



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 axial-flow

Authors : Hidajet HADZIC (Framatome GmbH), Mohammed Muaaz M. D. Hussain (NRG PALLAS), Sean Bistany (PSU), Thomas Norddine (EDF), Ewan Harris (EDF-Energy), Vladislav Filonov (IPP), Yaroslav Dubyk (IPP), Yuliia Filonova (IPP), Antoine Michel (CEA), Joachim Herb (GRS), Christoph Bläsius (GRS)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	Best practices for the analysis of FIV in single-phase axial-flow
Author(s)	Hidajet HADZIC, Mohammed Muaaz M. D. Hussain (NRG PALLAS), Sean Bistany (PSU), Thomas Norddine (EDF), Ewan Harris (EDF-Energy), Vladislav Filonov (IPP), Yaroslav Dubyk (IPP), Yuliia Filonova (IPP), Antoine Michel (CEA), Joachim Herb (GRS), Christoph Bläsius (GRS)
Number of pages	24
Document type	Deliverable
Work Package	WP2
Document number	D2.5
Issued by	Framatome GmbH
Date of completion	2026-06-05 11:32:15
Dissemination level	Public

Summary

This report presents a consolidated set of best practice guidelines for simulation of FIV in axial single-phase flow derived from the experience and lessons learned from the benchmarks analysed in the framework of work package 2. Across all investigated cases, it is shown that the accuracy of FIV predictions depends on the consistent treatment of turbulence, structural dynamics, and coupling methodology. It appears that the choice of a proper turbulence modelling approach has the greatest impact on accuracy, while the effect of coupling is less significant. Simplified, pragmatic, approaches can provide reliable engineering-level predictions in many cases, but the required model fidelity increases with the complexity of the flow and structural response. The guidelines therefore emphasize a balanced approach, where modelling choices are tailored to the physical problem and desired level of accuracy, keeping computational costs in mind.

Approval

Date	By
2026-06-05 11:41:34	Hidajet HADZIC (Framatome GmbH)
2026-06-05 11:44:24	Papukchiev ANGEL (GRS)



GO-VIKING deliverable D2.5:

Best practices for FIV analysis in single-phase axial flow

Version 1.0
May / 2026

Hidajet Hadžić¹, Mohammed Muaaz M. D. Hussain², Sean Bistany³, Thomas Norddine⁴, Ewan Harris⁵, Vladislav Filonov⁶, Yaroslav Dubyk⁶, Yuliia Filonova⁶, Antoine Michel⁷, Joachim Herb⁸, Christoph Bläsius⁸

¹Framatome GmbH, Paul-Gossen-Strasse 100, 91052 Erlangen, Germany

²NRG PALLAS, Westerduinweg 3, 1755 LE Petten, The Netherlands

³The Pennsylvania State University, Technology Center Bldg. 110, 16802 University Park, United States

⁴Electricité De France (EDF) R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatrou, France

⁵EDF ENERGY R&D UK CENTRE LIMITED, Street 90 WHITEFIELD STREET, W1T 4EZ LONDON, United Kingdom

⁶IPP Centre LLC, Bolsunovska Street 8, 01014 Kyiv, Ukraine

⁷Commissariat à l'énergie atomique et aux énergies alternatives, Rue Leblanc 25, 75015 Paris, France

⁸Gesellschaft für Anlagen- und Reaktorsicherheit (GRS), gGmbH, Schwertnergasse 1, 50667 Cologne, 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.

Document information

Grant Agreement / Proposal ID	101060826
Project Title	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
Project Acronym	GO-VIKING
Scientific Coordinator	Angel Papukchiev, mailto:Angel.Papukchiev@grs.de , GRS; Kevin Zwijsen, mailto:kevin.zwijsen@nrgpallas.com
Project starting date (duration)	1st June 2022 – 31st May 2026 (48 Months)
Related Work Package	WP2
Related Task(s)	Task 2.3 Best practices and input for reduced-order models
Lead Organisation	Framatome GmbH (FRA-G)
Contributing Partner(s)	FRA-G, EDF, NRG-PALLAS, PSU, EDF-Energy, IPP, CEA, GRS
Due Date	30-11-2025
Submission Date	31-05-2026
Dissemination level	PU

History

Date	Version	Submitted by	Reviewed by	Comments
31-05-2026	1.0	Hidajet Hadžić (FRA-G)	Angel Papukchiev (GRS) and Kevin Zwijsen (NRG PAL-LAS)	First submitted version

Table of contents

1	Purpose	8
2	Introduction	9
3	Overview of the WP2 benchmarks	10
4	CAD geometry and numerical mesh	11
4.1	Geometry representation	11
4.2	Numerical mesh	13
5	Flow field modelling.....	14
5.1	Flow field modelling in single rod configurations	15
5.2	Flow field modelling in fuel bundle simulations	17
6	Structural modelling	18
6.1	Structural modelling in single rod configurations.....	18
6.2	Structural modelling in bundle configuration	18
7	Coupling and FSI simulations	19
8	Additional aspects.....	21
8.1	Strategies to increase the efficiency of simulations	21
8.2	Validation and experimental data.....	22
9	Conclusions	22
10	Bibliography.....	24

List of figures

Figure 1: Setup of the WP2 benchmarks: Cantilever Rod (left) and ALAIN (right).	10
Figure 2: Vorticity distribution behind the spacer grid in a single span of the ALAIN experiment calculated with an LES approach by CEA [3].	12
Figure 3: Cantilever rod benchmark: High-resolution simulation - DNS by PSU - Computational domain [4].	13
Figure 4: Numerical mesh of a cantilevered rod for a LES simulation (EDF) left, numerical mesh around spacer grid (FRA-G) right.	14
Figure 5: DNS performed by Penn State University [4]: Q-criterion revealing coherent structures in the vicinity of the rod tip.	15
Figure 6: Beam model of the ALAIN 5x5 assembly with torsion springs and dampers, developed by GRS [8].	19

Abbreviations and acronyms

Acronym	description
BPG	Best Practice Guidelines
CFD	Computational Fluid Dynamics
CSM	Computational Structural Mechanics
CAD	Computer Aided Design
DOF	Degree Of Freedom
DES	Detached-Eddy Simulation
DNS	Direct Numerical Simulation
EVM	Eddy-Viscosity Model
FIV	Flow-Induced Vibration
FSI	Fluid–Structure Interaction
FA	Fuel assembly
GO-VIKING	G athering experience O n V ibration I mpaKt in N uclear power G eneration
LES	Large-Eddy Simulation
LDV	Laser Doppler Velocimetry
NPP	Nuclear Power Plants
PIV	Particle Image Velocimetry
PFM	Pressure Fluctuation Method
RSM	Reynolds-Stress Model
SAS	Scale Adaptive Simulation
SRS	Scale Resolving Simulation
SSG	Speziale-Sarkar-Gatski
TIV	Turbulence-Induced Vibration
TKE	Turbulent Kinetic Energy
(U)RANS	Unsteady Reynlods-Averaged Navier Stokes
WM	Wall-Modeled
WR	Wall-Resolved

Summary

This report presents a consolidated set of Best Practice Guidelines (BPGs) for the numerical simulation of Flow-Induced Vibration (FIV) under axial single-phase flow (e.g., specific for Fuel Assemblies (FA)), based on the findings of the GO-VIKING project deliverables D2.2 [1] that summarizes the results of the numerical simulations of the Cantilever Rod Benchmark, and D2.4 [2] that summarizes the results of the numerical simulations of the ALAIN Benchmark. By combining results from multiple partners and simulation approaches, including different turbulence models, structural representations, and coupling strategies, gained during the studies of the two benchmarks of the Work Package 2 (WP2), the report provides a consistent framework for selecting appropriate modelling choices for simulation of FIV of rods or fuel assemblies in single-phase axial flow. Key aspects addressed include CAD modelling, mesh generation, flow field modelling, structural representation, and Fluid–Structure Interaction (FSI) coupling.

Across all investigated cases, it is shown that the accuracy of FIV predictions depends on the consistent treatment of turbulence, structural dynamics, and coupling methodology. It appears that the choice of a proper turbulence modelling approach has the greatest impact on accuracy, while the effect of coupling is less significant. Simplified, pragmatic, approaches can provide reliable engineering-level predictions in many cases, but the required model fidelity increases with the complexity of the flow and structural response. The guidelines therefore emphasize a balanced approach, where modelling choices are tailored to the physical problem and desired level of accuracy, keeping computational costs in mind.

Keywords

Flow-Induced Vibration, Benchmark, Axial Flow, Rods, Fuel Assemblies, FIV, FSI, CFD, Single-Phase, LES, DES, SRS, URANS

1 Purpose

Vibration, induced by coolant flow, represents an important challenge for the integrity and operability of different components of the Nuclear Power Plants (NPP). The FIV leads to increased wear and tear and/or material fatigue, so that it has been and remains important contributor to failures of key NPP components. Such failures can degrade NPP safety features and fail confinement barriers. Some (typical) examples of the NPP components affected by FIV are steam generators and fuel assemblies.

The GO-VIKING (**G**athering experience **O**n **V**ibration **I**mpa**K**t **I**n **N**uclear power **G**eneration) project is aimed to contribute to an improvement of the current state-of-the-art knowledge on FIV and its analysis in support of a safe and reliable operation of NPP. The overall objective of GO-VIKING project is the increase of the expertise and improvement of the tools and skills of the European nuclear stakeholders in the analysis of complex FIV phenomena, which in turn will maintain and further enhance nuclear plant operation and safety. This was accomplished by making available existing and new experimental results and improved numerical approaches for the evaluation of FIV in nuclear power plants.

This document is the GO-VIKING deliverable D2.5, which is concerned with providing the best practice guidelines for the numerical simulation of FIV under single-phase axial flow, which is specific for nuclear fuel assemblies, fuel rods, control rods and other similar structures in NPP. The guidelines are derived from the numerical simulations performed by contributing partners that were involved in the numerical investigations of the two benchmarks within the work package WP2 of the GO-VIKING project.

The BPG provided in this report reflect the experience gathered by different institutions involved in the simulations of two specific benchmark cases that have been analysed in the frame of WP2. Therefore, these guidelines are to some extent specific and cannot and should not be considered as general. Nevertheless, the lessons learned, and the guidelines derived from this experience may still be of interest and helpful for the members of FIV community in further improving of their own simulation approaches. It is the decision of each user whether and to what extent these BPG can be applied to their own case.

2 Introduction

The prediction of FIV in fuel assemblies or other similar structures subjected to axial-flow is a challenging problem in nuclear engineering, nuclear fuel assembly design, and FSI modelling. Fuel assemblies exhibit highly complex flow behaviour, characterized by strong turbulence production by spacer grids or other structural components, which is propagated further along the fuel assembly, and excites its vibration. The vibration of fuel assemblies or other components subjected to axial flow is usually turbulence-induced which is characterized by stable low-amplitude vibration. Under certain conditions, the FIV may become more critical and exhibit a kind of instable behaviour which in turn can quickly lead to damage or even failure of the affected components. The accurate representation of such phenomena by numerical simulation requires careful consideration of modelling assumptions, numerical methods, and coupling strategies.

The present report provides a comprehensive set of BPG derived from the GO-VIKING WP2, which focuses on FIV under single-phase axial-flow. In particular, the BPG are derived from the work done by the involved partners on the simulation of the two benchmarks, Cantilever Rod and ALAIN, which are reported in project deliverables D2.2 [1] and D2.4 [2]. These documents include contributions from multiple partners and collectively cover a wide range of modelling approaches, including:

- High-fidelity simulations using Direct Numerical Simulation (DNS) and spectral element methods (e.g., using NekRS code),
- Industrial CFD tools: STAR-CCM+, ANSYS-CFX, OpenFOAM, Code_Saturne, TrioCFD,
- Turbulence modelling ranging from Unsteady Reynolds-Averaged Navier Stokes (URANS) Eddy-Viscosity Models (EVM) and Reynolds-Stress Models (RSM) to hybrid Detached-Eddy Simulation (DES) and Large-Eddy Simulation (LES),
- Structural modelling through three-dimensional finite elements representation, beam models, multiple Degrees Of Freedom (DOF) rigid-body models,
- One-way and two-way FSI coupling strategies as well as decoupled FIV analysis.

The aim of this report is to synthesize these diverse approaches into a coherent framework, identifying the modelling choices that most strongly affect prediction accuracy, computational cost, and robustness. The report focuses on the various aspects of FSI modelling separately. Since the two benchmarks are substantially different (see Figure 1), the simulation approaches used in two cases are also (significantly) different. Therefore, the BPG are reasonably provided separately for simulation of single rod and simulation of rod bundles (i.e. fuel assemblies) where appropriate.

The report is organized as follows: In Chapter 3 an overview of the two benchmark configurations analysed in D2.2 and D2.4 deliverables is recalled for reference. In Chapter 4, important points related to the geometry of the computational domain and numerical mesh are listed. In Chapters 5 and 6, best practices guidelines for the modelling of the flow field and structural side are discussed, respectively. Then in Chapter 7 important considerations for the coupled FSI simulations are given. In Chapter 8 some additional aspects related to experimental investigations are discussed. Finally, overall conclusions are summarized in Chapter 9.

3 Overview of the WP2 benchmarks

In this chapter a brief overview of the WP2 benchmarks is provided. The setup of each benchmark is provided in Figure 1 left for the Cantilever Rod benchmark and right for the ALAIN benchmark.

In the Cantilever Rod benchmark, the FIV of a single cantilever rod subjected to single-phase axial flow is investigated. The cantilever rod is clamped at the top and free at the bottom tip inside a confining vertical tube. Under the influence of the flow, the vibration of the free rod end takes place.

In the ALAIN benchmark a reduced scale 5x5 fuel assembly placed in a squared channel and subjected to single-phase axial flow is analysed. The fuel assembly consists of 23 fuel rods and 2 guide tubes arranged into a 5x5 square matrix. It incorporates 5 spacer grids distributed over the length and lower and upper tie plates at the bottom and top ends.

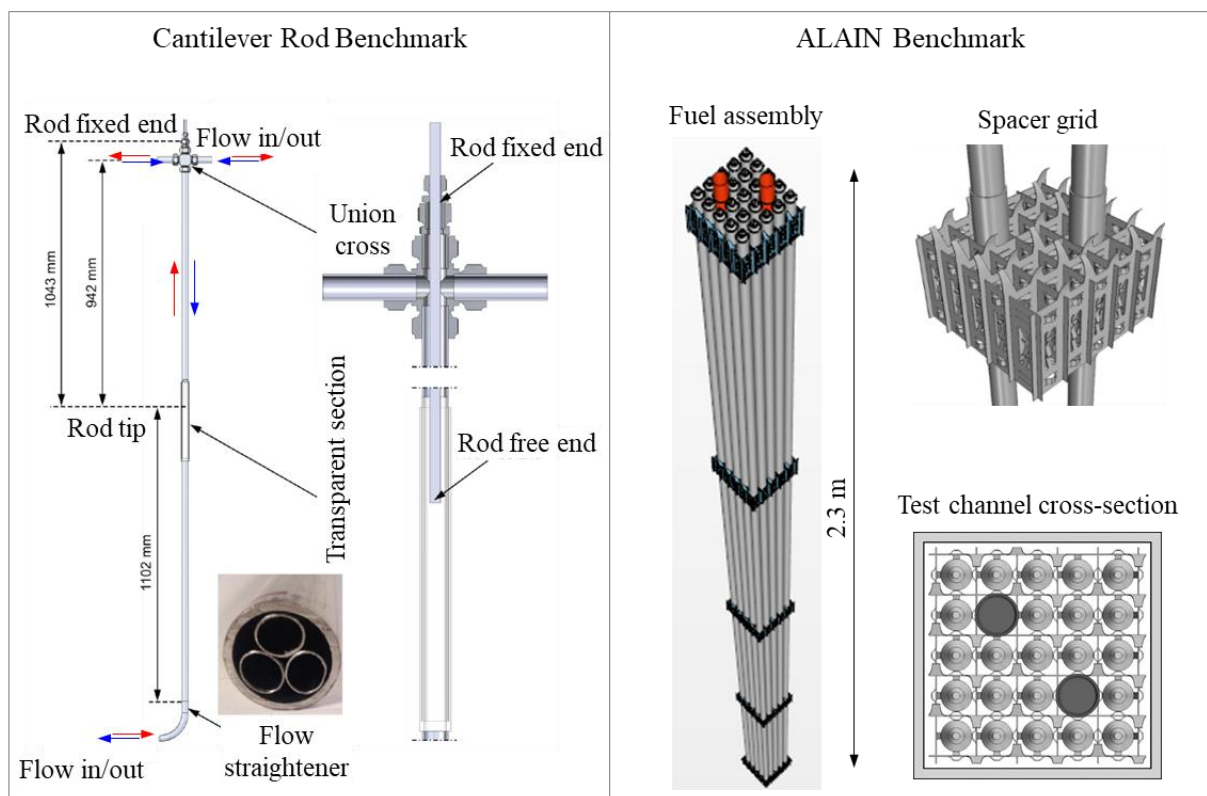


Figure 1: Setup of the WP2 benchmarks: Cantilever Rod (left) and ALAIN (right).

In the framework of Cantilever Rod benchmark, the following two types of FSI or stand-alone CFD simulations were performed:

- High-resolution simulations using Direct Numerical Simulation (DNS) and wall-resolved (WR) Large-Eddy Simulation (LES),
- Medium resolution simulations based on the use of different RANS EVM and RSM.

In the framework of the ALAIN benchmark, different simulations were performed using different Scale-Resolving Simulation (SRS) approaches in combination with different CFD codes and structural models.

4 CAD geometry and numerical mesh

The definition of the computational geometry and corresponding numerical mesh forms the foundation of any CFD or FSI simulation in general, and so also for simulation of FIV in axial flow. The main purpose of CAD (Computer Aided Design) model is to represent the geometry of the problem under investigation, while the purpose of the numerical mesh is to resolve the key flow features driving FIV. In this chapter, focus is put only on the geometry and numerical mesh related to CFD models. Structural modelling is addressed later in Chapter 6.

In the context of the numerical simulations performed in the framework of the WP2 deliverables D2.2 and D2.4, different partners employed a range of modelling tools and approaches that can be summarized into several consistent best practices regarding geometry representation, extent and simplification strategies, and mesh refinement.

4.1 Geometry representation

Regarding the geometry representation, following aspects are important: representation of the relevant geometrical features, extension of the computational domain to capture the flow field details relevant for the simulation, potential simplification or reduction of the geometrical details or geometry (computational domain) itself.

From the experience gathered with the simulations performed within WP2 benchmarks, following BPG regarding geometry can be made:

Make sure that relevant geometry features are included in the simulation: This it is usually well defined by the CAD input. Nevertheless, some additional consideration may be needed. In the case of cantilever rod, it is the geometry of the rod and of the confining tube. The area of primary interest is the rod tip. If coupled FSI simulation is performed, the whole rod length must be considered. In the case of CFD simulation only, just a part of the rod length above the rod tip may be sufficient to study all relevant features of the flow field. This was the case e.g. in the high-resolution simulation performed using Direct Numerical Simulation (DNS) (see Figure 3). In addition, the domain needs to be extended sufficiently in the front of the rod tip to capture properly the inflow conditions.

Extension of the domain to accommodate inflow and outflow – cantilever rod: Typically, the computational domain is extended in such a way to provide well defined / developed flow coming to the area of interest. Usually, one is interested in having a fully developed flow conditions around the rod tip. Common approach is to extend the flow domain far enough from the rod end and to specify there the inlet boundary. In the simulations of cantilever rod performed within the deliverable D2.2 the inlet boundary was placed at least 5-6 tube diameters upstream the rod tip. This shortening of the computational domain is especially important for high-resolution simulations where considerable savings in the computational effort can be achieved. Nevertheless, this strategy may also be used in combination with medium or low-resolution approaches. To have well-defined inlet boundary conditions it is important to prescribe corresponding quantities at the inlet boundary. If these are not known, which is often the case, a common practice is to provide these by a running in parallel an additional simulation that provides fully developed flow which is then mapped to the inlet. This was the approach used in the high-resolution simulation with wall-resolved LES.

Extension of the domain to accommodate inflow and outflow – ALAIN benchmark: In the case of ALAIN benchmark, at the bottom and top side of the FA there are components that significantly affect the flow field. Therefore, the sensitivity on the prescription of the inlet boundary conditions is much lower in this case. Typically, an extension of a few channel widths below the lower tie plate is sufficient for the inlet boundary and similar extension at the outlet. For the outlet it is a good strategy to extend the domain far enough to prevent flow recirculation.

Utilize geometrical simplifications: Geometrical simplifications can be applied for different purposes. This aspect is more relevant for the ALAIN benchmark since the geometry of the FA is quite complex. One possible purpose for geometry simplification can be the “smoothing” of some geometrical details e.g. where very sharp edges appear (typical situation is connection between fuel rods and spacer grids in a FA). In this case very small geometrical modifications are performed to ensure a sufficient mesh quality (see e.g. section 5.1.4.2 of D2.4 document [2]).

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 FA 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. 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 FA 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.

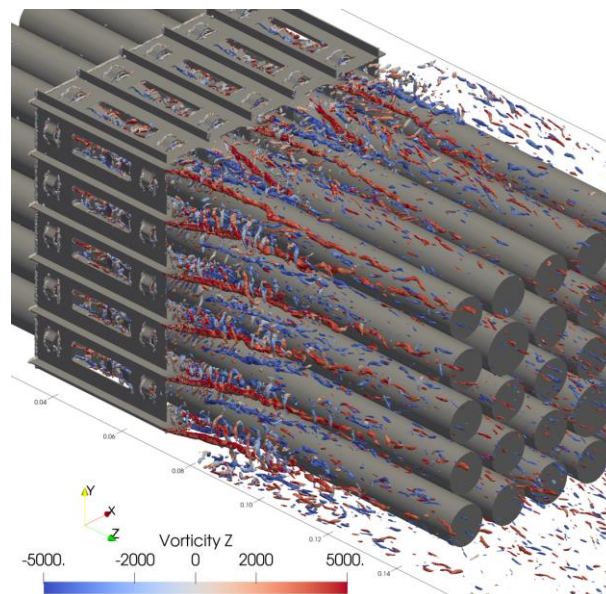


Figure 2: Vorticity distribution behind the spacer grid in a single span of the ALAIN experiment calculated with an LES approach by CEA [3].

Another example of the reduction of the geometry of the computational domain is the flow field analysis of the cantilever rod (without FSI) using the high-resolution approach based on DNS. In this case the computational domain extended for about 6 tube diameters upstream and about 10 tube diameters downstream the rod tip.

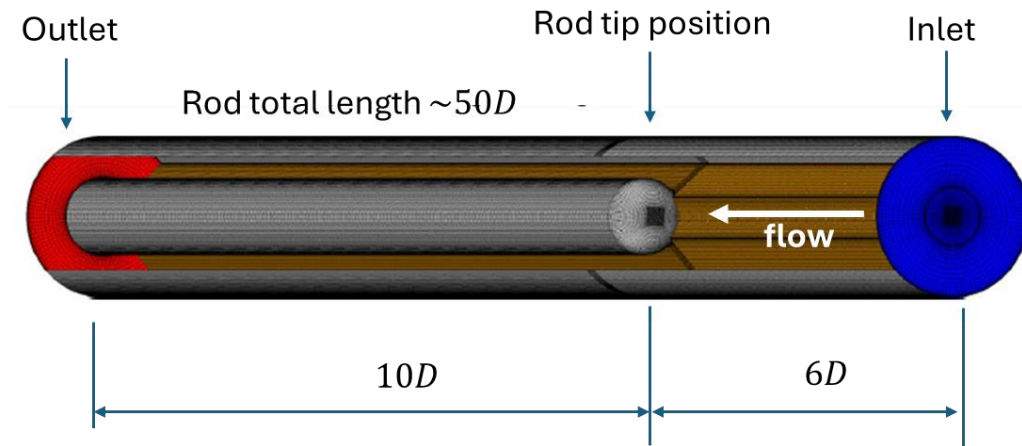


Figure 3: Cantilever rod benchmark: High-resolution simulation - DNS by PSU - Computational domain [4].

4.2 Numerical mesh

The numerical mesh is the central part of a CFD simulation. The topology, resolution, and quality of the numerical mesh play a decisive role in the success of a CFD simulation. The requirement on the mesh depends on different factors, but the main is the choice of the turbulence modelling approach. In this context, the scale-resolving simulation (SRS) approaches, which have been identified as generally more appropriate ones for the simulation of FIV for WP2 benchmarks, have higher demands than the approaches based on RANS models.

The topology of the mesh used in a CFD simulation depends strongly on the CFD code or/and associated tools used for mesh generation. In the WP2 benchmarks the meshes composed predominantly of hexahedral cells were used. The exception are the simulations performed by CEA using the TrioCFD code which uses the mesh composed of tetrahedral cells. According to the mesh topology, i.e. mesh generation tool and CFD code used, the associated specific meshing strategy may be different.

Some recommendations regarding mesh generation are summarized below:

Special care is necessary for the mesh preparation: While today an automatically generated mesh (Figure 4) is standard, one shall always check and further improve the mesh at the locations of interest. Especially is important to prevent bad cell quality that usually appear at the region with sharp edges, like is the case at the contact between fuel rods and spacer grids in a FA. “Smoothing” 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 which are using the non-diffusive but non-bounding discretization schemes which are more sensitive to mesh quality.

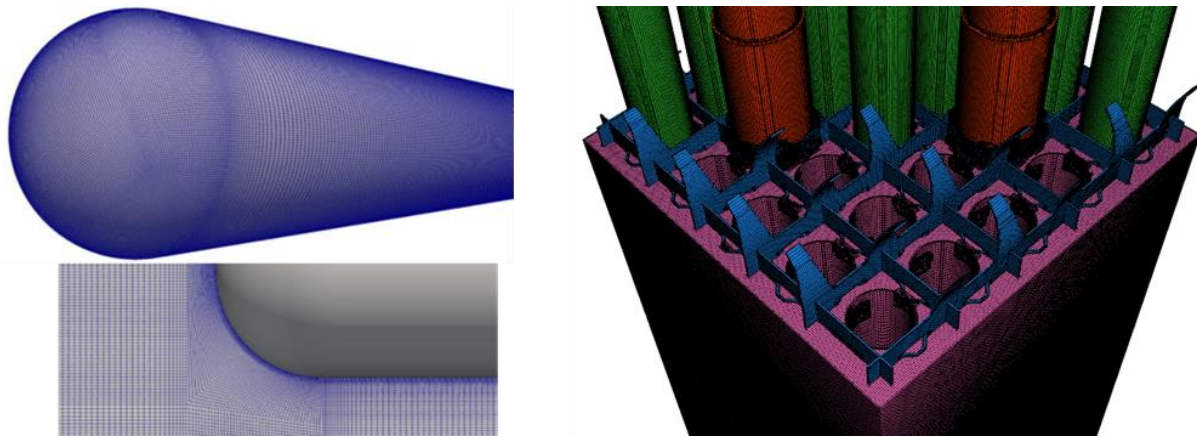


Figure 4: Numerical mesh of a cantilevered rod for a LES simulation (EDF) left, numerical mesh around spacer grid (FRA-G) right.

Utilize geometry repeatability for mesh generation and optimization: The local mesh quality can be optimized together with local geometry optimization described in previous section. Another good strategy for mesh generation and optimization is to use smaller domain blocks. Such smaller domain blocks are easier to mesh and to optimize the mesh. This can be very efficiently used in the case of FA geometry where several spacer grids are present. An optimized mesh generated around one spacer grid can be used also on other spacer grid locations.

Mesh resolution for FA configuration: Application of scale-resolving methods, like LES, typically requires the use of a numerical mesh much finer compared to RANS methods. A very fine mesh is required to resolve fine turbulent eddies being generated especially close to the solid wall boundaries. For 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 Tylor microscale) as reference to estimate roughly the required mesh cell size. For cell expansion the recommendations from the literature can be used, e.g. [5]. Close to wall boundaries, use multiple cell layers and adjust the y^+ value in a suitable range.

5 Flow field modelling

The accurate modelling of the flow field is of essential importance for reliable prediction of FIV in axial single-phase flow. The fluid forces driving the FIV in both benchmarks originate from the turbulent flow structures. Both deliverables D2.2 [1] and D2.4 [2] demonstrate that the choice of turbulence modelling approach and associated turbulence model in the CFD simulations play the key role for reliable prediction of FIV. In the following two sections the main best practices derived from the simulations performed in D2.2 and D2.4 are summarized. Due to significant differences in the two benchmark configurations the BSG are provided separately for each case.

5.1 Flow field modelling in single rod configurations

High-resolution approaches:

The 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 behaviour.

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

Following take aways can be taken from high-resolution simulations:

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 very fine mesh and small time steps required. In case of TIV, scale resolving simulations are irreplaceable.

Post-process further quantities: Post-processing first-order statistics is not always sufficient. It is advisable to evaluate the Q-criterion (Figure 5) and Turbulent Kinetic Energy (TKE) budget to carefully examine and analyse the flow phenomena that take place in the CFD domain. These statistics provide high-fidelity data that can also be used for benchmarking and validation of turbulence approaches with lower resolution.

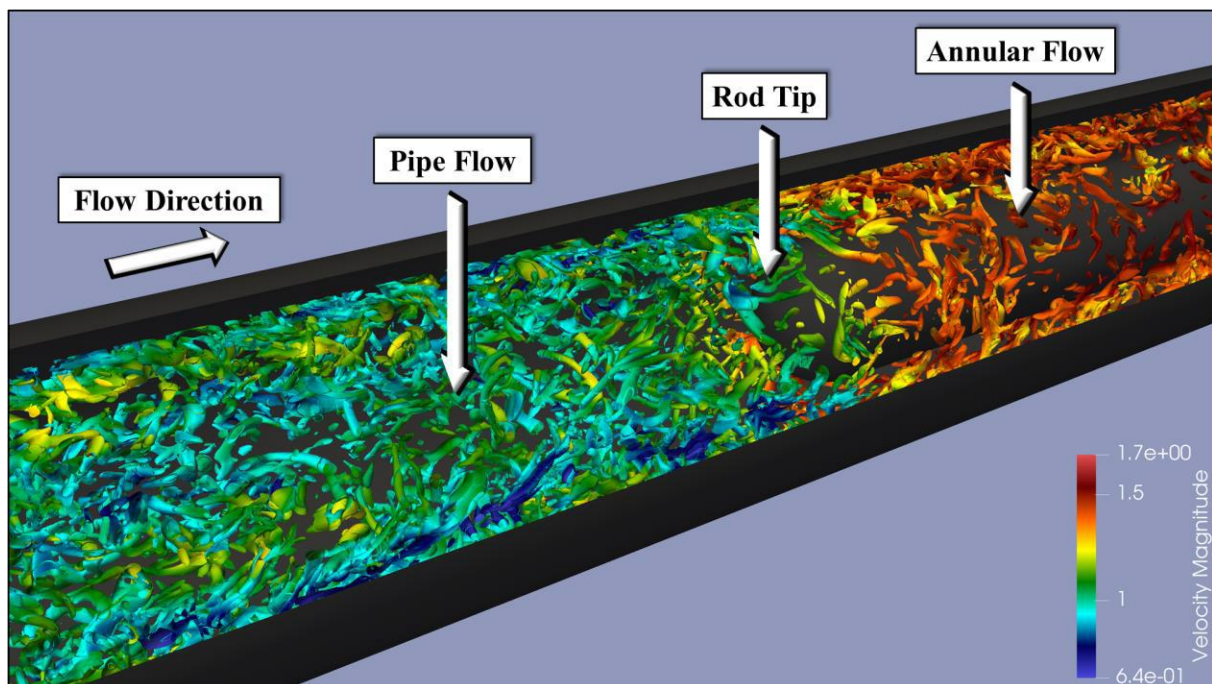


Figure 5: DNS performed by Penn State University [4]: Q-criterion revealing coherent structures in the vicinity of the rod tip.

Medium-resolution approaches

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

The SSG RSM model performed well in predicting FIV 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 with the $k - \varepsilon$ RANS EVM with cubic constitutive relations were also performed. The low Re formulation of the model is 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 $k - \omega$ 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 – more than a factor of 2 between them.

Unsteady simulations with RANS turbulence models struggle to accurately process the boundary layer and the generation of unsteady flow structures at the rod tip, which are the main driver of the rod vibration. 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 for the blunt-end 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 FIV not being predicted correctly.

Following can be derived from medium-resolution simulations with Cantilever Rod:

Pay attention to the choice of turbulence model, it has the largest impact on the accuracy of the results: It was found that the choice of turbulence model had the most impact on the accuracy of the results. The impact of this choice overshadowed any potential gain in accuracy of using sophisticated Computational Structural Mechanics (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. URANS approach could be used to some extent for the blunt-ended rod cases, since more turbulence is generated there through the impingement of the flow on the blunt rod end. 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 its 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.

5.2 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 modelling 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-modelled LES showed a better performance than the DES and SAS models.

The main drawback of the performed simulations was the computational effort 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 modelling 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 can resolve turbulent flow structures. The recommended approach is the wall-modelled LES.

Use wall-modelling as the only practical option for larger structures like FA: wall-resolving simulations of a single, complete FA is computationally extremely expensive, and hence, not feasible for industrial applications. Therefore, wall-modelling is the only reliable alternative in combination with LES or any other scale-resolving (also URANS) simulation approach. Even a wall-modelled LES is still very expensive when applied for a full FA.

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 (Anisotropic Pressure Fluctuation Model) 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 and solution of the equation for pressure fluctuations. More information can be found in [6] and [7].

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 microscale), 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 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.

6 Structural modelling

6.1 Structural modelling in single rod configurations

In single rod configuration the representation of the rod by a beam model was found a very good pragmatic solution. The use of finite element representation of the rod did not show any specific advantage. Especially the combination of the Beam model with the wall-resolved LES was found a very efficient way for FSI coupling.

The following BPGs for the structural modelling of the cantilever rod can be made:

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 quite often the case.

Keep in mind axial fluid force: Be aware that axial-flow (up-flow or down-flow) introduces compressive or tensile loads that can shift vibration frequencies of the rod.

6.2 Structural modelling in bundle configuration

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 FA may be modelled by just one or by multiple beams.

Include additional elements to capture some FA special features, such as the clamping of the fuel rods in the spacer grids: This is necessary to simulate the effects of these features on the overall FA structural dynamic properties. This typical approach for structural modelling

facilitates significantly the handling of the structural model and increases computational efficiency.

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 6).

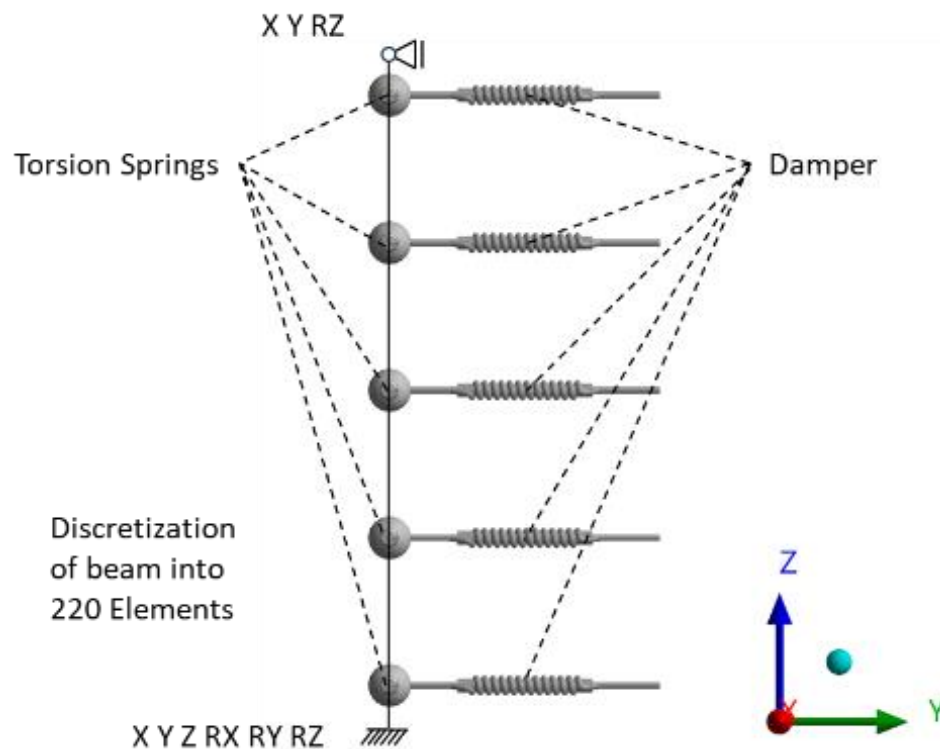


Figure 6: Beam model of the ALAIN 5x5 assembly with torsion springs and dampers, developed by GRS [8].

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 FA structure. Although not typical, this approach could also be followed for the representation of the FA structure. For more details the reader is referred to the D2.4 deliverable document.

7 Coupling and FSI simulations

For the ALAIN benchmark, coupled FSI simulation turned out to be very challenging for given configuration and could not be realized for most participants in the present project. Although, based on the very small vibration amplitude in the given case, the usefulness of a coupled FSI simulation may be questioned. CFD simulations performed with imposed mesh movement showed that there is no effect of this small-amplitude motion on the flow field. On the other hand, the decoupled approach showed the capability for reliable prediction of the vibration amplitude, given that all needed parameters are evaluated properly.

Based on this experience following recommendations can be made:

Analyse, if coupled FSI simulations are necessary and make sense: Are the expected changes in the geometry due to forces 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 in the fluid flow and therefore the simulations of the fluid and the structural domains of the model can be simulated separately. Then the forces 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 decoupled approach: In the case of decoupled simulations in which CFD (fluid flow) 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 the rod bundles, contrary to some simple structures such as a single rod where these effects are much smaller. Especially fluid damping may become significant with increasing axial flow velocity. One must 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 FA 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 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.

Make sure that the calculation of the forces is correct: A critical finding was that the calculation of forces 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. In general, one must make sure that the forces are computed correctly, and this should always be tested appropriately prior to the final simulation. One must be especially careful when using user libraries independently which CFD code is used.

8 Additional aspects

In this chapter some additional aspects derived from the numerical simulations performed in the framework of two WP2 benchmarks, which are not covered in previous chapters, are addressed.

8.1 Strategies to increase the efficiency of simulations

Here the focus is mainly on the simulations of rod bundles, e.g. fuel assemblies, resulting from the experience gathered with the ALAIN benchmark. Even the simulation of a small-scale fuel assembly turned out as very challenging with respect to computational effort. Therefore, any possibility for reduction of the computational effort would be of great benefit for such simulations.

One possibility to reduce the computational effort is by model simplifications. This can be realized through:

- Usage of element sizes larger than required accepting increased uncertainty,
- Performance of decoupled simulation in combination with added values for fluid effects from literature, experiments, experience.
- Reduction of the calculated CFD time interval and space by extending the results; Calculate only a rather short time signal of the excitation forces by CFD and extend this one to a length which is appropriate for mechanical simulation. Several methods may exist for an extension of CFD data, including staggered sequencing, cutting and reassembling, transfer to the frequency domain and resynthesis.
- The settling-in period of the mechanical model must be cut off when interpreting the results. Its length can be reduced by removing the static part from the applied data since it does not contribute to vibration but only to static displacement. The later, however, cannot be calculated correctly in the framework of a decoupled simulation and hence it is a better strategy to remove it from the input time histories.

Another possibility for reduction of the computational effort in a CFD simulation is by reduction of computational domain as described in section 4.1 and reduction of resolution inside the domain as described in this section above is the simplification of the geometry in general, e.g. by removing some features from the CAD combined with a low resolution through the numerical mesh. This strategy is typically connected to a kind of reduced-order (medium or low resolution) modelling to capture the global (mean) flow field. In such a case the reduction of the geometrical details must be in line with the intended modelling approach that should be sufficient to capture properly the flow features of interest. This kind of simplification can be supplemented with the detailed analysis on the reduced geometry models, such as one span in the case of ALAIN benchmark in combination with an SRS approach to provide detailed information about turbulence that can be mapped on the large-scale simplified model.

It must be emphasized that any simplification needs to be carried out carefully and accompanied by the additional validation and verification effort to make sure that it produces reliable results.

8.2 Validation and experimental data

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 sheds light on the flow phenomena in the CFD domain is very helpful. In this sense, force measurements of rigid structures are very useful as well.

Extend experimental investigations for further validation purposes: It is advisable to validate structural and fluid models with vibration tests in air and still water before moving to the cases with non-stagnant fluid. This would provide information on the main structural mechanics properties and effects of fluid on these properties.

Perform an extensive validation 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 should be performed for several cases with different boundary conditions, i.e. different Reynolds number.

Use high-fidelity high-resolution simulation data for further validation: Although the high-resolution simulations are very expensive from the computational point of view, they represent a valuable basis for validation of medium-resolution simulation approaches. This type of data should be used in addition to experimental data to extend the validation basis.

9 Conclusions

The turbulence modelling approach used in the CFD simulation plays obviously a significant role for the predictability of the vibration response. For both cases analysed in WP2 turbulence is the main driving vibration mechanism. Consequently, it is essential to use turbulence modelling approaches and turbulence models that can resolve relevant turbulent flow structures that are responsible for excitation of the structure.

LES was found to be the best performing approach in combination with wall-modelling, which is the only feasible option in combination with LES for the ALAIN (bundle) configuration. Wall-resolved LES, on the other hand, turned out as the best performing option for Cantilever Rod case with rounded rod end. Simulation approaches based on RANS turbulence models turned out as not performing well in the case of ALAIN benchmark and cantilever rod benchmark with the rounded end rod. In the case of the Cantilever Rod with blunt end the FIV predictions with RANS models were relatively good. The explanation for this discrepancy in the performance is that in the case of the blunt rod end different flow conditions are created around the rod tip which are capable to excite rod vibration. This is an important lesson learned that demonstrates the importance of turbulence modelling but also the importance of other parameters like the geometry of the rod end. Important to mention is also the fact that the $k - \varepsilon$ RANS model with cubic constitutive relations and in its low Re formulation showed certain advantages against the wall treatment based on high Re formulations in the case of cantilever rod with rounded end. It seems like the wall treatment plays a more important role for rounded geometry compared to geometry with sharp contours.

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. Computational costs play also a role for the simulation of cantilever rod. While providing high-fidelity results, the high-resolution simulations of the cantilever rod using DNS and wall-resolved LES were computationally significantly more expensive compared to medium-resolution approaches. The computational effort is in general a significant limitation factor for CFD simulations and any alternative solution that can reduce the computational effort while still providing sufficient accuracy is of great value for simulation of FIV.

For FIV with very small vibration amplitude decoupled simulation may be sufficient. However, one must keep in mind that additional parameters describing the effects of fluid on dynamic behaviour of the structure emerged in the fluid must be added to the mechanical model. An accurate evaluation of these parameters is essential for a reliable prediction of FIV since they may have a significant effect on the vibration response.

In the ALAIN benchmark the vibration amplitudes from the FIV tests are very small, just a few μm RMS. For such small displacements a decoupled simulation approach may be sufficient, i.e., a coupled FSI simulation may not be necessary. This was also shown by the simulation results, which on the one hand showed no influence of the movement of the structure on the flow and on the other hand showed a relatively good prediction of the vibration amplitudes.

Nevertheless, coupled FSI simulation is necessary in the case when instable FIV behaviour may be expected and therefore remains the preferred method for simulation of FIV.

10 Bibliography

- [1] M. M. M. D. Hussain et. al, GO-VIKING deliverable D2.2: Simulation of the single-phase Cantilever Rod benchmark. Version 1.0, December 2025.
- [2] H. Hadžić et. al. GO-VIKING deliverable D2.4: Simulation of ALAIN 5x5 PWR rod bundle benchmark. Version 1.0, May 2026.
- [3] A. Michel, M. A. Puscas, and A. Bergeron. A comparative analysis of the flow in a rod bundle with two guide-tubes and mixing grids, URANS simulation and LES of the ALAIN experiment. Nuclear Engineering and Design, 441, 114115, 2025.
- [4] S. Bistany, V. Coppo Leite, C. Bourdot Dutra, S. Okyay, A. Chaube, T. Nguyen, E. Merzari. Direct numerical simulation of a cantilevered rod experiment. International Journal of Heat and Fluid Flow, Volume 117, Part B, 2026, 110112, ISSN 0142-727X, <https://doi.org/10.1016/j.ijheatfluid-flow.2025.110112>. (<https://www.sciencedirect.com/science/article/pii/S0142727X25003704>).
- [5] U. Piomelli (2002). Large-eddy and direct simulations of turbulent flows. Department of Mechanical Engineering, University of Maryland.
- [6] A. Z. van Zuijlen. D5.1 Development of fast-running FSI methods. October 2024. https://go-viking.eu/images/resources/GO_VIKING_D5_1_development_and_implementation_of_fastrunning_fsi_methods_v1.pdf.
- [7] A. Z. van Zuijlen. D5.2 Validation of fast-running FSI methods. 2026. <https://go-viking.eu/results>.
- [8] C. H. Blaesius. Simulation of flow induced vibrations in the ALAIN fuel assembly mock-up experiment. Proc. of the FIV2026 conference, Paris, France. July, 2026