



GO-VIKING

Research and Innovation Action (RIA)

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

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



Best practices for the analysis of FIV in single-phase cross-flow

Authors : Kevin ZWIJSEN (NRG), Kevin Zwijsen(NRG PALLAS), Trevor Franklin (VCU), Lane Carasik (VCU), Sofiane Benhamadouche (EDF), Yuliia Filonova (IPP), Yaroslav Dubyk (IPP), Hemish Mistry (GRS), Angel Papukchiev (GRS)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	Best practices for the analysis of FIV in single-phase cross-flow
Author(s)	Kevin ZWIJSEN, Kevin Zwijsen(NRG PALLAS), Trevor Franklin (VCU), Lane Carasik (VCU), Sofiane Benhamadouche (EDF), Yuliia Filonova (IPP), Yaroslav Dubyk (IPP), Hemish Mistry (GRS), Angel Papukchiev (GRS)
Number of pages	24
Document type	Deliverable
Work Package	WP3
Document number	D3.5
Issued by	NRG
Date of completion	2026-05-18 17:53:53
Dissemination level	Public

Summary

This report presents a consolidated set of Best Practice Guidelines (BPGs) for the numerical simulation of flow-induced vibrations (FIV) in tube bundles subjected to cross-flow, based on the findings of the GO-VIKING project deliverables D3.2 (AMOVl) and D3.4 (GOKSTAD). By combining results from multiple partners and simulation approaches, including different turbulence models, structural representations, and coupling strategies, the report provides a consistent framework for selecting appropriate modeling choices. Key aspects addressed include CAD modeling, mesh generation, flow field modeling, structural representation, and fluid?structure interaction (FSI) coupling. Across all investigated cases, it is shown that the accuracy of FIV predictions depends strongly on the consistent treatment of turbulence, structural dynamics, and coupling methodology. While simplified approaches can provide reliable engineering-level predictions in many cases, the results highlight that the required model fidelity increases with the complexity of the flow and structural response. The guidelines therefore emphasize a balanced approach, where modeling choices are tailored to the physical problem and desired level of accuracy, keeping computational costs in mind.

Approval

Date	By
2026-05-18 17:55:59	Kevin ZWIJSEN (NRG)
2026-05-18 17:56:59	Papukchiev ANGEL (GRS)



D3.5 Best practices for the analysis of FIV in single-phase cross-flow

Version N°2
May / 2026

Kevin Zwijssen¹, Trevor Franklin², Lane Carasik², Sofiane Benhamadouche³, Yuliia Filonova⁴, Yaroslav Dubyk⁴, Hemish Mistry⁵, Angel Papukchiev⁵

¹NRG PALLAS, Westerduinweg 3, 1755 LE Petten, The Netherlands

²Virginia Commonwealth University (VCU), Richmond, Virginia, United States

³Electricité De France (EDF) R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatrou, France

⁴IPP Centre (IPP), Kiev, Ukraine

⁵Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Garching, Germany



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.

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	WP3
Related Task(s)	Task 3.3 Best practices and input for reduced-order models
Lead Organisation	NRG PALLAS
Contributing Partner(s)	NRG PALLAS, VCU, EDF, IPP, GRS
Due Date	30-11-2025
Submission Date	18-05-2026
Dissemination level	PU

History

Date	Version	Submitted by	Reviewed by	Comments
15-05-2026	1.0	K. Zwijsen	A. Papukchiev	
18-05-2026	2.0	K. Zwijsen		



Table of contents

1. Introduction	9
2. CAD and mesh	10
2.1 Geometry representation	10
2.2 Mesh topology and refinement	11
2.3 Near-wall resolution and turbulence scale considerations	12
3. Flow field modelling.....	14
3.1 Turbulence modeling	14
3.2 Inlet boundary conditions	15
3.3 Temporal resolution and convergence	16
4. Structural modeling	18
4.1 Structural representation and degrees-of-freedom	18
4.2 Structural properties and dynamic response.....	20
5. Coupling and FSI simulations	21
6. Conclusions	23
7. Bibliography	24

List of figures

Figure 1: 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.....	11
Figure 2: 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).....	12
Figure 3: 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).....	13
Figure 4: Resolved structures by GRS for AMOVI for single rigid tube in cross flow. Right: k- ω SST, left: SAS	14
Figure 5: The computational domain created by EDF for the single cylinder configuration of AMOVI, clearly showing the precursor used to generate proper inlet conditions.	16
Figure 6: Time evolution of the mean drag coefficient (left) and the standard deviation of the lift coefficient (right) as obtained by ED for the single rigid cylinder configuration.	17

Figure 7: 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..... 18

Figure 8: Tube displacements, calculated with linear and angular momentum algorithms (GRS).
..... 19

Figure 9: Lift displacement time series (left) and PSD (right) as obtained by NRG PALLAS for GOKSTAD..... 20

Figure 10: Test section with marked channel wall (left), Boundary condition for wall parallel to moving side of flexible tube..... 22

Figure 11: Volume rendered streamwise velocity contour plot of a reduced computational domain of the GOKSTAD tube bundle. 22

List of tables

No table of figures entries found.

Abbreviations and Acronyms

Acronym	description
ALE	Arbitrary Lagrangian–Eulerian
CAD	computer-aided design
CFD	Computational fluid dynamics
CFL	Courant–Friedrichs–Lewy number
DNS	Direct Numerical Simulations
DOF	Degree-Of-Freedom
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
PSD	Power Spectral Density
PWR	Pressurized Water Reactor
(U)RANS	(Unsteady) Reynolds-Averaged Navier Stokes
RMS	Root-Mean Square

RSM	Reynolds Stress Model (RSM)
TKE	Turbulent Kinetic Energy
VIV	Vortex-Induced Vibrations
WP	Work Package

Summary

This report presents a consolidated set of Best Practice Guidelines (BPGs) for the numerical simulation of flow-induced vibrations (FIV) in tube bundles subjected to cross-flow, based on the findings of the GO-VIKING project deliverables D3.2 (AMOVl) and D3.4 (GOKSTAD). By combining results from multiple partners and simulation approaches, including different turbulence models, structural representations, and coupling strategies, the report provides a consistent framework for selecting appropriate modeling choices. Key aspects addressed include CAD modeling, mesh generation, flow field modeling, structural representation, and fluid–structure interaction (FSI) coupling.

Across all investigated cases, it is shown that the accuracy of FIV predictions depends strongly on the consistent treatment of turbulence, structural dynamics, and coupling methodology. While simplified approaches can provide reliable engineering-level predictions in many cases, the results highlight that the required model fidelity increases with the complexity of the flow and structural response. The guidelines therefore emphasize a balanced approach, where modeling choices are tailored to the physical problem and desired level of accuracy, keeping computational costs in mind.

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 tube bundles subjected to cross-flow remains a critical and challenging problem in nuclear engineering, heat exchanger design, and fluid–structure interaction (FSI) modeling. These systems exhibit highly complex flow behavior 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 modeling assumptions, numerical methods, and coupling strategies.

The present report provides a comprehensive set of Best Practice Guidelines (BPGs) derived from the GO-VIKING Work Package 3 (WP3), which focuses on FIV of tube bundles in cross-flow. In particular, the BPGs are derived from work done by the involved partners on the modelling of the experimental facilities AMOVI and GOKSTAD, which is reported in project deliverables D3.2 (AMOVI benchmark) and D3.4 (GOKSTAD benchmark), respectively. These documents include contributions from multiple partners and collectively cover a wide range of modeling approaches, including:

- High-fidelity simulations using Large-Eddy Simulation (LES) and spectral element methods (e.g., Nek5000)
- Industrial CFD tools such as STAR-CCM+, ANSYS-based approaches, and Code_Saturne
- Turbulence modeling ranging from URANS eddy viscosity models and RSM to hybrid DES/IDDES and LES
- Structural modeling through finite element methods and rigid-body models, utilizing mass–spring–damper systems
- Both one-way and two-way FSI coupling strategies

The aim of this report is to synthesize these diverse approaches into a coherent framework, identifying the modeling choices that most strongly affect prediction accuracy, computational cost, and robustness. The report focuses on the various aspects of FSI modelling separately. In Chapter 2, important points for when creating the CAD and mesh are listed. In Chapters 3 and 4, BPGs for the modelling of the flow field and structural side are discussed, respectively. Then in Chapter 5 important considerations for the coupled FSI simulations are given. Finally, overall conclusions are listed in Chapter 6.

2. 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 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.

2.1 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 neighboring 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 modeling 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. 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 behavior 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 behavior.

2.2 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 and D3.4, all partners converged on the use of predominantly hexahedral meshes, with strong local refinement in physically important regions. This is illustrated in Figure 1, where mesh clustering is clearly visible around the tubes and in the inter-tube gaps.

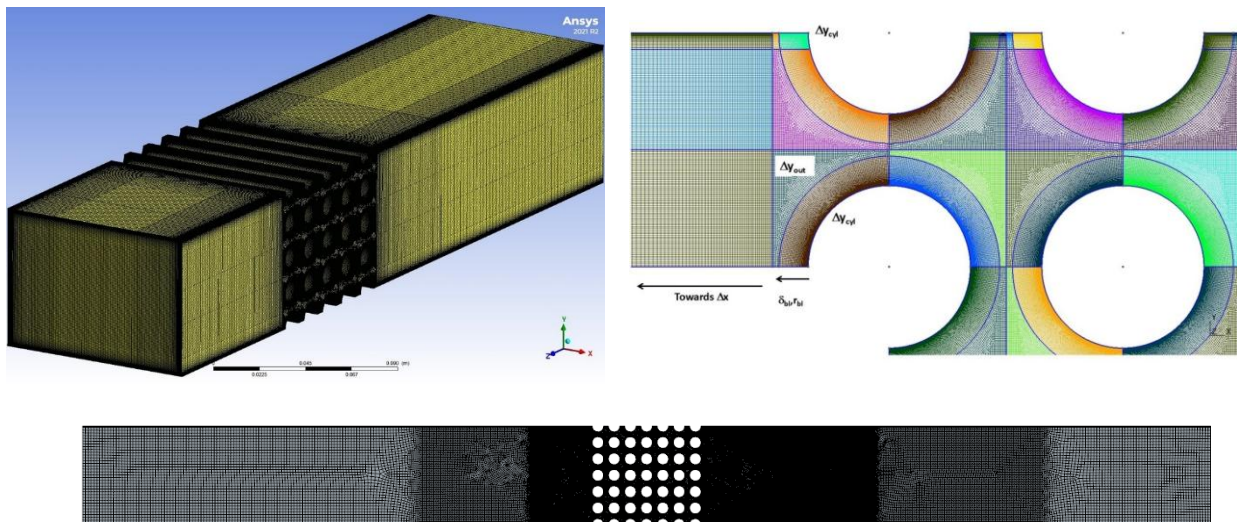


Figure 1: Examples of meshes used by the various partners within WP3, with top left: mesh GRS for AMOVI, top right: mesh EDF for AMOVI, and bottom: mesh NRG PALLAS for GOKSTAD.

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 1, 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 2.

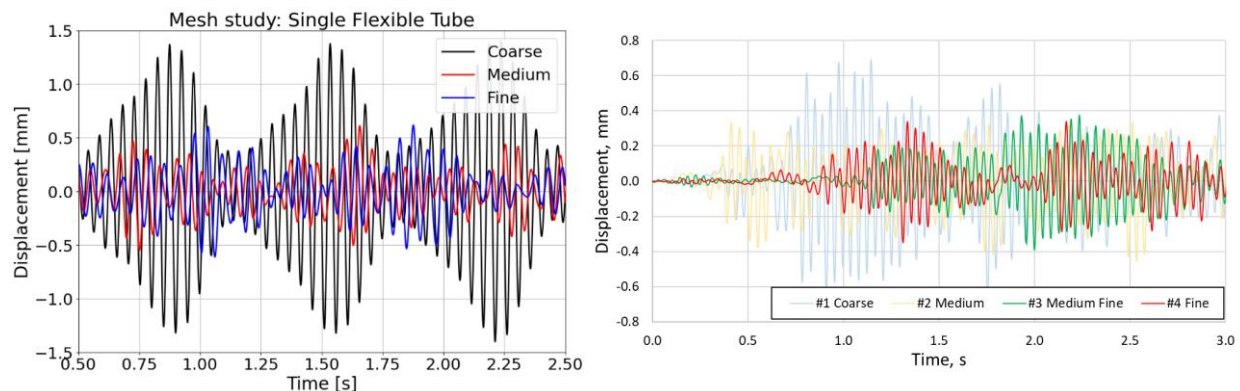


Figure 2: Example mesh independence studies performed for AMOVI, with left: for a single flexible tube (GRS) and right: for a flexible tube inside a rigid tube bundle (IPP).

2.3 Near-wall resolution and turbulence scale considerations

Accurate prediction of flow-induced forces requires proper resolution of near-wall behavior 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 are 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 aspect ratio of the near-wall computational cells should remain on 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 Reynolds Averaged Navier Stokes (RANS) simulation can be of great use. The Taylor microscale can be computed using turbulent variables from the RANS simulation, and compared against the mesh spacing in the LES mesh. Figure 3 shows an example of this comparison for the GOKSTAD tube bundle.
- **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.

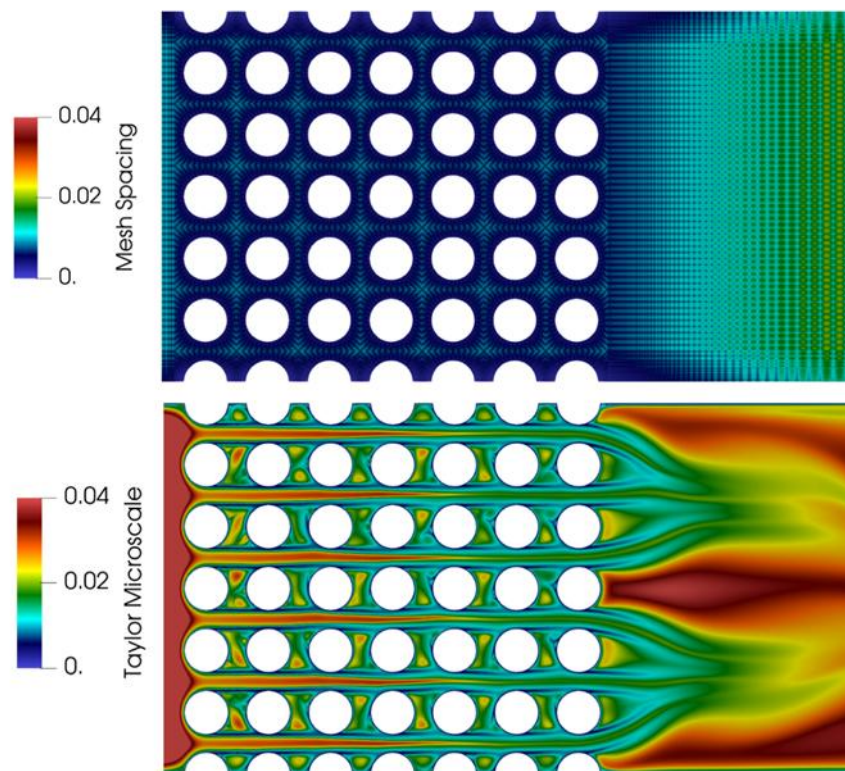


Figure 3: GLL spacing of a spectral element method mesh used in Nek5000 for LES (Top) and Taylor microscale computed from a RANS simulation using Nek5000 (Bottom).

3. Flow field 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 influences 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.

3.1 Turbulence modeling

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 wall-attached jet formation resembling the Coandă 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 4. 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.

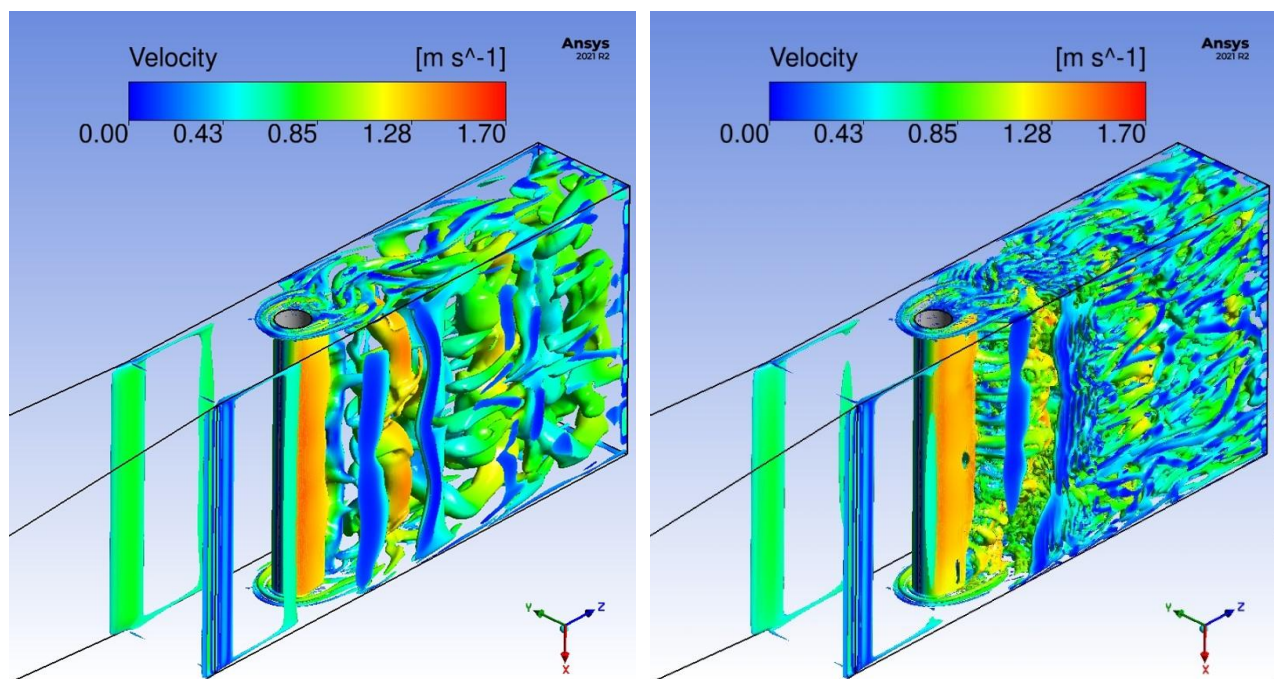


Figure 4: Resolved structures by GRS for AMOVI for single rigid tube in cross flow. Right: k- ω SST, left: SAS.

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 behavior is not of primary interest.

3.2 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—can 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 5. 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 several D_h . It is worth stating that D_h 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 $10 D_h$ 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.

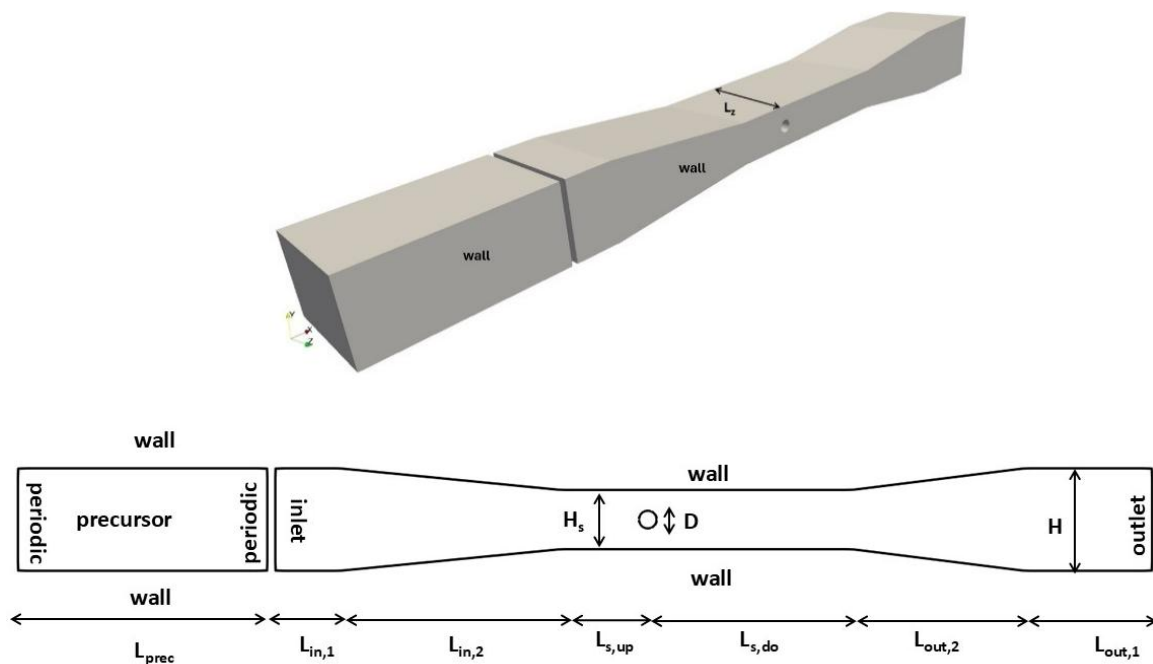


Figure 5: The computational domain created by EDF for the single cylinder configuration of AMOVI, clearly showing the precursor used to generate proper inlet conditions.

3.3 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 6. 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.

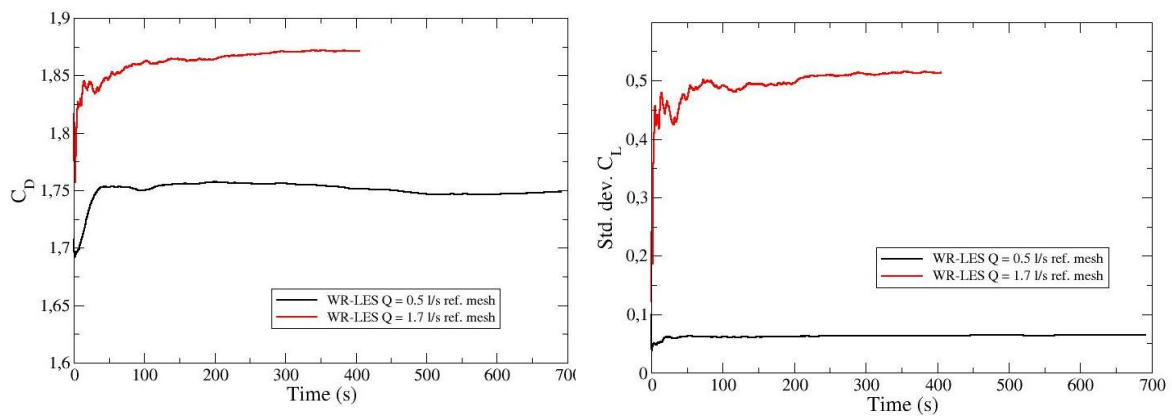


Figure 6: Time evolution of the mean drag coefficient (left) and the standard deviation of the lift coefficient (right) as obtained by EDF for the single rigid cylinder configuration.

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 behavior 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.

4. Structural modeling

The structural modeling of tubes in cross-flow conditions plays a crucial role in the accurate prediction of flow-induced vibrations. 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 behavior.

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 relatively 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

4.1 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 behavior. 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 behavior 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 behavior for small displacements, the AMOVI results indicate that the structural representation becomes more important as the dynamic response becomes more complex. Figure 7 shows two such possible representations for the moving cylinder in AMOVI.

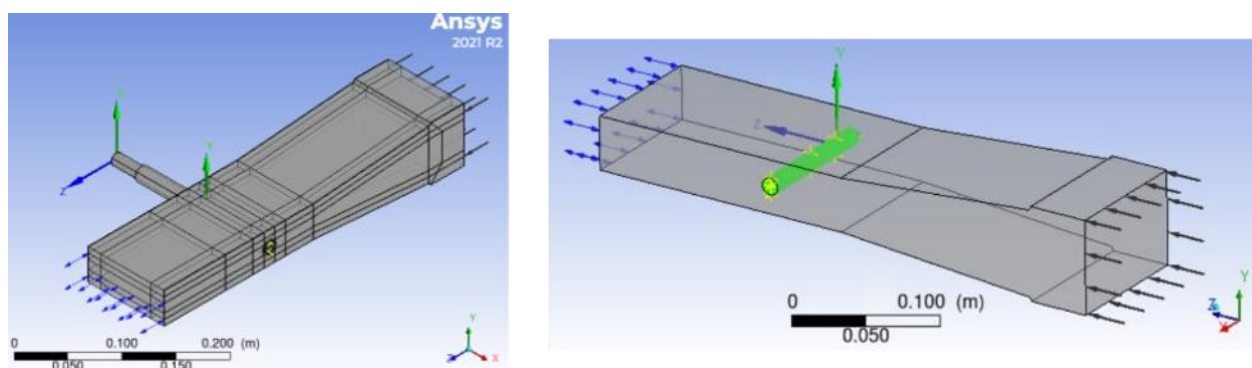


Figure 7: CAD models based on the used Rigid Body Motion approaches as used by GRS for AMOVI, with left: rotational motion model and right: translational motion model.

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. 2DOF 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.
- **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 behavior (e.g. through rotational formulations, as done by GRS) can improve the representation of the system dynamics compared to overly simplified assumptions, see Figure 8.
- **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.

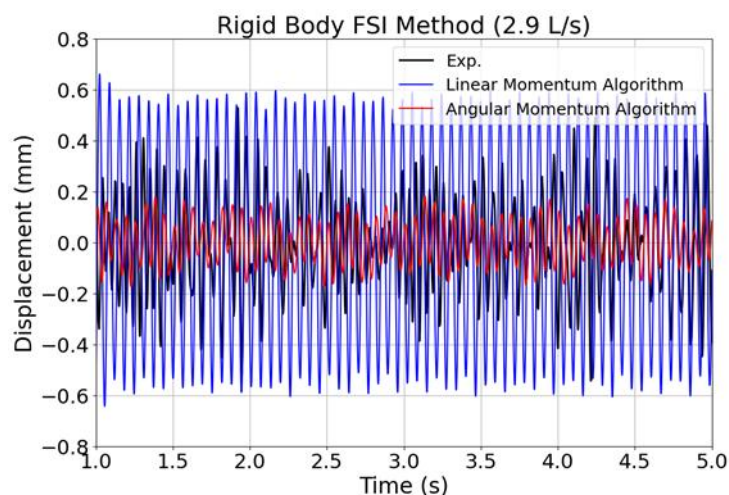


Figure 8: Tube displacements, calculated with linear and angular momentum algorithms (GRS).

4.2 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 modeling choices rather than inconsistencies in structural definition.
- **Ensure that the structural model reproduces the correct dynamic characteristics, particularly natural frequency and damping behavior:** 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 behavior, 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 9.

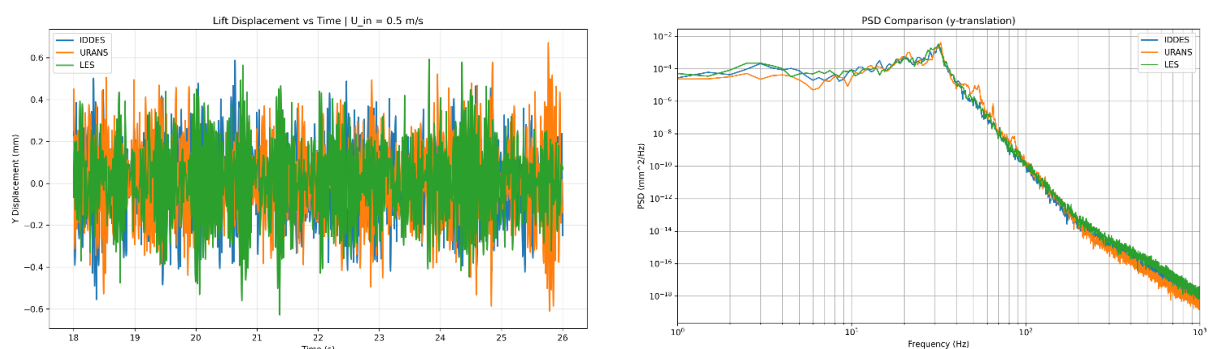


Figure 9: Lift displacement time series (left) and PSD (right) as obtained by NRG PALLAS for GOKSTAD.

5. 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 behavior.
- **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 10—are essential to preserve mesh quality and ensure numerical stability during FSI simulations.

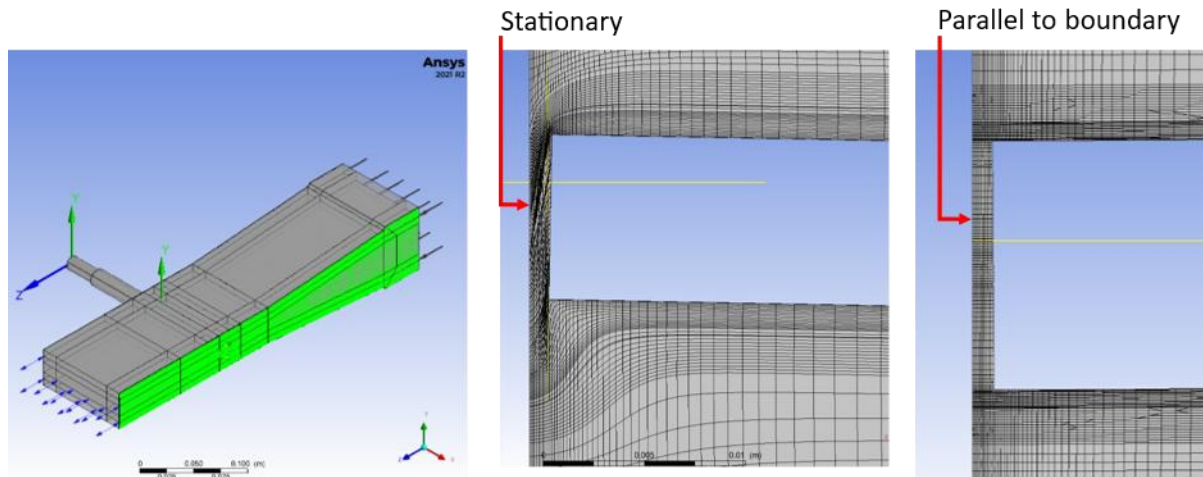


Figure 10: Test section with marked channel wall (left), Boundary condition for wall parallel to moving side of flexible tube

- **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 behavior 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 11 shows an example of a reduced computational domain of the GOKSTAD tube bundle using periodic boundary conditions.

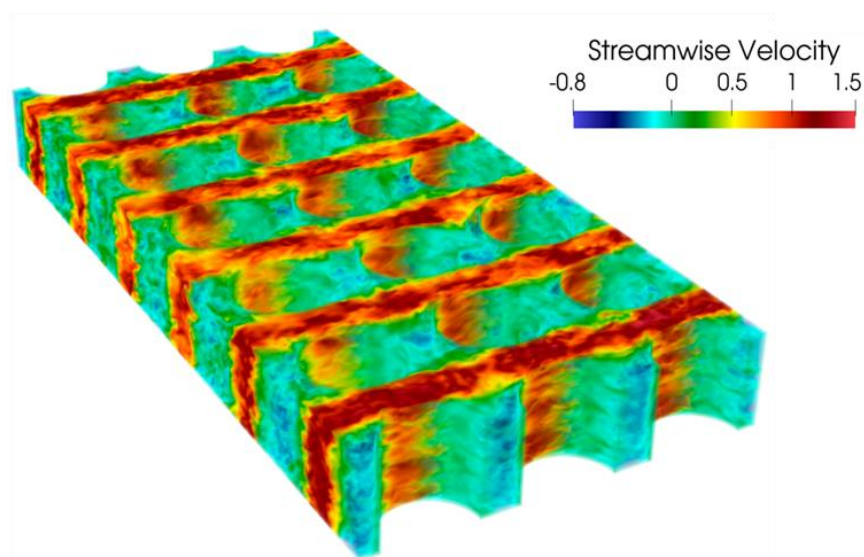


Figure 11: Volume rendered streamwise velocity contour plot of a reduced computational domain of the GOKSTAD tube bundle.

6. Conclusions

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 modeling 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 modeling 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 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 modeling 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 behavior is required. The Best Practice Guidelines presented in this report provide a structured framework to support such decisions, enabling users to select appropriate modeling strategies based on the specific requirements of their application.

7. Bibliography

Filonov, V., Filonova, Y., Dubyk, Y., Mistry, H., Papukchiev, A, and Benhamadouche, S. (2026). GO-VIKING deliverable D3.2: AMOVI model preparation and simulation results.

Zwjsen, K., Franklin, T., Carasik, L, and Benhamadouche S. (2026). GO-VIKING D3.4: GOKSTAD model preparation and simulation results.