water interfaces, causing divergence of the solution; this can be avoided at least by applying velocity limiting. It seems that fairly high values of the velocity limit are enough for stabilization, e.g. 10-20 times the maximum of mean mixture velocity. Possibly higher values than this could be used, although this was not tested. As the limit is high compared to the physically reasonable velocities, it is expected to have a minimal effect on the results as the limiting is typically temporarily applied only on a small group of cells. Secondly, with the bounded central differencing (BCD) scheme, the boundedness factor may require

adjustment such that stability (numerical diffusion) is increased at the expense of accuracy. Thirdly, in conjunction with LES, the sub-grid scale (SGS) model parameter may need to be increased somewhat, in order to have higher eddy viscosity and thus a more stable solution.

Employment of hybrid two-fluid approaches: Methods that are somewhere in-between VOF and the classical two-fluid models are also available and could be used for FIV evaluations. These hybrid two-fluid approaches, developed and used by NRG PALLAS within OpenFoam, are capable of resolving large-scale, two-phase structures in a VOF-like manner when the grid resolution is sufficient. For smaller structures below the grid size, the model transitions to a classical two-fluid method.

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

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

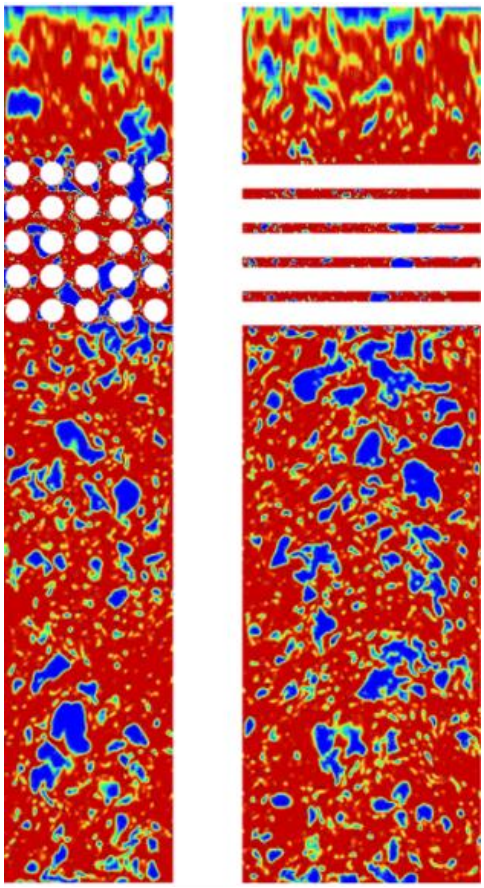

Figure 16: Void distribution in the TREFLE experiment, calculated with FLUENT using VOF in combination with LES (VTT).

Perform fast-running orientation simulations: Simple 1D calculations are highly recommended to provide order-of-magnitude estimates for the simulation. These do not need to be done exclusively with 3D CFD programs. System codes like ATHLET, CATHARE, RELAP can be used to provide some first orientation in the investigated flow regime. These results can be used in combination with analytical correlations to get a rough first estimation of the expected vibration frequencies, displacements, etc.

It can be concluded that two-phase FIV still remains a challenging analysis case, mainly due to the error margins involved in predicting the flow field and structural loads, particularly if phase change is present.

6.2 Mesh and resolution of the gas-liquid interface

Properly address the mesh refinement: Sufficient prism layers are necessary around the tube walls for both the FIV modelling and bubble break-up effects. Adaptive mesh refinement significantly increases computational cost, but can improve computational accuracy. However, remeshing the CFD domain after each time-step requires large computational effort that in the end might be too expensive to afford and can have limited added value given its costs.

Use mesh smoothing when using ALE: for ALE simulations, smoothing the internal CFD mesh at each time step (sometimes called mesh morphing) can be expensive, so that the computing time may grow e.g. by a factor of 3-4 compared to fixed geometry (although this depends on the mesh smoothing method). It is, thus, advisable to deform the mesh only at regions where required and to optimize the smoothing algorithm settings for faster computations (e.g. number of iterations for diffusion-based methods). When taking into account all of the factors which add to the computational load, it can be said that two-way FSI simulations of vibrations issues may become very expensive compared to just obtaining flow field statistics on fixed geometry. Furthermore, the simulation setup becomes more demanding. Therefore, it is also worthwhile to pursue one-way coupled FSI calculations with added fluid mass and flow-induced damping if the case allows it.

Ensure good overall mesh quality: Mesh refinement is critical near the structure and in regions of strong void fraction gradients. Poor resolution leads to inaccurate force predictions and vibration amplitudes. Upstream mesh quality is also essential, therefore:

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

Without proper upstream resolution, structural loading predictions are unreliable. Grid convergence studies are mandatory. Without them, quantitative conclusions are not reliable. Figure 17 shows a high-quality mesh, generated and used by EDF for the simulations of the TITAN experiment.

Use Immersed Boundary method if you want to avoid strong coupling: In terms of vibration, the validated beam model from neptune_cfd coupled to the immersed boundary method was used to predict the deformation of the beam. This method appears to be

interesting due to avoiding a strong coupling. For TITAN, an ALE method was developed in order to reduce the time of computation and the size of the mesh.

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

The consideration that large gas-liquid interface tracking and resolution plays the primary role in terms of a consistent numerical prediction of the flow is also supported by the observation that one-fluid VOF simulations realized good results, in terms of both void fraction profiles and force/vibration spectra, for both bundle experiments. The VOF simulations realized used mesh refinements unable to correctly track the smallest bubbles.

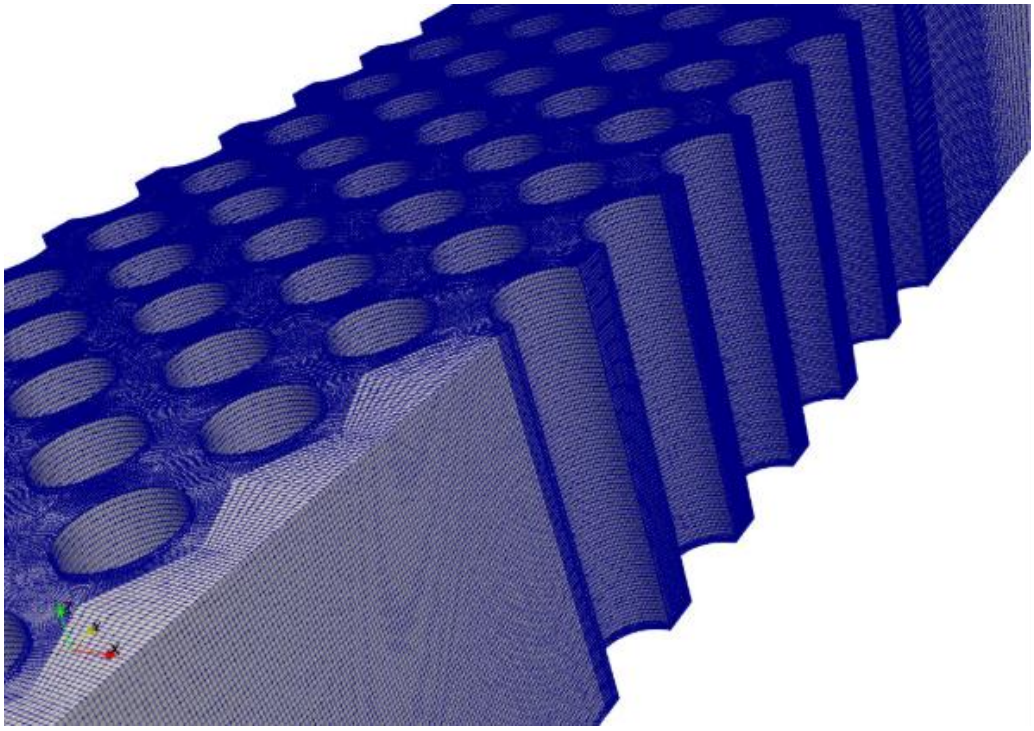

Figure 17: High-quality numerical grid for the TITAN experiment, generated by EDF.

Assure sharp bubble interfaces: For having sharp bubble interfaces, the explicit geometric reconstruction VOF method can be recommended instead of the implicit algebraic method. Also, the VOF modelling of FIV might favour scale-resolving turbulence models (e.g. LES) compared to RANS, as the latter may yield too high turbulent viscosity and thus too large bubble sizes.

Use compression methods for the gas-liquid interface: To maintain numerical setups including compression methods for the void fraction seem to help the correct prediction

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

6.3 Inlet boundary conditions

Specify inlet boundary conditions with great care: Inlet boundary conditions play a major role in configurations with a limited number of tube rows. The flow pattern crossing the tube bundle region is mainly affected by the pattern found upstream of it. In this case, the simulation of the flow upstream of the bundle becomes fundamental: this means simulating as close as possible to the experimental configuration the gas-liquid injection and mixing, and the flow evolution between the injection/mixing position to the tube bundle. Numerical results for a tube with less rows seem significantly influenced by these aspects. On the other hand, a bundle with larger number of rows seems significantly less affected by the numerical inlet boundary conditions employed.

Gas size distribution at the inlet: For high void fraction cases, there is close to no effect of the gas size distribution at the inlet, because the tube bundle drives the gas size distribution. In comparison, for lower void fraction (below 50%), the inlet size distribution is of interest since bubbles are smaller than the gap, therefore, their evolution depends on the inlet.

The necessity of resolved gas structures at the domain inlet resulted in testing of different inlet conditions. In particular, the Synthetic Bubble Model (SBM) from Ghent University and air injectors at the inlet were implemented (Figure 18). The first method was promising, but dependent on the bubble generation length and duration in the precursor. That appears to be observed in inlet mass flow rate fluctuations and therefore, influences the tube vibration. This kind of method is needed to reduce the size of the domain, but has to be further improved. Finally, air injectors were modelled at the inlet. These were able to generate resolved gas structures, however, no sensitivity has been performed to the injector diameter.

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

Ensure sufficiently long entrance region when applying the hybrid two-fluid model for flow-induced vibration studies: It is essential to include a sufficiently long entrance section

(20–100 hydraulic diameters) with high enough resolution upstream of the vibrating structure to allow flow structures to develop. Proper placement of the object downstream, local mesh refinement, and monitoring of phase fraction, velocity, and forces help ensure accurate statistical characterization.

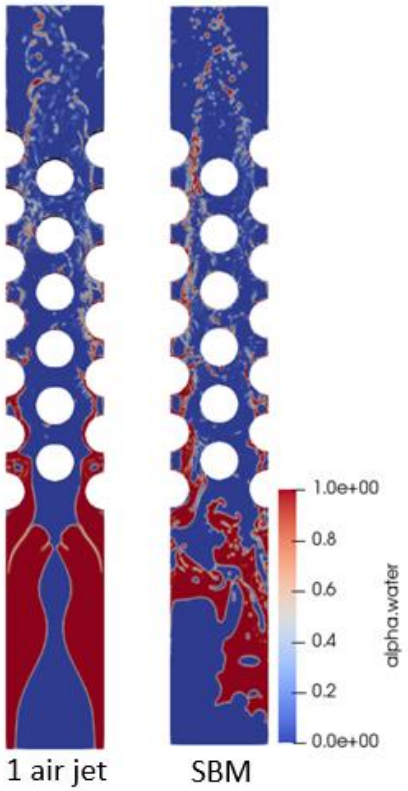

Figure 18: Difference between a single air jet and SBM as inlet for a tube bundle in multiphase cross flow (University of Ghent).

6.4 Coupling and FSI simulations

Use implicit coupling, when explicit coupling fails: Explicit FSI coupling, where the CFD load and structural displacement data is exchanged only once in time step, can become numerically unstable. Parameters promoting the instability are e.g. low structure to fluid mass ratio and high structural flexibility. The numerical instability can be usually cured by using iterative implicit FSI coupling, however, the semi-implicit or implicit methods may be computationally expensive and with some codes it might not be available. For explicit coupling, using smaller time steps typically improves stability.

Check carefully if two-way coupling is really necessary: The two-way coupling increases the overall CPU cost of the multiphase FSI run that is expensive by default. Therefore, it would make sense to first check if for the evaluation of the expected phenomena the two-way coupling will be necessary or a one-way might be sufficient. Figure 19 shows PSD results calculated with one- and two-way coupling approaches that are compared with experimental data from the TREFLE experiment. It is obvious that the vibration frequency is very well captured by both coupling strategies.

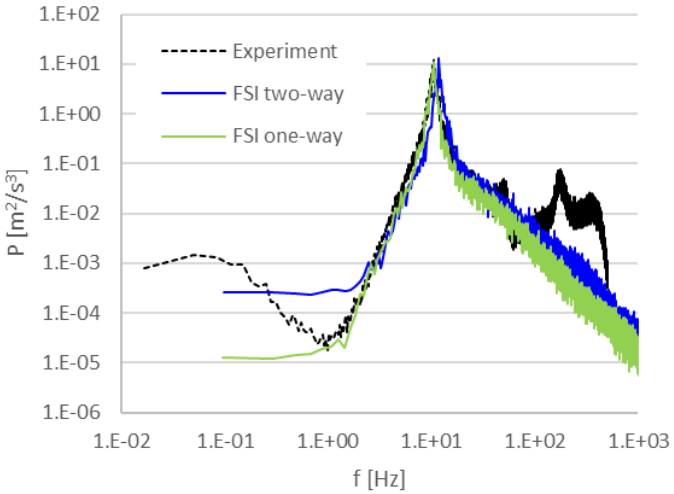

Figure 19: Measured and calculated PSDs with one- and two-way coupling by VTT for the TREFLE experiment.

Check the convergence level: FSI simulations were run in the two-phase turbulence- (or buffeting-) induced force regime (meaning not in the fluid-elastic response regime). In this regime, the fluid-structure coupling was observed to converge after very few sub-iterations within each time step: at the second sub-iteration, the evolution of the tube displacement (compared to the first sub-iteration) was practically negligible. Fast convergence is often a sign of an accurate and stable solution. Of course, this behaviour applies for a numerical time-step low enough compared to the cylinder natural period and the two-phase flow.

Use velocity limiting to stabilize VOF: Velocity limiting may stabilize VOF. One could introduce limits up-to 10-20 times the max. of mean mixture velocity. This is important, because sometimes divergence occurs locally, leading to unexpectedly high, unphysical velocities in just a few cells of the computational domain. As a result, the whole simulation crashes, although the solution in the rest of the CFD domain is still plausible and stable. Specifying limiters may help in such cases. These could be also introduced for other quantities.

Longer sampling times necessary compared to single-phase analyses: FSI calculations in two-phase flows of vibration problems require longer sampling times for converged statistics (e.g. mean and RMS) compared to the flow field with fixed geometry. This is especially true if a reliable structural fatigue estimation is needed. This depends to a large extent on the structural eigenfrequencies. The computing time requirements can become large when the time scale of the structural motions is long compared to the relevant flow time scales.

Perform consistent spectral analysis: Spectral analysis is highly sensitive to time step, total simulation time, windowing method, sampling frequency. A meaningful comparison between experimental and numerical spectra requires strict consistency: same sampling frequency, time window, windowing technique and cut-off frequency. If experimental data include a cut-off frequency, the numerical signal must be processed identically (including re-interpolation if needed). The spectra represent distributions of integrated quantities.

Agreement with data can be artificially improved through post-processing choices, and this shall not be done. Always align post-processing methodologies before drawing physical conclusions. Comparing spectra obtained with different processing is equivalent to comparing quantities expressed in different functional bases. All of this holds also for FSI analyses in single-phase flows.

Monitor relevant quantities: Monitoring only the void fraction is insufficient. The relevant conserved quantity is the phase flux (e.g. αUg). A correct void fraction with inconsistent velocities is not physically reliable. Monitoring the phase flux helps also to assess the code's ability to conserve mass, which is essential.

Sometimes, the instability with explicit coupling shows later in the simulation when monitoring only the structural displacements. However, monitoring the structural acceleration signals can reveal the instability much earlier; this can be done also by numerically differentiating the displacement signal.

Mind possible low-frequency content, when analysing results generated with the hybrid two-fluid model: Force measurements in two-phase flows using the hybrid model often contain low-frequency content, necessitating long simulation times (100+ seconds) to achieve accurate averaging. Given that such low-frequency, slow trends are not present in the experimental signals, these transients likely stem from numerical artifacts, such as overestimated bubble sizes on coarse meshes. Mesh refinement and sensitivity analyses are recommended to discern physical effects from numerical artifacts.

6.5 Structural modeling

Add realistic Rayleigh damping: Adding realistic Rayleigh damping to the structure at the frequency range of interest may also prevent high-frequency numerical instabilities. The stiffness proportional contribution of the Rayleigh damping damps effectively very high frequencies. The frequency of the numerical instability corresponds often to the time step size and is thus typically much higher than the structural eigenfrequencies of interest. The stiffness proportional damping grows linearly with frequency, and thus, a fairly low damping value may be sufficient for stabilization when small time steps are applied.

6.6 Conclusions on FIV analysis in multiphase flows

GO-VIKING WP4 tackled the difficult phenomenon of two-phase flow-induced vibration for NPP applications. A progress has been made in the simulation of the interplay between structures and fluids in multiphase environment, demonstrating that despite the complex nature of these phenomena, the numerical models today are able to predict to some extent the vibration behaviour of long slender structures and tube bundles.

The numerical simulations highlighted that it is easier to simulate high void fraction cases than dispersed ones. Intermittent flows might be challenging for both main approaches: Eulerian-Eulerian and VOF. All-regime Eulerian-Eulerian two-phase flow methods can help in these situations, but sometimes getting converged solution might not be straightforward.

The information at the inlet of the domain proved to be crucial. When this information is not available, the dependency of the numerical results becomes strongly related to the choice of inlet boundary conditions (bubble size distribution, etc.). Therefore, sufficient information about the bubble size distribution shall be provided by the experimentalists, when code validation is to be performed. For low void fraction cases, the dependency to the inlet gas size distribution is even stronger. Innovative injecting methods for the inlet can be used, such as the SBM, but further improvement is still necessary.

It was found that often the mesh sizes were generated on the information of the gas size distribution. Nevertheless, mesh sensitivity studies remain necessary.

It appears important to resolve large gas structures with LES or URANS approaches to predict accurately forces and vibration. It was found that the large gas structures are the ones that drive the structural displacements in experiments with high gas volume fractions. For such cases VOF provided good results due to its ability to resolve the large gas structures.

7. Best practices for the utilization of fast-running and uncertainty quantification methods

This chapter focuses on the experiences gained through the development, implementation and validation of fast-running methods. Within GO-VIKING WP5, three reduced-order models (ROM) were developed for the fluid domain and one for the solid domain that already existed was validated on experimental data. Further, it will also summarize the main findings from the development and application of uncertainty quantification (UQ) methods within the GO-VIKING project.

It should be mentioned that some of the lessons learned mentioned here and the provided best practices may be directly applicable to standard FSI analyses that were discussed in the previous chapters of this document.

7.1 Reduced-order model for the structural domain

The structural ROM ANSYS MOR (Model Order Reduction) was developed by ANSYS (Einzinger, 2014). Within GO-VIKING, GRS validated this approach on multiple AMOVI cases (D. Panunzio, 2023). The major findings and guidelines are summarized below:

Accelerate your FSI analyses using ROMs for the structural domain: Although the reduction in the necessary CPU time for a FSI run using structural reduced-order model is smaller compared to the one of a FSI run with ROM of the fluid domain, it was demonstrated in (van Zuijlen A. Z., 2026) and (van Zuijlen A. Z., 2024) that still significant speed-up of the FSI simulation (by factor of 6 faster than the full-order model) could be achieved, while not remarkably reducing the accuracy of the results (Figure 20).

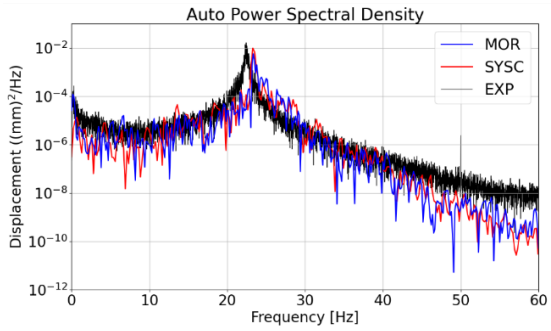

Figure 20: PSDs of tube displacements calculated with FOM (red) and ROM (blue) (GRS).

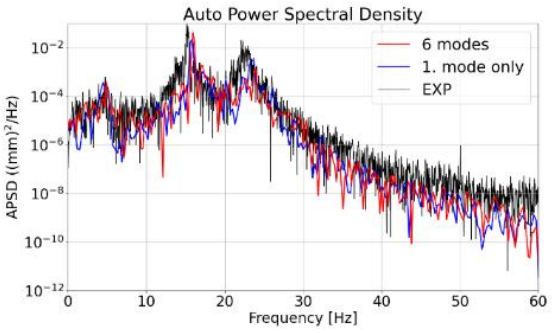

Figure 21: Comparison of two simulations of that consider 6 and just the first tube vibration mode (GRS).

Consider the number of modes in ANSYS MOR that really have influence on the results: The MOR approach is a reduced-order model for the structural domain. The user can determine the number of modes to be considered in the coupled FSI simulation. In the AMOVI simulations with a single flexible tube and tube bundle, the first six eigenmodes of the vibrating structure with their respective mode shapes and frequencies were used. A

test with the consideration of only the first bending mode was carried out as well. This led to very similar results, proving that the structure primarily vibrates in its first mode (Figure 21). The influence of the higher modes is negligible on the vibration amplitude and frequency. This means that for such simplified cases the higher vibration modes can be neglected, thus increasing the overall efficiency of the numerical analysis. Note, that in more complex applications higher modes might play an important role, therefore, additional investigations on the number of modes impacting the results shall be made.

Use hybrid models to increase the accuracy of the calculated displacements: The experience shows that both SST (URANS) and Zonal LES (ZLES, hybrid model) turbulence approaches correctly predict the vortex-shedding and the reduced eigenfrequencies of a single flexible tube in a channel. Differences emerge when comparing the tube displacements with the measured data. Although both simulations underpredict the measured tube displacements, the more sophisticated hybrid model deviates less from the data. The RMS displacement increased by 80% when using ZLES in comparison with SST on the same mesh and setup, thus being much closer to the AMOVI data (Figure 22). The vibration frequencies were predicted well by both simulations (Figure 23). Similar findings were made also for a bundle case.

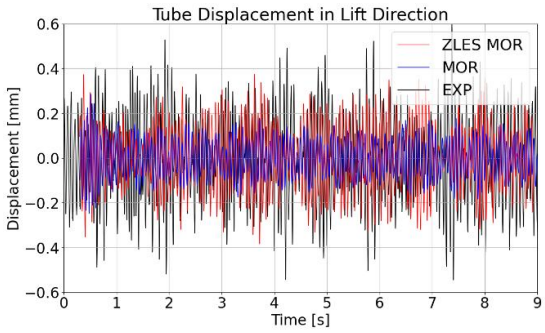

Figure 22: Tube displacements calculated with ZLES (red) and SST (blue) (GRS).

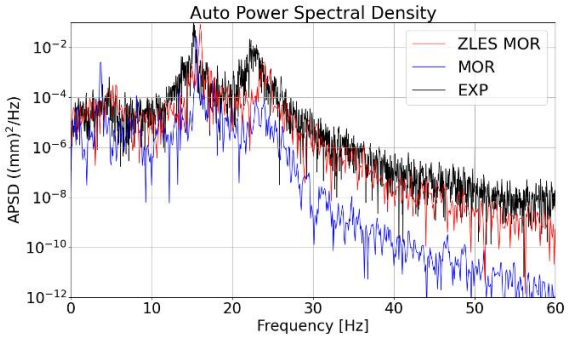

Figure 23: PSDs calculated with ZLES (red) and SST (blue) (GRS).

Position of the URANS/LES interface upstream the zone of interest: In the ZLES model, the user hast to specify the LES zone by using CEL expressions in ANSYS CFX (Figure 24). The further the interface is away from the vibrating structure, the larger the LES zone and the numerical mesh will be. Therefore, it is important to know the minimum distance upstream of the vibrating structure, where the LES zone should begin. At that location ANSYS CFX will put a harmonic generator for the turbulent structures. The investigations showed that moving the LES zone size further upstream did not change the RMS value for the tube displacement. This proved that the exact position of the turbulent generator in the domain is not crucial for the vibration amplitudes and do not artificially trigger these in this particular case. Nevertheless, the presentation of Florian Menter from ANSYS during the GO-VIKING Final Workshop (Piliuhina, 2026) revealed that special attention is necessary by the analyst when specifying the position of the URANS-LES interface.

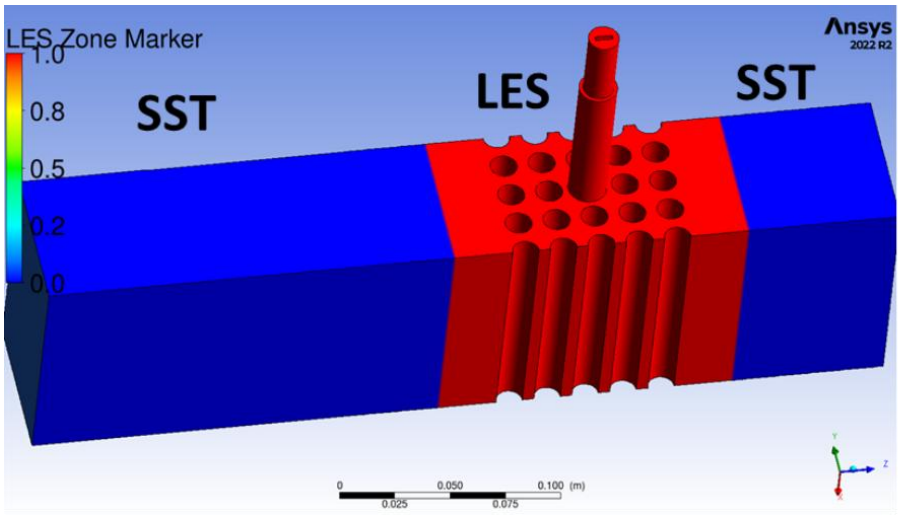

Figure 24: Distribution of the URANS (SST) and LES zones in the CFD domain that represents the AMOVI experiment (GRS).

Improve the calculated displacements using dedicated meshing strategy: The current experience shows that the most challenging parameter to predict is the tube displacement, which is often underpredicted. Overall refinement of the numerical mesh increases the computational time significantly, and therefore, should be avoided. It was found that a refinement of the mesh only around the vibrating structure and especially reducing the size of the first cell remarkably improves the displacement predictions.

Guarantee mesh-independence of the FSI results: Mesh-independence studies are usually done in CFD analyses to guarantee that the results do not depend on the mesh resolution. This is a standard procedure. It was found that a mesh-independent CFD result does not necessarily mean that the FSI result will be also mesh independent, even when using the same structural models. Therefore, for accurate predictions, it is advisable to perform the studies with the coupled FSI model. This, in turn, is very expensive in terms of CPU time. The recommendation here is to perform detailed sensitivity study on the CFD mesh and then a limited number of FSI simulations with different CFD and CSM meshes.

Preserving good mesh quality during an FSI run: Keeping good mesh quality is essential in FSI runs, since it often degrades due to the mesh deformation. In ANSYS CFX, the option "*Displacement Relative To = Initial Mesh*" significantly improved the mesh quality during the run. Further, increasing the mesh stiffness with the regions of interest is also advisable. The option *"Blended Distance and Small Volumes"* provided the highest stability with tube displacements up to 4 mm in the simulation of a fluid-elastic instability (FEI) before the simulation crashed with negative volumes found in the domain.

Code validation on multiple runs: Code validation shall be performed on multiple experimental runs with different boundary conditions. To prove the predictive capabilities of the FSI tool, one shall use the same setup, but with the respective new boundary conditions. Introducing, for example, an extra damping or further non-justified changes in the inputs of the program may lead to improved agreement with the experimental data in other cases, but will not support the evaluation of the predictive capabilities of the code, which is the ultimate goal.

7.2 Reduced-order model for the fluid domain

7.2.1 AniPFM

Based on the experience of the work on the development and application of the AniPFM model (van Zuijlen A. Z., 2024), developed at NRG PALLAS, a number of best practices were identified. These are listed below based on the code/tool/case involved. Detailed information can be found in (van Zuijlen A. Z., 2026). Figure 25 presents rod tip displacements calculated with AniPFM, LES, and compared with experimental results.

Deal II

Use the appropriate parallelization option: Working with Deal II, by virtue of the possibility of tweaking the setup code provides the user (with the right know-how) immense flexibility to setup the same case n-number of ways. This includes parallelization of the working code. Two options are possible: Threads and MPI. The current code used threads since the structural mesh (and thereby degrees of freedom) was large, but not so large that there was a need to split the problem across multiple interconnected servers. The advice is to pick the right one that suites your needs, or if you are dealing with a very coarse mesh, a single core could do the job as was the case in past validation works.

Keep computational cost low: Another Deal II advice, in a more general sense, is to make sure tensors are built and called only as many times as absolutely necessary to keep computational costs down. For example, for the linear elasticity model, the mass, stiffness and damping matrices need to only be built once. Likewise, the direct method solver relies on performing an LU decomposition of the coefficient matrix 'A' to solve Ax=b. Since this does not change in time, it makes sense to only decompose this once at the start of the calculation and only rely on vector multiplication for the remainder of the calculation owing to the expensive nature of the operation.

Synthetic turbulence (ST)

Focus on resolving the relevant scales: The advice here for those developing a fast-working method via generation of synthetic turbulence is to not lose sight of the ultimate goal – predict vibration amplitudes accurately. By extension, this means the primary interest is to obtain 'correct' pressure fluctuations on the structure from (synthetic) turbulence in the flow. In terms of capturing the turbulent structures and scales, it is expected that the vibration contribution of the tiny scales very close to the wall are insignificant owing to the extremely high frequencies involved – the structure won't be able to react. Rather the intermediate/larger scales passing by the structure are the ones more critical and the ones that a synthetic turbulence model should predict accurately.

General Advice

Separate validation of fluid and structural solvers: It is good practice to validate structural and fluid models before applying them to the final application. This is done via validation against free vibration tests in air/vacuum and still water.

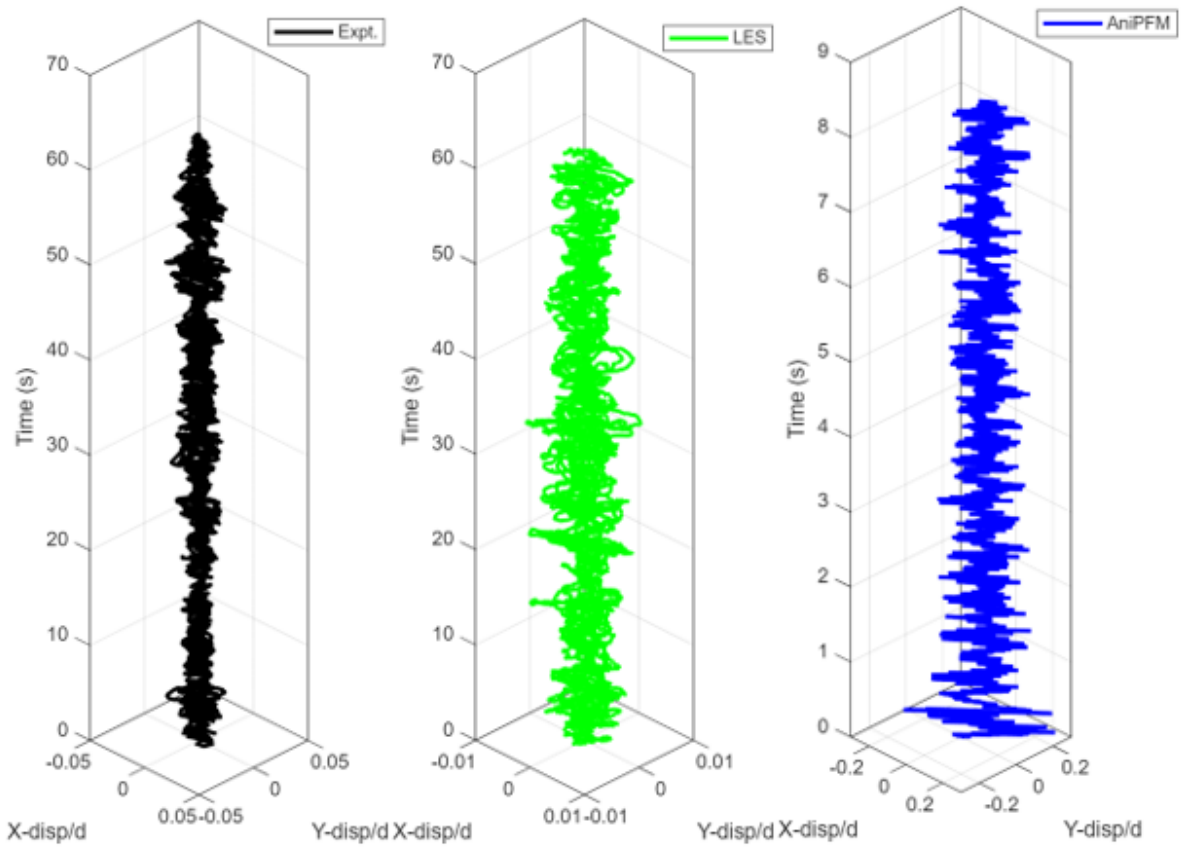

Figure 25: 3D time history of non-dimensional rod tip displacements calculated with LES and AniPFM (NRG PALLAS).

7.2.1 Synthetic Turbulence Method

The Synthetic Turbulence Method was developed and validated within GO-VIKING WP5 by IPP Centre (van Zuijlen A. Z., 2026) (van Zuijlen A. Z., 2024). The major findings and guidelines are summarized below:

The more turbulence parameters are used, the more accurate and physical the result is: Potentially fast methods for engineering practice of FSI calculations could be those that impose artificial fluctuation fields on the main mean flow field within the URANS framework. These fluctuations are calculated using so-called Synthetic Turbulence Generators (STGs), the number, mathematical and physical content of which has been significantly expanded in the last 15 years. The simplest STGs produce fluctuations in the form of random white noise, while the most complex STGs rely on information about the statistical characteristics of turbulence, which can be determined using URANS models. In

this case, the more turbulence parameters are used, the more accurate and physical the result is.

Apply a synthetic turbulence model to moving solid walls with caution to avoid self-induction of fluctuations: The overall velocity field must be decomposed into three components: the average velocity, turbulence fluctuations, and fluctuations caused by the rigid wall's motion. As theoretical estimates show, in the case of expected resonances under given operating conditions, it is important to prevent self-induction of fluctuations from one type to another. This means that the concept of using volumetric sources based on synthetic fields requires careful consideration.

A fast method for calculating coupled oscillations consists of two fundamental subtasks: calculating the mean flow (including non-stationary flow) and estimating the velocity and pressure pulsations with subsequent calculation of the total force.

Apply small time steps when using ANSYS CFX in combination with Synthetic Turbulence method: oscillations may not occur if the time step size is greater than 1 ms or may have a significantly smaller amplitude. When coupled oscillations are established, the displacements amplitudes agree quite well for the cases where the flow unevenness (and hence the magnitude of the force) is determined directly by the moving boundary, and the oscillation modulation is determined by the turbulent nature. Figure 26 visualizes the main instantaneous fields calculated using ANSYS CFX + ST with different time step sizes.

Use Synthetic Turbulence with care in bundle configurations: For the calculation of a complex flow configurations (rod bundles), the fields obtained by synthetic turbulence induce significantly greater hydrodynamic forces than the average URANS fields, which determines the sensitivity to fluid - elastic instability.

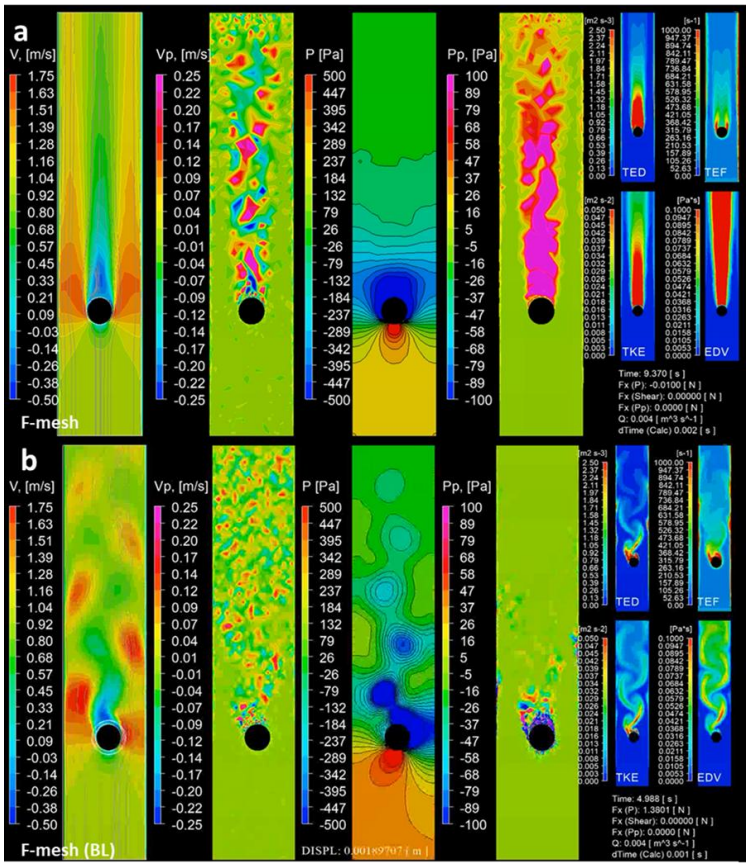

Figure 26: Visualization of the main instantaneous fields calculated using ANSYS CFX + ST for the single flexible tube case with Q = 4.0 l/s (a – integration time step 2.5 ms, K10; b – integration time step 1.0 ms, K31)

Increase efficiency using the Stable Fluid Algorithm (SFA): The Stable Fluid Algorithm procedure was developed specifically for the fastest possible calculation of coupled vibrations. This method is quite diffusion-based, so the use of a turbulence model is optional and necessary specifically for STG, which is implemented in 2D formulation. From the perspective of the general theory of vortex formation in separated flows, this approximation is limiting, but is acceptable only for two-dimensional problems. In the case of the computational procedure, the basic option for calculating pressure pulsations is solving the Poisson equation.

Synthetic Turbulence (ST) approach generates higher forces on the structures: For the calculation of complex flow configurations (rod bundles), the fields obtained by synthetic turbulence induce significantly greater hydrodynamic forces than the average URANS fields, which determines the sensitivity to FEI.

Freeze the flow field: The results obtained, especially with the use of SFA+ST, show that for some calculation cases it is possible to completely "freeze" the URANS field and determine the force solely using synthetic pressure pulsations. Such results will reproduce both displacement amplitudes and frequency characteristics with sufficient accuracy for engineering practice.

It is advisable to solve the Poisson equation for both the average flow (SFA) and the Synthetic Turbulence using the FFT transform, similar to precise LES/DNS: The method for calculating the force, especially from a synthetic field, is fundamentally sensitive to the computational mesh at the moving wall, similar to LES models (for this reason, calculations are typically performed with predefined stresses). This means that the proportionality coefficients for pressure pulsations or the hydrodynamic force multipliers themselves are functions of the characteristic mesh dimensions. This has the greatest impact on low-velocity regimes, where the forces are determined by the turbulent nature rather than by the unevenness of the mean flow. This also applies to transient regimes.

Calibrate a fast calculation procedure on available experimental data and then validate on some other experimental runs from the same campaign. Thereafter, the procedure can be applied with greater reliability to other practical problems.

ANSYS CFX in combination with ST requires approximately 5 times more time steps to calculate the same physical time as SFA+ST: In integral terms, this means that for a single flexible tube (Q = 4.0 l/ s, see Figure 26 and Figure 27) , the 10 seconds calculation in SFA+ST procedure takes 6800 seconds, while ANSYS CFX+ST for this case requires ~50 000 seconds on 4 cores (taking into account the required time step and the possibility of exciting oscillations on a given mesh).

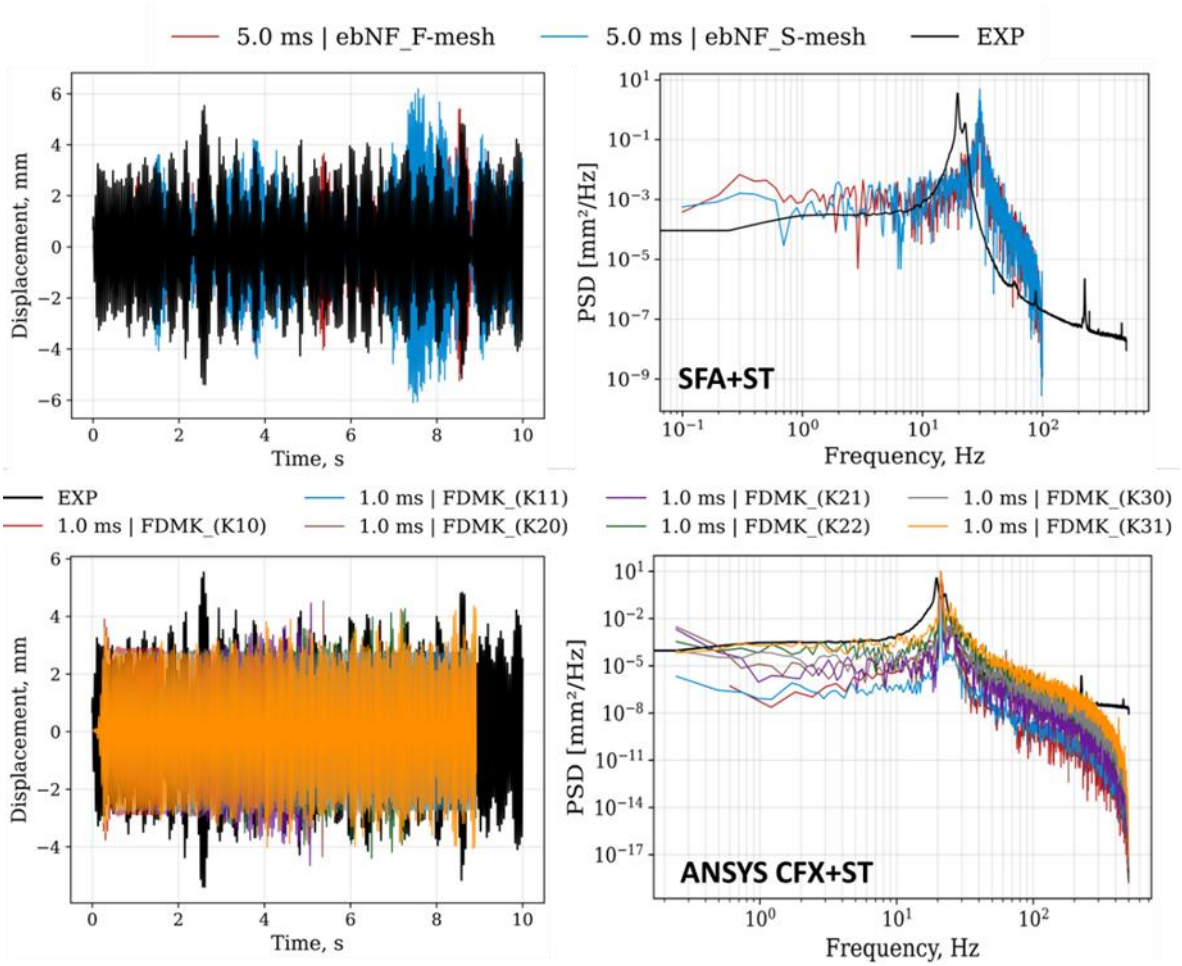

Figure 27: Results of displacements calculation for the single flexible tube with Q = 4.0 l/s

7.3 Best practices on the implementation and utilization of uncertainty quantification methods

The work on uncertainty quantification (UQ) methods was performed within WP5 of the GO-VIKING project. Detailed information can be found in (Dolfen, D5.3 "Development of UQ approaches for FIV analyses", 2024) (Dolfen, D5.4 "Application and validation of UQ approaches", 2025).

7.3.1 Polynomial Chaos Expansion

The work on the implementation of Polynomial Chaos Expansion (PCE) method was performed by University of Ghent. The major findings and guidelines are summarized below:

Be aware not only of the efficiency of the UQ methods, but also of their challenges: For UQ of FSI simulations, one needs efficient methods. Monte Carlo approaches usually require a high number of computational runs, which makes them not practical for UQ of FIV simulations. Therefore, techniques that require less runs are of interest. Methods such as Polynomial Chaos Expansion (PCE) are best suited for outputs that vary smoothly as a function of the inputs. These techniques are based on sampling of the input-output relation, so effects like the incomplete convergence as a function of the simulated physical time can cause wiggles in the input-output relationship, which is in turn making the PCE more challenging. Hence, ensuring there are no wiggles in the output due to numerical parameters is important for the results themselves as mentioned in above, and for their fitting using PCE. Also be aware that the phenomena and simulation case to be treated with PCE need to be conducive for PCE, e.g. because the system response to disturbances is known to be smooth.

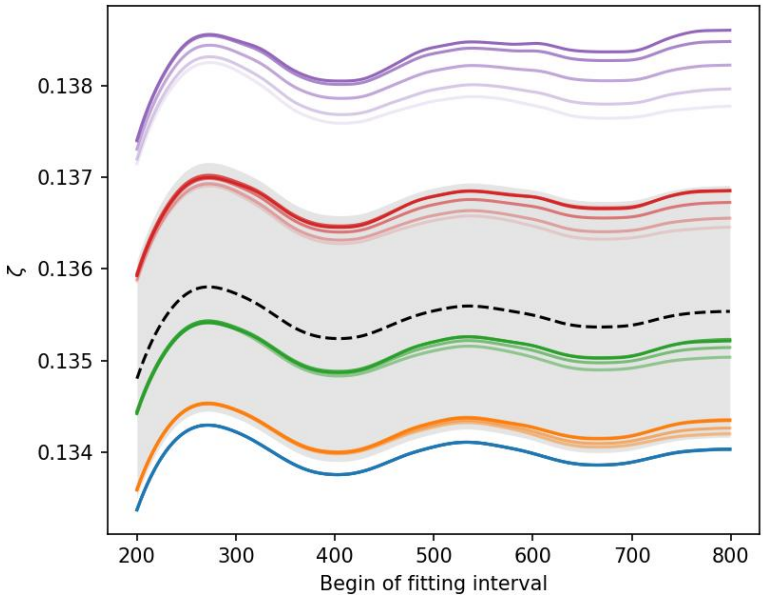

Figure 28: Effect of the begin of the fitting window on the damping for multiple free vibration signals after an initial excitation, each indicated with a different colour (Universtiy of Ghent).

Run simulations sufficiently long: Simulations should be executed for a sufficient number of vibration periods, even as this obviously increases the computational cost. The effect of the time window over which the results are analysed should be checked by systematically varying the length of this period. For example, when fitting a damped oscillation formula to free vibration data after an initial excitation, the damping is influenced by the begin of the fitting interval as shown in Figure 28. Similarly, the effect of the end of the time interval on the RMS value of a vibration is shown in Figure 29.

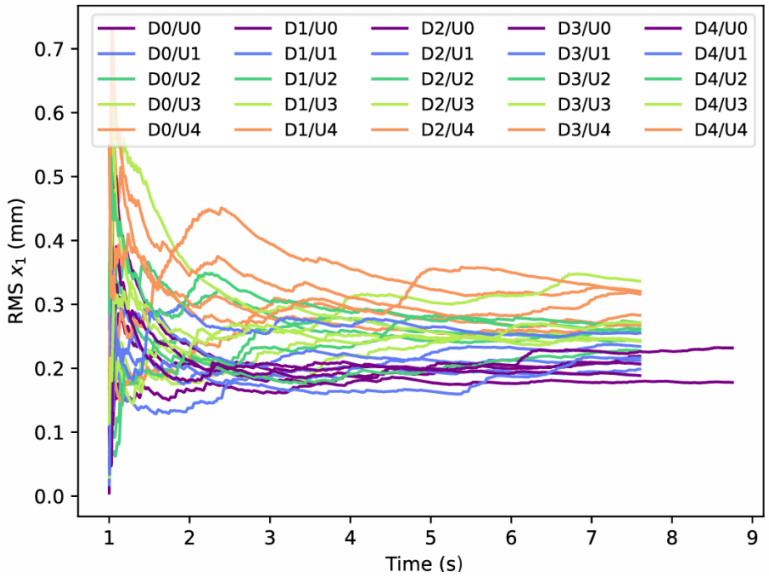

Figure 29: Convergence of the RMS vibration amplitude for a cylinder as a function of the final time step included in its calculation. The legend indicates different diameters and velocities (University of Ghent).

First validate solvers separately: When simulating flow-induced vibration with a partitioned strategy, i.e. coupling a flow solver to a structural solver, it is advised to validate both solvers separately first. This should be done in conditions that mimic the expected conditions during the coupled simulation. For example, if the flow solver uses a deforming grid using the ALE formulation, then a simulation with imposed deformation at the fluid-structure interface is recommended, using an analytical function for the motion in the expected mode and with the maximal expected amplitude. Similarly, the structure should be loaded with a spatially and temporally varying load, as close as available to what is expected in the fluid-structure interaction simulation. After that, validate the coupled solver against suitable test data.

Minimize the number of iterations in the FSI coupling algorithm: As flow-induced vibration simulations are typically time-consuming, it is important to minimize the number of coupling iterations if a partitioned simulation with coupling iterations is used. Based on benchmarks with different coupling algorithms, quasi-Newton algorithms like IQN-ILS and IQN-ILSM are typically faster than (Aitken) relaxation. Reuse of data from coupling iterations from previous time steps can significantly reduce the average number of coupling iterations per time step, and the performance of the algorithms is quite insensitive to the number of reused steps. Note that in IQN-ILS this is only the case if filtering of linear dependencies is present while in IQN-ILSM this is always the case. Both mentioned algorithms also have a computational cost and memory requirement that scales linearly with the number of degrees of freedom on the interface, which is important for large cases.

Pay attention to the supports in the structural domain: The support of the vibrating structure as present in the experiments needs to be represented accurately in the simulations. In vibrating tube bundle experiments with cross flow, the tubes can be supported on one side using a bending beam. This means the tubes will move up and down, but also rotate, so the motion at the end of the cylinder closest to the support will move less than the free end. In three-dimensional simulations, this can be taken into account, while in two-dimensional simulations a choice has to be made for the position along the cylinder for the determination of the supporting stiffness and for the comparison with experimental data.

Inlet conditions in multiphase flows: In FIV simulations with multiphase flow, the multi-phase inlet conditions are important for realistic force and vibration results. If the liquid and the gas are entering the domain at fixed parts of the inlet, then a sufficient distance between the inlet of the domain and the region of interest, typically the tube bundle, should be present. This long domain will increase the computational cost of the simulation, but is necessary to let the inlet develop into a realistic two-phase flow. Alternatively, a SBM can be used to impose gas bubbes coming into the domain in a liquid stream, according to an imposed distribution. This reduces the required length of the domain upstream of the region of interest, and hence the computational cost. However, current implementations of the SBM are characterized by a high memory requirement and the presence of a spurious frequency related to the inflow velocity and the length of the pre-cursor domain used by the SBM.

7.3.1 Uncertainty Quantification of Turbulence Models

Within GO-VIKING WP5, TU-Delft focus on methods that can be used to quantify the uncertainties in turbulence models. These were identified to have one of the major impacts on the FSI results at all. Therefore, it was of particular interest to evaluate these uncertainties. To enable efficient calibration, surrogate models were constructed to emulate the CFD solver. These models reproduce the relationship between turbulence-model parameters and flow quantities at a fraction of the computational cost. Both Gaussian-process (Kriging) models and Polynomial Chaos Expansion were used, and they produced consistent results. Bayesian inference was then applied to estimate parameter values that best match high-fidelity DNS or LES data. This approach not only provides calibrated parameter values but also quantifies uncertainty and reveals how strongly each parameter is constrained by the data. Below are some key findings:

Turbulence model uncertainty is effectively low-dimensional: Across different geometries and flow conditions, the same small subset of parameters controls most of the variability. This allows a significant reduction in complexity without loss of predictive capability. Calibration improves predictions, but only to a certain extent. For some quantities—such as streamwise velocity or spanwise Reynolds stress—the agreement with high-fidelity data becomes very good. However, for others—particularly turbulent kinetic energy and wall-normal stresses—noticeable discrepancies remain even after calibration. These differences are not due to parameter uncertainty, but to structural limitations of the turbulence model itself.

Parameter tuning alone cannot fully correct a model: Parameter tuning alone can only provide the best possible prediction within the limits of the chosen modelling approach. Another important observation is that results are robust with respect to the choice of surrogate model. Both Kriging and PCE lead to nearly identical calibrated parameters and predictive uncertainty, indicating that the conclusions are driven by the physics and data rather than the surrogate methodology.

Main lesson learned: UQ should not be treated as an additional step after CFD simulations, but as an integral part of the modelling process. First, performing a sensitivity analysis at an early stage is essential. It reveals which parameters actually matter and prevents unnecessary computational effort. Second, surrogate models are indispensable for enabling Bayesian calibration, especially in high-cost simulations. Third, predictive uncertainty should always be evaluated in terms of physical quantities of interest, not only in parameter space. Most importantly, calibration must be interpreted carefully. When predictive uncertainty bands fail to capture high-fidelity data, this is a clear indication of model-form error rather than insufficient parameter tuning. Recognizing this distinction helps avoid overconfidence in calibrated models.

Recommendation for engineers and researchers dealing with turbulence modelling: first analyse simplified flow configurations to identify dominant parameters, then apply a reduced-parameter calibration to more complex geometries using surrogate models. This approach provides a good balance between computational cost and predictive reliability. The overall UQ workflow adopted in the TU-Delft study is summarised in Figure 30.

Figure 30: UQ framework for turbulence model calibration in CFD (TUD).

7.4 Conclusions on fast-running and uncertainty quantification methods

The work performed within GO-VIKING WP5 demonstrated that ROMs are capable of increasing the computational speed of FSI analyses. The generation of a ROM for the solid domain is generally simpler, but the gain in terms of CPU time savings is smaller. Nevertheless, compared to the full-order model, the ROM needed a factor of 6 less CPU time, while the generated results of both simulations were very similar. It was also shown that the number of considered modes in the AMOVI case was not important, since the tube vibrated primarily in its first bending mode. It can be concluded that ANSYS CFX - MOR can be used for the analysis of tube vibrations in a cross-flow. It would be also interesting to test this method for more complex configurations, for example, for bundles with larger number of vibrating tubes.

To generate turbulent pulsations, the synthetic turbulence generation procedure was used. The proposed procedures are simplified, since these are based on local parameters that take into account only the averaged flow history. In the IPP development, the turbulent characteristics (correlations of velocity pulsations) were considered as isotropic in contrast to the NRG PALLAS' AniPFM that accounted also for the anisotropy in the turbulent characteristics. Despite significant simplifications compared to procedures that consider the transfer of pulsation characteristics, IPP tests with the Synthetic Turbulence Method showed a good degree of agreement between the qualitative picture of the synthetic turbulence generator and the application of the scale-resolving method, which is satisfactory for engineering practice. The pulsation component of pressure was calculated based on a simple algebraic relationship, where it is assumed that pressure pulsation is a dynamic consequence of velocity pulsation. In order to organize more efficient

calculations, a one-parameter turbulence model of effective kinematic viscosity transfer was proposed, which is obtained by combining two k-ε RNG equations. To achieve this, it became necessary to express the kinetic energy of turbulence and its dissipation through eddy viscosity.

Concerning AniPFM, the pure CFD results revealed good predictions of the 1st order statistics, i.e the mean axial and radial components of velocity at several elevations in the vicinity of the rod tip. This was directly an output of the base URANS k-ω SST. For validation of AniPFM, 2nd order statistics, namely the mean u'v' Reynolds stress tensor component and the mean turbulent kinetic energy were compared. The experimental results coming from PIV were considered mainly to realize trends while the high-fidelity data was taken as the standard for the magnitudes. The comparison revealed that the AniPFM predicted significantly different trends when compared on a location basis, but it could also be interpreted as a different evolution of the quantities in a holistic sense moving downstream where ultimately the match becomes decent as the flow becomes purely annular. The comparison revealed the current gap that must be improved with future work. The main limitation was reasoned to come not from the AniPFM methodology, but the exclusion of introducing anisotropy either via the Wilcox correction or using a 2-equation model that uses the Boussinesq approximation for the Reynolds stress tensor which the generated synthetic turbulence replicates.

The 1-way FSI study revealed vibration predictions that were, unexpectedly, directional instead of chaotic, and at an offset instead of centered. Furthermore, the vibration amplitudes were grossly overpredicted by ~8 times compared to the experiment. The frequency of vibrations, however, were correctly predicted to be the same for front and side displacements and only slightly higher than the experiment (within 2%). It is concluded that the vibration amplitudes are more affected by the p' generated by AniPFM than the vibration frequencies, which appear to be more influenced by the chosen structural model.

Concerning the application of UQ methods for FSI runs, it was shown, that efficient approaches are necessary due to the high cost of a single FSI run. Such UQ approaches shall be based on analyses with limited number of FSI runs. It turned out that PCE is a good candidate for the evaluation of UQ in FSI analyses. Further, it was found that UQ should not be treated as an additional step after CFD simulations, but as an integral part of the modelling process. Sensitivity analysis at an early stage can reveal which parameters actually matter and prevents unnecessary computational effort and the utilization of surrogate models are indispensable in high-cost simulations. It was suggested that simplified flow configurations shall be analysed first in order to identify dominant parameters. In a next step, a reduced-parameter calibration to more complex geometries using surrogate models shall be performed.

8. Overall conclusions

This chapter briefly summarizes the conclusions on the best practices for the use of FSI analyses for FIV evaluation.

Analysis of FIV in FA: It was found that the turbulence modelling approach used in the CFD simulation plays a significant role for the predictability of the vibration response. Based on the very small vibration amplitudes measured in the FIV tests, turbulence is assumed to be the main driving vibration mechanism. According to this assumption, it is essential to use turbulence modelling approaches and turbulence models that are capable of resolving relevant turbulent flow structures that are responsible for excitation of the FA structure. LES was found to be the best performing in combination with wall-modelling, which is the only feasible option for bundle configurations. The modelling approaches, based on RANS turbulence models, show poor performance in prediction of turbulent flow fluctuations and therewith also in prediction of FIV for given cases. Therefore, this class of turbulence models is not suitable for application to TIV.

The computational effort turned out as to be a significant limitation in the analysis of the ALAIN benchmark. Therefore, a reduction of the computational domain may be a useful simplification. For given configuration a single span model represents such a useful geometrical reduction. Given the fact that the vibration amplitudes measured in the FIV tests were very small, just a few micrometres RMS, a decoupled simulation approach seems to be sufficient for the simulation of FIV in this case. However, it needs to be noted in the context of decoupled simulation approach that the effects of fluid in terms of added (hydrodynamic) mass, fluid damping and fluid stiffening play also an essential role for reliable prediction of the fuel assembly vibration response.

With respect to mechanical modelling of the FA structure, the use of beams as the basic structural elements in combination with additional elements (e.g. springs) for the simulation of a fuel assembly was proven as to be a good choice. However, this approach may affect the strategy for coupled FSI simulations. Most of the standard tools for FSI coupling are based on the common sharing boundary surface between fluid and structural domain. This boundary does not exist explicitly when the fuel assembly is modelled using beam elements. Hence, an alternative strategy is needed to establish the interface between fluid and structural domain in a coupled FSI simulation according to discretization of the structural dynamic model. This, in turn, requires in general special non-standard solutions for FSI coupling.

Analysis of bundles in steam generators: the combined analysis of the GOKSTAD and AMOVI benchmarks demonstrated that reliable FIV prediction requires a consistent treatment of both the fluid and structural domains, as well as their interaction. The studies confirm that relatively simple modelling approaches—such as rigid-body mass–spring–damper systems combined with appropriately chosen turbulence models—can provide satisfactory predictions for cases with limited structural deformation and weak coupling. This makes such approaches well-suited for engineering applications, where computational efficiency is often a key requirement.

At the same time, the results clearly show that increasing physical complexity—such as strong turbulence, flow asymmetry, structural deformation, or proximity to resonance—requires higher model fidelity. In the fluid domain, this is reflected in the need for more advanced turbulence modelling and realistic inlet conditions, while in the structural domain, more detailed representations may be required to capture the full dynamic response. Similarly, the selection of coupling strategy must reflect the strength of FSI, with two-way coupling becoming essential when structural motion significantly influences the flow.

A key overarching conclusion is that many of the largest discrepancies between simulations do not arise from the choice of solver itself, but rather from modelling assumptions and setup choices. In particular, inlet boundary conditions, mesh resolution, and temporal convergence were identified as dominant sources of uncertainty. Simplified models can be used effectively for screening studies and initial design assessments, while higher-fidelity approaches should be applied when detailed understanding of flow dynamics, force fluctuations, or resonance behaviour is required.

Analysis of bundles in a multiphase flow: Progress has been made in the simulation of the interplay between structures and fluids in multiphase environment, demonstrating that despite the complex nature of these phenomena, the numerical models today are able to predict the vibration behaviour of long slender structures and tube bundles. The numerical simulations highlighted that it is easier to simulate high void fraction cases than dispersed ones. Intermittent flows might be challenging for both main two-phase flow simulation approaches: Eulerian-Eulerian and VOF. All-regime Eulerian-Eulerian two-phase flow methods can help in these situations, but sometimes getting converged solution might not be straight forward.

The information at the inlet of the domain proved to be crucial. When this information is not available, the dependency of the numerical results becomes strongly related to the choice of inlet boundary conditions (bubble size distribution, etc.). Therefore, sufficient information about the bubble size distribution shall be provided by the experimentalists, when code validation is to be performed. For low void fraction cases, the dependency to the inlet gas size distribution is even stronger. Innovative injecting methods for the inlet can be used, such as the SBM, but further improvement is still necessary.

It appears important to resolve large gas structures with LES or URANS approaches to predict accurately forces and vibration. It was found that the large gas structures are the ones that drive the structural displacements in experiments with high gas volume fractions. For such cases VOF provided good results due to its ability to resolve the large gas structures.

Fast-running and UQ methods: The work performed within GO-VIKING WP5 demonstrated that ROMs are capable of increasing the computational speed of FSI analyses. The generation of a ROM for the solid domain is generally simpler, but the gain in terms of CPU time savings is smaller. Nevertheless, compared to the full-order model, the coupled simulation with the structural ROM needed a factor of 6 less CPU time, while the generated results of both simulations were very similar. It can be concluded that structural ROMs can be successfully used for the analysis of tube vibrations in a cross-flow.

The generation of turbulent pulsations is essential for fluid ROMs, based on synthetic turbulence generation procedures. In the IPP development, the turbulent characteristics (correlations of velocity pulsations) were considered as isotropic in contrast to the NRG PALLAS' AniPFM that accounted also for the anisotropy in the turbulent characteristics. Despite significant simplifications compared to procedures that consider the transfer of pulsation characteristics, the IPP tests with the Synthetic Turbulence Method showed a good degree of agreement between the qualitative picture of the synthetic turbulence generator and the application of the scale-resolving method, which is satisfactory for engineering practice. The pulsation component of pressure was calculated based on a simple algebraic relationship, where it is assumed that pressure pulsation is a dynamic consequence of velocity pulsation.

Concerning AniPFM, the pure CFD results revealed good predictions of the 1st order statistics, i.e the mean axial and radial components of velocity at several elevations in the vicinity of the rod tip. For validation of AniPFM, 2nd order statistics, namely the mean u'v' Reynolds stress tensor component and the mean turbulent kinetic energy were compared. The comparison revealed that the AniPFM predicted significantly different trends when compared to experiment for local quantities. The comparison revealed the current gap that must be improved with future work. The main limitation was reasoned to come not from the AniPFM methodology, but the exclusion of introducing anisotropy either via the Wilcox correction or using a 2-equation model that uses the Boussinesq approximation for the Reynolds stress tensor which the generated synthetic turbulence replicates.

Concerning the application of UQ methods for FSI runs, it was shown, that efficient approaches are necessary due to the high cost of a single FSI run. Such UQ approaches shall be based on analyses with limited number of FSI runs. It turned out that PCE is a good candidate for the evaluation of UQ in FSI analyses. Further, it was found that UQ should not be treated as an additional step after CFD simulations, but as an integral part of the modelling process. Sensitivity analysis at an early stage can reveal which parameters actually matter and prevents unnecessary computational effort and the utilization of surrogate models are indispensable in high-cost simulations. It was suggested that simplified flow configurations shall be analysed first in order to identify dominant parameters. In a next step, a reduced-parameter calibration to more complex geometries using surrogate models shall be performed. In addition, the validation of surrogate models in line with good practice in IAEA SSG-2 (Rev. 1) (IAEA, 2019) needs to be ensured. Finally, the range of applicability of specific surrogate models needs to be taken into account when using them in analyses.

The paragraphs above described the main lessons learned within the project. In the different chapters, best practices were derived from these lessons learned. It is important to mention that all these were derived on numerical activities of 7 different experiments, performed with different codes and numerical methods. These BPGs can be applied when analysing similar FIV cases relevant to reactor cores and steam generators. Nevertheless, these might be applicable and interesting for the analysis of FIV occurring also in other industries (oil&gas, automotive, energy, etc.). All project results are available on https://go-viking.eu/results.

9. Bibliography

Bistany, S. L. (2024). Direct numerical simulation of a cantilevered rod experiment. DOI: 10.1016/j.ijheatfluidflow.2025.110112.

Blaesius, C. H. (2026, July). Simulation of flow induced vibrations in the ALAIN fuel assembly mock-up experiment. Proc. of the FIV2026 conference, Paris, France.

D. Panunzio, L. P. (2023). Single-Phase AMOVI Experiments and Benchmark Setup, GO-VIKING Deliverable D3.1. https://go-viking.eu/images/resources.

Degroote, J. T. (2020). Computational Methods for FSI/FIV. Fluid-Structure Interaction for Industrial Applications, Von Karman Institute Lecture Series.

Dolfen, H. D. (2024, June). D5.3 "Development of UQ approaches for FIV analyses". https://go-viking.eu/results.

Dolfen, H. D. (2025, December). D5.4 "Application and validation of UQ approaches". https://go-viking.eu/results.

Einzinger, J. F. (2014). Bi-directional fluid-structure interaction with model order reduction. ANSYS Germany GmbH.

IAEA. (2019). Deterministic Safety Analysis for Nuclear Power Plants, Specific Safety Guide. IAEA Safety Standards Series, SSG-2 (Rev. 1), ISBN 978-92-0-102119-9.

Michel, A. B. (2025). A comparative analysis of the flow in a rod bundle with two guide-tubes and mixing grids, URANS simulation and Wall-Modeled LES of the ALAIN experiment. Nucl. Eng. and Des., Vol. 441, open-access., DOI: 10.1016/j.nucengdes.2025.114115.

OECD/NEA. (2025). Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Applications- Update 2024, Nuclear Safety, NEA/CSNI/R(2022)10 . https://www.oecd-nea.org/jcms/pl_102607/best-practice-guidelines-for-the-use-of-cfd-in-nuclear-reactor-safety-applications-2024-update?details=true.

Piliuhina, K. Z. (2026, May). D6.5 "GO-VIKING Final Workshop Proceedings". https://go-viking.eu/results.

Piomelli, U. B. (2022). Wall-Layer Models for Large-Eddy Simulations. Annular Review of Fluid Mechanics, Vol. 34:349-374.

van Zuijlen, A. Z. (2024). D5.1 Development of fast-running FSI methods. https://go-viking.eu/images/resources/GO_VIKING_D5_1_development_and_implementation_of_fastrunning_fsi_methods_v1.pdf.

van Zuijlen, A. Z. (2026). D5.2 Validation of fast-running FSI methods. https://go-viking.eu/results.