



Final Workshop Proceedings

Deliverable 6.5

Version N°1
May / 2026



Kateryna PILIUHINA¹, Angel PAPUKCHIEV², Kevin ZWIJSEN³

¹ Kateryna PILIUHINA - European Nuclear Education Network

² Angel PAPUKCHIEV - GRS gGmbH

³ Kevin ZWIJSEN – NRG PALLAS



Funded by
the European Union

Disclaimer

Funded by the European Union. Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union or European Atomic Energy Community. Neither the European Union nor the granting authority can be held responsible for them.

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
Project starting date (duration)	1st June 2022 31st May 2026 (48 Months)
Related Work Package	WP6; WP1-5
Related Task(s)	Task 6.2; All research tasks
Lead Organisation	ENEN
Contributing Partner(s)	ENEN, VKI, GRS, NRG PALLAS, LGI
Due Date	30.05.2026
Submission Date	28.05.2026
Dissemination level	Public

History

Date	Version	Submitted by	Reviewed by	Comments
26.06.2026	1	Kateryna Piliuhina		

Table of contents

Contents

1. Introduction	10
1.1 Project summary	10
1.2 Purpose of this document	10
1.3 WP6 as a bridge between technical results and knowledge transfer.....	10
1.4 Scope and structure of the deliverable	11
1.5 Lessons learned	11
2. GO-VIKING Final Workshop	11
2.1 Workshop objectives.....	11
2.2 Workshop organisation and venue	12
2.3 Participants and target audience	12
2.4 Programme of the workshop	12
2.5 Overview of workshop sessions	12
3. Abstracts of workshop contributions	13
3.1 Day 1 May 20, 2026: Project Introduction and Results	13
3.1.1 Opening by Project Coordinator GO-VIKING overview objectives	13
3.1.2 Operational experience with FIV in NPPs	14
3.1.3 Results of the Stakeholders' Workshop and literature reviews	16
3.1.4 The ALAIN experiment and its analysis using different FSI approaches	17
3.1.5 Outcome of the ALAIN benchmark	21
3.1.6 Single- and two-phase Cantilever Rod experiments on FIV in axial flow.....	23
3.1.7 Results of the Cantilever Rod benchmark.....	24
3.1.8 High-resolution DNS simulations of the Cantilever Rod experiment	25
3.1.9 The AMOVI experiment on FIV in cross-flow	26
3.1.10 High-fidelity LES analysis of AMOVI	27
3.1.11 Validation of ANSYS CFX using the Rigid Body Motion approach on the AMOVI experiment.....	28
3.1.12 Comparison of different numerical approaches within the AMOVI benchmark 29	
3.1.13 Tube vibrations in the GOKSTAD experimental bundle	30
3.1.14 High-resolution GOKSTAD computations at VCU.....	32
3.1.15 The GOKSTAD benchmark: comparison of numerical results and data.....	33
3.2 Day 2 May 21, 2026: Project Results.....	34

3.2.1	Uncertainty quantification and Bayesian calibration for turbulence model parameters.....	34
3.2.2	Uncertainty quantification for flow-induced vibration using polynomial chaos methods	35
3.2.3	Simulations of the Cantilever Rod experiment	36
3.2.4	Results of the two-phase Cantilever Rod benchmark.....	38
3.2.5	The TITAN experiment: vibrations in a tube bundle exposed to a two-phase cross-flow	38
3.2.6	Results from the TITAN simulations at ASNR.....	39
3.2.7	The TITAN benchmark: presentation of the numerical results.....	40
3.2.8	TREFLE: Experimental setup and campaigns.....	41
3.2.9	Presentation of the TREFLE simulations at CEA	43
3.2.10	Main results from the TREFLE benchmark	44
3.2.11	Development and validation of the Projection-Based Reduced-Order Model for the analysis of FIV.....	45
3.2.12	Development and validation of the Synthetic Turbulence Model for FIV analysis	46
3.3	Day 3 May 22, 2026: Education and Best Practices	47
3.3.1	Education and dissemination activities.....	47
3.3.2	Best Practice Guidelines on the use of FSI methods for FIV evaluation	49
Annex 1 – Final Workshop Programme		50
Annex 2 – Workshop presentations		50

Abbreviations and Acronyms

Acronym	description
AB	Advisory Board
AIAD	Algebraic Interfacial Area Density
ALAIN	Framatome experimental test loop / benchmark used in GO-VIKING
AMOVl	Experimental benchmark on FIV in cross-flow
ANSYS CFX	ANSYS Computational Fluid Dynamics code
ASNR	French Authority for Nuclear Safety and Radiation Protection
BP3	Address reference used for ASNR Cadarache site
BPG	Best Practice Guidelines
CEA	Commissariat à l'énergie atomique et aux énergies alternatives
CFD	Computational Fluid Dynamics
CFD-CSM	Coupled Computational Fluid Dynamics – Computational Structural Mechanics
CFD-FSI	Coupled Computational Fluid Dynamics – Fluid-Structure Interaction
CFX	ANSYS CFX computational fluid dynamics solver
CNSC	Canadian Nuclear Safety Commission
CSM	Computational Structural Mechanics
DES	Detached Eddy Simulation
DNS	Direct Numerical Simulation
EDF	Électricité de France
EDF-E	EDF Energy
EBRSM	Elliptic Blending Reynolds Stress Model
EB	Elliptic Blending
ENEN	European Nuclear Education Network
EPRI	Electric Power Research Institute
EU	European Union
FA	Fuel Assembly
FDM	Finite Difference Method
FEI	Fluid-Elastic Instability
FIV	Flow-Induced Vibration
FRA	Framatome
FRA-G	Framatome GmbH
FSI	Fluid-Structure Interaction
GOKSTAD	Experimental benchmark / test facility used in GO-VIKING
GO-VIKING	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
GPU	Graphics Processing Unit
GPR	Gaussian Process Regression
GRS	Gesellschaft für Anlagen- und Reaktorsicherheit gGmbH
HRIC	High-Resolution Interface Capturing

IAEA	International Atomic Energy Agency
IATE	Interfacial Area Transport Equation
IPP	IPP Centre LLC
IRSN	Institut de radioprotection et de sûreté nucléaire
LDA	Laser Doppler Anemometry
LDV	Laser Doppler Velocimetry
LES	Large Eddy Simulation
LGI	LGI Sustainable Innovation
NPP	Nuclear Power Plant
NRC	Nuclear Regulatory Commission
NRG	Nuclear Research and Consultancy Group
NSSS	Nuclear Steam Supply System
PALLAS	PALLAS research reactor / NRG PALLAS organisation
PCE	Polynomial Chaos Expansion
PDF	Portable Document Format
PhD	Doctor of Philosophy
PIV	Particle Image Velocimetry
PSD	Power Spectral Density
PSU	Pennsylvania State University
PWR	Pressurised Water Reactor
RANS	Reynolds-Averaged Navier–Stokes
RMS	Root Mean Square
ROCOM	Rosendorf Coolant Mixing Test Facility
ROM	Reduced-Order Model
RPV	Reactor Pressure Vessel
SAS	Scale-Adaptive Simulation
SBM	Synthetic Bubble Model
SBES	Stress-Blended Eddy Simulation
SG	Steam Generator / Spacer Grid, depending on context
SGS	Sub-Grid Scale
SST	Shear Stress Transport
STAR-CCM+	Siemens CFD simulation software
STG	Synthetic Turbulence Generator
TKE	Turbulent Kinetic Energy
TITAN	Experimental benchmark on two-phase cross-flow-induced vibrations
TREFLE	Experimental setup / benchmark for two-phase flow simulations
TSO	Technical Safety Organisation
TUD	Delft University of Technology
TVO	Teollisuuden Voima Oyj
UK	United Kingdom
UMR	Unité Mixte de Recherche
UoM	University of Manchester
UQ	Uncertainty Quantification
U-RANS / URANS	Unsteady Reynolds-Averaged Navier–Stokes

USA	United States of America
VCU	Virginia Commonwealth University
VKI	von Karman Institute for Fluid Dynamics
VOF	Volume of Fluid
VTT	Technical Research Centre of Finland
WALE	Wall-Adapting Local Eddy-viscosity model
WFLES	Wall-Function Large Eddy Simulation
WMLES	Wall-Modelled Large Eddy Simulation
WP	Work Package

Summary

This deliverable D6.5, Final Workshop Proceedings, documents the outcomes of the GO-VIKING Final Workshop on Flow-Induced Vibrations, held on 20–22 May 2026 at GRS gGmbH in Garching near Munich, Germany. The workshop was organised as the final knowledge-transfer and dissemination event of the GO-VIKING project and brought together consortium partners, external experts, research organisations, universities, technical safety organisations and industrial stakeholders.

The Final Workshop provided a structured overview of the main scientific, technical and educational achievements of the project. The programme covered operational experience with FIV in nuclear power plants, stakeholder needs and literature review outcomes, experimental and numerical benchmarks for fuel assemblies and steam generator-related configurations, multiphase flow benchmarks, uncertainty quantification, fast-running methods, education and dissemination activities, and Best Practice Guidelines on the use of FSI methods for FIV evaluation.

WP6 played a cross-cutting role in the project. It did not operate as an isolated communication package, but built directly on the research results and technical achievements generated in all other technical work packages. The experimental results from WP2-WP4, the uncertainty quantification and fast-running methods developed in WP5, the stakeholder needs and literature review work of WP1, and the Best Practice Guidelines synthesised in WP5 were transformed through WP6 into educational material, lecture activities, dissemination products and the final international workshop.

This deliverable therefore has two complementary purposes. First, it preserves the content of the final workshop by compiling the programme, abstracts and presentation-based summaries. Second, it demonstrates how the workshop contributed to the wider dissemination and exploitation of GO-VIKING outcomes, ensuring that the project results remain accessible to the nuclear research, industrial, regulatory and education communities after the end of the project.

The workshop proceedings are intended to serve as a reusable reference document for stakeholders interested in FIV phenomena, FSI modelling, benchmark validation, uncertainty treatment, fast-running tools and practical guidance for nuclear applications. The report also supports the continuity of knowledge transfer to students, young professionals and

practitioners working on nuclear safety, reactor component integrity and advanced simulation methods.

Keywords

Flow-Induced Vibration; Fluid-Structure Interaction; Nuclear Safety; Computational Fluid Dynamics; Experimental Benchmarks; Uncertainty Quantification; Fast-Running Methods; Best Practice Guidelines; Education and Training; Knowledge Transfer.

1. Introduction

1.1 Project summary

GO-VIKING, Gathering expertise On Vibration ImpaKt In Nuclear power Generation, addresses flow-induced vibration phenomena relevant to nuclear power reactors. The project builds on the earlier VIKING initiative and brings together vendors, utilities, technical safety organisations, universities, research organisations and associated international partners to improve understanding and prediction of FIV phenomena in nuclear systems.

Flow-induced vibrations can affect the safe and reliable operation of key components of the nuclear steam supply system. They may contribute to wear, fatigue, component damage and operational limitations if not properly understood, predicted and addressed through design, analysis and operational measures. GO-VIKING therefore contributes to nuclear safety by improving experimental databases, numerical tools, benchmark exercises, uncertainty quantification approaches, fast-running methods and Best Practice Guidelines.

The technical work of the project covered several complementary areas: stakeholder needs and literature review, FIV in fuel assemblies, FIV in steam generators, FIV in multiphase flows, uncertainty quantification, fast-running methods and best practice guidance. The results of these work packages formed the scientific basis for the final workshop and for the knowledge-transfer activities documented in this deliverable.

1.2 Purpose of this document

The purpose of this document is to provide the proceedings of the GO-VIKING Final Workshop and to document the role of the workshop as the final dissemination and knowledge-transfer event of the project.

In line with the Grant Agreement, D6.5 includes proceedings of the presentations delivered during the workshop and a summary of the impact of communication and dissemination actions. The deliverable consolidates the technical contributions, abstracts, discussion points and educational/dissemination outputs of the event in a structured and reusable format.

The document is intended for the GO-VIKING consortium, the European Commission, nuclear stakeholders, researchers, students, young professionals and practitioners interested in FIV and FSI applications in nuclear systems.

1.3 WP6 as a bridge between technical results and knowledge transfer

WP6 Education, training and dissemination was designed as a cross-cutting work package supporting the transfer of GO-VIKING knowledge to the wider community. Its objectives included training of young scientists, master and PhD students on FSI related to nuclear applications, development of educational material for FIV, presentation and distribution of the Best Practice Guidelines developed in WP5, dedicated code training with code benchmarks, communication and dissemination activities, preparation of communication material, and organisation of an international event with key stakeholders.



A key feature of WP6 is that it built on all other technical work packages and their research results. WP6 did not generate technical content in isolation; instead, it translated the outcomes of WP1-WP5 into educational, training and dissemination outputs. This ensured that the scientific results of the project could be reused by students, young professionals, researchers, industry representatives, technical safety organisations and other stakeholders.

WP1 provided the stakeholder needs, literature review and context for the project. WP2 contributed experimental and numerical results on FIV in fuel assemblies. WP3 provided results related to steam generator configurations and associated benchmarks. WP4 contributed the multiphase-flow experimental and numerical basis. WP5 delivered methods for uncertainty quantification, fast-running approaches and Best Practice Guidelines. WP6 integrated these results into educational material, lectures, code benchmark lessons, dissemination activities and the Final Workshop proceedings.

The Final Workshop therefore represents the synthesis point of the project. It made the technical achievements of the project visible, allowed discussion of their practical significance, and supported the long-term availability of the results through proceedings, presentations and related communication material.

1.4 Scope and structure of the deliverable

The deliverable is structured around the workshop logic. Section 2 introduces the workshop objectives, organisation, venue, target audience and programme. Section 3 provides placeholders for the abstracts of all workshop contributions, using the titles and speakers listed in the final programme. Final Workshop is summarised in the Conclusions. Annexes are reserved for the Final Workshop programme and presentations.

1.5 Lessons learned

The Final Workshop created a direct link between the technical results of the project and the Best Practice Guidelines on the use of FSI methods for FIV evaluation, consolidated in GO-VIKING Deliverable D5.5 “BPGs on the use of FSI methods for stakeholders” and made available through the official GO-VIKING project website <https://go-viking.eu/>. The presentations and discussions provided important input for understanding modelling choices, validation evidence, uncertainty treatment, benchmark interpretation and practical limitations. These elements, together with the lessons learned captured in D5.5, are central for guidance aimed at vendors, operators, regulators and researchers.

2. GO-VIKING Final Workshop

2.1 Workshop objectives

The Final Workshop was the concluding international event of the GO-VIKING project. It aimed to present, discuss and preserve the main outputs of the project and to make them accessible for future use by the nuclear community.

- present the overall achievements of GO-VIKING and the project-level outcomes;

- share experimental results and numerical benchmark outcomes generated in the technical work packages;
- discuss the relevance of the results for FIV prediction, design evaluation and nuclear safety;
- exchange experience on operational FIV issues in nuclear power plants;
- present uncertainty quantification and fast-running approaches for FIV analysis;
- disseminate education, training and communication achievements of the project;
- present and discuss Best Practice Guidelines on the use of FSI methods for FIV evaluation;
- identify lessons learned and possible future collaboration opportunities.

2.2 Workshop organisation and venue

The workshop was held on 20–22 May 2026 at Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Boltzmannstr. 14, 85748 Garching near Munich, Germany. The programme was organised over three days in a hybrid mode. Day 1 focused on project introduction, operational experience, WP1 outcomes, fuel assembly configurations and steam generator-related FIV. Day 2 addressed uncertainty quantification, multiphase flow benchmarks and fast-running methods. Day 3 focused on education, dissemination, Best Practice Guidelines and future collaboration.

2.3 Participants and target audience

The workshop targeted GO-VIKING partners, members of the Advisory and User Group, nuclear industry representatives, technical safety organisations, research organisations, universities, students, young professionals and other external experts. The programme was designed to support both expert-level technical exchange and wider knowledge transfer to communities that can use the results after the end of the project.

2.4 Programme of the workshop

The full workshop programme is provided in Annex 1. The programme combined open lectures, project-result sessions, benchmark presentations, technical discussions, education and dissemination sessions, and closing reflections on key outcomes and future collaboration opportunities.

2.5 Overview of workshop sessions

The workshop sessions followed the technical progression of the project. The first sessions placed the project in context, presented operational experience and introduced stakeholder requirements. The following sessions moved through the main technical domains: FIV in fuel assemblies, steam generator configurations, uncertainty quantification, multiphase flow, fast-running methods, education and Best Practice Guidelines. This structure allowed the participants to see how individual technical results contribute to a broader project-level understanding of FIV phenomena and their modelling in nuclear applications.

3. Abstracts of workshop contributions

This section provides the abstracts of all workshop contributions. The titles and speakers are corresponding the Final Workshop programme. Presentations are provided in Annex 2.

3.1 Day 1 May 20, 2026: Project Introduction and Results

3.1.1 Opening by Project Coordinator GO-VIKING overview objectives

Speaker(s): GRS / NRG PALLAS

Session: Welcome & Project Overview

Overview of the Horizon Europe Gathering expertise On Vibration ImpaKt In Nuclear power Generation (GO-VIKING) project

Angel Papukchiev¹, Kevin Zwijsen²

*¹ Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Boltzmannstraße 14,
85748 Garching near Munich, Germany*

² NRG PALLAS B.V., Westerduinweg 3, 1755 LE Petten, The Netherlands

Within the European GO-VIKING (Gathering expertise On Vibration ImpaKt In Nuclear power Generation) project, experimental and numerical efforts are performed to improve the understanding of FIV and force measurements in reactor cores and steam generators, as well as to develop and validate advanced numerical approaches for their prediction. This presentation gives a brief overview on project objectives, structure, timeline and concept. It also provides some background information on FIV in nuclear power reactors.

3.1.2 Operational experience with FIV in NPPs

Speaker(s): P. Moussou / P. Veber, EDF / Vattenfall

Session: Open Lecture on FIV

Operational experience with FIV in NPPs

Pascal Veber

Vattenfall AB, SE-169 92 Stockholm, Sweden

Fluid structure interaction (FSI) in reactor vessel and internals are common and can cause excessive force or stress to the structures resulting in mechanical damage that may threaten the structural integrity of the vessel and internals and thus the safety of the nuclear power plants.

FSI problems have resulted in severe vibrations, noise, wear and fatigue and failure of components. To reduce the occurrence of FSI damages and risk for unscheduled shutdown, experiences on previous problems have been gathered and descriptions of the possible issues due to FSI have been presented.

Flow induced vibration (FIV) in nuclear reactor vessels has been extensively studied during the mechanical design of reactor vessel. Thus, the power upgrade that many power plants have undergone has resulted in changes of operation parameter as mass flow, temperature, dimension that may induce FIV problems.

The study on these upcoming problems during power upgrade has increased the interest on FIV computational models as modern fluid and solid mechanics codes can be coupled, which has facilitated the understanding of the phenomena causing damage on components in nuclear power plants.

In nuclear power plants, the four dominant vibration-excitation mechanisms are: (1) fluid-elastic instability; (2) periodic wake shedding; (3) turbulence-induced excitation; and (4) acoustic resonance.

Computational simulations of vibrations caused by fluid-elastic instability, turbulence-induced excitation, periodic wake shedding and acoustic resonance are presented in the following chapters and are shortly described below.

- Turbulence-induced excitation of the core barrel has been simulated to determine the displacement of the core barrel due to the non-asymmetric flow field in the downcomer.
- Concerning fluid-elastic instability, simulation of an experiment designed for FSI codes validation has been performed. A slender structure in axial flow representing a simplified geometry of the in-core neutron flux detector system guide tubes, located in between the fuel bundles, is used to simulate flow induced vibrations.
- Following a power upgrade and increase of velocity in the steam line, vortex shedding frequency at a T-junction and coupling to piping acoustic resonance was analysed.

The above phenomena are included in the IAEA guidelines of FSI as vortex induced vibrations, forced vibrations (turbulence-induced) and self-excited vibrations.

Operational experience with FIV in NPPs

Pierre Moussou

Électricité de France (EDF) R&D, EDF Lab Chatou, 6 quai Watier, 78400 Chatou, France

Flow-induced vibrations (FIV) remain a major concern in nuclear power plant (NPP) components, particularly when associated with vortex shedding and acoustic coupling mechanisms. This work presents an overview of vortex-shedding-induced instabilities observed in industrial configurations, with a focus on orifices, valves, and cavity flows. Field observations from EDF and international partners highlight that such phenomena can generate strong tonal noise, high vibration levels, and, in some cases, significant structural damage.

Several representative configurations are discussed, including multi-stage orifice assemblies, gate valves exhibiting shallow cavity effects, and branch connections where deep cavity interactions occur. These cases demonstrate the importance of vortex shedding interactions with the acoustic modes of the system, sometimes leading to lock-in conditions. Experimental evidence shows that this type of instability is typically confined to specific velocity ranges and is characterized by sharp spectral peaks and harmonics, often associated with Strouhal numbers in the range 0.2–0.5.

A simplified theoretical framework is proposed to interpret these phenomena using a feedback-loop analogy between an acoustic amplifier and a reverberating system. The Nyquist stability criterion is used to identify the conditions for self-sustained oscillations. This approach successfully reproduces key experimental features, including the existence of multiple instability branches, their dependence on flow velocity, and the dominant role of the acoustic field.

Finally, modeling strategies are discussed. While simplified approaches based on U-RANS and reduced-order models with unsteady inlet conditions are effective for identifying instability onset, fully coupled simulations remain challenging due to their computational cost and sensitivity to boundary conditions.

3.1.3 Results of the Stakeholders' Workshop and literature reviews

Speaker(s): D. Vivaldi, ASNR

Session: Outcome of WP1

Results of the Stakeholders' Workshop and literature reviews

Daniele Vivaldi¹, Sofiane Benhamadouche²

*¹ Autorité de sûreté nucléaire et de radioprotection (ASNR), 31 avenue de la Division
Leclerc, 92260 Fontenay-aux-Roses Cedex, France*

² Électricité de France (EDF) R&D, EDF Lab Chatou, 6 quai Watier, 78400 Chatou, France

One of the GO-VIKING WP1 was to identify industry needs and regulatory expectations in terms of tools and methods for the analysis of FIV phenomena in NPPs. To do that, a stakeholders' workshop was organized. The workshop took place on the 16th February 2023 at EDF Lab Chatou. The following utilities accepted to contribute: EDF (France), EDF UK (UK), TVO (Finland), Vattenfall and Uniper Energy (Sweden, involving 3 nuclear power plants), EPRI (USA). The following vendors accepted to contribute: Framatome (France-Germany) and Westinghouse (USA). The following safety authorities and TSOs accepted to contribute: CNSC (Canada), GRS (Germany), IRSN (France), NRC (USA). A state of the art of numerical methods for FIV in fuel assemblies and steam generators was provided, together with the related industry needs and expectations from GO-VIKING. FIV issues in different RPV internals and in steam/water lines were presented. Finally, numerical tools requirements and examples of FIV analysis were provided by regulatory.

3.1.4 The ALAIN experiment and its analysis using different FSI approaches

Speaker(s): H. Hadzic / J. Herb / Ch. Bläsius / Y. Dubyk, FRA / GRS / IPP

Session: Session 1 FIV in FA (WP2)

ALAIN experiment and its analysis using different FSI approaches Experimental studies and numerical simulations by Framatome

Hidajet Hadžić

Framatome GmbH, Paul-Gossen-Straße 100, 91052 Erlangen, Germany

To study the vibration behavior of pressurized water reactor (PWR) fuel assemblies subjected to axial single-phase flow, Framatome performed extensive experimental investigations on reduced-scale fuel assemblies. Experiments were carried out at the technical center of Framatome GmbH in the mini test loop “ALAIN” with a test channel designed to accommodate a 5x5 rod bundle configuration. The objectives of the experimental research were a quantitative evaluation of natural and flow-induced vibrations of reduced-scale fuel assemblies equipped with a specific spacer grid (SG) design and determination of the fluid flow behavior within the fuel assembly in presence of the vibrating solid structures. One set of the experimental data related to a specific fuel assembly design and one specific flow rate used in the tests for which flow-induced vibration (FIV) and flow field measurements were performed was made available for the GO-VIKING project. This test setup and experimental data provided built the basis of the ALAIN Benchmark that was analyzed in the frame of the Work Package 2 (WP2) of the GO-VIKING project. Presentation gives an overview of the experimental setup, type of tests and data provided as well as a brief overview of the numerical simulation results achieved by Framatome in the frame of the ALAIN Benchmark.

The test loop is operated as a low-pressure loop with operating conditions corresponding to 4 bar pressure and 40°C temperature with water as working fluid. The test section consists of a squared flow channel made of Plexiglas designed to accommodate a 5x5 rod bundle. The main component of the test setup is the 2.3 m long reduced-scale fuel assembly equipped with 5 spacer grids and a lower and upper tie plate which are utilized to fix the fuel assembly inside the test channel. Between test channel and fuel assembly (spacer grids) there is a gap of 1.5 mm at each side providing space for fuel assembly vibration. In the test campaign three type of tests have been performed: mechanical impact tests to determine the natural frequencies of the fuel assembly, FIV tests to measure the vibration response induced by the flow and LDA measurements of the flow field. For mechanical impact tests and FIV tests the lateral displacements of the fuel assembly were measured for spacer grids 2, 3 and 4. For the flow field measurements were performed along two traverses across the

test channel in the middle of subchannels at the two sides of the central rod row. Flow field profiles were measured at five axial locations behind the second spacer grid. The measurements provide the data for mean flow and for the flow fluctuations of axial and cross flow velocity components. For the cross flow only one component was measured which is aligned with the orientation of the mixing vanes. The results from impact tests are presented in form of frequency spectra showing the natural frequencies of the fuel assembly. The results from FIV tests are presented in form of power density spectra (PSD) showing the characteristic of the vibration response of the fuel assembly to flow-induced excitation. The results of flow field LDA measurements are presented in form of profiles at corresponding locations. Finally, the objectives of the benchmark are outlined with reproduction of the flow field profiles and vibration response of fuel assembly from the FIV test being specified as the main targets.

At the end a brief overview of the numerical simulations results achieved by Framatome is given. First an overview of the CFD models based on a reduced-scale single-span and full-scale full fuel assembly configurations is given with the main outcome from the CFD simulations performed on single-span configuration. The CFD simulations on single-span configuration were performed with two scale-resolving turbulence modeling approaches, large eddy simulation (LES) and detached eddy simulation (DES) in combination with different sub-grid scale (SGS) i.e. turbulence models. For LES two SG models were used: Smagorinsky and wall-adapting local eddy (WALE) model, while DES was used with three turbulence models: Spalart-Almaras, elliptic blending (EB) and shear stress transport (SST) model. Based on the comparison with experimental flow field profiles the LES with WALE SGS model was selected as preferred model that predict best both mean and fluctuation flow. The LES with WALE model was then used in the final CFD simulation on full-scale geometry. The main results of the full-scale CFD simulation are shown in terms of fluid forces which are used in the subsequent mechanical simulation aimed at calculation of FIV. Finally, the mechanical model is shown together with the main results from the mechanical simulation. The mechanical model is based on the Euler-Bernoulli beam with additional rotational stiffness being added at the nodes corresponding to spacer grid positions to consider the effect of the clamping of the fuel rods in spacer grids. The effects of the fluid on the dynamic behavior of the fuel assembly submerged in water flow are evaluated from a small-scale FSI model reflecting a span of the fuel assembly. The added mass and fluid damping parameters derived from these simulations are implemented in the mechanical model. The results of mechanical simulation are shown in terms of PSD of the displacements at spacer grids 2, 3 and 4 and in terms of RMS amplitudes. Comparison with experimental results show a reasonable agreement.

Simulation of flow induced vibrations in the ALAIN fuel assembly mock-up experiment

Christoph Bläsius¹, Joachim Herb²

¹ *Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Schwertnergasse 1, 50667 Cologne, Germany*

² *Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Boltzmannstraße 14, 85748 Garching near Munich, Germany*

The prediction of flow-induced vibrations of fuel assemblies is an important contribution to maintaining a high safety standard of nuclear power plants. This paper describes a numerical simulation of Framatome's ALAIN experiment of a 5x5 fuel assembly mock-up exposed to axial fluid flow, which was performed by GRS as part of the European GO-VIKING project using a simulation chain of the CFD code OpenFOAM and the CSM code ANSYS Mechanical. The flow within a sub-section was simulated transiently using periodic boundary conditions. The applicability of different turbulence models (two-equations models and a coarse-mesh Large-Eddy Simulation (LES)) model was analyzed. The transient CFD results of the forces on the spacer grids of the mock-up sub-section were split into shorter intervals, randomly recomposed and applied to the mechanical model of the full mock-up. This mechanical model consisted of a beam with additional springs that was calibrated against measured natural frequencies and mode shapes. Different fluid damping factors were used based on data from similar experiments on shaker table tests. The analyses proved that the simulation chain was able to capture the main vibration phenomena observed in ALAIN. For the simulations using the coarse-mesh LES turbulence model, the power spectral density of the simulated vibrations revealed good agreement with the experiment for the mean displacement amplitude and the lower natural frequencies.

Flow-Induced Vibration Analysis of the ALAIN 5x5 Fuel Assembly: A Coupled CFD–Structural Approach

*Vladislav Filonov, Yuliia Filonova and Yaroslav Dubyk
IPP Centre LLC, 8 Bolsunovska Street, 01014 Kyiv, Ukraine*

Flow-induced vibrations (FIV) in nuclear fuel assemblies represent a critical factor in reactor safety and operational reliability. Turbulent coolant flow past closely-packed rod bundles generates non-stationary hydrodynamic forces that can drive structural vibrations, with amplitudes ranging from negligible to potentially damaging depending on the excitation mechanism. This work presents the approach developed by IPP-CENTRE within WP2 of the GO-VIKING project, focused on the ALAIN 5x5 PWR fuel assembly benchmark.

For the structural model, a 3-Degree-of-Freedom (3-DOF) lumped-parameter system was adopted, with measurement nodes at the centres of the three intermediate spacer grids. The physical mass, stiffness, and damping matrices were reconstructed from experimental impact-test data using inverse modal identification. Three matrix reconstruction methods were implemented and compared: Singular Value Decomposition (SVD), Direct Least

Squares (Equation Error), and the Berman–Baruch finite element model updating technique. Natural frequencies of {6.0, 14.4, 23.9} Hz, along with corresponding mode shapes and damping ratios, were used as input. Modal Assurance Criterion (MAC) checks confirmed the fidelity of the reconstructed models. To account for fluid–structure interaction effects, hydrodynamic added mass and damping coefficients were computed for each bending mode by solving the time-harmonic Stokes equations on the 5×5 rod bundle cross-section using the FreeFEM++ finite element solver. Modal projection over the rod length yielded generalised hydrodynamic coefficients that were superimposed onto the structural matrices.

Coupling between the CFD solver and the structural model was implemented via a custom Fortran library using the User Junction Box interface. The library reconstructs the assembly displacement profile as a piecewise-linear function between the three reference grid locations, computes hydrodynamic forces from the CFD solution at each time step, and advances the structural equations of motion using either the Duhamel integral or the Newmark method. An explicit coupling scheme was adopted for all production calculations.

Transient FIV simulations were performed using both SST and SAS-SST turbulence models. Results demonstrate a fundamental difference in predicted behaviour: SST fully suppresses assembly vibrations by rapidly damping transient perturbations, while SAS-SST sustains persistent oscillations throughout the simulation — in qualitative agreement with the experimental observations. However, calculated RMS velocity fluctuations are approximately 35–60% below experimental values across all calculation variants, indicating a systematic underestimation of turbulent pulsation intensity, particularly in the near-grid wake region. Consequently, predicted vibration amplitudes and power spectral density are significantly lower than measured, although the low-frequency spectral character is qualitatively reproduced.

These results confirm that turbulent pulsations behind spacer grids are the dominant excitation mechanism for fuel assembly vibrations in this configuration, and that standard URANS approaches are insufficient to quantitatively reproduce them. From a scientific standpoint, accurate prediction would require Scale-Resolving Simulation (SRS) methods with approximately 10^8 mesh cells and low-dissipation advection schemes. For engineering applications, it is proposed to supplement URANS/SAS models with empirical or synthetic turbulence inputs — such as geometry-corrected Vortex Confinement — to achieve acceptable estimates of vibration amplitude within practical computational times.

3.1.5 Outcome of the ALAIN benchmark

Speaker(s): H. Hadzic, FRA

Session: Session 1 FIV in FA (WP2)

ALAIN Benchmark Outcome

Hidajet Hadžić

Framatome GmbH, Paul-Gossen-Straße 100, 91052 Erlangen, Germany

The outcome of the ALAIN Benchmark is presented. In the framework of the benchmark numerical simulations have been performed to analyze flow-induced vibration (FIV) of a reduced-scale 5x5 fuel assembly subjected to axial single-phase flow. The benchmark is based on the experimental setup and experimental results provided by Framatome. The experimental data include the results from FIV tests describing the vibration response of the fuel assembly and the results from flow field LDA measurements in terms of flow velocity profiles at several locations. The objectives of the benchmark are to reproduce the flow field profiles and vibration response of fuel assembly from the flow field measurements and FIV test. The benchmark setup consists of a squared flow channel and reduced-scale 5x5 fuel assembly placed inside the flow channel. The fuel assembly is 2.3 m long and equipped with 5 spacer grids and a lower and upper tie plate which are utilized to fix the fuel assembly inside the test channel. Between test channel and fuel assembly (spacer grids) there is a gap of 1.5 mm at each side providing space for fuel assembly vibration. For FIV tests the lateral displacements of the fuel assembly were measured for spacer grids 2, 3 and 4 and the flow field measurements were performed along two traverses across the test channel in the middle of subchannels at the two sides of the central rod row. Flow field profiles were measured at five axial locations behind the second spacer grid. The measurements provide the data for mean flow and for the flow fluctuations of axial and cross flow velocity components. For the cross flow only one component was measured which is aligned with the orientation of the mixing vanes.

Four organizations participated to the benchmark: FRA-G (Framatome GmbH), CEA (Commissariat à l'énergie atomique et aux énergies alternatives), GRS (Gesellschaft für Anlagen- und Reaktorsicherheit gGmbH), IPP (IPP Centre LLC). All participants contributed with numerous of different numerical simulations, all of them following a general approach consisting of the use of computational fluid dynamics (CFD) for fluid flow analysis and use of a mechanical model of fuel assembly for structural dynamic analysis. CFD simulations are performed using two geometrical configurations: full-scale model (whole setup) and single-span model (only one span modeled). For mechanical modeling all participants used an approach based on a reduced representation of the fuel assembly, such as beam elements. In terms of prediction of FIV two approaches were followed: decoupled approach in which CFD and mechanical simulations are performed separately and coupled FSI in which CFD and mechanical simulations are performed simultaneously.

The case considered in this benchmark is based on a reduced scale 5x5 fuel assembly. From the point of view of a real world this fuel assembly is quite small (only a half of the length and less than one third of the width of a real full-scale fuel assembly). Nevertheless, this configuration is still very challenging for numerical simulation, especially for CFD modeling which requires significant computational resources even for a moderate resolution. The experimental results from FIV tests show that the vibration amplitude was just a few μm . Such small vibration amplitude is typical for turbulence-induced vibration which is assumed as the driving FIV mechanism in this case. These aspects played an important role in the selection of the simulation strategy.

For CFD simulations following codes have been used: STAR-CCM+ (by FRA), TrioCFD (by CEA), OpenFoam (by GRS) and ANSYS CFX (by IPP). Simulations are performed for single-span and /or full-scale configuration. In the CFD simulations different turbulence modeling approaches were used. These include large eddy simulation (LES), detached eddy simulation (DES), scale adaptive simulation (SAS) and Reynolds averaged Navier Stokes (RANS) equations in combination with different turbulence or sub-grid scale models. For mechanical modeling the option used by FRA and GRS is based on a beam model with additional elements added at the spacer grid locations that increase the rotational stiffness thus simulating the effect of furl rod clamping in the spacer grids. On the other hand, IPP used 3 degree of freedom model.

The comparison of the flow velocity profiles obtained by simulations by all participants with the experimental results shows generally a reasonable agreement with some larger or smaller local deviations which depend on the model and resolution used in corresponding simulations. The profiles obtained by SAS model show certain level of damping in both mean flow and flow fluctuations compared to other results obtained by LES. This may be attributed to the setting of the model used in the simulations that may not be optimized for present application.

The vibration response obtained by simulations shows in general a reasonable agreement with experimental results. The results obtained by FRA generally overestimate the vibration amplitude by 10-50% except for spacer grid position 4 in y direction (parallel to orientation of mixing vanes). The results obtained by GRS are in the range or underestimate the vibration amplitude depending on the level of fluid damping implemented in the model. In the case of results provided by IPP the vibration amplitude is generally underestimated which is most likely attributed to the turbulence model used in the simulation that shows significant damping. It should be noted that FIV results provided by FRA and GRS are obtained from a decoupled simulation and IPP performed coupled FSI simulations.

3.1.6 Single- and two-phase Cantilever Rod experiments on FIV in axial flow

Speaker(s): M. Nabawy, UoM

Session: Session 1 FIV in FA (WP2)

Single- and two-phase cantilever rod experiments on FIV in axial flow

Mostafa R A Nabawy and Hector Iacovides

Department of Mechanical and Aerospace Engineering, School of Engineering, The University of Manchester, Oxford Road, Manchester M13 9PL, United Kingdom

The flow-induced vibration (FIV) of a cantilever rod confined in a pipe and exposed to an axial water flow has been experimentally investigated using a simple test piece designed to be informative for water-cooled nuclear reactor fuel rods. The first set of experiments were for a rod with a semi-spherical tip end shape exposed to single-phase water flow in both clamped free and free-clamped configurations, representing opposite flow directions. The Reynolds number was varied between 7.43k and 82.5k: a range that significantly extends the scope of previously available data on axial-flow-induced vibration of cantilever rods. Measurements were carried out using non-contact techniques: fast video imaging was used to track the rod motion, whilst particle image velocimetry was employed to resolve the flow field. The sources of excitation were found to include turbulent buffeting, movement-induced excitation, and flow separation at the rod tip, whilst the observed dynamic responses of the rod comprised a fuzzy period-1 motion, a flutter-like large-amplitude oscillation, and deflection. Since both the rod vibration and the flow field were measured, the generated data are particularly suited for numerical methodologies development and code validation.

Next, the influence of rod tip shape on FIV of the cantilever rod subject to axial water flow was experimentally investigated using the same experimental techniques. The rod tip shapes considered included a blunt tip and cones with height-to-diameter ratios of 0.5, 1, and 2. The experiments were conducted across a Reynolds number range of 20k to 100k. The rod vibration dynamics exhibited a primarily fuzzy period-1 behavior, characterized by a periodic motion with a chaotic component. Flutter-like oscillation and buckling were also observed at higher Reynolds numbers, depending on the flow direction. The mechanisms of fluid-structure interaction involved turbulent buffeting and movement-induced excitations. Unsteady flow separation around the rod tip was identified as a further contributing mechanism to flow excitation. In the clamped-free configuration, unsteady flow separation was more pronounced for the cone tips due to the increased rod surface area in the wake region, leading to larger vibration amplitude. In the free-clamped configuration, flow separation effects were more prominent for the blunt tip, as the

streamlined cone shapes were less prone to flow separations. Overall, the rod with the blunt tip resulted in smaller displacement in the clamped free configuration, while the rods with the cone tips led to smaller displacement in the free clamped configuration.

Finally, the experiments were conducted in two-phase air-water mixtures. For such gas liquid flows, video imaging becomes ineffective; hence, fundamental mechanistic understanding and predictive modelling of FIVs remained hindered by the lack of non-intrusive diagnostic tools. As such, a Hall-effect-based electromagnetic sensing technique was introduced, for the first time, that enabled comprehensive resolution of the axial-FIV dynamics of a cantilevered rod with different tip geometries over a range of air-water flow regimes. Experimental results revealed that increasing the void fraction amplifies chaotic vibrations while suppressing periodic oscillations, a transition driven by the increased intensity of stochastic-forcing induced by gas-liquid interactions and bubble impacts. As such, a dual regime response emerges where vibration amplitudes increase at low Reynolds numbers but decrease/plateau at high Reynolds numbers. Strikingly, beyond a critical void fraction of 0.2, amplitudes converge across Reynolds numbers, signalling two-phase stochastic force dominance. These findings elucidated the mechanistic competition between stochastic and periodic excitations in two-phase axial-FIVs with a simplified paradigmatic configuration that provides valuable preliminary information for nuclear reactor applications. However, the developed novel technique also provides an enabling tool for real-time, non-intrusive FIV diagnostics, with potential applications extending beyond nuclear engineering.

3.1.7 Results of the Cantilever Rod benchmark

Speaker(s): M. Hussain, NRG PALLAS

Session: Session 1 FIV in FA (WP2)

Cantilever Rod: Benchmark Results

Mohammed Muaaz M. D. Hussain

NRG PALLAS B.V., Westerduinweg 3, 1755 LE Petten, The Netherlands

The GO-VIKING deliverable D2.2 and its corresponding workshop presentation is concerned with providing the results of the numerical simulations for the Benchmark 1 of Work Package 2 (WP2) (Cantilever Rod). This involves making predictions of the Flow Induced Vibrations (FIV) of a free-clamped rod exposed to axial water flow. The experimental facility is that of the University of Manchester, which has been described in detail along with obtained experimental data in the GO-VIKING deliverable D2.1. The area of interest for the numerical simulations is the test section which comprises of a vibrating rod confined inside a tube.

Five contributing partners: PSU, EDF, NRG PALLAS, EDF-Energy and IPP, performed numerical simulations of this test section for rods having different end shapes such as hemispherical end and the blunt end, and subjected to axial flow at different annular Reynolds numbers. One of these simulation cases: rod with hemispherical end subjected to $Re=21.2k$, has been selected as a benchmark case and is used to compare the setup, computational costs and results of the involved participants.

Key results from the benchmark indicate that the choice of turbulence model for the fluid has an immense impact on the accuracy and involved computational cost of the simulations. Results also indicate that simplifications can also be made with little-to-no impact on the solution accuracy while reducing computational costs, either by virtue of simplifying the physics (beam model vs full CSM for the rod) or by making simplifications to the geometry and reducing the mesh requirement (single inlet-single outlet configuration, no explicit modelling of lead shots inside the rod).

3.1.8 High-resolution DNS simulations of the Cantilever Rod experiment

Speaker(s): E. Merzari, PSU

Session: Session 1 FIV in FA (WP2)

High-Fidelity Simulations of Cantilever Beam Experiment

Elia Merzari, Sean Bistany

Ken and Mary Alice Lindquist Department of Nuclear Engineering, The Pennsylvania State University, 206 Hallowell Building, University Park, PA 16802-4400, United States

This work presents high-fidelity direct numerical simulations (DNS) of a cantilever beam experiment relevant to fluid-structure interactions and turbulence phenomena in nuclear engineering applications. Simulations were performed using GPU-enabled exascale computing resources to investigate turbulent flow behavior in the complex transition region between pipe and annular geometries at Reynolds numbers up to approximately $\sim 21,000$. The computational campaign employed high-order spectral element methods with meshes exceeding 1.35 billion grid points, enabling fully resolved turbulence statistics and detailed analysis of flow acceleration effects near the beam tip.

The study includes extensive verification and validation activities through comparisons against experimental particle image velocimetry (PIV) measurements, velocity profiles, and turbulent kinetic energy (TKE) statistics. Excellent agreement between simulations and experiments was observed for both mean flow fields and turbulence quantities. Detailed analyses of TKE production and dissipation budgets revealed the presence of negative turbulence production in the transition region caused by strong flow acceleration effects. The simulations further highlighted complex anisotropic turbulence structures and the

emergence of pronounced two-component turbulence behavior near the beam-end geometry.

Time-series analyses and power spectral density evaluations were also performed to characterize the unsteady turbulent dynamics. Additional studies examined the influence of beam-end geometry and simplified fluid-structure interaction effects, demonstrating limited impact of prescribed oscillatory motion on first- and second-order turbulence statistics. The results provide valuable insight into turbulence behavior in accelerated annular flows and demonstrate the capability of modern exascale DNS methodologies to resolve highly complex thermal-hydraulic phenomena relevant to advanced reactor systems and flow-induced vibration problems.

3.1.9 The AMOVI experiment on FIV in cross-flow

Speaker(s): D. Panunzio, CEA

Session: Session 2 SG (WP3)

The AMOVI experiment on FIV in cross-flow

Domenico Panunzio

*Commissariat à l'énergie atomique et aux énergies alternatives (CEA), CEA Saclay, 91191
Gif-sur-Yvette Cedex, France*

The GO-VIKING project investigates flow-induced vibration (FIV) phenomena in nuclear reactor components under single- and two-phase flow conditions. Within this framework, Work Package 3 (WP3) focuses on FIV occurring in steam generator (SG) tube bundles subjected to single-phase cross-flow. This study presents experimental results obtained at CEA Saclay using the AMOVI facility, a dedicated hydraulic loop designed to analyse fluid-structure interactions in tube bundles at intermediate Reynolds numbers ($Re_{max} \sim 23000$).

The test section consists of a square tube bundle composed of 3×5 tubes supplemented by two columns of semi-tubes. The tubes have a diameter of approximately 12.5 mm, a length of 99 mm, and a pitch-to-diameter ratio of 1.44. The central tube, made of stainless steel, can be configured either as rigid—allowing force measurements via a tri-axial sensor—or flexible—enabling displacement measurements through a strain-gauge instrumented support. The surrounding plexiglas tubes are rigid and modular, allowing different geometric configurations.

Two main configurations were investigated: a single tube in a reduced channel and a full tube bundle. For each configuration, both rigid and flexible setups of the central tube were tested, resulting in four experimental cases. Experiments were conducted at four flow rates ranging from 0.5 to 4 L/s. Force and displacement signals were analysed in terms of root mean square (RMS) and power spectral density (PSD).

Results for the single rigid tube show a clear vortex shedding signature in the lift force spectra, with a reduced frequency around 0.25, and of increasing magnitude with flow velocity. In contrast, the drag component does not exhibit a dominant frequency. In the full bundle configuration, force spectra are smoother and lack distinct peaks, indicating a more complex and distributed flow interaction. In all cases, RMS force levels increase with flow rate, with lift forces consistently exceeding drag forces.

For the flexible tube, displacement spectra reveal two dominant frequencies: the vortex shedding frequency and the natural frequency of the structure (approximately 23 Hz in water). At high flow rates, these frequencies approach each other, leading to a resonance and significantly amplified vibrations. In the bundle configuration, vibration amplitudes increase sharply beyond the critical flow rate, eventually causing tube-to-tube shocks.

Flow characterization using Particle Image Velocimetry (PIV) highlights flow acceleration within inter-tube gaps and the formation of jet-like structures, with acceleration factors consistent with geometric constraints. Turbulence is particularly intense in the wake region and near tube surfaces, where localized shear stress peaks are observed. Laser Doppler Velocimetry (LDV) measurements upstream reveal non-uniform velocity profiles and significant turbulence levels, attributed to large-scale three-dimensional flow structures.

Overall, this study provides a comprehensive experimental dataset on FIV mechanisms in simplified SG tube bundle configurations. The results contribute to a better understanding of flow–structure coupling, vortex-induced vibrations, and turbulence effects, and serve as a valuable benchmark for the validation of numerical models developed within the GO-VIKING project.

3.1.10 High-fidelity LES analysis of AMOVI

Speaker(s): S. Benhamadouche, EDF

Session: Session 2 SG (WP3)

High-fidelity LES analysis of AMOVI

Sofiane Benhamadouche

Électricité de France (EDF) R&D, EDF Lab Chatou, 6 quai Watier, 78400 Chatou, France

Within the European GO-VIKING (Gathering expertise On Vibration ImpaKt In Nuclear power Generation) project, experimental and numerical efforts are performed to improve the understanding of FIV and force measurements in reactor cores and steam generators, as well as to develop and validate advanced numerical approaches for their prediction. In GO-VIKING WP3 EDF performed high-fidelity LES analysis on the AMOVI experiment. The

measurements carried out in the CEA's facility were used for the validation of the code `code_saturne` to predict fluid forces and structural displacements in 5x5 tube bundle. Analyses were carried out for cases with rigid or flexible central tube in the AMOVI tube bundle. For the rigid tube bundle configuration, the results for the standard deviation and the spectra of the lift and drag forces compare with data better as the Reynolds number increases. They are however satisfactory and exhibit the right trends. It was found that multiple solutions are possible in the bundle and its wake. Further, the results obtained with wall-resolved LES for a tube bundle with a central flexible tube are overall satisfactory both for the displacements amplitudes and the frequencies. The configuration of the moving cylinder in a bundle at $Q = 1.95$ l/s has been successfully run with a coarsened mesh to allow larger displacements and to predict the observed fluid-elastic instability. A lock-in at $Q = 1.1$ l/s is found and the computation is also diverging at $Q = 1.05$ and 1.15 l/s. This has to be more deeply investigated.

3.1.11 Validation of ANSYS CFX using the Rigid Body Motion approach on the AMOVI experiment

Speaker(s): H. Mistry, GRS

Session: Session 2 SG (WP3)

Validation of ANSYS CFX using the Rigid Body Motion approach on the AMOVI experiment

Hemish Mistry, Angel Papukchiev

*Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Boltzmannstraße 14,
85748 Garching near Munich, Germany*

The integrity of the nuclear steam supply system (NSSS) is essential for the safe operation of a nuclear power plant (NPP). Key components such as fuel assemblies (FA) and steam generator (SG) tubes interact with cooling fluid and their solid structures (rods, tubes) in NPPs may lead to flow-induced vibrations (FIV), causing material fatigue, fretting wear, and eventually loss of component integrity. Thus, it is essential to investigate the fluid forces that act on the structural components and the resulting displacements. Within the European GO-VIKING (Gathering expertise On Vibration Impact In Nuclear power Generation) project, experimental and numerical efforts are performed to improve the understanding of FIV and force measurements in reactor cores and steam generators, as well as to develop and validate advanced numerical approaches for their prediction. Within the third work package of the project, GRS investigated fluid forces and vibration response of tubular structures exposed to a cross-flow. The measurements of CEA's AMOVI experiment were used for the validation of the code ANSYS CFX to predict fluid forces and structural displacements in two configurations: single tube and tube bundle. In this work,

we investigate different turbulence models for the simulation of the AMOVI experiment and compare the numerically calculated fluid forces and displacements with the experimental data. The comparison shows a good agreement in terms of the force magnitude, vortex shedding frequency and displacement amplitude for different flow rates.

3.1.12 Comparison of different numerical approaches within the AMOVI benchmark

Speaker(s): Y. Filonova, IPP

Session: Session 2 SG (WP3)

COMPARISON OF DIFFERENT NUMERICAL APPROACHES WITHIN THE AMOVI BENCHMARK

*Vladislav Filonov¹, Yuliia Filonova¹, Yaroslav Dubyk¹, Hemish Mistry², Angel Papukchiev²,
Sofiane Benhamadouche³*

¹ IPP Centre LLC, 8 Bolsunovska Street, 01014 Kyiv, Ukraine

*² Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Boltzmannstraße 14,
85748 Garching near Munich, Germany*

³ Électricité de France (EDF) R&D, EDF Lab Chatou, 6 quai Watier, 78400 Chatou, France

This work is dedicated to the development and investigation of methods and models to provide reliable assessment of structural flow induced vibrations (FIV), occurring in heat exchangers and steam generators under cross-flow conditions.

The development of numerical models is inseparably accompanied by their validation on existed experimental data. This work is concerned with providing the results of the numerical simulations for the AMOVI benchmark in order to investigate the applicability of the models and methods used by GRS, IPP and EDF for the analysis of cross-flow induced vibrations in bundles in the framework of GO-VIKING project.

High-resolution AMOVI simulations performed by EDF in Code_Saturne using a wall-resolved LES model as well as the medium-resolution simulations performed by GRS and IPP in ANSYS CFX with a 3D/2D formulation using hybrid (e.g. ZLES) and URANS turbulence models are described. Each partner aimed to simulate the time dependent vibration amplitudes and forces acting on the rods for AMOVI experimental cases with four different configurations: single rigid tube, rigid tube bundle, single flexible tube and flexible tube in a rigid tube bundle.

A detailed description of the models used by partners and an analysis of the results obtained are supplemented by a general analysis devoted to comparative benchmarking. The

predictive capabilities as well as the computational costs of the described models are compared within the framework of simulations of selected AMOVI experimental cases. Performed analysis with the comparison of approaches that are fundamentally different in their level of detail, allowed to formulate an understanding of their applicability to the modeling of FIV, both in terms of the reconstruction and analysis of the vortex structures formation and appropriate spectral characteristics of forces on the rods and as well as for the resulting displacements, thereby advancing the state-of-the-art of FIV analysis of complex heat transfer equipment for engineering purposes.

The provided comparison of computing resources needed to perform both simulations with rigid tube and flexible tube configurations allows to analyze and conclude not only the prediction abilities of different methods, but also their combination with computational costs, which is very useful in terms of understanding the necessary depth and objectives of analysis, which will differ when it comes to studying a phenomenon or engineering assessments of real equipment.

3.1.13 Tube vibrations in the GOKSTAD experimental bundle

Speaker(s): L. Koloszar, VKI

Session: Session 2 SG (WP3)

Tube vibrations in the GOKSTAD experimental bundle

Lilla Koloszar

von Karman Institute for Fluid Dynamics (VKI), Waterlooesteeweg 72, B-1640 Sint-Genesius-Rode, Belgium

The GOKSTAD facility was designed, built, and commissioned at the von Karman Institute within the GO-VIKING project to provide a high-quality experimental benchmark for the study of single phase cross flow and flow-induced vibration phenomena in tube bundle geometries. The facility consists of a fully closed loop water circuit with a 7×7 square rod-bundle test section, representative of configurations relevant to nuclear steam generator applications. The setup was developed to provide desired hydraulic conditions, optical access, and reliable measurements for both flow characterization and fluid–structure interaction studies.

The first part of the experimental campaign focused on fixed rod flow characterization using high resolution Particle Image Velocimetry (PIV). A dedicated optical setup was implemented, including a telecentric lens to minimize perspective distortion and improve measurement accuracy and accessibility of the measurement plane in the densely packed rod bundle. PIV measurements were performed in three regions of the facility: inlet region (upstream of the bundle), inside the rod bundle, and outlet region (downstream of the bundle). More than 40 experimental cases were carried out over a range of bulk velocities

and measurement planes. These measurements provided detailed information on inlet boundary conditions, internal bundle flow structures, turbulence levels, recirculation zones, jet development, and outlet flow behavior.

The PIV campaign showed that the inlet flow was generally repeatable but contained a slight persistent asymmetry, likely linked to upstream hydraulic effects. Within the bundle, recirculation regions were observed at low flow rates, while increasing velocity led to stronger crossflow and more distributed flow structures. At the outlet, the measurements captured the evolution of jet patterns and the collapse of central recirculation zones at intermediate velocities. Repeatability tests confirmed the robustness of the measurement procedure and the reliability of the generated data.

The second part of the work extended GOKSTAD from a fixed rod flow investigation to a vibration capable experimental setup. A new mechanical system was developed to allow selected rods to move under crossflow excitation while preserving the original hydraulic and optical characteristics of the facility. The movable rods were spring mounted and dynamically characterized before installation. In this part, three vibration configurations were investigated: a single vibrating rod, two vertically aligned vibrating rods, and two horizontally aligned vibrating rods.

High speed optical tracking was used to measure the motion of the vibrating rods. The recorded images were processed frame by frame using image processing techniques based on edge detection, circular Hough transform, and circle fitting to determine the instantaneous rod center position with subpixel accuracy. Additional optical markers placed on the rod faces were used to estimate rotational motion. Single and two camera arrangements enabled the extraction of time resolved rod displacements, vibration amplitudes, dominant frequencies, spanwise coherence, rocking motion, and rotational motion.

The results showed that the rod motion remained predominantly rigid and spanwise coherent, with very limited rocking and torsional deformation. Mean displacements increased with flow velocity, while vibration amplitudes remained relatively small compared with the rod diameter. Frequency analysis showed dominant vibration frequencies close to the expected structural response of the rod–spring system, with a possible local amplification near $U_b \approx 0.3 \text{ m/s}$, especially in the vertical configuration.

Overall, the resulting datasets, including processed PIV results and vibration measurements, provide valuable validation material for CFD and FSI models developed within the GO-VIKING framework. Moreover, the facility and data support improved understanding of flow-induced vibration mechanisms and contribute to the assessment of vibration related issues in nuclear related thermal-hydraulic systems.

3.1.14 High-resolution GOKSTAD computations at VCU

Speaker(s): L. Carasik, VCU

Session: Session 2 SG (WP3)

High-resolution GOKSTAD computations at VCU Reduced Domain and Full Experimental Domain Studies

Trevor Franklin and Lane Carasik, Ph.D.

Fluids in Advanced Systems and Technology (FAST) Research Group, Department of Mechanical and Nuclear Engineering, Virginia Commonwealth University, Engineering East Hall, 401 West Main Street, Richmond, VA 23284, United States

This presentation presents a comparison between LES performed using Nek5000 and experimental measurements obtained for the GOKSTAD tube bundle at the lowest flowrate 15.5 m^3/hr . Overall, good agreement is observed in the mean velocity field between the LES and the experiment results, both in terms of spatial distribution and magnitude. While the LES also captures the general spatial trends of TKE, it consistently overpredicts the magnitude throughout the bundle. Time series data of lift and drag forces were extracted on the center tube from the LES and as a input to a one-way spring-mass model to estimate the structural displacement response of the center tube. The resulting structural response shows good agreement with the experimental measurements in terms of mean displacement. However, the LES-based forcing leads to underprediction of the RMS displacement compared to the experiment. In conclusion, the LES framework demonstrates strong capability in capturing mean flow behavior, but further analysis is needed to accurately capture turbulence intensity and its impact on fluid-structure interaction response.

3.1.15 The GOKSTAD benchmark: comparison of numerical results and data

Speaker(s): K. Zwijsen, NRG

Session: Session 2 SG (WP3)

The GOKSTAD benchmark: comparison of numerical results and data

Kevin Zwijsen¹, Trevor Franklin², Lane Carasik², Sofiane Benhamadouche³

¹ NRG PALLAS B.V., Westerduinweg 3, 1755 LE Petten, The Netherlands

² Department of Mechanical and Nuclear Engineering, Virginia Commonwealth University, Engineering East Hall, 401 West Main Street, Richmond, VA 23284, United States

³ Électricité de France (EDF) R&D, EDF Lab Chatou, 6 quai Watier, 78400 Chatou, France

The current presentation is concerned with providing the results of the numerical simulations for the GOKSTAD benchmark of Work Package 3 (WP3). This involves making predictions of the Flow Induced Vibrations (FIV) of a tube bundle of circular rods arranged in a square lattice and subjected to single-phase cross-flow. The experimental facility used for validation is located at the von Karman Institute (VKI) in Belgium and is described in detail, together with the corresponding measurement data, in deliverable D3.3. The numerical investigations focus on the test section, which consists of a tube bundle with one or two centrally located flexible tubes, allowing for direct comparison between simulations and experimental observations.

Three contributing partners—VCU, EDF, and NRG PALLAS—performed numerical simulations of the tube bundle subjected to various inlet velocities. The modelling approaches and corresponding results from each institution are documented in dedicated chapters. One specific case, with an inlet velocity of 0.125 m/s, was selected as the primary benchmark case and is used for a detailed comparison of model setup, computational cost, and predictive performance across all contributors.

While all partners addressed the same physical problem, they employed different numerical tools and modeling strategies. VCU applied a high-fidelity Large Eddy Simulation (LES) approach, focusing on detailed resolution of turbulence structures and inflow conditions. EDF used the Code_Saturne solver with an Elliptic-Blending Reynolds Stress Model (EB-RSM) combined with two-way FSI coupling, providing an intermediate level of fidelity with improved turbulence anisotropy representation. NRG PALLAS adopted a wall-resolved LES approach on a coarser, URANS-quality mesh, aiming to balance accuracy and computational cost within an industrial workflow. Despite these differences in fluid modeling strategies, all partners consistently used a rigid-body structural representation, justified by the relatively simple and limited motion of the cylinder in the GOKSTAD setup. This consistency in

structural modeling ensures that observed differences in results are primarily associated with the fluid modeling and coupling approaches rather than structural assumptions.

Key results from the benchmark indicate that the choice of turbulence model for the fluid has a considerable impact on both the accuracy and computational cost of the simulations. High-resolution approaches generally yield more accurate flow field predictions, particularly in capturing turbulence structures and velocity distributions, but at significantly higher computational expense. Medium-resolution approaches can provide a practical alternative, offering slightly reduced accuracy at substantially lower cost. Despite these differences in flow field prediction, the displacement statistics of the vibrating tube are found to be relatively consistent across all approaches, provided that the relevant turbulence scales are sufficiently resolved.

3.2 Day 2 May 21, 2026: Project Results

3.2.1 Uncertainty quantification and Bayesian calibration for turbulence model parameters

Speaker(s): A. Eidi, TUD

Session: Session 3 Uncertainty quantification

Uncertainty quantification and Bayesian calibration for turbulence model parameters in URANS-based FIV analyses

Ali Eidi and Richard P. Dwight

Faculty of Aerospace Engineering, Delft University of Technology, Kluyverweg 1, 2629 HS Delft, The Netherlands

Flow-induced vibration (FIV) remains a major challenge in nuclear engineering applications, where turbulent pressure and velocity fluctuations can excite structural components and affect long-term reliability. Although scale-resolving simulations such as LES and DNS provide detailed turbulence information, their computational cost is often prohibitive for routine engineering FSI analyses. As a result, URANS simulations remain attractive for practical applications, but their predictive reliability is affected by uncertainty in turbulence-model parameters and structural model-form limitations. Within WP5 of the GO-VIKING project, we developed a surrogate-based uncertainty quantification framework for assessing and calibrating SST $k-\omega$ turbulence model parameters in wall-bounded turbulent flows relevant to FIV. The framework combines global Sobol sensitivity analysis, surrogate modeling, and Bayesian calibration using high-fidelity DNS or LES reference data. In the

canonical channel-flow and annular-flow cases, URANS simulations are coupled with an anisotropic pressure fluctuation model to reconstruct unresolved fluctuation quantities. These relatively affordable configurations are used to identify robust and transferable parameter hierarchies before moving to the substantially more expensive blunt-end cantilevered-rod geometry. The sensitivity analysis shows that the SST coefficients $\alpha\omega_1$ and β^* consistently dominate the variability of key quantities of interest across the investigated wall-bounded configurations. This finding enables a major reduction of the parameter space for the cantilevered-rod case, from the original nine uncertain SST coefficients to two dominant parameters. Gaussian process regression and polynomial chaos expansion surrogates are then trained on a limited set of URANS simulations and used for Bayesian calibration and posterior predictive uncertainty quantification. The results demonstrate that Bayesian calibration can substantially reduce parametric uncertainty and constrain the dominant SST parameters toward consistent regions of the admissible parameter space. In the cantilevered-rod case, both GPR and PCE surrogates produce consistent posterior distributions and predictive uncertainty estimates, indicating that the main conclusions are robust with respect to surrogate formulation. At the same time, remaining discrepancies between calibrated URANS predictions and LES reference data show that structural model-form uncertainty can dominate once the most influential parameters have been calibrated. Overall, the proposed multistage workflow provides a computationally feasible route for uncertainty-aware URANS-based FIV analysis. By exploiting transferable sensitivity information from canonical flows and using surrogate-based Bayesian inference, the framework supports calibrated and uncertainty-aware predictions for expensive engineering configurations relevant to future FSI applications.

3.2.2 Uncertainty quantification for flow-induced vibration using polynomial chaos methods

Speaker(s): J. Degroote, UGent

Session: Session 3 Uncertainty quantification

Uncertainty Quantification for Flow-Induced Vibration using Polynomial Chaos Methods

Henri Dolfen, Joris Degroote

Department of Electromechanical, Systems and Metal Engineering, Ghent University, Tech Lane Ghent Science Park - Campus A, Technologiepark 131, B-9052 Gent, Belgium

Flow-induced vibration (FIV) in nuclear reactor components remains a safety concern due to the coupling between turbulent flow, structural response and thermal-hydraulic conditions. Within the GO-VIKING project, uncertainty quantification (UQ) methodologies have been developed to assess how uncertainties in geometry, operating conditions and multi-phase flow propagate through high-fidelity computational fluid dynamics and solid

mechanics (CFD CSM) models, coupled into fluid-structure interaction (FSI) simulations, and affect vibration predictions. A stochastic framework based on generalized Polynomial Chaos Expansion (gPC) was developed and applied to two representative problems. In axial flow, fuel rod deformation is propagated through a coupled CFD-CSM model to quantify its impact on modal properties such as frequency and damping. The approach provides fast convergence compared to Monte Carlo sampling and shows that, for the considered configuration, deformation-induced variability remains limited, although sensitivity may increase in more complex scenarios. Subsequently, UQ was applied to two-phase cross-flow in a tube bundle representative of steam generator conditions. A key development is the use of a Synthetic Bubble Model (SBM) to generate realistic transient inlet conditions for Volume-of-Fluid (VOF) simulations without costly precursor domains. Uncertainty in the inlet velocity is propagated through a transient FSI model, showing that moderate input variability can result in amplified uncertainty in vibration amplitudes. The results demonstrate that non-intrusive Polynomial Chaos methods provide an efficient and robust framework for UQ in computationally demanding CFD-FSI applications with low-dimensional uncertainty. They also highlight the importance of accurate inlet modelling, careful numerical treatment and large-scale parallelisation to obtain reliable vibration statistics. Overall, the work supports the integration of probabilistic approaches into CFD-based tools for nuclear thermal-hydraulics, enabling more realistic and robust assessments of FIV-related risks.

3.2.3 Simulations of the Cantilever Rod experiment

Speaker(s): E. Harris, EDF-E

Session: Session 4 Multiphase Flow (WP4)

FSI in two-phase Cantilever Case

Ewan Harris

*EDF Energy R&D UK Centre Limited, Nova North, 11 Bressenden Place, London SW1E 5BY,
United Kingdom*

EDF R&D UK Centre investigated multiphase flow-induced vibration within the GO-VIKING WP4 cantilever-rod benchmark, using the University of Manchester POT3 case as the validation basis. The configuration consists of a vertical rod clamped at the top and free at the bottom, subjected to upward air-water flow through an annulus. EDF UK focused on the test condition with liquid superficial velocity $U_{ls} = 1.7$ m/s and gas superficial velocity $U_{gs} = 0.35$ m/s, for which experimental rod-tip displacement data were available. The objective was to determine whether a practical CFD-fluid-structure interaction approach could reproduce the measured vibration amplitudes and dominant frequencies at reasonable computational cost, being valuable when modelling FSI effects within the core of the reactor.

A Synthetic Bubble Method was explored as an inlet strategy, based on a transient bubble-injection approach and modified to impose representative bubble-size and Gaussian distributions targeting bubbly/intermittent regimes. However, the lack of measured inlet bubble-size distribution and void-fraction profile introduced significant uncertainty in the prescribed gas injection and inlet volume fraction. Consequently, the final EDF UK calculation did not rely on the SBM inlet. Instead, a mixture velocity inlet was used, with the upstream inlet length extended from 5D to 15D to allow bubble structures, void fraction and velocity profiles to develop through inertia, gravity, shear and surface-tension effects before interacting with the rod.

The final model was developed in STAR-CCM+ v2024.6 using a Volume-of-Fluid URANS (VOF) formulation, $k-\omega$ SST turbulence closure, HRIC interface capturing with a sharpening factor of 0.9, and enabled sub-models for surface tension, interface turbulence damping and interaction area density. The fluid was solved using finite volumes and the rod using finite elements through the Solid Stress model. The rod was idealised as a homogeneous solid with density 7490 kg/m^3 and Young's modulus 76.4 GPa . Two-way FSI was applied through a mapped interface; coupling was activated after 2s of flow development, with boundary added mass included to improve numerical stability.

The simulation reproduced the main flow features around the rod tip and gave encouraging vibration predictions. After mean subtraction, predicted RMS displacements were 0.1968 mm in the front/X direction and 0.2724 mm in the side/Y direction, compared with experimental values of 0.2278 mm and 0.3628 mm respectively. This corresponds to errors of -13.6% and -24.9% . The frequency-domain response captured the main structural modes, with peaks around 3.64 Hz and 23 Hz , although the numerical PSD peaks were broader and flatter than experiment, partly reflecting the shorter simulation duration of 8s compared with 60s experimentally. Overall, EDF UK demonstrated that the STAR-CCM+ VOF two-way FSI approach provides credible agreement for this challenging two-phase benchmark, while identifying inlet-condition uncertainty as the dominant limitation. Future work should prioritise improved experimental characterisation of bubble-size distribution, inlet void fraction and time-resolved phase structure, alongside longer simulations and sensitivity studies on inlet development length, mesh refinement, turbulence modelling sensitivities and high-resolution approaches.

Keywords: GO-VIKING WP4; two-phase flow; flow-induced vibration; cantilever rod; STAR-CCM+; VOF; FSI; Synthetic Bubble Method

3.2.4 Results of the two-phase Cantilever Rod benchmark

Speaker(s): F. Baraglia, EDF

Session: Session 4 Multiphase Flow (WP4)

Presentation of the TREFLE simulations at CEA

Federico Baraglia, William Benguigui

Électricité de France (EDF) R&D, EDF Lab Chatou, 6 quai Watier, 78400 Chatou, France

Multiphase flow-induced vibration of elongated structures is a common issue in many engineering systems and can affect structural integrity through deformation and wear. This report presents CFD simulations of a deformable rod subjected to axial air–water multiphase flow, aiming to improve understanding of the vibration mechanisms and support engineering design and assessment. Three numerical approaches were used, combining finite-element structural models with finite-volume multiphase flow solvers in Star-CCM+, ANSYS Fluent, and neptune_cfd. Multiphase flow was modeled using either the Volume of Fluid (VoF) method or a multi-Eulerian approach, while turbulence was treated with RANS or LES models. Two-way fluid-structure coupling was applied and results were compared with experimental data for specified inlet mass fluxes. The report describes the experiment, numerical methods, computational setup, and resulting comparisons.

3.2.5 The TITAN experiment: vibrations in a tube bundle exposed to a two-phase cross-flow

Speaker(s): D. Panunzio, CEA

Session: Session 4 Multiphase Flow (WP4)

The TITAN experiment: vibrations in a tube bundle exposed to a two-phase cross-flow

Domenico Panunzio

Commissariat à l'énergie atomique et aux énergies alternatives (CEA), CEA Saclay, 91191 Gif-sur-Yvette Cedex, France

The GO-VIKING project investigates flow-induced vibration (FIV) phenomena in nuclear reactor components under single- and two-phase flow conditions. Within this framework, Work Package 4 (WP4) focuses on FIV occurring in multiphase flows. This study presents experimental results obtained at CEA Saclay using the TITAN facility, a dedicated loop designed to analyse fluid–structure interactions in tube bundles subjected to air–water cross-flows.

The test section consists of a rotated triangular tube bundle composed of 50 tubes supplemented by two columns of semi-tubes. The tubes have a diameter of 30 mm, a length of 300 mm, and a pitch-to-diameter ratio of 1.44. Made of stainless steel, the tubes can be

configured either as rigid—allowing force measurements via a tri-axial sensor—or as flexible—enabling displacement measurements through a strain-gauge-instrumented support. An optical probe can also be installed inside the tube bundle to perform local void fraction measurements.

For displacement measurements, two main configurations were investigated: two flexible tubes located in the same column within a rigid bundle, and a cluster of seven flexible tubes in a rigid bundle. Flexible tubes were allowed to vibrate only in the lift direction. Experiments were conducted at constant homogeneous void fractions of 35% and 55%, respectively, while the flow velocity was progressively increased up to the onset of fluid-elastic instability (FEI). Signals were analysed in terms of root mean square (RMS) values and power spectral density (PSD). The RMS displacement increased with flow velocity up to the onset of FEI. The spectra exhibited dominant peaks corresponding to the flow-coupled modes of the system.

For force and void fraction measurements, six tests were performed with homogeneous void fractions ranging from 40% to 90%. Force signals were analysed in terms of power spectral density (PSD), and both lift and drag spectra exhibited a distinct broadband bump. In addition, the measured void fraction deviated from the homogeneous flow model.

Overall, this study provides a comprehensive experimental dataset on FIV mechanisms in simplified steam generator (SG) tube bundle configurations. The results contribute to a better understanding of flow–structure coupling and turbulence effects in two-phase cross-flows, and provide a valuable benchmark for the validation of numerical models developed within the GO-VIKING project.

3.2.6 Results from the TITAN simulations at ASNR

Speaker(s): D. Vivaldi, EDF

Session: Session 4 Multiphase Flow (WP4)

The TITAN benchmark: presentation of ASNR numerical results

Daniele Vivaldi

Autorité de sûreté nucléaire et de radioprotection (ASNR), Cadarache, BP 3, 13115 Saint-Paul-lez-Durance Cedex, France

In the frame of GO-VIKING WP4, ASNR realized numerical simulations of the TITAN experimental campaigns, featuring air-water cross-flow in a rotated triangular tube bundle. The specific experimental tests simulated lie in the churn/intermittent two-phase cross-flow regimes. The simulated gas and liquid flow rates generate two-phase buffeting forces and are lower than the ones experimentally observed to trigger fluid-elastic instability. The study employed and assessed the morphology-adaptive CFD approach AIAD (proposed by

Hohne et al., 2020), implemented inside Ansys Fluent. Numerical simulations of the full rigid tube configuration were first performed: results were compared to experimental results in terms of two-phase induced forces in the drag and lift directions. Sensitivity of the results on bubble diameter (constant or obtained by a population balance) and mesh refinement were preliminary assessed. The calculated force spectra are overall in good agreement with the experimental ones. Then, the experimental configuration featuring 7 vibrating cylinders was simulated, employing a strong fluid-structure coupling approach. The numerical simulations are found to calculate tube displacement of the same order of magnitude as the experimental measurements, however with a certain underestimation.

3.2.7 The TITAN benchmark: presentation of the numerical results

Speaker(s): W. Benguigui, EDF

Session: Session 4 Multiphase Flow (WP4)

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

William Benguigui^{1,2}, Daniele Vivaldi³, Sina Tajfirooz⁴, Kevin Zwijsen⁴, Constantinos Katsamis⁵, Ehimen Iyamabo⁵, Théo Ondoua¹, Fanny Beltran¹, Samuel Gaucher¹

¹ *Électricité de France (EDF) R&D, EDF Lab Chatou, 6 quai Watier, 78400 Chatou, France*

² *Institut des Sciences de la Mécanique et Applications Industrielles (IMSIA), UMR 9219 EDF/ENSTA Paris/CNRS/CEA, 7 boulevard Gaspard Monge, 91120 Palaiseau, France*

³ *Autorité de sûreté nucléaire et de radioprotection (ASNR), Cadarache, BP 3, 13115 Saint-Paul-lez-Durance Cedex, France*

⁴ *NRG PALLAS B.V., Westerduinweg 3, 1755 LE Petten, The Netherlands*

⁵ *EDF Energy R&D UK Centre Limited, Nova North, 11 Bressenden Place, London SW1E 5BY, United Kingdom*

The GO-VIKING project investigates FIV phenomena occurring in nuclear reactors under single and two-phase flow conditions. The focus of WP4 of the project is on FIV phenomena induced by two-phase flows, such as in steam generator. The present presentation, as part of WP4, reports simulations of experiments performed at CEA Saclay (France) on a triangular tube bundle facility, named TITAN, subjected to an air-water cross-flow. The triangular bundle consists of 30 tubes with two columns of half-tubes at both lateral plates to minimize wall effects with a pitch ratio of $P/D = 1.44$, representative of French steam generators. A rigid and a flexible configuration are simulated by four different partners with different numerical approaches in 4 different codes. Numerical approaches and calculation domains are presented. Considering the variety of numerical models and calculation domains, having such quantitative and qualitative similarities between simulations strengthens confidence in the numerical predictions. The triangular arrangement coupled to a slip between phases close to 1 are probably responsible of the stability of the flow and the agreement between simulation, since the flow is more restricted by the triangular

arrangement geometry than in a square arrangement. Thus, having a different inlet gas size distribution than the experiment appears to be less predominant after a few rows in the tube bundle than for the square arrangement. This finding is of great interest since there is no information on the gas size distribution from the experiment at the inlet except the flow rates. Finally, the present result shows for different homogenous void fraction and mass flow rates a correct agreement from three different partner contributions with the experiment in terms of void fraction and force prediction. Further, the use of resolved approach for large gas structure appears to be the main parameter to get correct force spectra. The conclusions of the flexible configuration are in agreement and highlight that it is possible to predict two-phase flow-induced vibration in tube bundles.

3.2.8 TREFLE: Experimental setup and campaigns

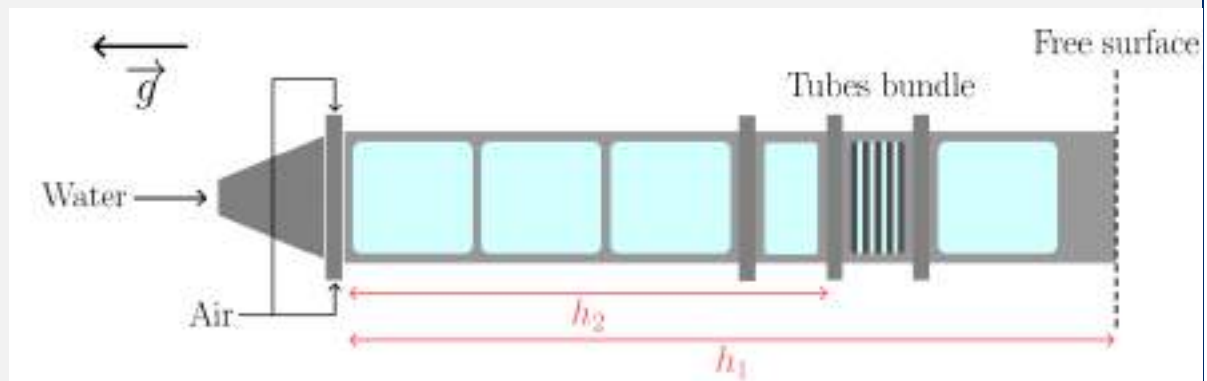
Speaker(s): L. Rotily, ASNR

Session: Session 4 Multiphase Flow (WP4)

TREFLE: Experimental setup and campaigns

Lucas Rotily

Autorité de sûreté nucléaire et de radioprotection (ASNR), Cadarache, BP 3, 13115 Saint-Paul-lez-Durance Cedex, France



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

A water circulation is established from bottom to top with a hydraulic pump, and the flowrate is kept constant at 2500 g/s for the different experiments at the facility maximum flow rate. The air is injected, with a flow rate starting from 0.4 g/s to about 50g/s, from the

bottom of the section after the convergent through seven adjustable-size injectors. The total height of the water column measures approximately 3.5 meters (h_1 in the figure above). The facility has a large flow homogenization zone of approximately 1.5 m between the air injection and the obstacle zone (h_2 in the figure above). Different windows allow flow visualization at different heights. The test section of the facility contains an in-line, square pitch 5×5 tube bundle of fixed (i.e. non vibrating) tubes with pitch-to-diameter ratio (P/D) of 1.44. The tube length is 10 times the tube diameter, which measures 30 mm.

Various instruments allow the two-phase cross flow to be characterised at different air flow rates: a differential pressure sensor, a wire mesh sensor, a dual optical probe and a high-speed camera. The differential pressure sensor, which is located at the bottom of the TREFLE test section, is used to calculate the void fraction across the entire test column. More localised void fraction measurements can be performed using an optical probe, which can measure the void fraction inside the 5×5 tube bundle. A wire mesh sensor is used to obtain high-frequency void fraction measurements either upstream or downstream of the tube bundle, covering the entire TREFLE section. High-speed cameras are used to visualise bubbly flow through the various windows. However, as the air flow rate increases, it becomes impossible to see into the section in depth, therefore limiting the measurement to qualitative characterisation.

To study fluid-structure interactions, some tubes may be replaced with flexible tubes consisting of a metallic blade and a plastic casing to maintain the shape and dimensions of the original tubes. These tubes are designed to oscillate at their free end and their displacement is recorded using a strain gauge sensor and an accelerometer. The tip end is tracked using a high-speed camera. These different measurements have allowed various findings to be made during the GO-VIKING project, such as an increase in displacement amplitude with air flow rate and the measurement of the power spectrum for both single and double flexible tube configurations. More recently, analyses have been conducted on the cross-correlation of the displacement signals of neighbouring tubes in order to study the synchronisation of oscillations.

For force measurements on the tube, which were taken during another campaign, the central tube of the bundle was replaced with a shorter tube that was connected to the section via a piezoelectric force sensor. These two sensors provided information on the fluctuations in the forces applied to the tube by the two-phase flow. The forces increased initially as the air flow rate rose in the bubbly regime and then reached a plateau at a higher void fraction in the intermittent regime. The power spectrum density showed a dominant frequency that remained fairly constant throughout the explored air flow rate, which is consistent with previous findings regarding the frequency at which large air pockets pass through the bundle, estimated to be around 5 Hz.

3.2.9 Presentation of the TREFLE simulations at CEA

Speaker(s): C. Bazin, CEA

Session: Session 4 Multiphase Flow (WP4)

Presentation of the TREFLE simulations at CEA

Clement Bazin

*Commissariat à l'énergie atomique et aux énergies alternatives (CEA), CEA Saclay, 91191
Gif-sur-Yvette Cedex, France*

The TREFLE experiment reproduces two-phase upward flow through a tube bundle with the aim of studying flow-induced tube vibrations. Numerical simulations of this configuration were carried out at CEA using Neptune_CFD code. The modelling approach relies on an Euler-Euler URANS three-field framework, solving 22 coupled differential equations for three phases: liquid, small dispersed bubbles (group 1), and large or hybrid bubbles (group 2). The model handles interfacial area transport equations (IATE), an Elliptic Blending Reynolds Stress turbulence model (EBRSM), and several interfacial force closures. Experimental data from the TREFLE facility were carefully processed to derive realistic inlet boundary conditions, combining void fraction PDF analysis for bubble group separation with 1D analytical pre-calculations for consistency checks. Two-dimensional simulations were performed for multiple flow conditions, covering both fixed and free cylinder configurations. Results show qualitative agreement with experimental observations in fixed configuration, notably reproducing the lateral preferential gas paths through the bundle. However, the 2D simulations systematically underestimate flow fluctuations, a limitation attributed to the low liquid turbulence regime characteristic of the TREFLE configuration, which promotes stable gas preferential paths and suppresses lateral fluctuations. Three-dimensional simulations are identified as the critical next step to resolve larger flow structures, increase fluctuation levels, and improve the quantitative predictive accuracy of the model.

3.2.10 Main results from the TREFLE benchmark

Speaker(s): W. Benguigui, EDF

Session: Session 4 Multiphase Flow (WP4)

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

*William Benguigui^{1,2}, Daniele Vivaldi³, Lucas Rotily³, Sina Tajfirooz⁴, Kevin Zwijsen⁴,
Constantinos Katsamis⁵, Ehimen Iyamabo⁵, Antti Timperi⁶, Clément Bazin⁷*

¹ *Électricité de France (EDF) R&D, EDF Lab Chatou, 6 quai Watier, 78400 Chatou, France*

² *Institut des Sciences de la Mécanique et Applications Industrielles (IMSIA), UMR 9219
EDF/ENSTA Paris/CNRS/CEA, 7 boulevard Gaspard Monge, 91120 Palaiseau, France*

³ *Autorité de sûreté nucléaire et de radioprotection (ASNR), Cadarache, BP 3, 13115 Saint-
Paul-lez-Durance Cedex, France*

⁴ *NRG PALLAS B.V., Westerduinweg 3, 1755 LE Petten, The Netherlands*

⁵ *EDF Energy R&D UK Centre Limited, Nova North, 11 Bressenden Place, London SW1E 5BY,
United Kingdom*

⁶ *VTT Technical Research Centre of Finland Ltd, Kivimiehentie 3, FI-02044 VTT, Espoo,
Finland*

⁷ *Commissariat à l'énergie atomique et aux énergies alternatives (CEA), CEA Saclay, 91191
Gif-sur-Yvette Cedex, France*

The GO-VIKING project investigates FIV phenomena occurring in nuclear reactor under single and two-phase flow conditions. The focus of the WP4 of the project is on the FIV phenomena induced by two-phase flows which is the case in steam generators. The present document, as part of WP4, reports simulations of an experiments performed at ASNR (Cadarache, France) on a square tube bundle facility, named TREFLE, subjected to an air-water cross-flow. The square bundle consists of 25 tubes with no half-tubes at wall. A rigid and a flexible configuration are simulated by 5 different partners with different numerical approaches from 4 different codes. First numerical approaches and calculation domains are presented. Considering the range of numerical models and calculation domains, having quantitative and qualitative similarities between simulations strengthens confidence in the numerical predictions. However, it appears in the present case that there is less agreement with the experiment than for the triangular arrangement (see D4.6). Moreover, the high slip in the present cases also pose a challenge for the two-phase flow prediction. Thus, having a different void distribution than the experiment appears to have a significant role in the tube bundle. Most of the partners simulated two cases, with a reasonable agreement found upstream and downstream the cylinder by at least two of the partners. For those having the correct flow prediction, it can be seen that vibrations are in a correct agreement with experiment. Once again, similarly to the triangular case (D4.6), the force fluctuation in two phase flow numerical simulation appears to be driven by the resolved gas structure, highlighting correct prediction from VOF approaches without any treatment for the dispersed phase (oppositing to hybrid models). This is true since the present cross

comparison was performed on an intermittent flow where the force is driven by large gas structures.

3.2.11 Development and validation of the Projection-Based Reduced-Order Model for the analysis of FIV

Speaker(s): A. van Zuijlen, TUD

Session: Session 5 Fast-running methods (WP5)

Development and validation of the Projection-Based Reduced-Order Model for the analysis of FIV

A. van Zuijlen

Faculty of Aerospace Engineering, Delft University of Technology, Kluyverweg 1, 2629 HS Delft, The Netherlands

In many engineering applications, flow induced vibrations play a pivotal role in the safety and longevity of these applications. For example in nuclear power plants, turbulence induced vibration of (bundles of) fuel rods can cause excessive wear when vibrations become too large, leading to expensive outages/maintenance of the power plant. A good prediction of the systems turbulence induced vibration sensitivity is of utmost importance.

Modelling turbulence induced vibration is challenging: typically, the total length of the rod (bundles) versus the small scale turbulence that is required, excludes the use of high fidelity turbulence modelling such as LES to the used on the application scale, in particular when used within the design phase where many design changes and iterations are needed. On the other hand the use of less computational demanding approaches such as (U)RANS, fails in predicting the turbulence induced vibration.

In the present research we propose to derive a reduced order model for the fluid that predicts the mean flow response to a reduced set of interface kinematics. This model can be used to model very inexpensively the interaction between a structural (modal) model and the mean flow. The turbulence forcing is then obtained by projection of a turbulence pressure fluctuation field onto the same modal basis. These fluctuations could come from e.g. a turbulence pressure fluctuation model, or in this case a higher fidelity simulation on a reduced, simplified domain (2.5cm length vs a rod length of 1m).

The method is applied to a single cantilever rod case. The results show that the reduced order model captures the typical added mass and added damping effects within the interaction between the flow and the structure. Also, when turbulence pressure fluctuations are imposed that were sampled on a much smaller, periodic domain, the tip displacement RMS is matching well to experimental data when the pressure fluctuations are simply duplicated over the whole rod length, leading to a full correlation of each 2.5cm

section over the rod's length. The tip RMS is underpredicted by almost a factor of 10 when the sampled pressure fluctuations are imposed with random correlation between each 2.5cm section. Using an assumed advection speed of the turbulence fluctuations to create a temporal correlation between sections, shows an underprediction of the tip RMS for low advection speeds which is reduced for higher advection speeds.

3.2.12 Development and validation of the Synthetic Turbulence Model for FIV analysis

Speaker(s): Y. Filonova, IPP

Session: Session 5 Fast-running methods

Development And Validation Of The Synthetic Turbulence Model For Fiv Analysis

Vladislav Filonov^{1,2}, Yuliia Filonova¹

¹ IPP Centre LLC, 8 Bolsunovska Street, 01014 Kyiv, Ukraine

² Institute of Hydromechanics, National Academy of Sciences of Ukraine, 8/4 M. Kapnist Street, Kyiv, 03057, Ukraine

This work is dedicated to the development and implementation of methods for reconstruction of the turbulent pulsation field for one-directional and two-directional FSI with providing reliable assessment of structural flow induced vibrations (FIV) with reasonable computational costs. The main criterion for selecting approaches is their speed and sufficient accuracy for engineering practice. The basic idea of this work is to develop the procedures, which generate a physically based field of turbulent pulsations based on RANS simulations.

For that purpose two different implementations of Synthetic Turbulence Generator (STG) were proposed and developed. The first one implies the supplementing the ANSYS CFX by introducing both FORTRAN libraries for the FIV analysis and for STG. The library for FIV analysis consists of three different options for the beam representation: two analytical solution options (Euler-Bernoulli and Timoshenko beams) and the third one is the FDM based solver of Euler-Bernoulli Beam which doesn't consist any predetermined spectrum of natural frequencies. The transfer of synthetic fluctuations of velocity fields in the general three-dimensional case is carried out on the basis of the wave function without using advection equations. The influence of the method for calculating the force from the flow acting on a solid body and the need to consider the tangential component of average and fluctuation stresses in addition to the dominant pressure contribution is discussed. Particular emphasis is placed on finding a special algebraic relationship for estimating pressure pulsations, as an alternative to the Poisson equation to avoid the effect of "self-

induction of turbulence”. In this case, the most promising relationships turned out to be those constructed on the gradients of fluctuation velocity without involving the diffusion of Reynolds stresses. The second method of STG implementation is based on adapting the Stable Fluid Algorithm for engineering calculations. This specialized, certainly stable algorithm is generally used for graphical visualization of flows in almost real-time mode, which makes it a potential candidate for a “fast running method”. Along with the main advantage of the “end-to-end” calculation for the time step, a significant disadvantage is that it generally distorts the physical features associated with vortex formation, and therefore, in its original formulation, is not suitable for problems related to FSI. These issues are overcome by extending the capabilities of the method based on deterministic vorticity source and generation of synthetic turbulence fields.

Validation of the developed and described methods is performed using AMOVI Benchmark cases. The main results of the prediction of rod displacements, as well as the spectral characteristics of the associated vibrations are shown. The computational advantages and physical limitations, as well as further possible steps to improve the proposed approaches in terms of computational efficiency are discussed.

3.3 Day 3 May 22, 2026: Education and Best Practices

3.3.1 Education and dissemination activities

Speaker(s): L. Koloszar, VKI

Session: Session 6 Education and BPGs

GO-VIKING Education, training, dissemination

Philippe Planquart, Lilla Koloszar and Maria Teresa Ramandi

von Karman Institute for Fluid Dynamics (VKI), Waterlooesteeweg 72, B-1640 Sint-Genesius-Rode, Belgium

Flow-induced vibration (FIV) remains a critical issue for the safe operation of nuclear power plants, particularly under long-term operation conditions where vibration-induced degradation may affect key components such as fuel assemblies and steam generators. The GO-VIKING project has addressed this challenge through an integrated approach combining experimental investigations, numerical modelling, and structured knowledge transfer.

On the scientific side, the work combined experimental data generation and advanced numerical simulations, including computational fluid dynamics (CFD) and computational structural mechanics (CSM), enabling improved characterization and prediction of FIV phenomena across representative conditions. This combined approach supports more

reliable evaluation methodologies and contributes to enhanced safety and performance assessments of nuclear systems.

A central outcome of the project is the development of a structured and accessible educational framework covering the full spectrum from fundamental concepts to advanced applications. This framework includes a set of at least 12 educational materials, combining lectures, literature reviews, and technical presentations, together with curated video content integrating numerical and experimental results. These materials have been made available online to ensure long-term accessibility and reuse.

Education and training activities reached a broad audience through multiple formats. A dedicated lecture series was organized, comprising 10 lectures, an industry panel session, and a facility visit, with a total of 36 participants, including lecturers, internal participants, and external attendees. In addition, open lectures attracted larger audiences, with 63 and 36 participants respectively, and recordings were published online to extend their reach. Practical training was delivered through hands-on sessions on OpenFOAM and TrioCFD tools, as well as through the definition of a benchmark case based on a cantilever rod configuration to support cross-comparison and collaborative learning.

Dissemination activities ensured broad visibility and engagement across stakeholders. Digital outreach exceeded initial targets, with social media channels surpassing key performance indicators and the project website attracting a total of 8,971 visitors, with strong interest in education and training-related content. Scientific dissemination included at least 15 papers and 1 poster presented to the research community, including recognition through a best poster award. Communication efforts were supported by structured materials and a coordinated multi-channel strategy, including online platforms and physical dissemination tools.

Overall, the work demonstrates the effectiveness of combining research, education, training, and dissemination in a coherent framework. By integrating quantitative outputs—ranging from educational resources and training activities to measurable outreach indicators—the project has strengthened knowledge transfer, supported skill development in the next generation of engineers, and contributed to a broader and more operational understanding of FIV phenomena in nuclear engineering.

3.3.2 Best Practice Guidelines on the use of FSI methods for FIV evaluation

Speaker(s): A. Papukchiev, GRS

Session: Session 6 Education and BPGs

Best Practice Guidelines on the Use of FSI Methods for FIV Evaluation

Angel Papukchiev¹, Kevin Zwijsen²

*1 Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Boltzmannstraße 14,
85748 Garching near Munich, Germany*

2 NRG PALLAS B.V., Westerduinweg 3, 1755 LE Petten, The Netherlands

The experience gained throughout the GO-VIKING project on the analysis of FIV phenomena in key nuclear steam supply system (NSSS) components with modern 3D programs is synthesized in best practice guidelines. These are based on the lessons learned by the partners throughout the four years of collaborative work and are structured according to the project's work packages. These guidelines shall not be considered prescriptive (e.g. to teach how to design a component), but rather help other researchers or professionals dealing with similar problems. The document focuses on best practices when analysing flow-induced vibrations (FIV) in single-phase axial-flows, single-phase cross-flows, and multiphase flows. Further, some best practices were derived from the development and validation of fast-running and uncertainty quantification methods. All guidelines are based on seven different experiments and the application of multiple FSI numerical approaches and codes. The best practices are intended to help vendors, regulators, TSOs, and industry to obtain a deeper understanding on the FIV nature and their impact on the safety of nuclear reactors, and provide them with knowledge and confidence in their analysis in this very challenging interdisciplinary field.

Conclusion

The GO-VIKING Final Workshop provided a comprehensive overview of the main scientific, technical and educational outcomes of the project. By bringing together operational experience, stakeholder needs, experimental campaigns, numerical benchmarks, uncertainty quantification, fast-running methods and best practice guidance, the workshop demonstrated the integrated nature of the project results.

The proceedings documented in this deliverable preserve the content and value of the workshop and make it available for future use. The deliverable also shows that WP6 built on the work of all other technical work packages and translated their results into education, training, dissemination and stakeholder engagement outputs.

Through this integration, the Final Workshop contributed to the long-term exploitation of GO-VIKING results and supported the development of a more knowledgeable community able to address FIV challenges in nuclear applications. The deliverable therefore represents not only a record of the final event, but also a knowledge-transfer instrument supporting future research, education, industrial application and regulatory dialogue.

Annexes

Annex 1 – Final Workshop Programme

Final Workshop Programme

Annex 2 – Workshop presentations

Final Workshop presentations

Annex 1 Final Workshop Programme



Funded by
the European Union

GO-VIKING FINAL WORKSHOP

20-22.05.2026

Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH

Boltzmannstr. 14, 85748 Garching n. Munich, Germany

Final Program

Time (CET)	Session Title	Format / Speaker(s)	Duration
Day 1 - May 20, 2026: Project Introduction & Results			
09:00-09:20	Welcome & Project Overview	Opening by Project Coordinator (GRS/NRG PALLAS) – GO-VIKING overview objectives	20 min
09:20-10:20	Open Lecture on FIV	Operational experience with FIV in NPPs (P. Moussou/P. Veber, EDF/Vattenfall)	60 min
10:20-10:50	Outcome of WP1	Results of the Stakeholders' Workshop and literature reviews (D. Vivaldi, ASNR)	30 min
Break			15 min
11:05-11:40	Session 1: FIV in FA (WP2)	The ALAIN experiment and its analysis using different FSI approaches (H. Hadzic/J. Herb/Ch. Bläsius/Y. Dubyk (FRA/GRS/IPP))	35 min
11:40-12:05	Session 1: FIV in FA (WP2)	Outcome of the ALAIN benchmark (H. Hadzic, FRA)	25 min
Lunch Break			55 min
13:00-13:25	Session 1: FIV in FA (WP2)	Single- and two-phase Cantilever Rod experiments on FIV in axial flow (M. Nabawy, UoM)	25 min

13:25-13:55	Session 1: FIV in FA (WP2)	Results of the Cantilever Rod benchmark (M. Hussain, NRG PALLAS)	30 min
13:55-14:20	Session 1: FIV in FA (WP2)	High-resolution DNS simulations of the Cantilever Rod experiment (E. Merzari, PSU)	25 min
14:20-14:40	Session 2: SG (WP3)	The AMOVI experiment on FIV in cross-flow (D. Panunzio, CEA)	20 min
14:40-15:05	Session 2: SG (WP3)	High-fidelity LES analysis of AMOVI (S. Benhamadouche, EDF)	25 min
15:05-15:25	Session 2: SG (WP3)	Validation of ANSYS CFX using the Rigid Body Motion approach on the AMOVI experiment (H. Mistry, GRS)	20 min
Break			15 min
15:40-16:10	Session 2: SG (WP3)	Comparison of different numerical approaches within the AMOVI benchmark (Y. Filonova, IPP)	30 min
16:10-16:35	Session 2: SG (WP3)	Tube vibrations in the GOKSTAD experimental bundle (L. Koloszar, VKI)	25 min
16:35-16:55	Session 2: SG (WP3)	High-resolution GOKSTAD computations at VCU (L. Carasik, VCU)	20 min
16:55-17:25	Session 2: SG (WP3)	The GOKSTAD benchmark: comparison of numerical results and data (K. Zwijsen, NRG)	30 min
17:25	Adjourn		
Day 2 - May 21, 2026: Project Results			
09:00-10:00	Open Lecture	How GPUs change industrial CFD simulation strategies (F. Menter, ANSYS Germany)	60 min
10:00-10:25	Session 3: Uncertainty quantification	Uncertainty quantification and Bayesian calibration for turbulence model parameters (A. Eidi, TUD)	25 min
10:25-10:50	Session 3: Uncertainty quantification	Uncertainty quantification for flow-induced vibration using polynomial chaos methods (J. Degroote, UGent)	25 min
Break			15 min

11:05-11:30	Session 4: Multiphase Flow (WP4)	Simulations of the Cantilever Rod experiment (E. Harris, EDF-E)	20 min
11:30-11:55	Session 4: Multiphase Flow (WP4)	Results of the two-phase Cantilever Rod benchmark (F. Baraglia, EDF)	30 min
Lunch Break			60 min
13:00-13:20	Session 4: Multiphase Flow (WP4)	The TITAN experiment: vibrations in a tube bundle exposed to a two-phase cross-flow (D. Panunzio, CEA)	20 min
13:20-13:45	Session 4: Multiphase Flow (WP4)	Results from the TITAN simulations at ASNR (D. Vivaldi, EDF)	25 min
13:45-14:15	Session 4: Multiphase Flow (WP4)	The TITAN benchmark: presentation of the numerical results (W. Benguigui, EDF)	30 min
14:15-14:40	Session 4: Multiphase Flow (WP4)	TREFLE: Experimental setup and campaigns (L. Rotily, ASNR)	25 min
Break			15 min
14:55-15:15	Session 4: Multiphase Flow (WP4)	Presentation of the TREFLE simulations at CEA (C. Bazin, CEA)	20 min
15:15-15:45	Session 4: Multiphase Flow (WP4)	Main results from the TREFLE benchmark (W. Benguigui, EDF)	30 min
15:45-16:10	Session 5: Fast-running methods (WP5)	Development and validation of the Projection-Based Reduced-Order Model for the analysis of FIV (A. van Zuijlen, TUD)	25 min
16:10-16:40	Session 5: Fast-running methods	Development and validation of the Synthetic Turbulence Model for FIV analysis (Y. Filonova, IPP)	30 min
16:40	Adjourn		

18:30	Dinner at the Augustiner Stammhaus (Munich)		
Day 3 - May 22, 2026: Education and Best Practices			
09:00-09:30	Session 6: Education and BPGs	Education and dissemination activities (L. Koloszar, VKI)	30 min
09:30-10:15	Session 6: Education and BPGs	Best Practice Guidelines on the use of FSI methods for FIV evaluation (A. Papukchiev, GRS)	45 min
10:15-10:45	Session 6: Education and BPGs	Discussion on the BPGs (all)	30 min
10:45-11:30	Session 6: Education and BPGs	Closing remarks by coordinators - summary of key outcomes and future collaboration opportunities	45 min
11:30	Adjourn and Lunch		

Annex 2 Final Workshop Presentations



Funded by
the European Union



Overview of the Horizon Europe *Gathering expertise On Vibration ImpaKt In Nuclear power Generation (GO-VIKING)* project

Angel Papukchiev (GRS), K. Zwijsen (NRG)

GO-VIKING Final International Workshop, 20-22.05.2026, Garching, Germany



Funded by
the European Union

Outline

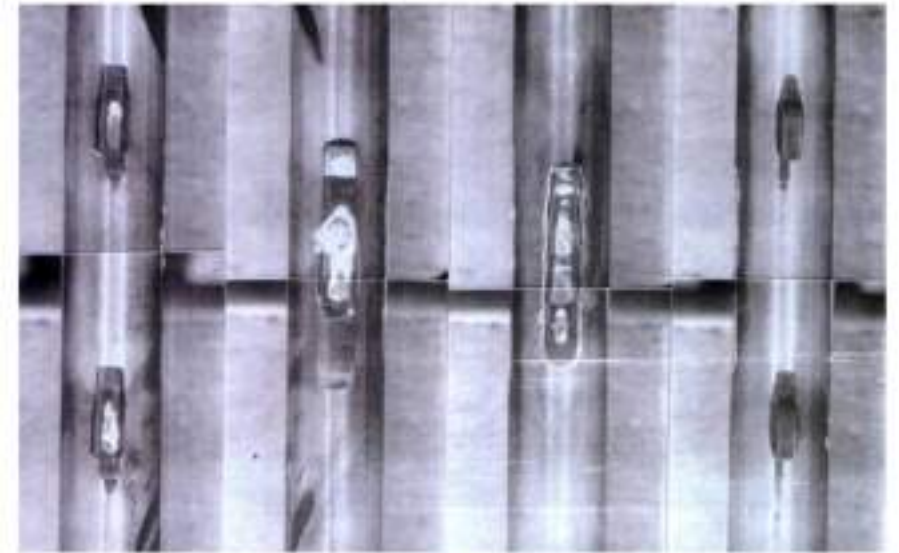


- Introduction
- Background
- Objectives
- Concept and structure
- Timeline
- Final Workshop

Introduction



- Many NPP structures (rods, tubes, etc.) are exposed to a highly energetic flow
- Mechanical forces exerted by the fluid on the structure's surface may lead to structural displacement and eventually to flow-induced vibrations (FIV)
- The structural displacement impacts the flow itself
- Fluid-Structure Interaction weakly or strongly coupled
 - may lead to large structural vibrations/displacements
 - evolve over time due to:
 - altering components' nominal conditions
 - transient or accident scenarios
 - aging and degradation
- FIV may lead to unplanned outages → plant availability
- Knowledge of FIV already in the design phase important

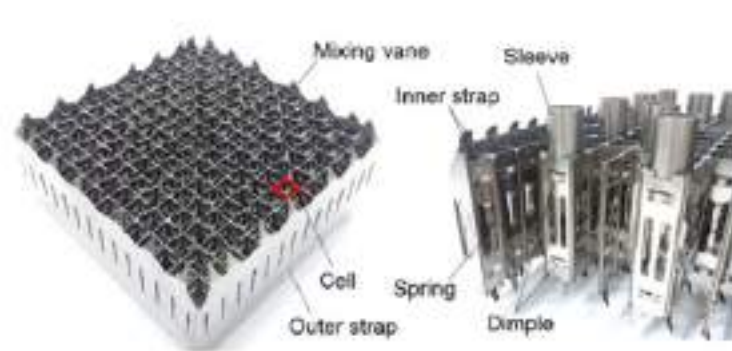
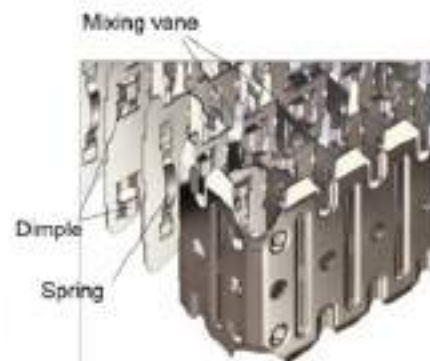
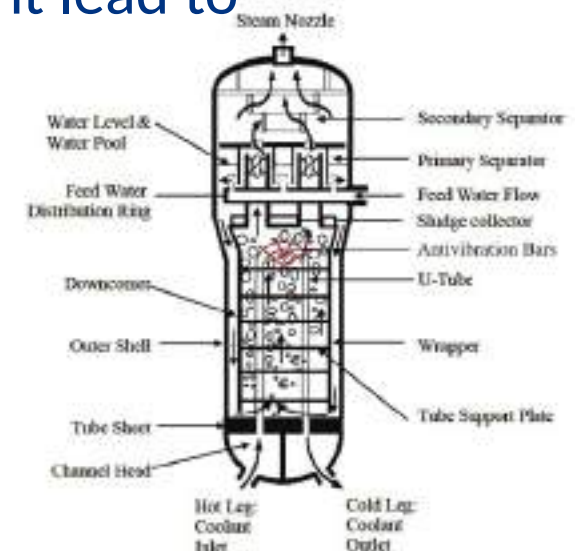


Failed KOFA fuel rod (K.-T. Kim, NED, 2010)

FIV in fuel assemblies and steam generators



- Reactor core and SG are key components of the nuclear steam supply system (NSSS) in the NPPs
- Grid-to-rod fretting largest source of fuel failure in PWRs and one of main sources in BWRs – 58% for the period 2006-2015 (IAEA, 2019)
- Flow-induced forces act on the U tubes of the SG and eventually might lead to steam generator tube rupture (SGTR) accident



The need for GO-VIKING

- In the beginning of 2020 the European VIKING initiative was launched (F. Roelofs, NRG)
- Objective: use experimental and hi-res numerical data to further develop and validate numerical tools and methods for FIV issues
- Focus on the analysis of fretting in FA and FIV in tube bundles
- VIKING addressed these issues, but also identified the need for further investigations within a larger collaboration in the Horizon Europe framework
- State-of-the art in FIV analysis, FIV in multiphase flows, fast-running FSI methods, best practices and training in FIV analysis are very important and need attention
- GO-VIKING proposal prepared and submitted in autumn 2021



VATTENFALL

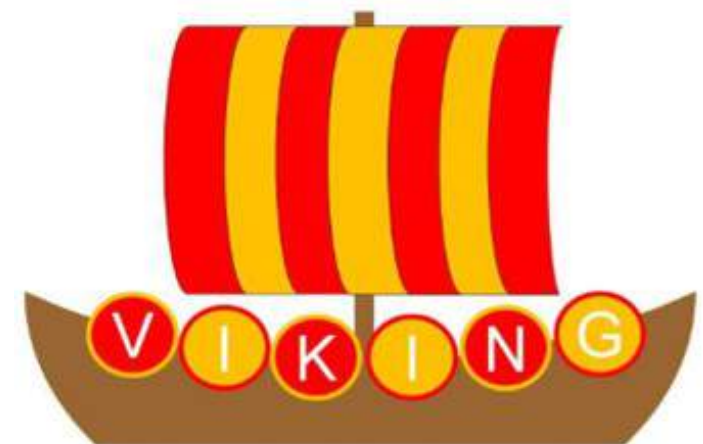


framatome



IRSN

IMSIA



The GO-VIKING project



- The GO-VIKING proposal accepted for funding by the EC in February 2022 Coordinator GRS, co-coordinator NRG
- Grant amount: 2.82 million €
- Sixteen partners from Europe and two from US
- Project duration: 01.06.2022 – 30.05.2026
- Main objective: develop, improve and validate FSI methods, for FIV analysis and generate guidelines for the prediction and assessment of FIV phenomena in nuclear reactors

Partners



Associated Partners



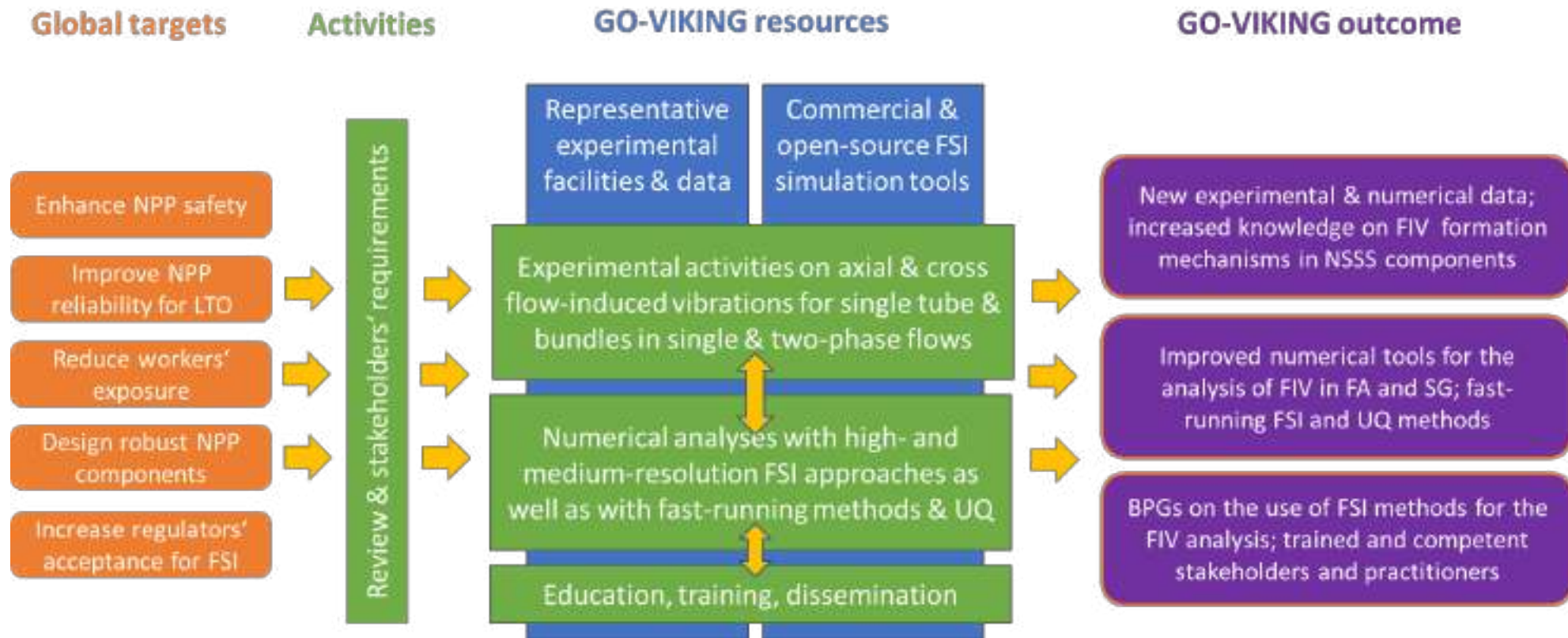
* EDF R&D UK Center

GO-VIKING specific objectives

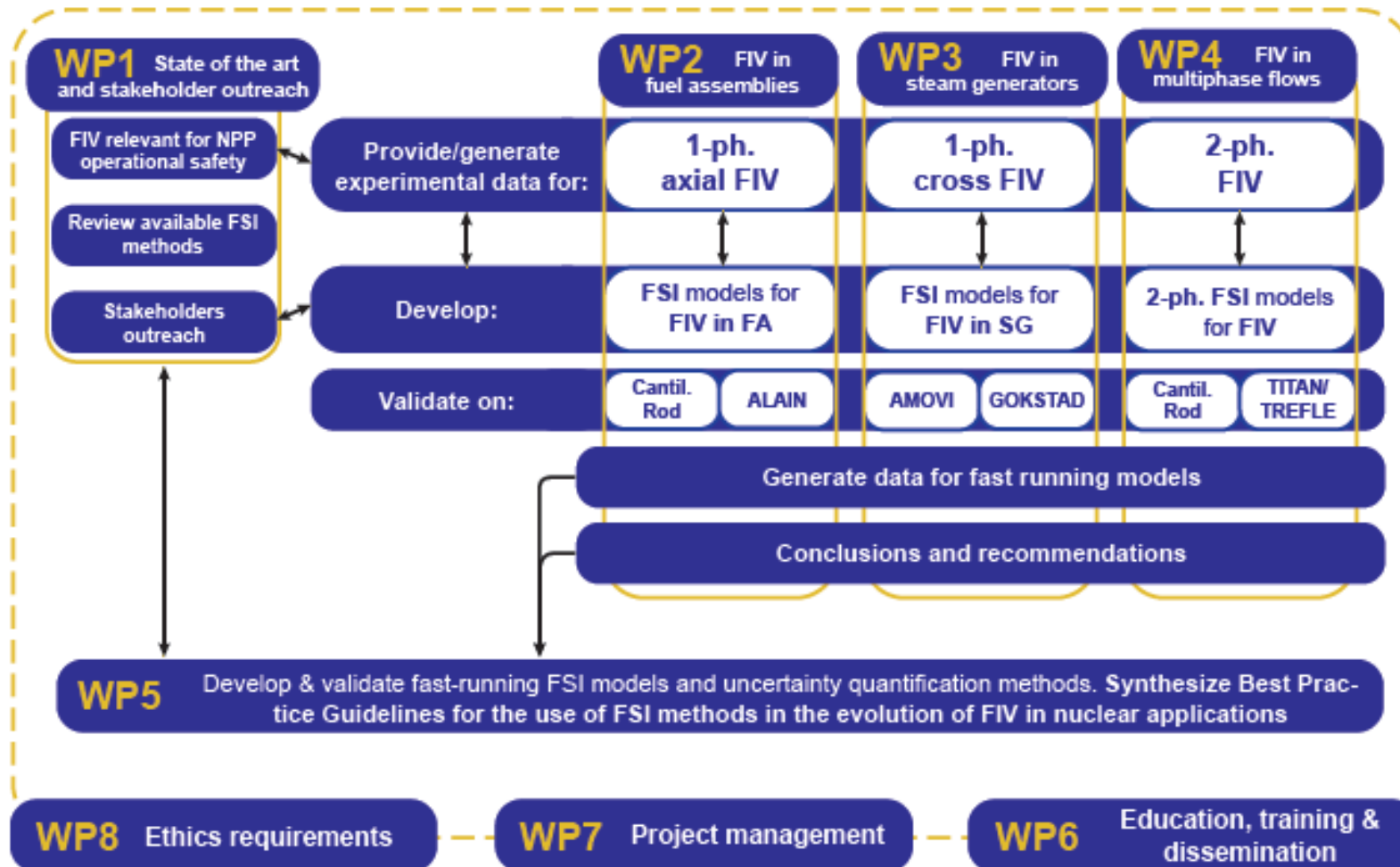


- **SO1:** Generation of new experimental and high-resolution numerical data, relevant for nuclear FA and SG tubes
- **SO2:** Expanded knowledge on efficiency, accuracy and reliability of FSI methods
- **SO3:** Provision of validated fast-running FSI tools with uncertainty quantification methods
- **SO4:** Highly increased expertise of and awareness on FIV phenomena in NPPs
- **SO5:** Synthesis of best practices and training of stakeholders and graduates in numerical FIV analysis

GO-VIKING concept



GO-VIKING structure



[illegible]



Thank

Angel Papukchiev, Kevin
Zwijssen **you!**



GRS, NRG PALLAS



**angel.papukchiev@grs.de,
kevin.zwijssen@nrgpallas.com**

www.go-viking.eu



Funded by the
European Union



Operational experience with FIV in NPPs

Pascal Veber

Vattenfall AB, Sweden

Outline

- FSI/FIV background
- FIV in RPV
 - Annular gap turbulence
 - Upper/lower plenum
- FIV in steam line
 - T-junction
 - Valve
 - Acoustic
 - Turbulence

Introduction

Ringhals - a part of Vattenfall BA Generation

Ringhals 1 (R1)

BWR, ASEA ATOM

Commercial operation 1975

Shut down 2020

Ringhals 2 (R2)

PWR, Westinghouse

Commercial operation 1975

Shut down 2019

Ringhals 3 (R3)

PWR, Westinghouse

Commercial operation 1981

Planned service life 60 years

Ringhals 4 (R4)

PWR, Westinghouse

Commercial operation 1983

Planned service life 60 years



FSI/FIV in reactor vessel and internals

FIV in nuclear reactor vessels has been extensively studied during the mechanical design of reactor vessel

Upcoming problems during for example **power upgrade, components replacement, flexible operation...** has increased the interest on FIV computational models

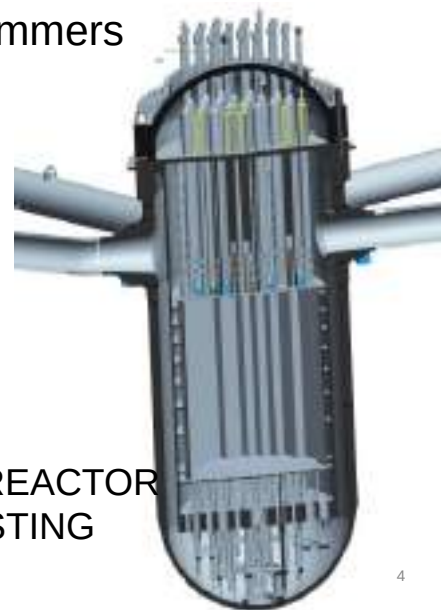
IAEA guidelines of FSI: vortex induced vibrations, forced vibrations (turbulence-induced), self-excited vibrations and water hammers

Four dominant vibration-excitation mechanisms

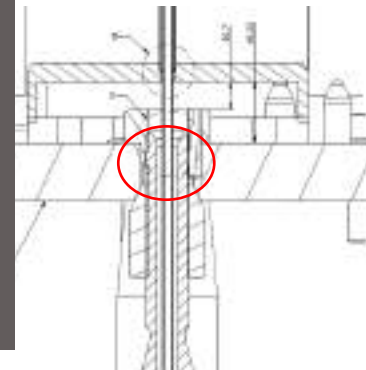
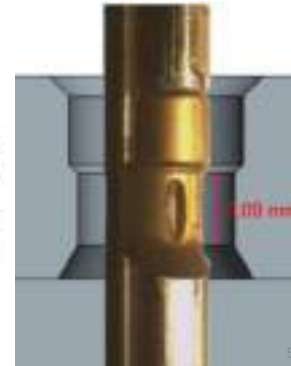
- fluid-elastic instability
 - periodic wake shedding
 - turbulence-induced excitation
 - acoustic resonance
- (M.J. Pettigrew, C.E.)

REGULATORY GUIDE 1.20

COMPREHENSIVE VIBRATION ASSESSMENT PROGRAM FOR REACTOR
INTERNALS DURING PREOPERATIONAL AND STARTUP TESTING



FIV in RPV

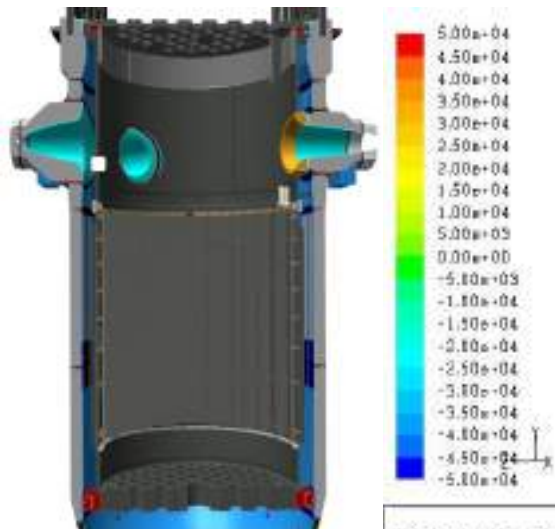
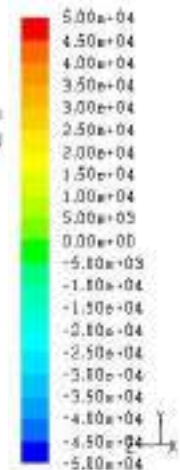
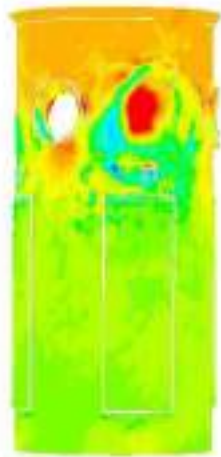
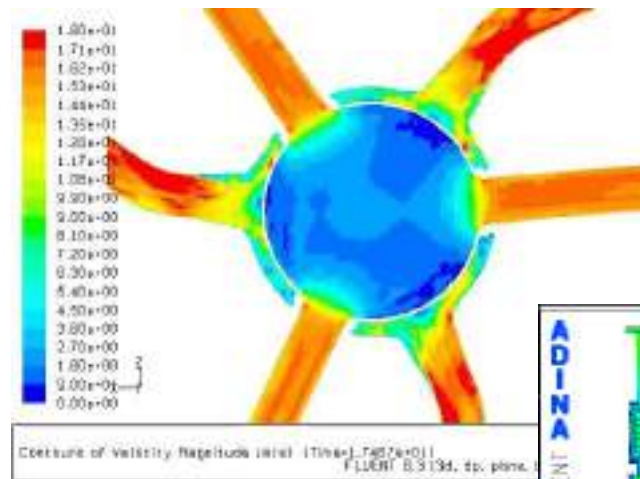
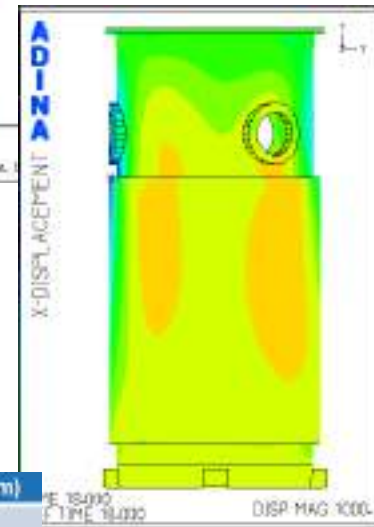


Flow turbulence induced vibration in RPV

Excessive core barrel movement, caused by flow- induced vibration, may lead to many detrimental effects including damage to reactor internals and interference with control rod movement. Problems resulting from excessive core barrel movement have been encountered in at least one plant.

NRC Item C-12: Primary System Vibration Assessment

Turbulent excitation from random fluctuations

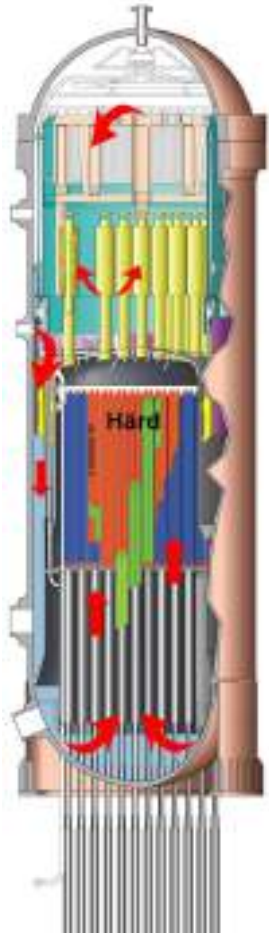
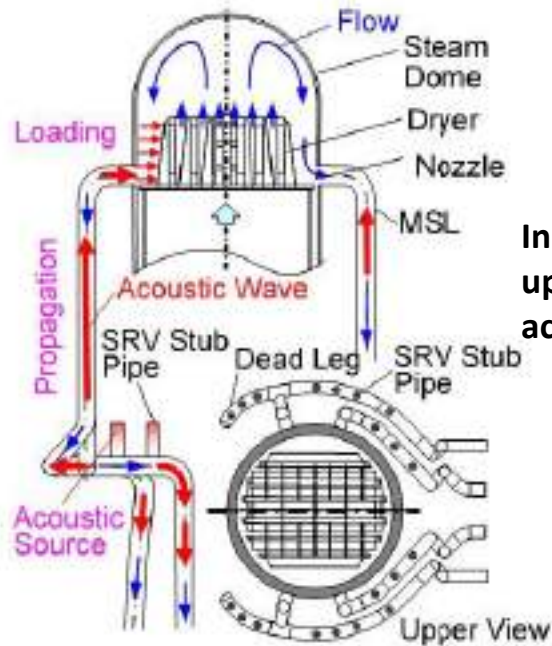


Load	RPV (mm)	LSP (mm)
4M	0.029	0.7
16M	0.023	0.64
80M	0.014	0.37

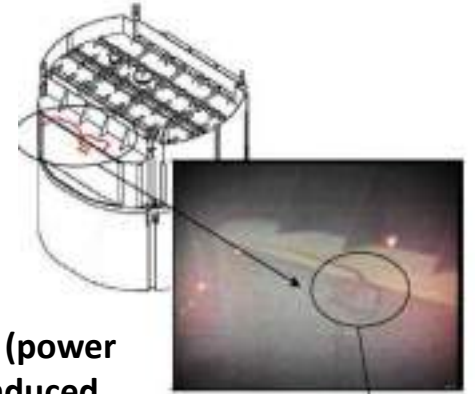
FIV for steam line vibration

Steam dryer failure at Quad Cities Unit 2

Load mechanism: vortex shedding coupled to acoustic waves



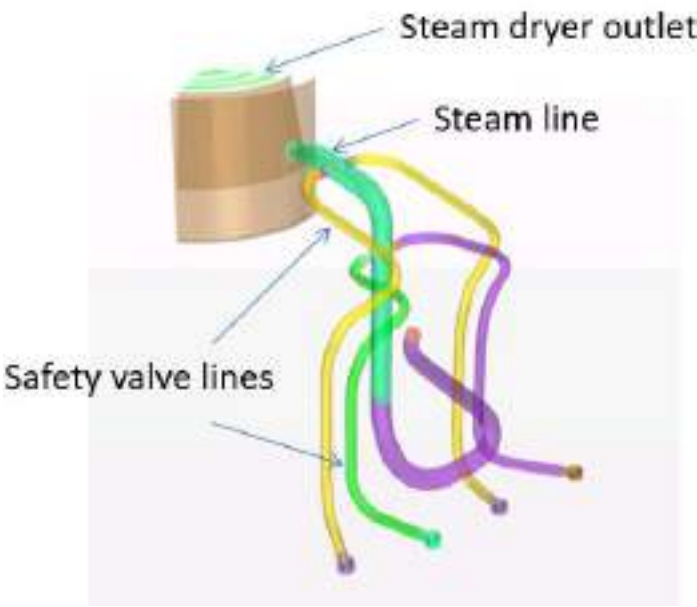
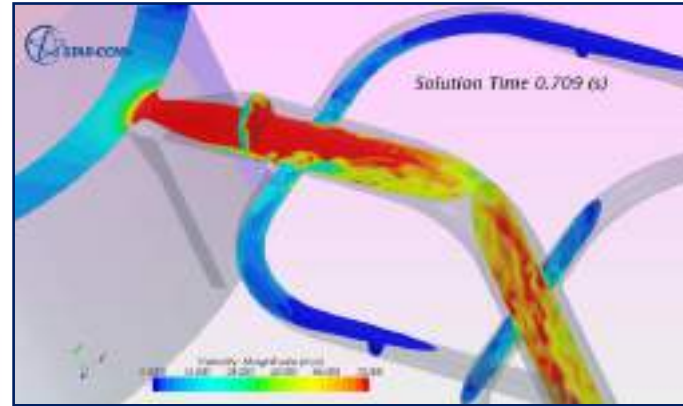
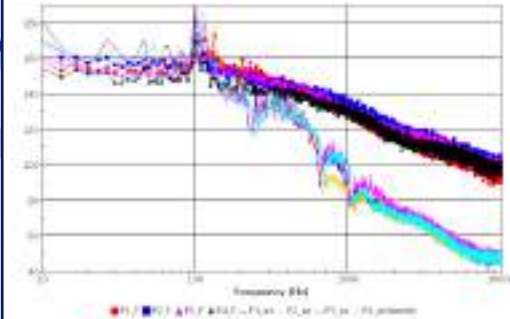
Increase of the flow velocity (power up-rate) resulted in a flow-induced acoustic resonance in the SRV pipes



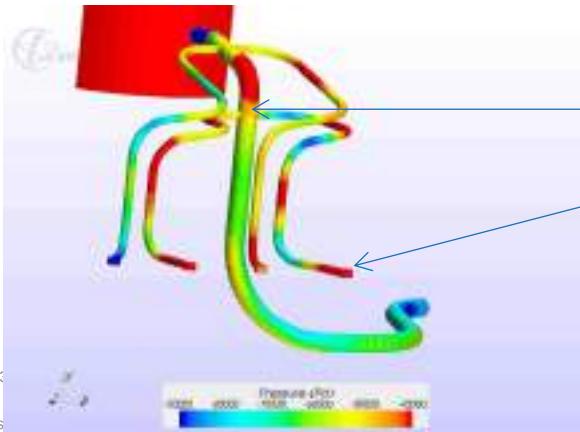
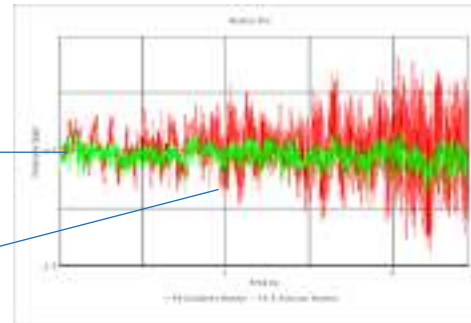
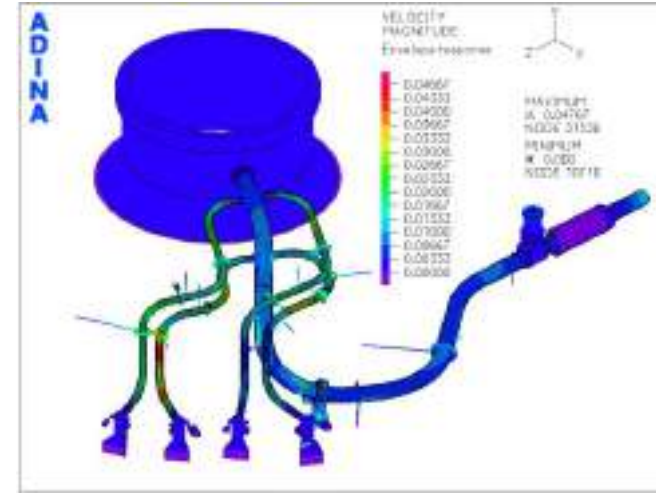
Schematic and photos of cracks and subsequent loose parts in original QC-dryers (from GEN SIL 648, 2006).



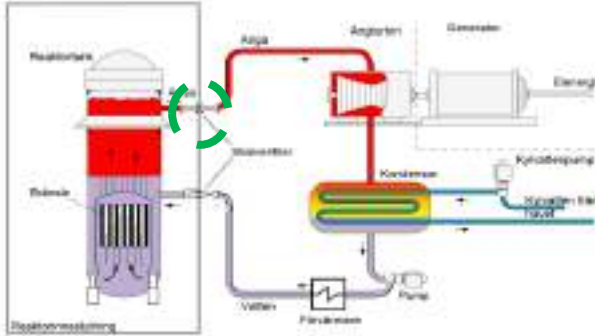
Star-ccm+ Detached Eddy Simulation model



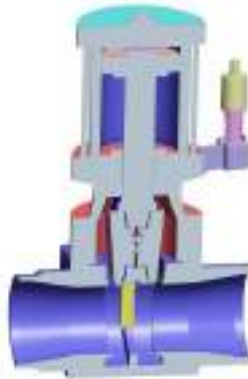
Maximum envelope response of RMS velocity [m/s].



Main Steam Isolation Valve vibration



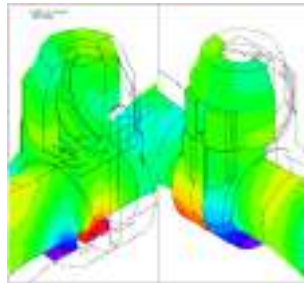
New MSIV due to power uprate



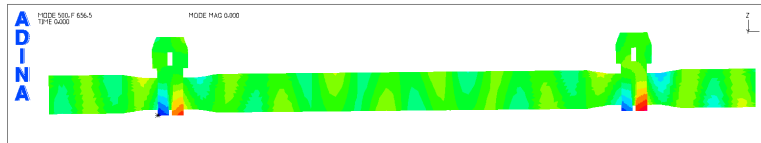
Acoustical analysis Model



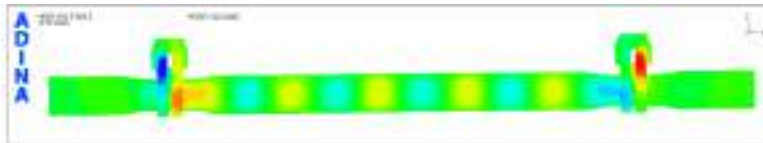
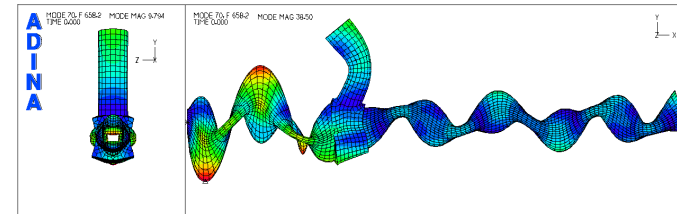
Mode 25 660 Hz



Mode 500 at 657 Hz

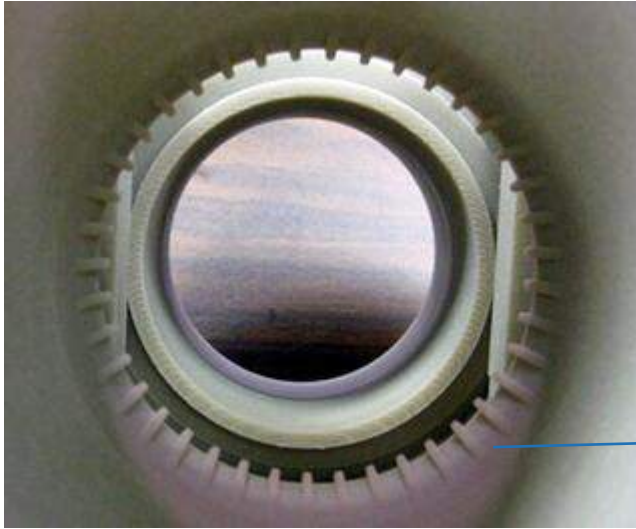
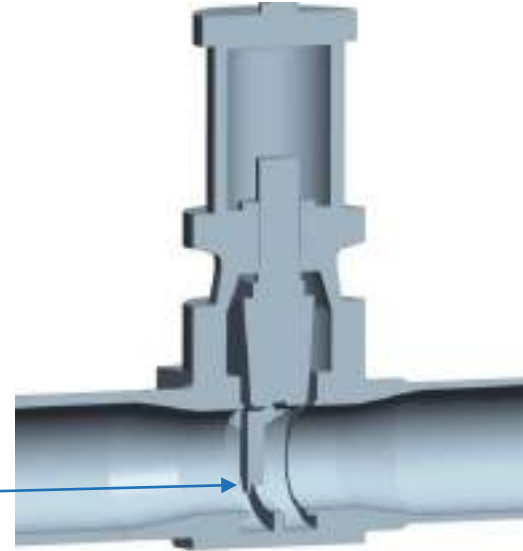


Mode 509 at 663 Hz



Mode 70 658 Hz

Solution

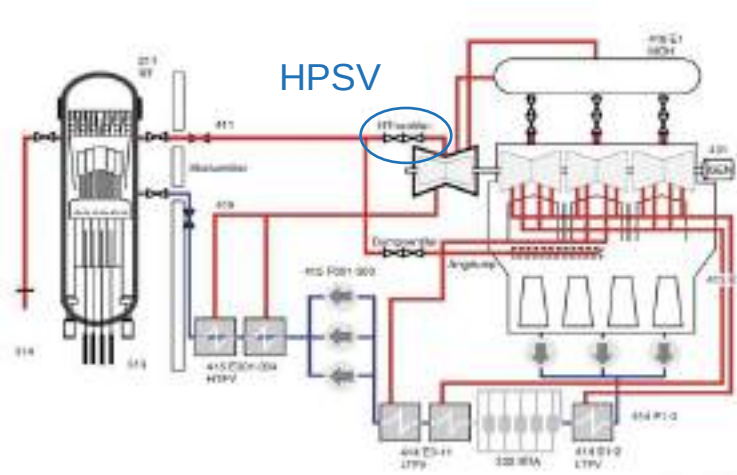
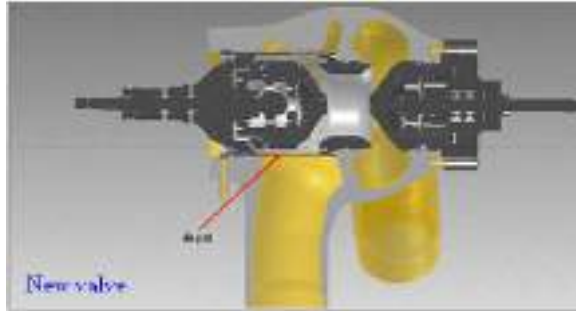
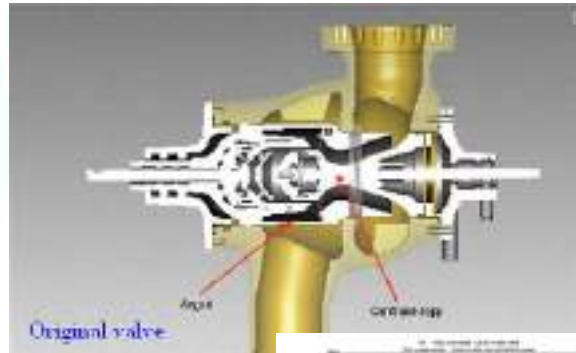
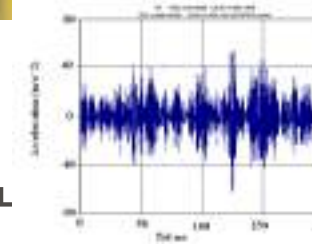


<https://www.fauske.com/blog/revisiting-mitigation-options-for-acoustic-resonances-in-pipe-attachments>

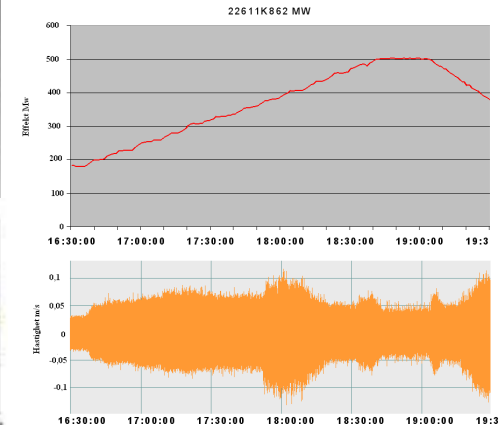
Numerical and experimental analysis of flow-acoustic interactions in an industrial gate valve *Romain Lacombe, Philippe Lafon, Frédéric Daude and Fabien Crouzet*

Flow induced vibrations in HPSV

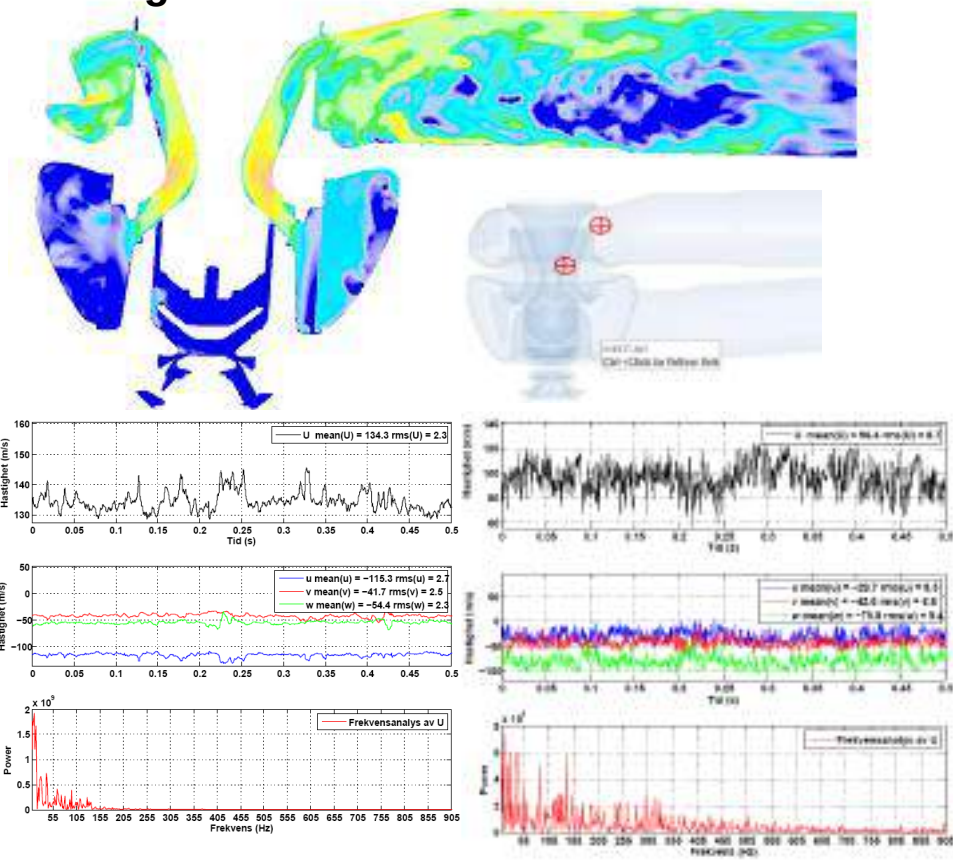
Turbine HPSP valves was refurbished due to a power uprate



High vibration levels on the steam lines and valves with accompanying hammer strokes

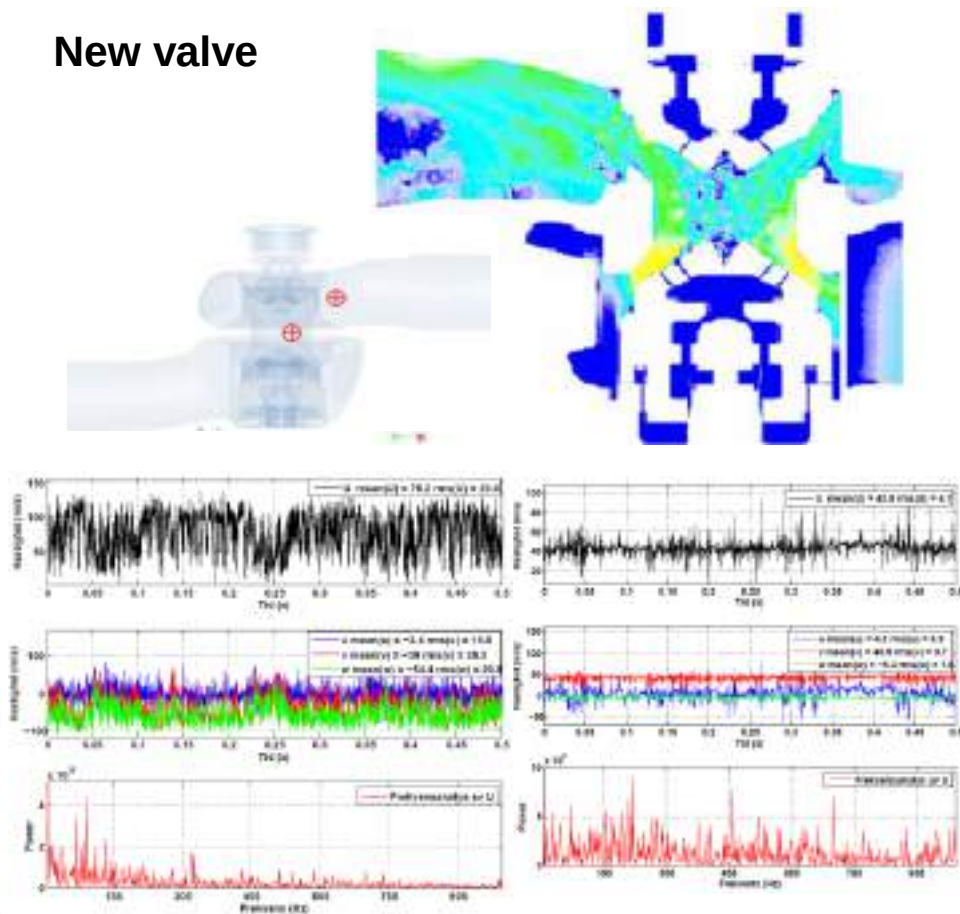


Original valve



- Small fluctuations
- Narrow bandwidth
- max frequency 300-400 Hz

New valve



- Large velocity fluctuations
- High frequencies, broad bandwidth
- High vibration levels (> 35 mm/s)

Ref.: NURETH14-222

Discussion

Operational experience

- “LTO vibration” → wear → monitoring, testing, calculation,...
- Design, power-uprate, component replacement → higher vibration level → damage?

CFD calculations to study FIV great potential to understand the cause of vibrations and is becoming more common in root cause analysis.

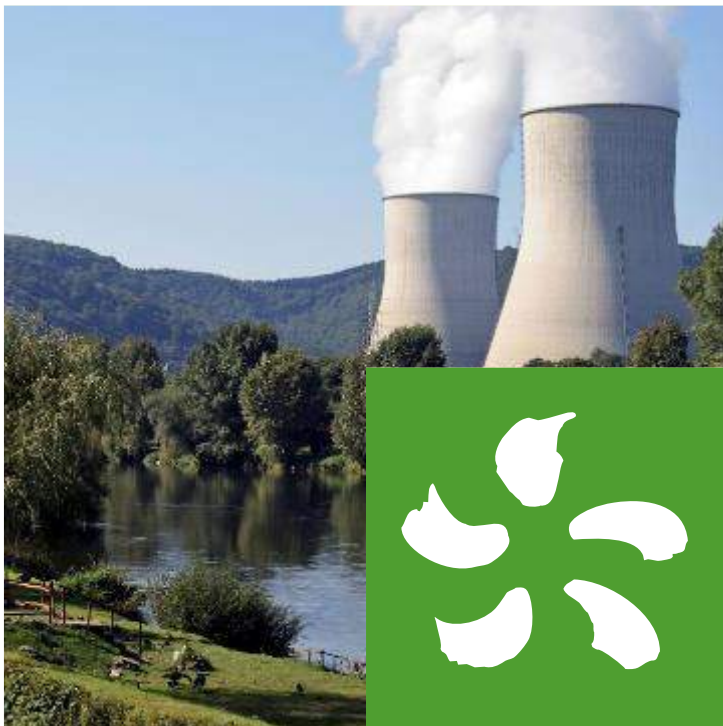
Predictive use of CFD simulations for FIV much more complex

Two-way FSI calculations more and more available with increase of computer capacity and the development of more cost-effective turbulence model

→ **Development of FSI fast running method as in GO-VIKING project**

Questions?





Flow-induced Vibration in NPPs

A focus on vortex- shedding issues

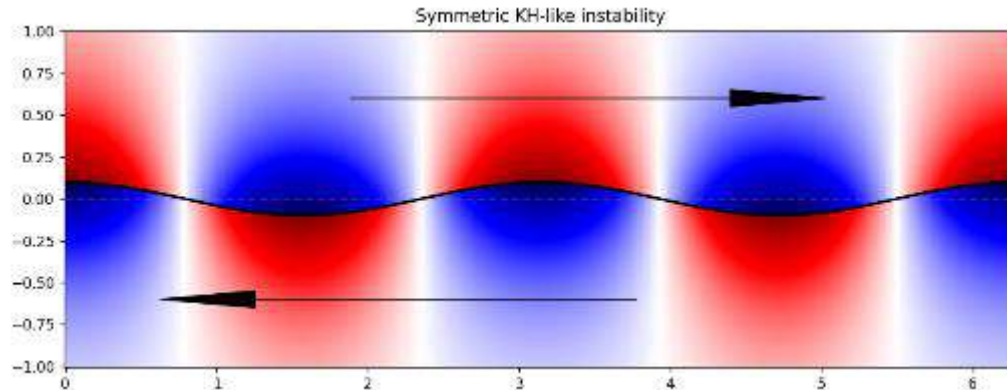
P. Moussou, EDF

Go-Viking final workshop
GRS, Garching, 20 May 2026

Vortex Shedding with Lock-in

Shear flows may undergo self sustained oscillations

Driving mechanism: pressure redistribution after displacement of the interface.



Occurs in:

- Shear flows: Kelvin-Helmholtz mechanism
- Flow against deformable structures: axial flow mechanism

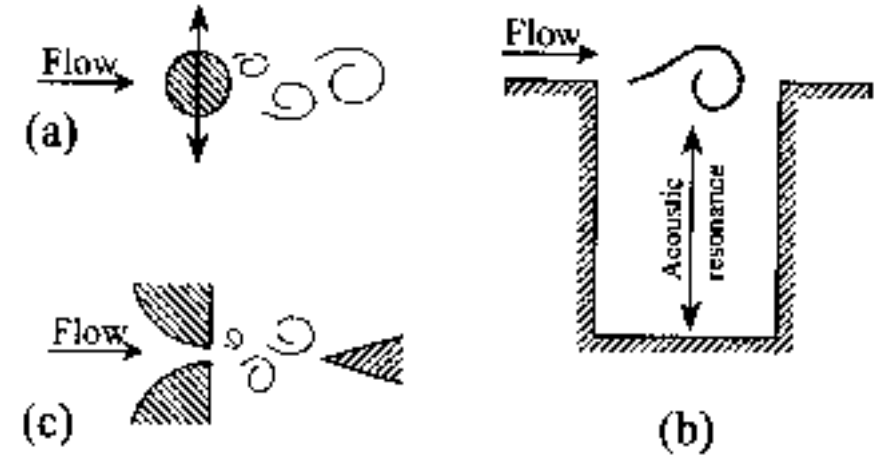


Fig. 3 Basic feedback mechanisms enhancing vorticity (a) structural resonance feedback, (b) acoustic resonance feedback, (c) flow impingement feedback

Flow Induced Vibrations in Power and Process Plant Components - Progress and Prospects

Weaver et al. (2000) *Journal of Pressure Vessel Technology*, ASME, **122** 339



1.

Field observations

From EDF and others

Vortex shedding in orifices

Flexible multihole orifice between a pump and a reducer

A new 'home made' orifice designed to increase the flow velocity of the Spent Fuel Cooling system

Tonal noise caused by vortex-related mechanism observed during commissioning

New design delayed by several months



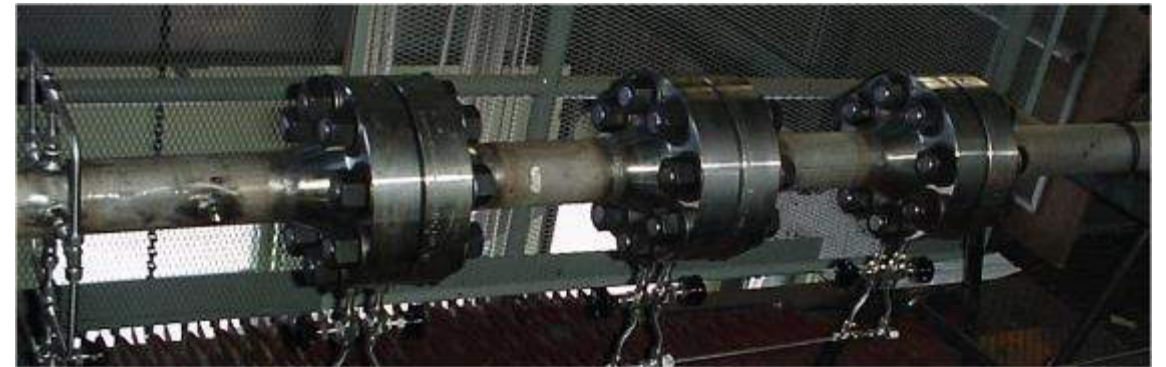
(from Moussou et al. PVP Conf. 2003)

High-Energy Multi-Stage Orifice Assemblies

Multi-stage orifice assemblies for controlled flow with large pressure reduction

Tonal noise caused by vortex-related mechanism

Has caused piping failures within hours



(courtesy of V. Janzen, Canadian National Labs, during a IAEA meeting / ASME meeting, 2014)

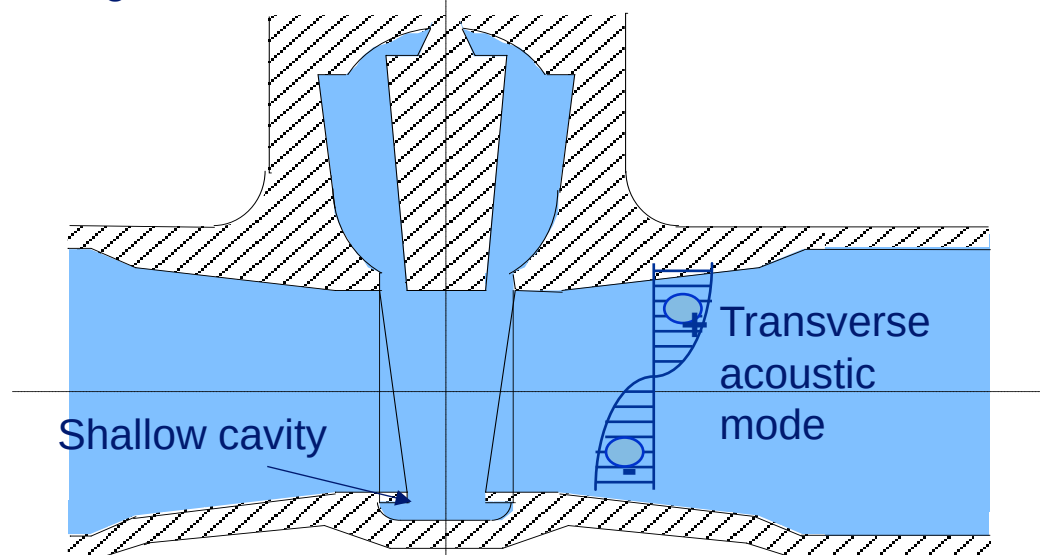
Vortex shedding in gate valves

A shallow cavity effect observed by EDF (and others)

The seat region of a gate valve behaves as a cavity.

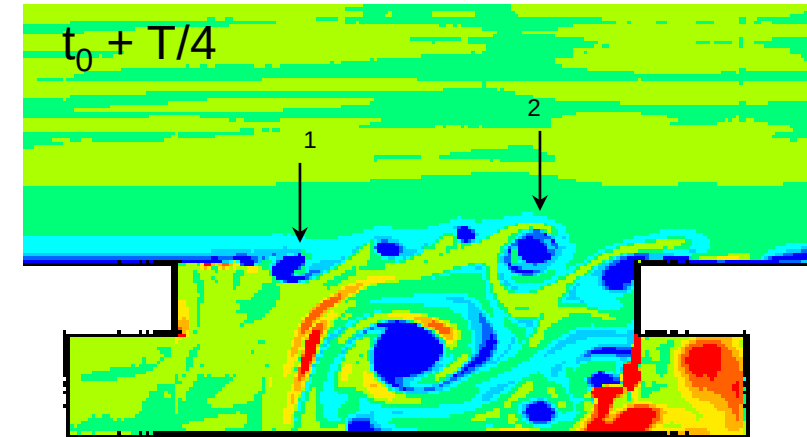
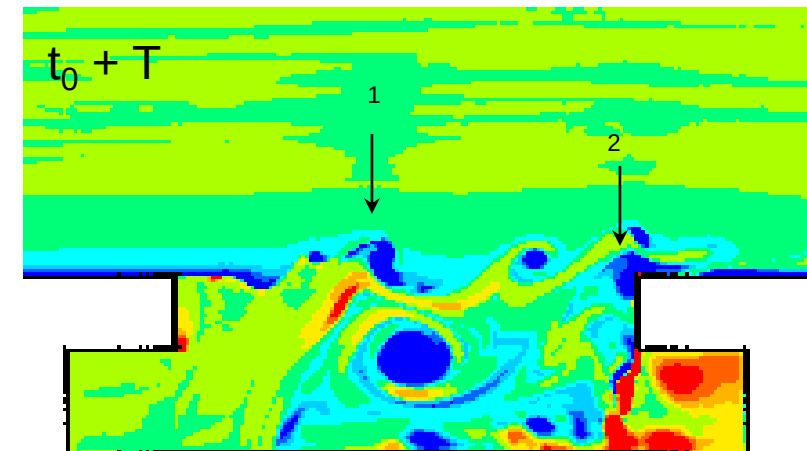
Tonal noise was observed at nominal flow regime with a high acoustic level in the surroundings

Adding a chamfer resolved the issue



See *Aeroacoustical coupling in a ducted shallow cavity and fluid/structure effects on a steam line*, Lafon Ph., Caillaud S., Devos J. P. and Lambert C., *Journal of Fluids and Structures*, 2003, 18

Vorticity field simulation



Vortex shedding in deep cavities

A deep cavity effect observed by Westinghouse (and others)

Vortex shedding across the branch line nozzle

Solved by rounding the nozzle profile (and adding a strut)

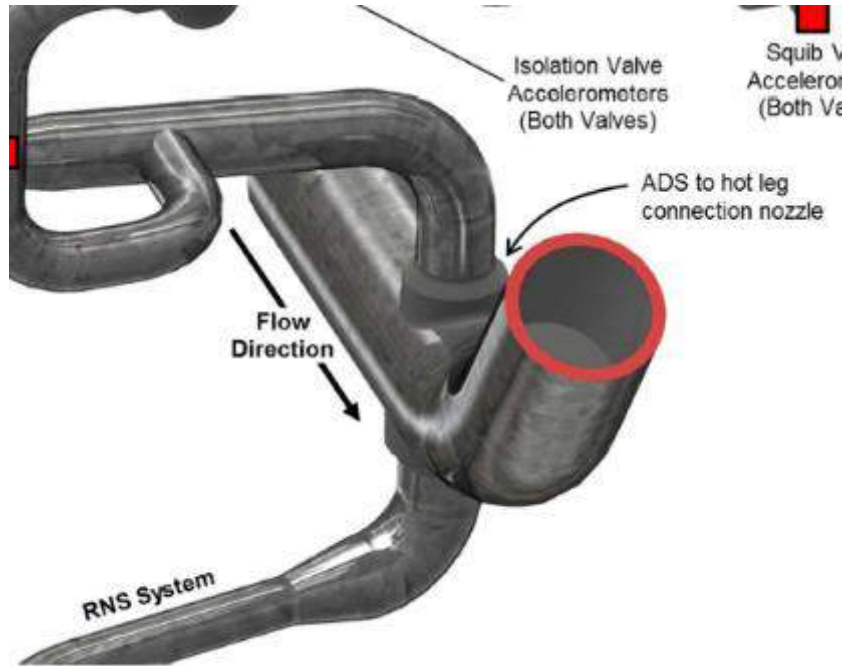
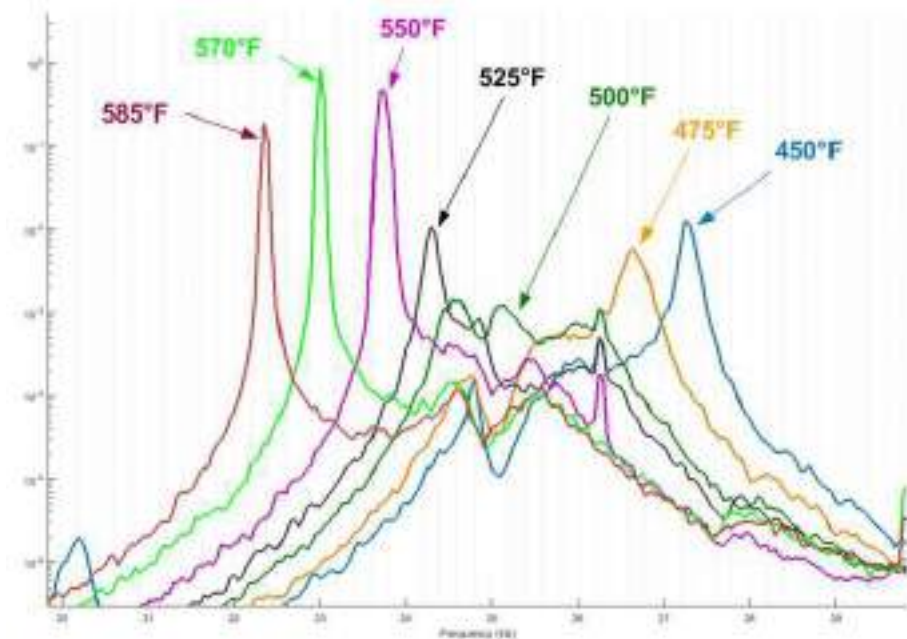


Figure 1 PWR Automatic Depressurization Piping System (ADS)

See *Vibration of a High Energy PWR Piping System Due to Vibro-Acoustic Coupling: Evaluation and Mitigation*, Meyer G. A., Meneely T. K. Koether J. R. Smith S.D. DiBasilio D. C., ASME PVP Conf. 2019, San Antonio

A strong temperature effect

Changing the temperature alters the speed of sound, suggesting a possible acoustic effect



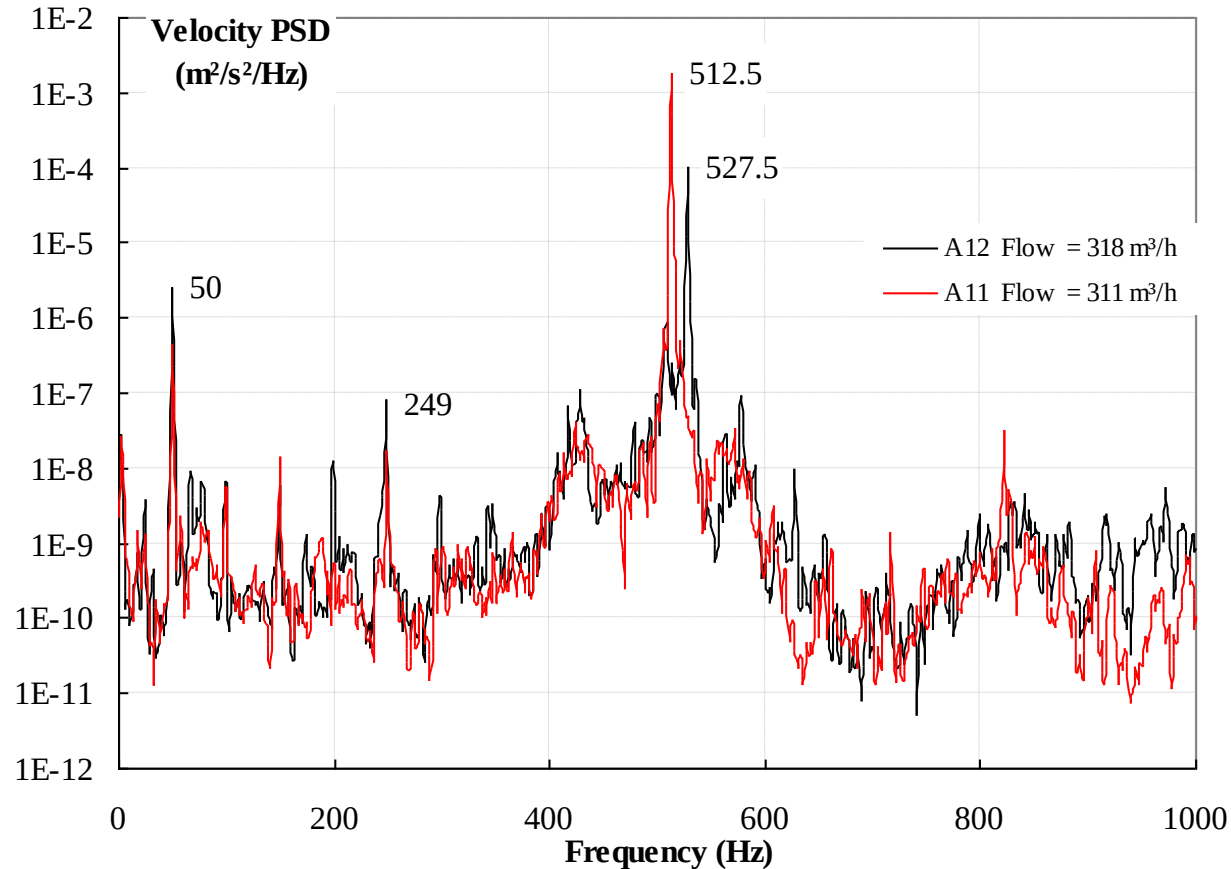


2.

Analysis and experiments

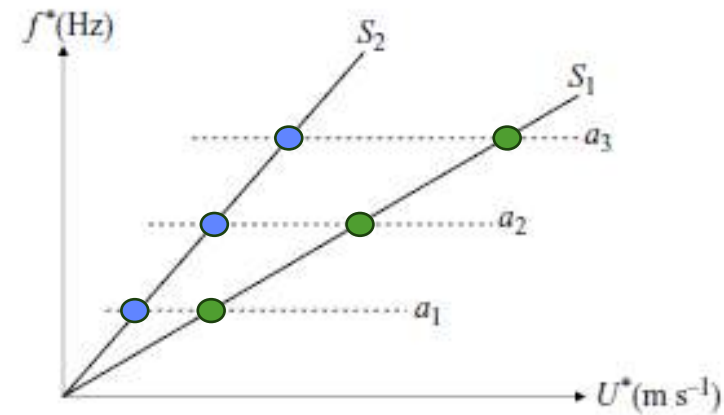
Vortex shedding signature

A sharp peak dependent on the flow velocity



From *Acoustic resonances and trapped modes in pipes and tunnels*,
Hein S. and Koch W., J. Fluid Mech. 2008) 605

Different instability branches ($\text{St} \approx n \times 0.2$) intersect the natural frequencies of the system

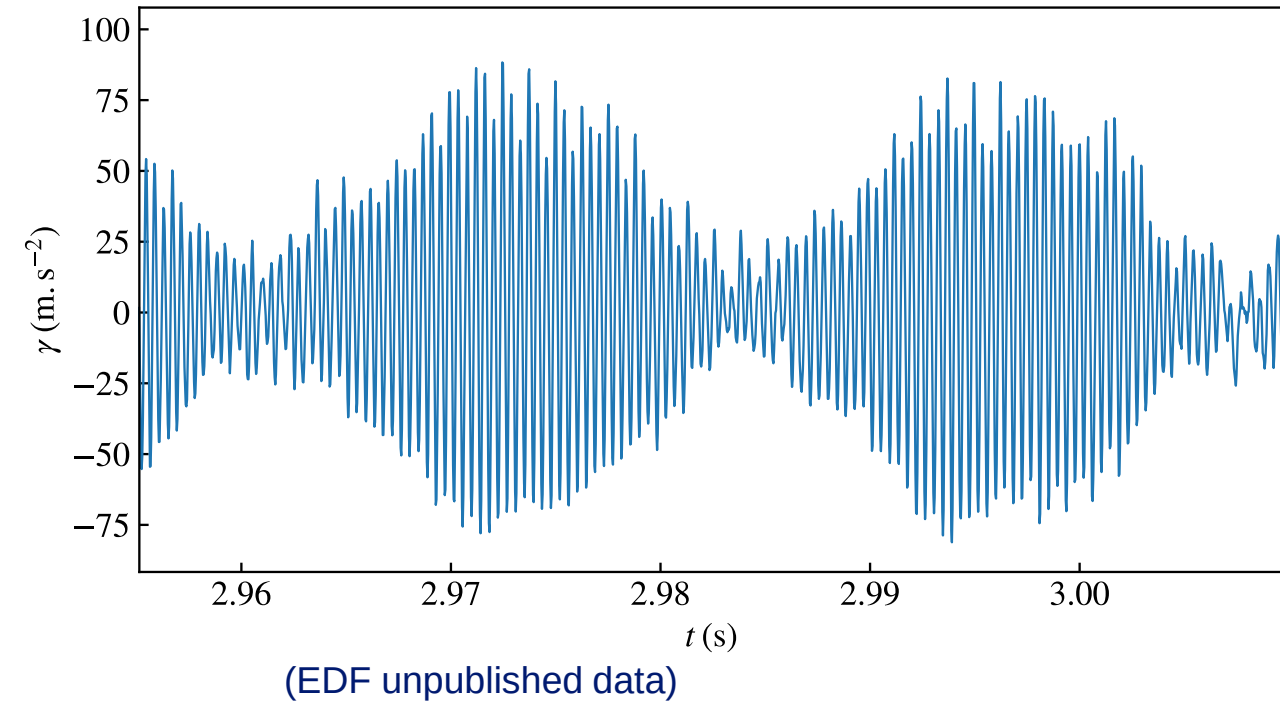


Frequency f^* versus flow velocity U^* for Strouhal sources S and acoustic resonances a (schematically).

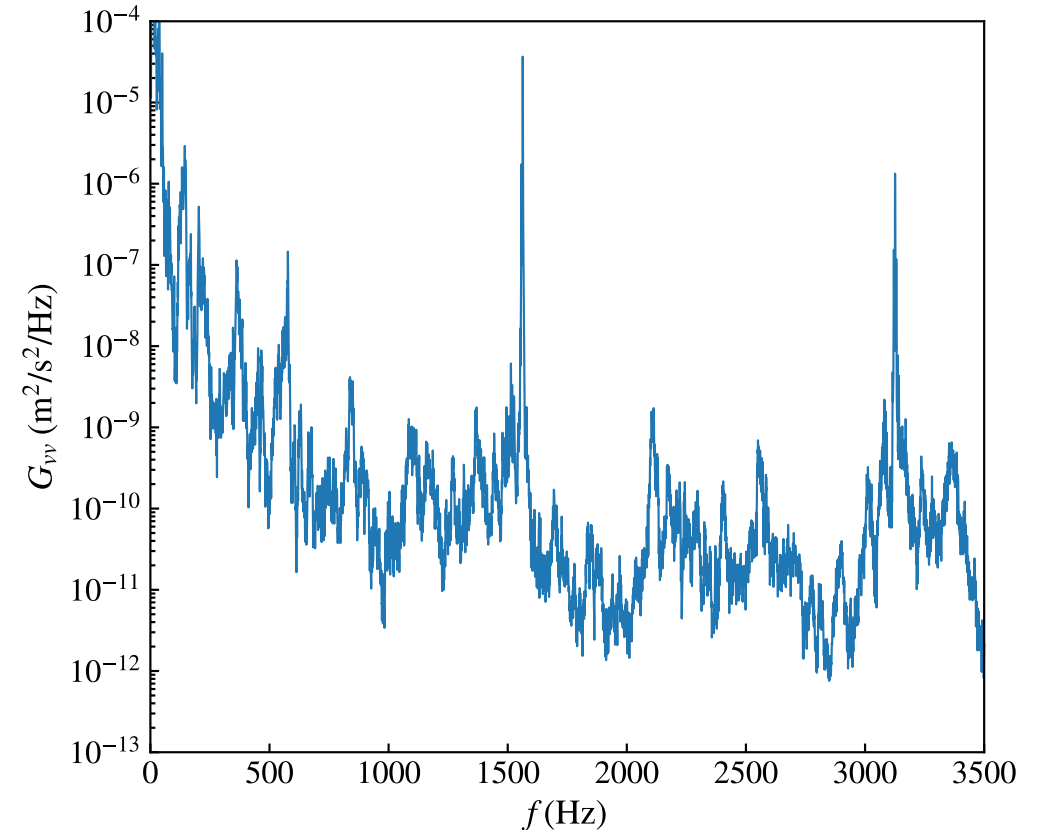
From *Acoustic resonances and trapped modes in pipes and tunnels*,
Hein S. and Koch W., J. Fluid Mech. '2008) 605

Vortex shedding signature

Often visible in the time signal



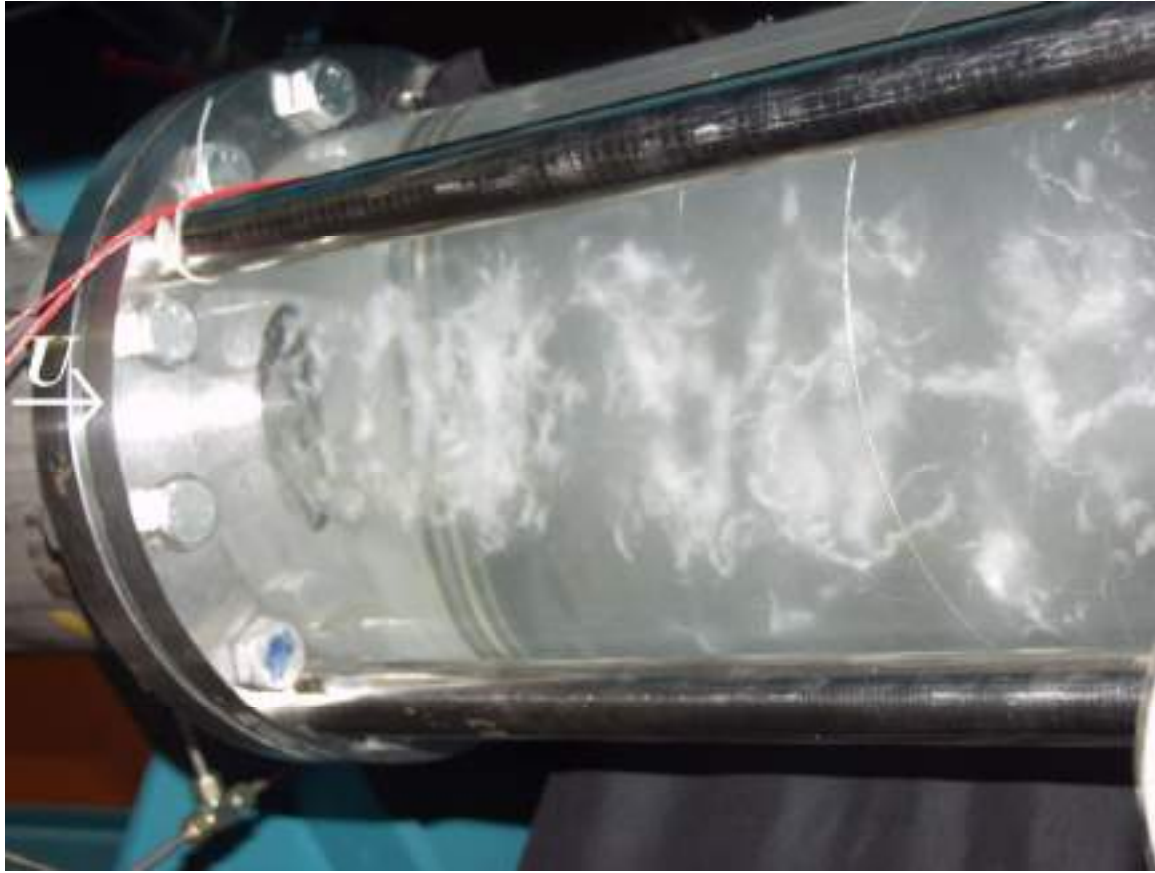
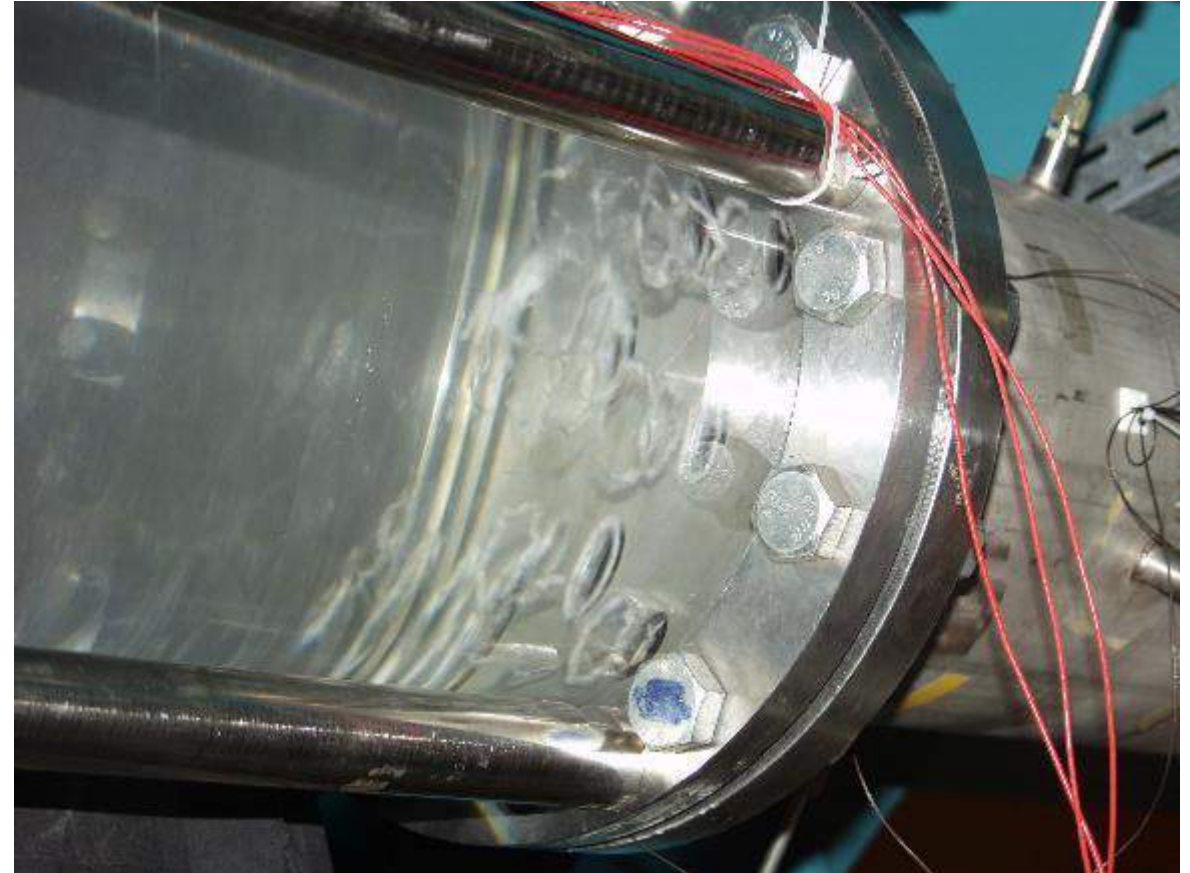
and generating sharp peaks with harmonics



Occurs in a reduced range of flow velocities associated with a Strouhal number $\frac{fL}{U_c} = 0.2 - 0.5$

Synchronized vortex rings downstream an orifice

Cavitation can be used to visualize high vorticity areas: application to a single hole and a multihole orifices



(EDF unpublished measurement, by A. Archer et al.)



3.

**A formal analogy for
illustrating the
feedback effect**

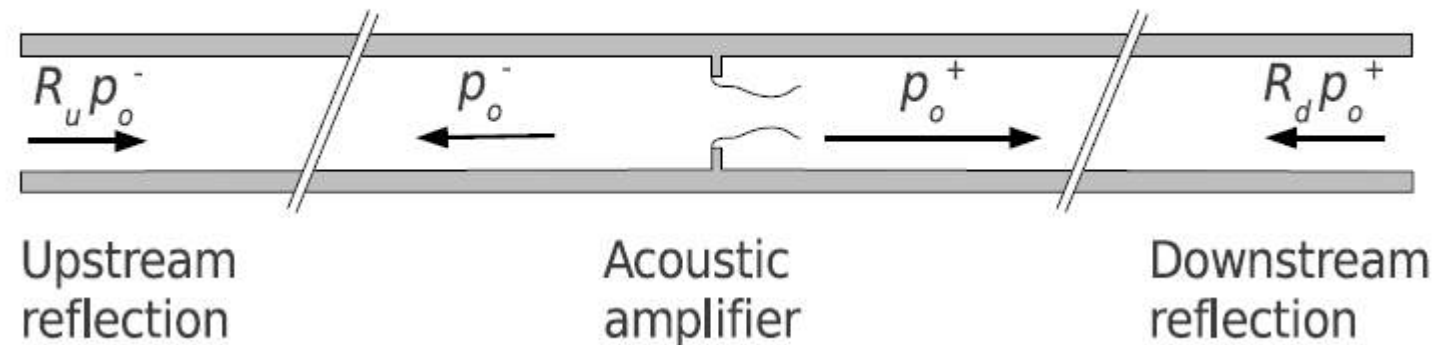
Instability toolbox: an amplifier and some feedback

Just like the Larsen effect, an instability can be triggered by an acoustic amplifier (an orifice) and some feedback (acoustic reverberation)



From P. Moussou, ASME PVP 2014, Anaheim

Plane acoustic waves inside a pipe



Instability toolbox: a criterion and a test case

The Nyquist criterion

Coupling a gain G to a feedback F triggers self-sustained oscillations if one complex root s of

$$G(s) \times F(s) = 1$$

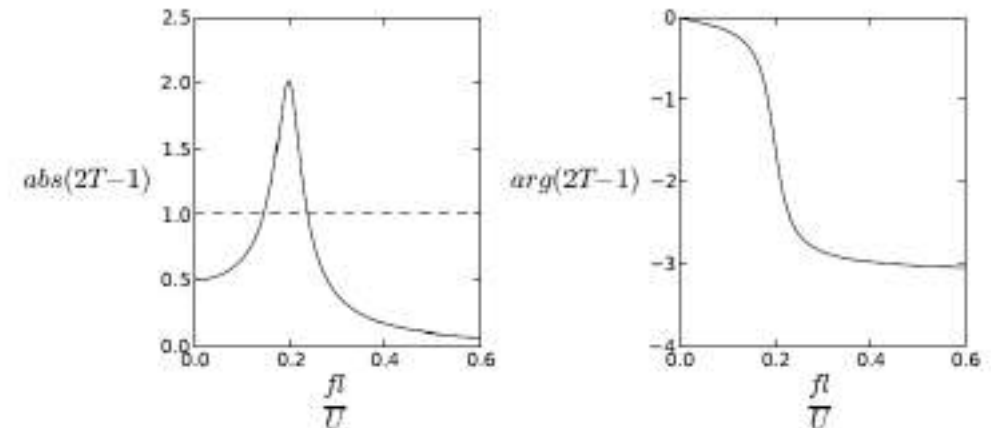
has a positive real part (exponentially diverging solution).

Nyquist criterion: the existence of such a root can be graphically inferred from the loci of $G(\omega) \times F(\omega)$ in harmonic regime.

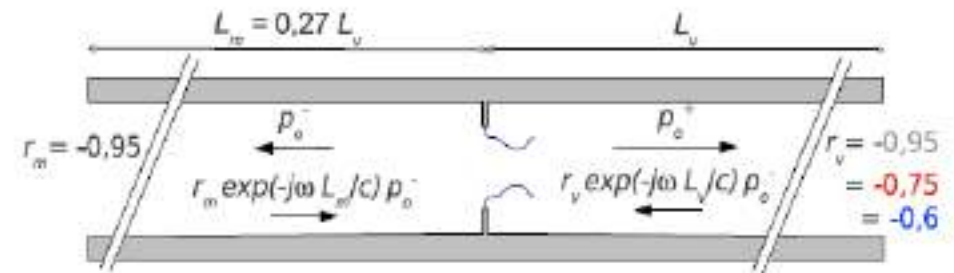
Refs: Nyquist 1935, Sattelmayer and Polifke 2003, Karlson and Åbom 2011

Piping system elementary representation

The gain is equal to $2T - 1$ where T is the acoustic wave transmission coefficient of the orifice, here modeled as single DOF oscillator (inspired by actual measurements)



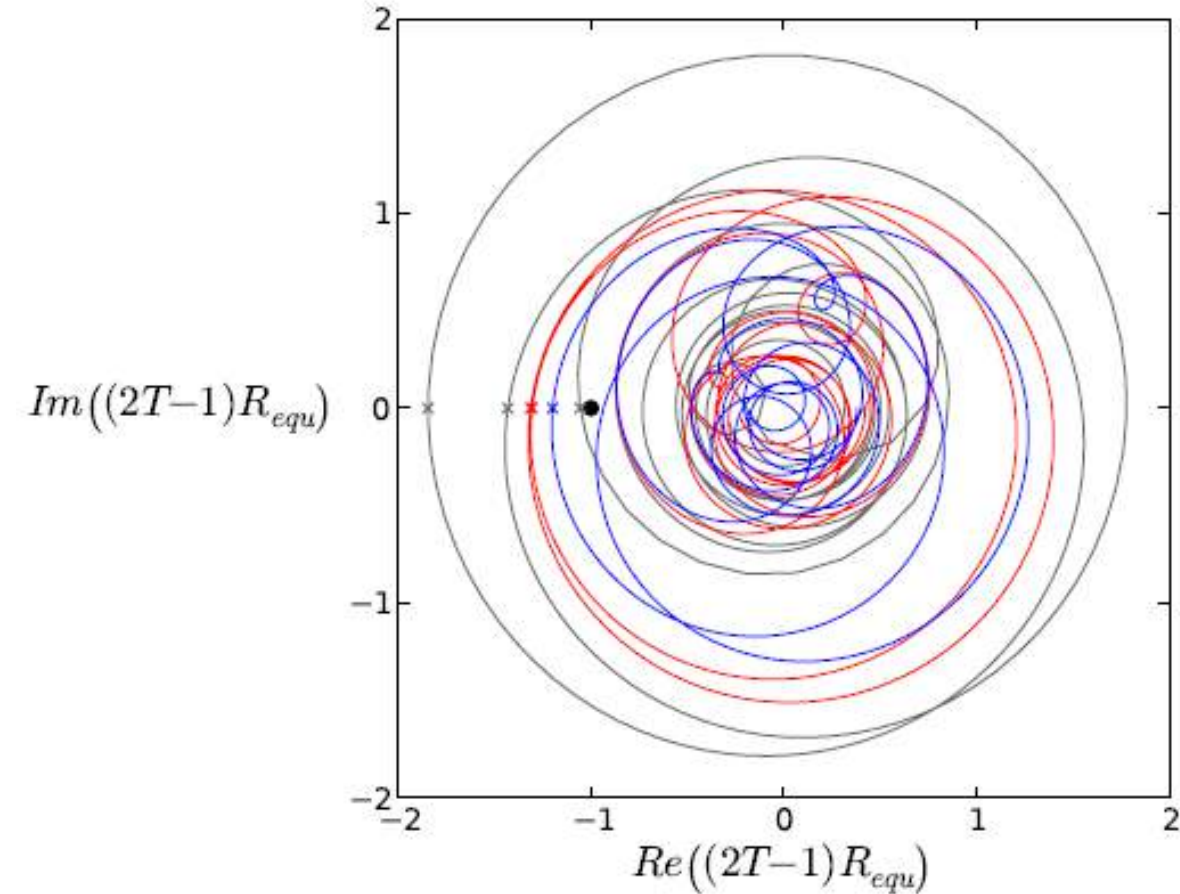
Simple reflection coefficients with variable modulus are arranged upstream and downstream



Instability toolbox: calculation

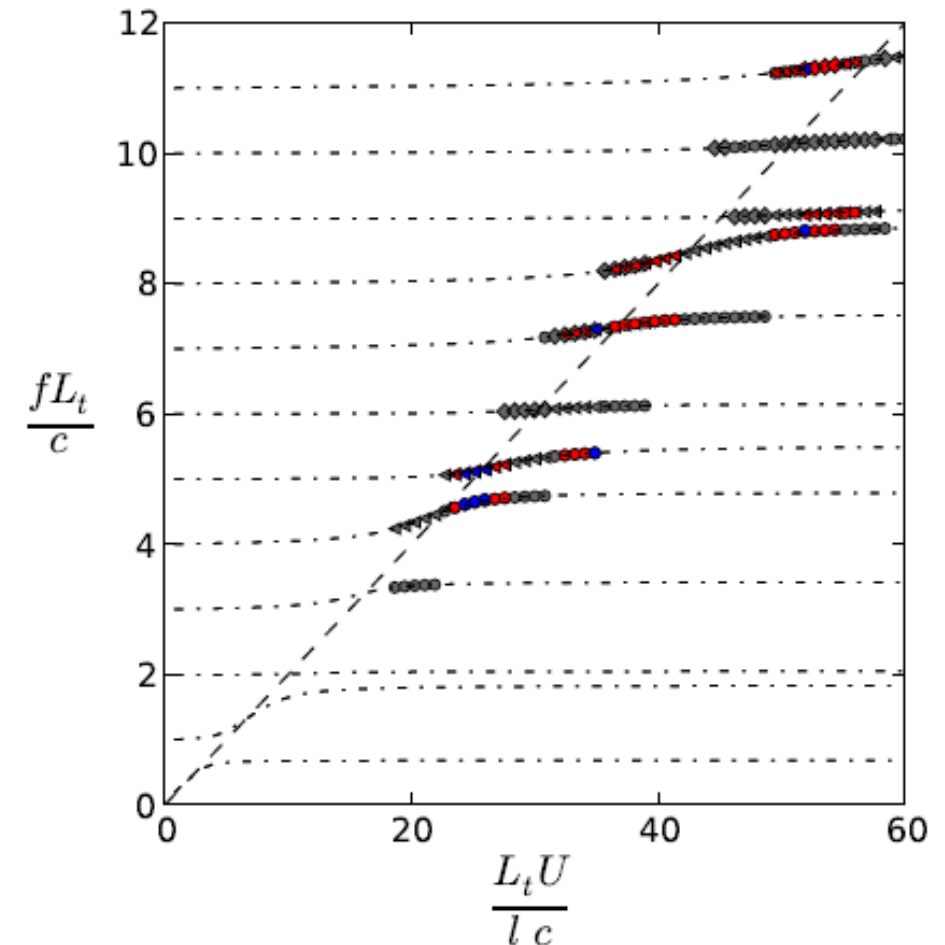
Complex plot of $(2T - 1) \times R_{equ}$

Instability occurs if the plot encircles the complex point $(-1, 0)$.



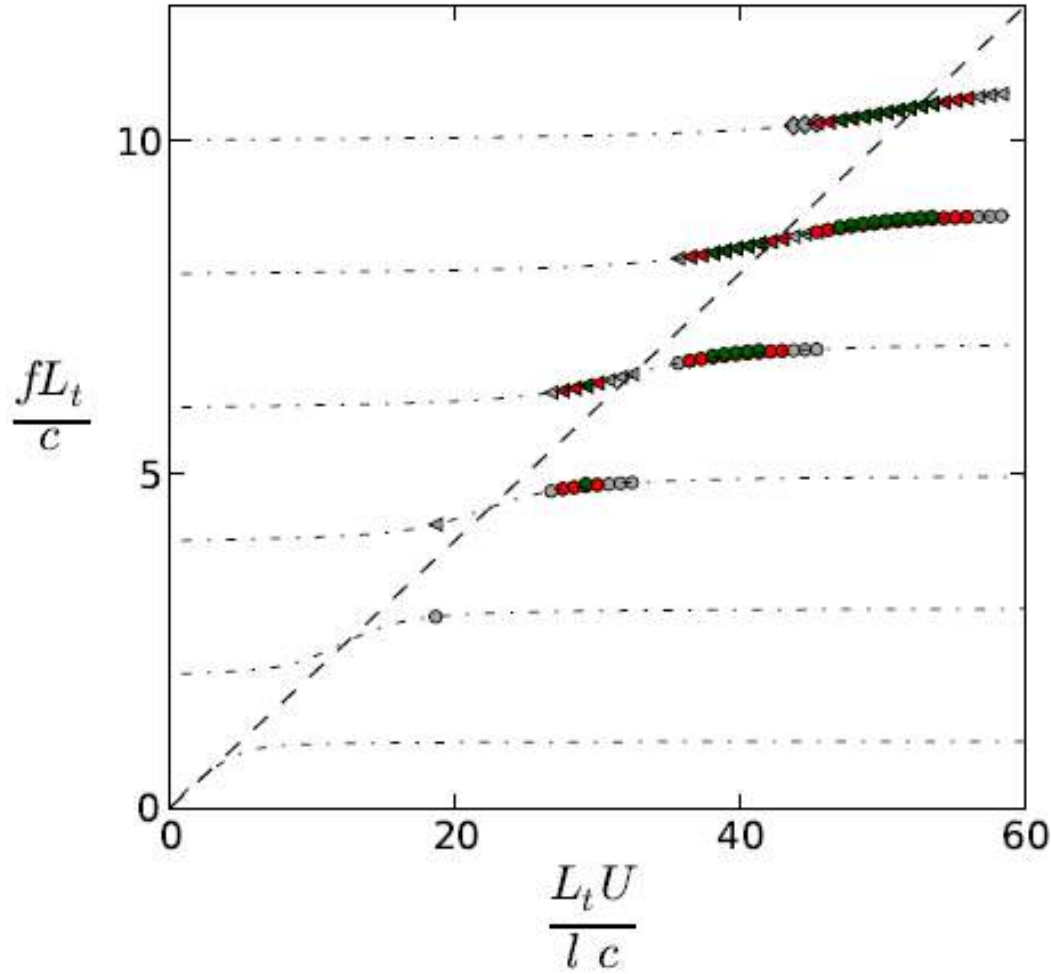
Instability frequencies against the reduced velocity

Everything aligns along $St \sim 0.2$



Instability toolbox: results

Results for different upstream and downstream lengths



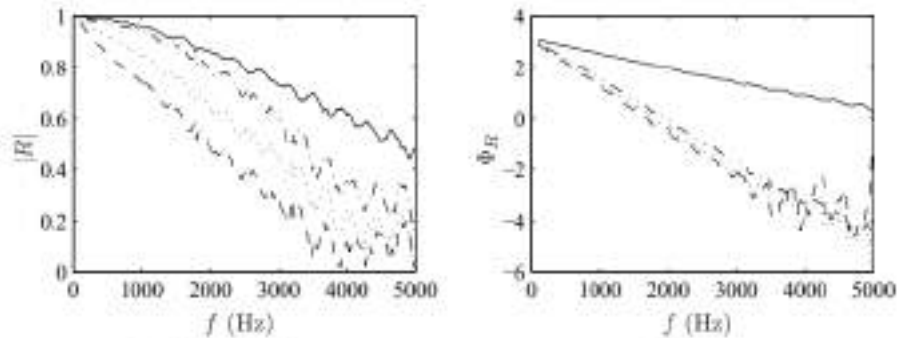
Several experimental results are retrieved

- Multiple instability branches are observed
- Transitions to higher branches occur as the fluid velocity increases
- Instability frequencies cluster around Strouhal ≈ 0.2 (as designed)
- Instability frequencies are governed by the acoustic field
- Within each branch, the frequency varies with flow velocity due to device-induced frequency shifts
- No instability is observed when the device is located near an acoustic velocity node

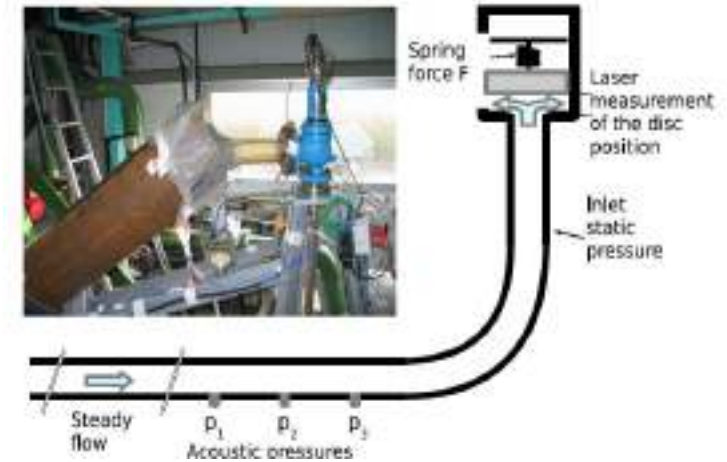
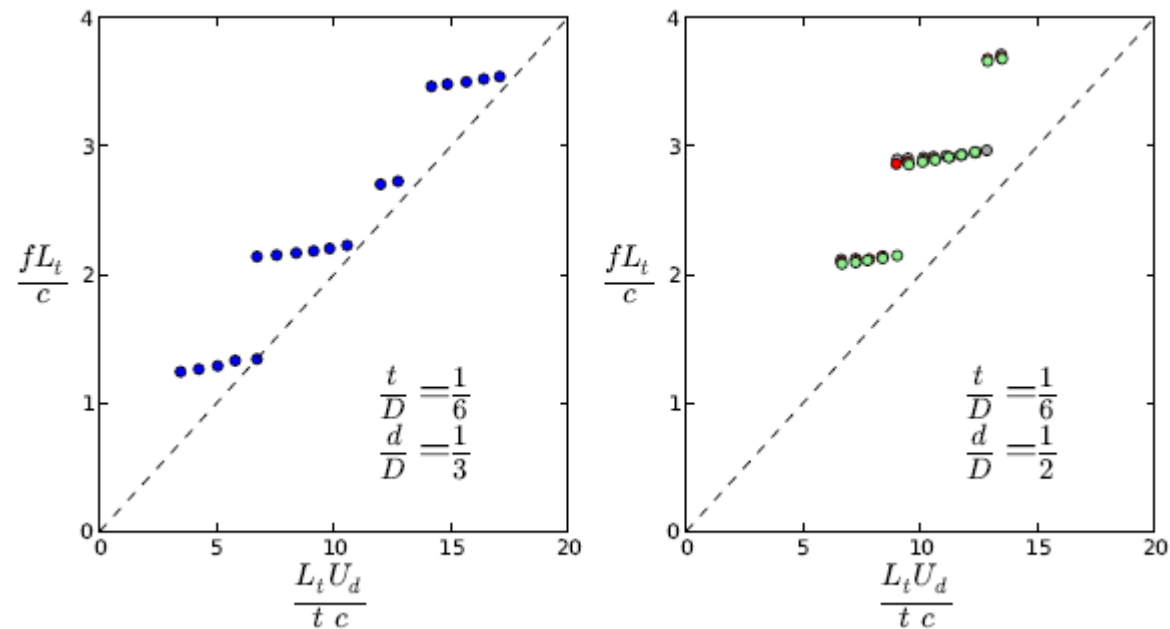
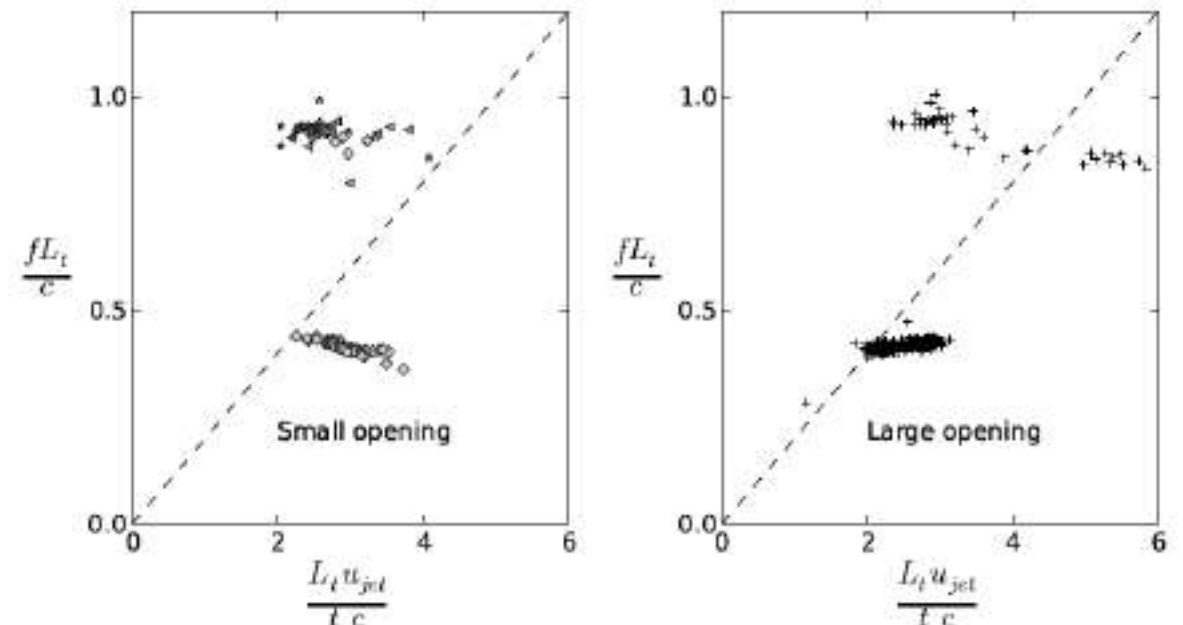
Instability toolbox: applications

Whistling of thin orifices in a wind tunnel

Chatter of a safety relief valve at partial opening



Example of reverberating termination used to trigger the instability





4.

A few recommendations for CFD

Several options for instabilities

Focus on the passive response (gain estimation)

U-RANS is applicable to orifices and tee junctions under plane-wave acoustic excitation

Potentially applicable using incompressible solvers

Difficult to apply to shallow cavities subjected to non-planar acoustic modes

Requires a Nyquist stability analysis

Does not predict the limit-cycle amplitude

→ **Well suited to determine the range of possible whistling conditions**

Directly resolves acoustic-flow interactions

Requires modeling of the entire system

Often necessitates LES in practical configurations

Requires a compressible solver

Boundary conditions can be challenging to define (beyond simple Sommerfeld conditions)

Varying the boundary conditions requires a new computation each time

→ **Represents a significant computational and modeling challenge!**

Merci

Contact

Pierre Moussou
pierre.moussou@edf.fr



D1.1: Stakeholders' outreach

WP 1

D. Vivaldi (ASNR), S. Benhamadouche (EDF)



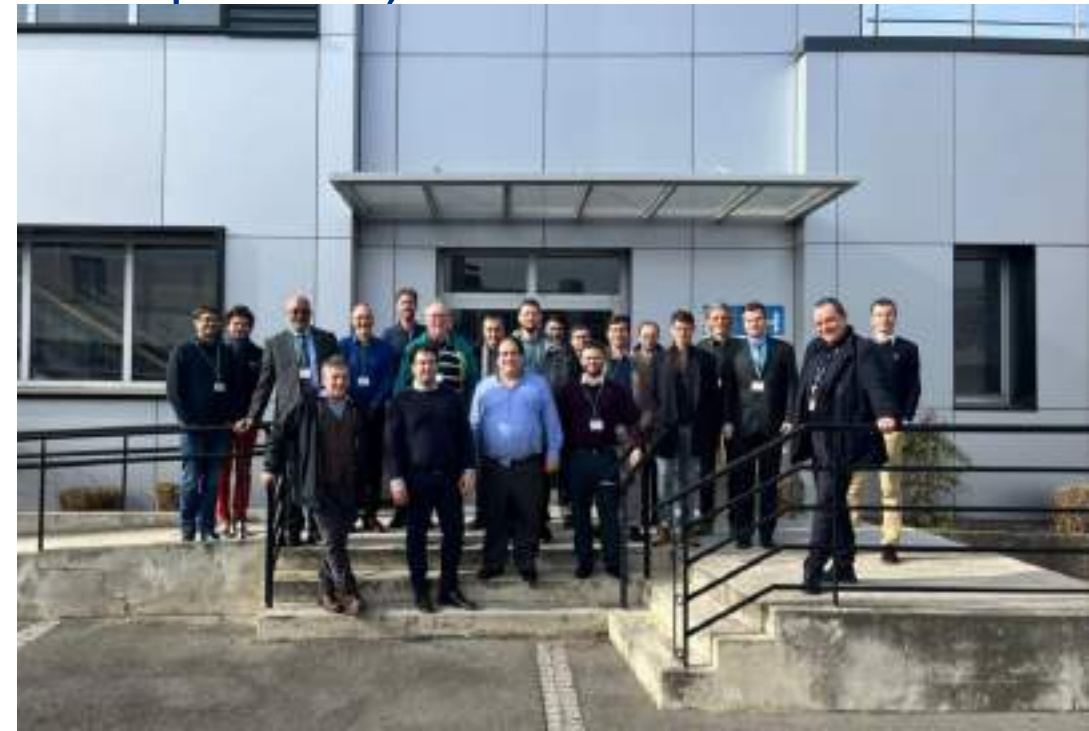
Funded by
the European Union

Stakeholders' workshop



The workshop was a “deliverable” of WP1

- Goal: collect needs and expectations from stakeholders in terms of FIV for NPPs;
- Organized by EDF and ASNR (former IRSN) starting from Sept. 2022;
- Hosted by EDF R&D Chatou, on 16 Feb. 2023.
- Who accepted to join and do a presentation:
 - **Utilities**: EDF (France), EDF UK (UK), TVO (Finland), Vattenfall and Uniper Energy (Sweden, involving 3 nuclear power plants), EPRI (USA).
 - **Vendors**: Framatome (France & Germany), Westinghouse (USA)
 - **Regulatory/TSOs**: CNSC (Canada), GRS (Germany), IRSN (France), NRC (USA)



Associated deliverable

Resumes the content of the presentations given by participants:

- Organized into 4 main paragraphs:
 1. FIV in fuel assemblies;
 2. FIV in steam generators;
 3. FIV in other configurations;
 4. FIV and regulator
- All presentations available for the GO-VIKING community as a separate folder in Flexx.
- Agreement received by all workshop participants for the content of the deliverable and for sharing their presentations.
- Submitted on time (M12).

FIV in fuel assemblies



- **Mechanisms discussed:**

- Fretting in PWRs
- Baffle-jetting in PWRs
- Pin-brace wear in AGRs

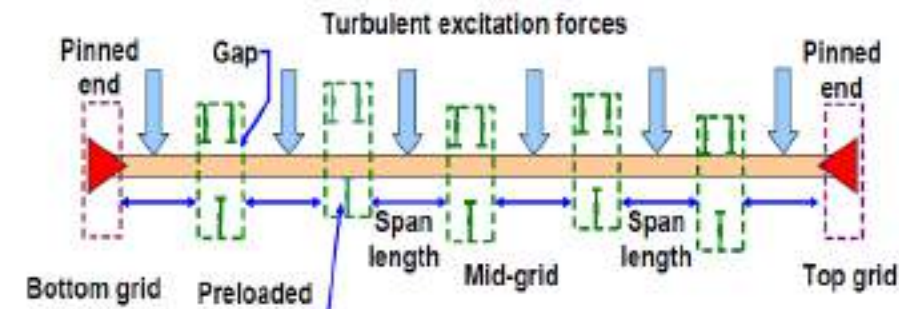
- **State of the art of num. sim.:**

- Numerical tools feasible for turbulence induced vibr., since no strong FSI coupling required;
- Non-linear beam models for the structure → estimation of work rate;
- **Must be fed with the turbulent excitation:**
 1. Reference spectra;
 2. **LES (with wall function) results feasible for FA spans or inlet zones.**
- 1-way FSI coupling proved to lead to reasonable results;
- LES not applicable at the core scale and CPU-time high compared to design process;
- **CFD potential use to establish/extend database that can replace reference spectra.**

- **Industry needs, requirements for new num. tools, expect. vs GO-VIKING:**

- Exp. data for LES validation:
 - **Time-resolved PIV of Cantilever Beam (WP2);**
 - **Time-averaged LDA of ALAIN (WP2).**
- FIV mechanisms requiring strong coupling (self-induced resonance, baffle-jetting, ...) still an open field in R&D:
 - **FSI coupled methods** planned to simulate WP2 benchmarks.

VITRAN FA MODEL:

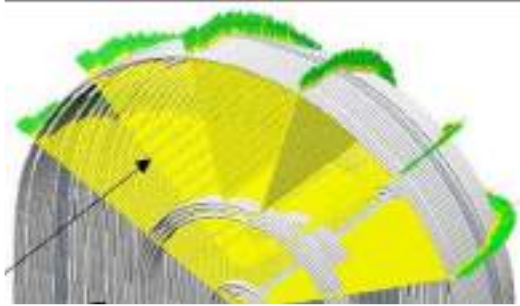


FIV in steam generators

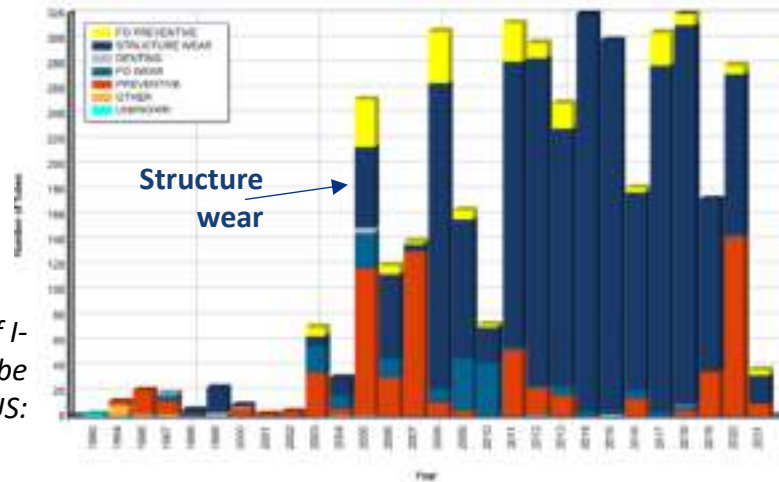


• Mechanisms discussed:

- Tube fatigue cracking (fluid-el. instab.)
- Tube wear



Number of I-690TT SG tube repairs in the US:



• State of the art of num. sim.:

- Porous media approach component scale codes (~30,000 cells) calculating local flow conditions (vel., dens.) along the tube length;
 - Exp. results largely employed to determine FIV empirical constants, lift coeff., added mass, damping, etc.;
 - Different support conditions taken into account.
- R&D in Canada:
 - Mono-dimensional flow channel approach coupled to beam models;
 - R&D on support condition impacts (models, impact of clearance, ...);
 - URANS drift-flux 3D approach for two-phase bubbly flow.
- R&D in France:
 - VISCACHE experiments (EDF);
 - CFD (single and two-phase), strong FSI coupling (EDF).

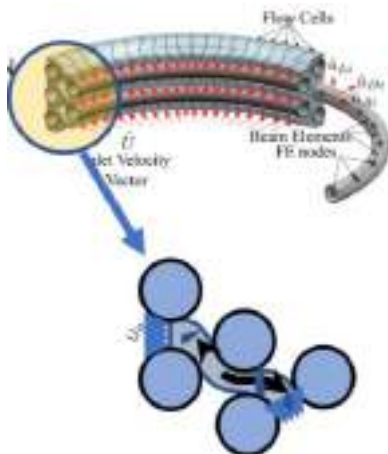
• Industry needs, requirements for new num. tools, expect. vs GO-VIKING:

- Exp. CFD-grade results for void fraction higher than bubbly-flow to assess CFD results:
 - TREFLE and AMOVI (WP4)
 - Two-phase CFD “all-flow-regime” approaches (WP4)
- Turbulence modelling in single-phase:
 - TITAN and GOKSTAD (WP3)
- Validation of flow resistance models in porous media component scale codes:
 - Measurements of vel. and void fraction beneficial (TREFLE and AMOVI – WP4)
- Study of the tube interaction with supporting structures;
- Develop. of SG FIV standards and guidelines.

Tube fatigue cracking at top TSP



Tube-to-AVB wear



neptune.cfd

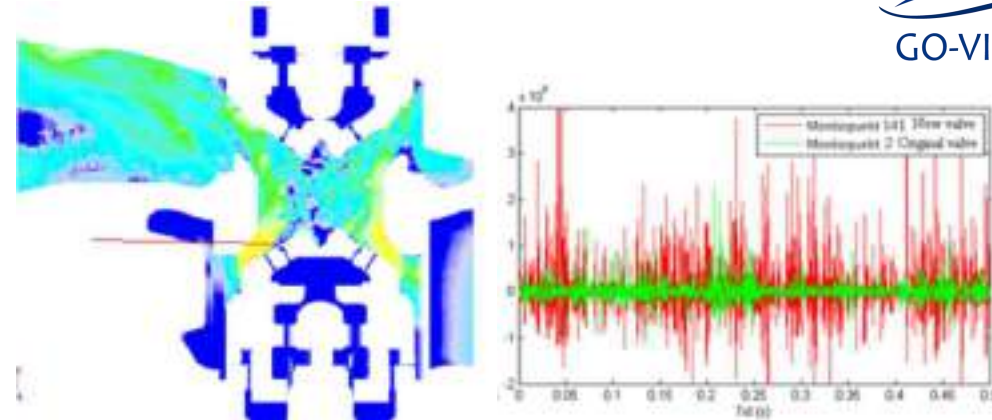
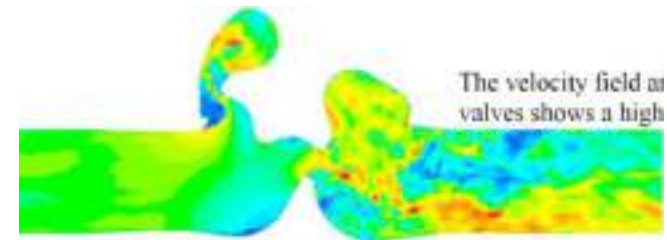
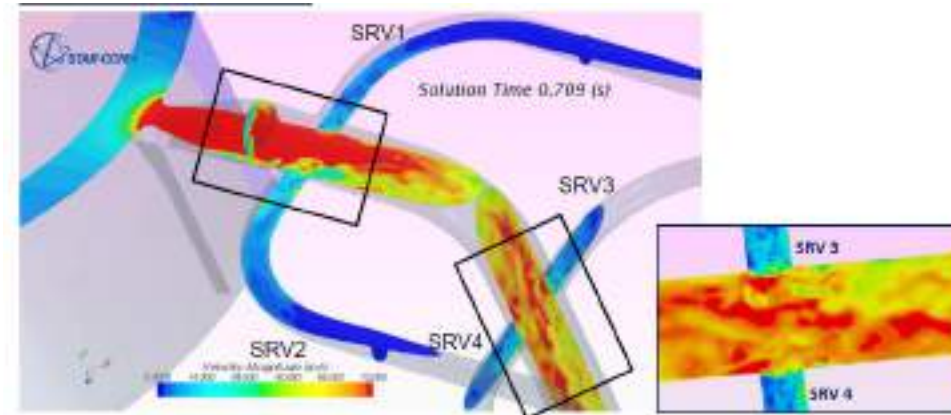


FIV in other configurations



- Water/steam lines:

- Vibr. of water feed line: pump pulsation supposed to excite the acoustic modes;
- Vibr. of steam line downstream of a PWR SG: pressure pulsation created in the SG outlet nozzle;
- Vibr. in the main steam line of a BWR: shear layer instability causing pressure peaks;
- Vibr. in steam stop control valves in a BWR: high vel. fluctuations inside the valve;
- **CFD often employed** to support the root cause analysis.

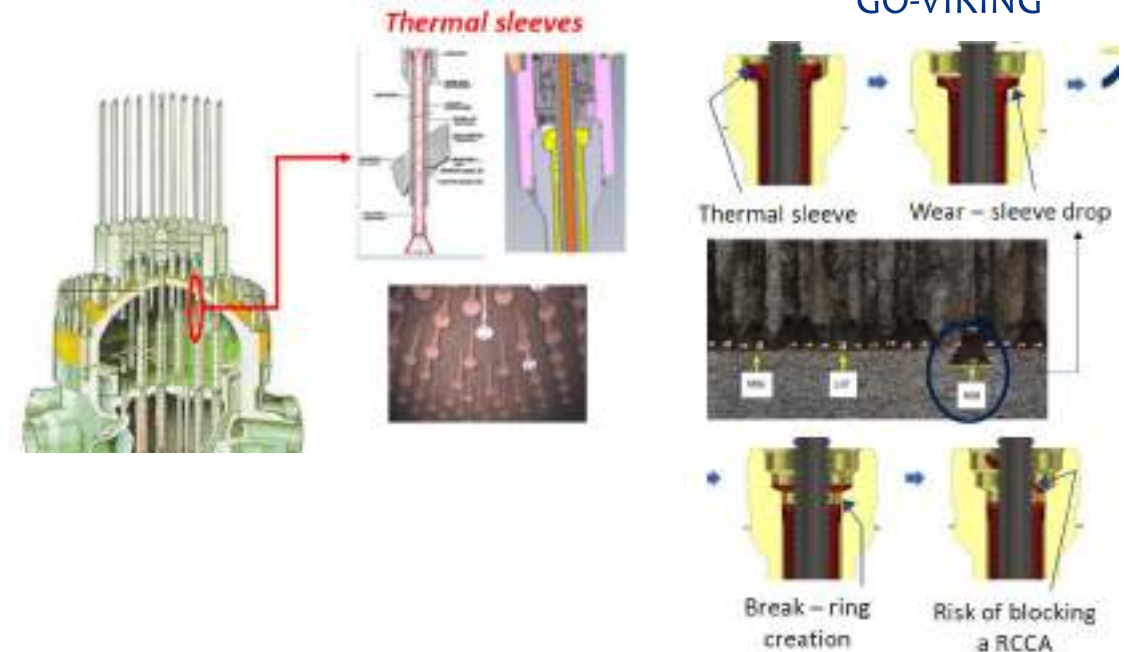
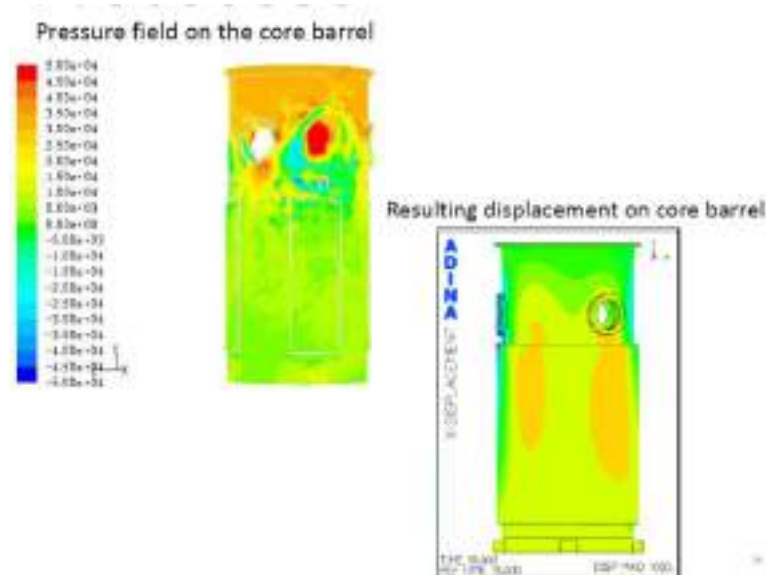
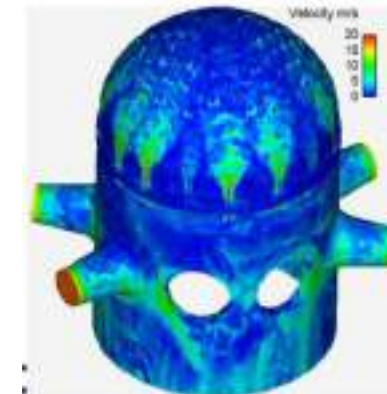
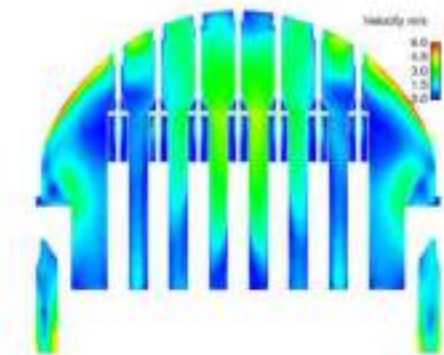


FIV in other configurations



- **RPV internals:**

- Wear of thermal sleeves in PWRs: CFD analysis in support;
- Guide tube wear: preliminary CFD;
- Vibr. of the core barrel.: 1-way FSI simulations (CFD + FEM).



FIV and regulators

- Regulatory requirements for num. tools

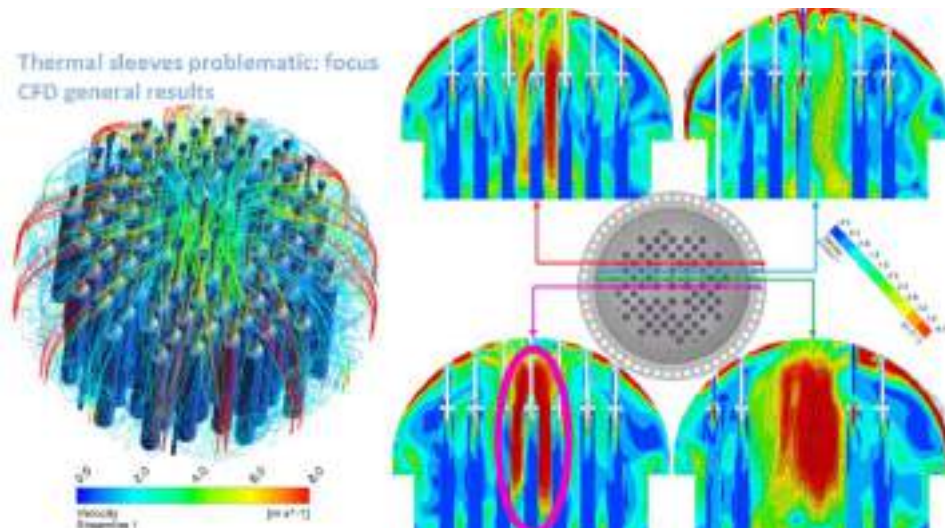
- **NRC (USA)** requires tools developed under software quality assurance plan and undergo V&V-UQ activities:
 - Graded sets of guidelines exist for different applications.
- **ASN (France)** requires calculation and modelling tools qualified for the fields in which they are used (V&V and UQ):
 - “Guide 28”

- CFD and regulatory:

- German CFD network: FIV sim. with CFD-CSM tools is mentioned;
- FIV num. analyses realized by GRS starting from 2015;
- Current challenges: lack of guidelines, CPU-time, V&V through CFD-grade exp., regulatory acceptance...
- IRSN: preliminary study of thermal sleeve wear realized by IRSN: 1-way CFD-FEM analysis.

- Examples of FIV in the German safety requirements:

- « FIV » mentionned at several levels of the regulatory hierarchy:
 - Ex. « [...] *mechanical loading coming from long-term influences of vibrational stresses and from vibrations caused by fluid forces and earthquakes*” in the KTA document “Components if the Reactor Pressure Boundary of Light Water Reactors, Part 2: Design and Analysis”





D1.2: Literature review

WP 1

NRG, GRS, EDF, ASNR



Funded by
the European Union

Deliverable D1.2



FIV phenomena in NPPs

1) FIV in Fuel Assemblies (ASNR, GRS)

- Grid-to-rod fretting wear
- Bow
- Baffle jetting

2) Experiments and numerical methods for Fas FIV (NRG)

- Experimental results on FIV in Fas
- Analytical methods
- Numerical simulations

1) FIV in Steam Generators (EDF)

- Experimental results in single-phase
- Experimental results in two-phase
- Numerical simulations

Published articles



Experimental and numerical contributions on flow-induced vibration in steam-generator-like tube bundles: A review

W. Benguigui^{a,b,*}, S. Benhamadouche^c, F. Beltran^d, M. Hassan^e

^a Fluid Mechanics, Energy and Environment Department, EDF R&D, 6 Quai Watier, 78400 Châtenay, France

^b IMHM UMR EDF/CNRS/CEA/ENSTA ParisTech 9219, EDF R&D, 7 boulevard Gaspard Monge, 91120 Palaiseau, France

^c School of Engineering, University of Guelph, Guelph, ON, Canada



Fluid–structure interactions in nuclear power plants: Review of some main phenomena and analysis of the related safety aspects

Daniele Vivaldi^{a,*}, Angel Papukchiev^b, Clément Viron^c, Stéphanie Graff^d, Mikael Kountchou^e

^a Autorité de sûreté nucléaire et de radioprotection (ASNR), PSN-RS/SEMMA/ASMA F-33115, St Paul-la-Forêt, France

^b Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Germany

^c Autorité de sûreté nucléaire et de radioprotection (ASNR), PSN-RS/SEMMA/DMFC F-92265, Fontenay-aux-Roses, France

^d Autorité de sûreté nucléaire et de radioprotection (ASNR), PSN-RS/SEMMA/BSMC F-92260, Fontenay-aux-Roses, France







GO-VIKING

Thank you!



www.go-viking.eu



Funded by the
European Union



ALAIN experiment and its analysis using different FSI approaches

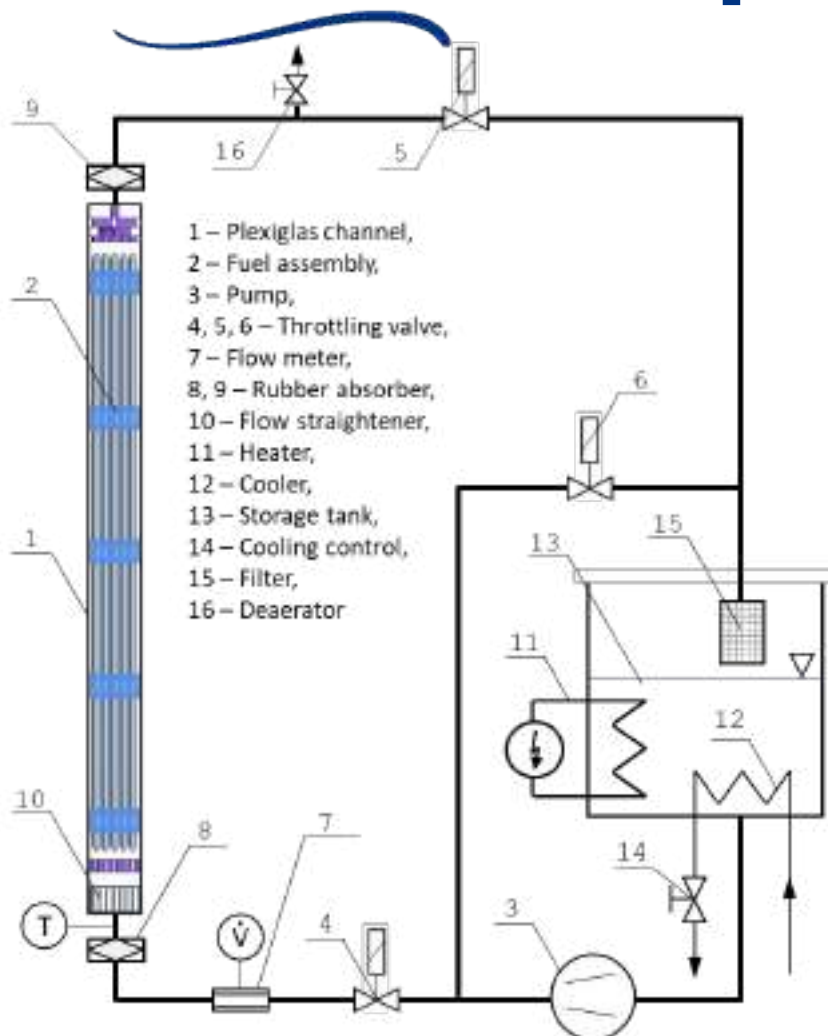
Experimental studies

GO-VIKING Final Workshop Garching 20-22 May 2026, Hidajet Hadžić, Framatome GmbH



Funded by
the European Union

ALAIN loop overview



- FIV tests performed by Framatome in old mini test loop in 2007
- Objective to study effects of different SG design on FIV
- Reduced-scale fuel assemblies used to facilitate the studies
-

ALAIN Test Loop
Technical Center Erlangen
Framatome

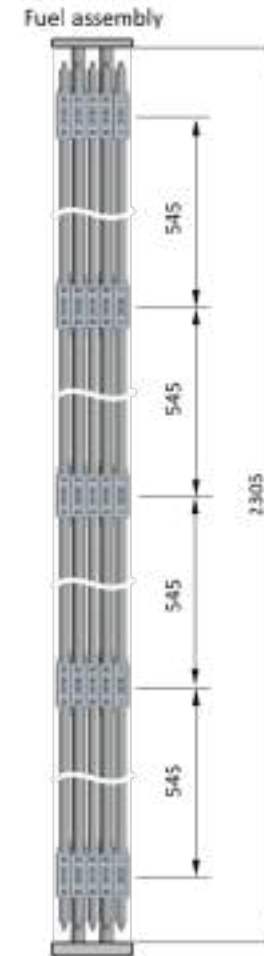
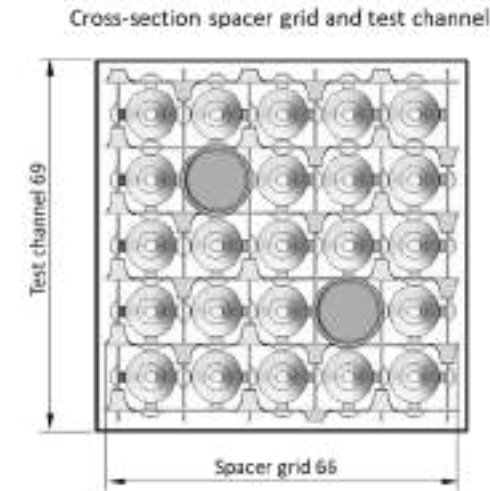
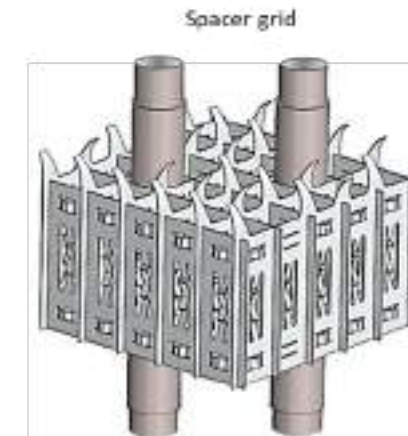
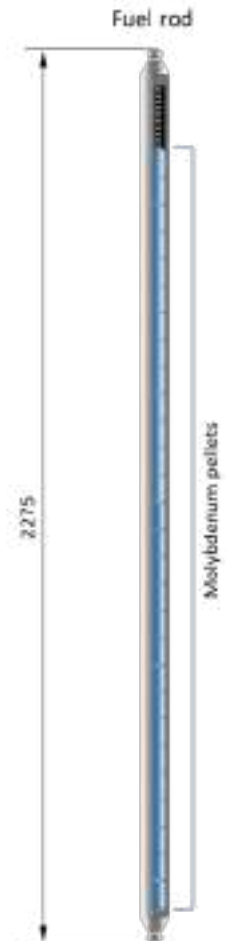


Fuel assembly overview

Rod arrangement	5x5
Number of fuel rods	23
Number of guide tubes	2
Bundle length	2305 mm
Rod length	2275 mm
Cladding tube outer diameter	9.5 mm
Cladding tube inner diameter	8.22 mm
Guide tube outer diameter	11.96 mm
Guide tube inner diameter	11.05 mm
Rod filling – Mo pellets column	8.05 x 2160 mm
Spacer grid	66 x 38 mm

Material	Density [g/cm ³]	E-modulus [GPa]	Parts
Zircaloy	6.6	98.1	cladding tubes, guide tubes
Alloy 718	8.2		spacer grids
Molybdenum	10.2	-	pellets

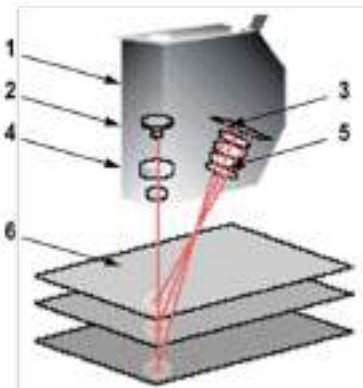
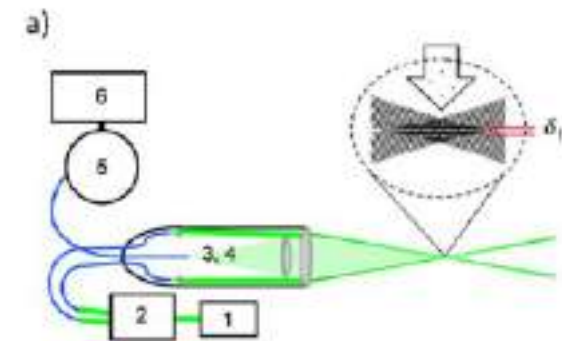
	Mass (single part) [kg]	Mass (total) [kg]
Fuel rod	1.4011	32.224
Guide tube	0.3366	0.673
Spacer grid	9.183E-02	0.4592



Test program

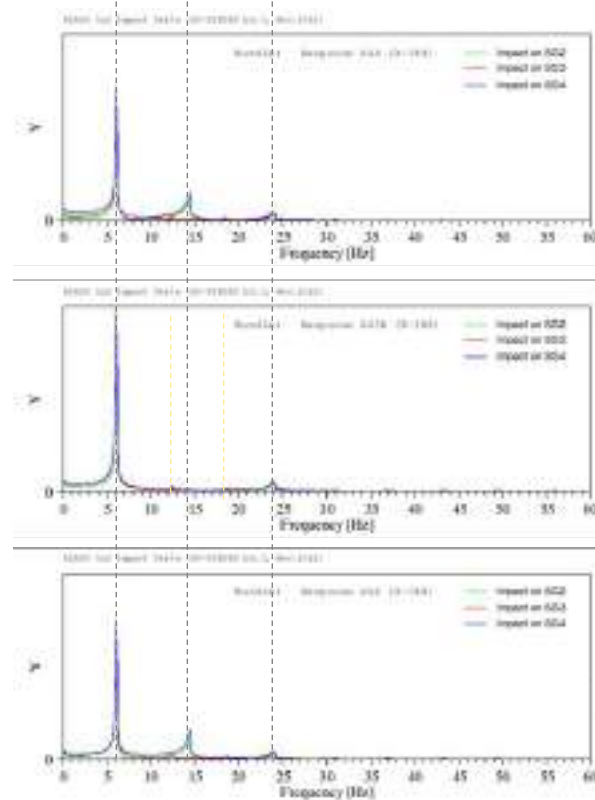
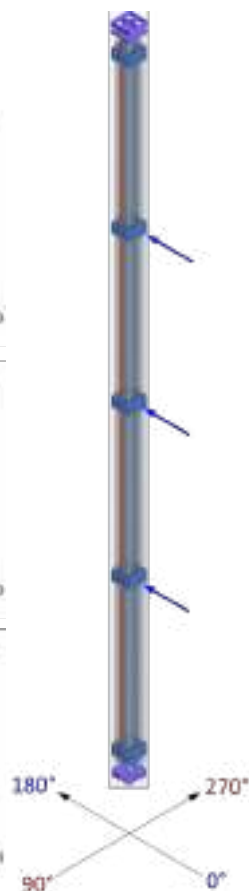


- Three types of tests performed
 - Mechanical impact tests to determine main mechanical properties
 - FIV tests (using laser triangulation system)
 - Flow field measurements using LDA technique

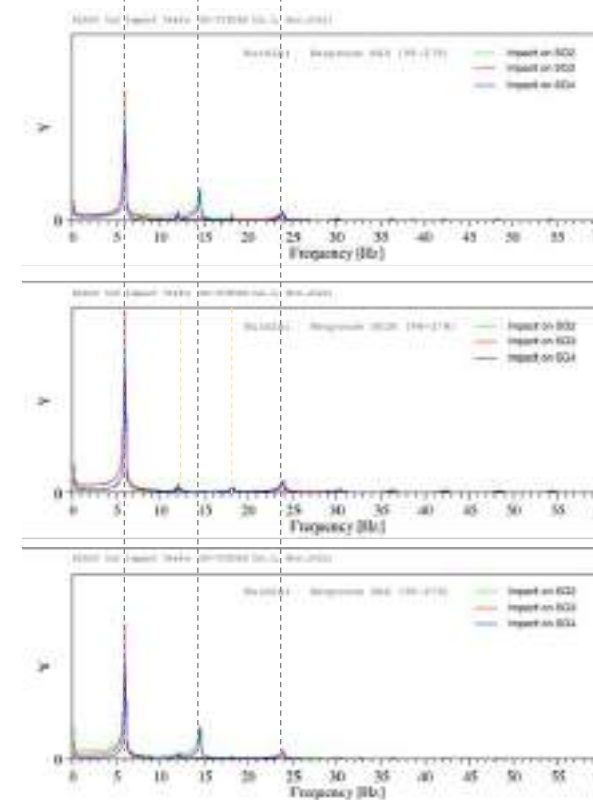


Impact tests results

	1	2	3	4	5	6
Bundle 0°	6.0	14.4	23.9			



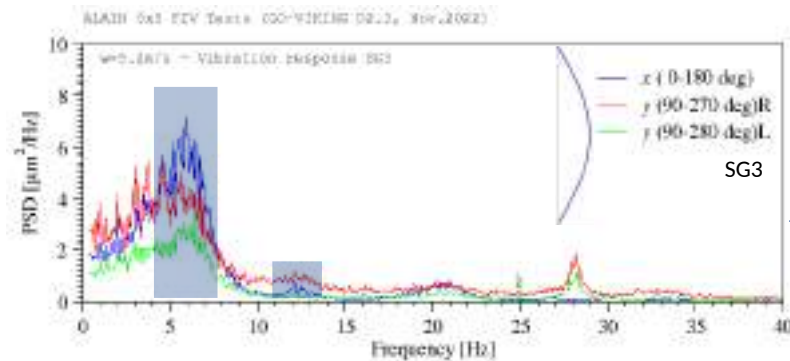
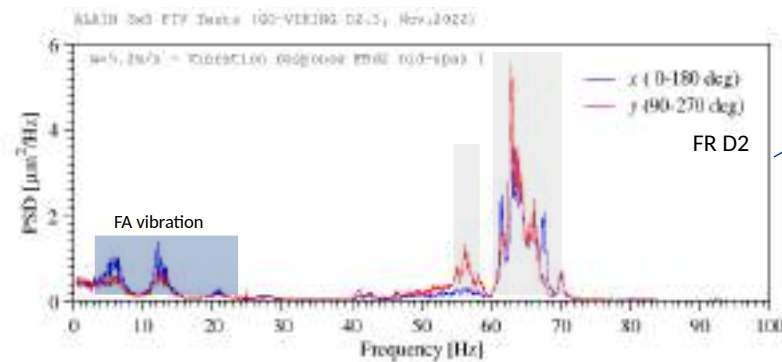
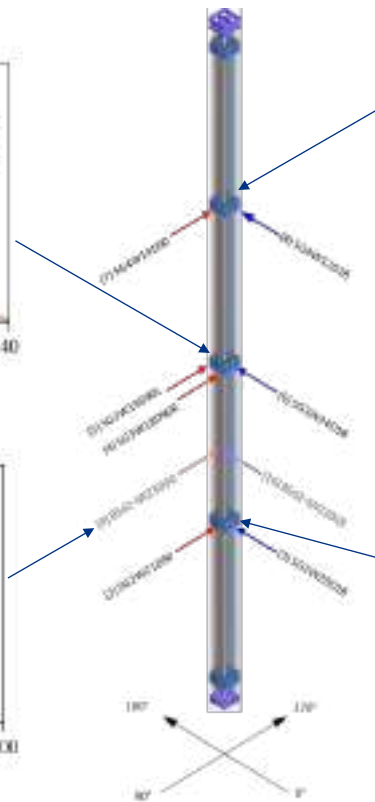
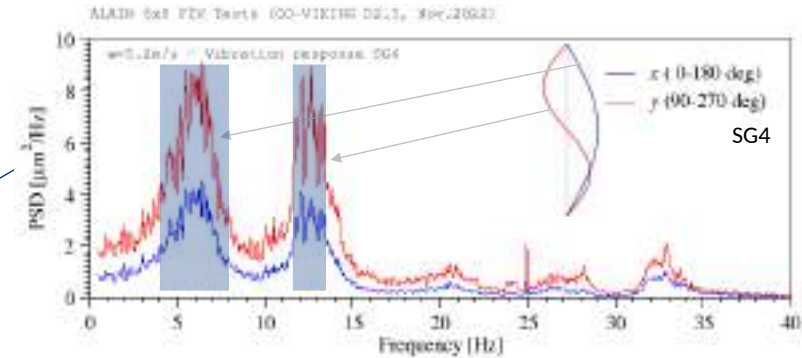
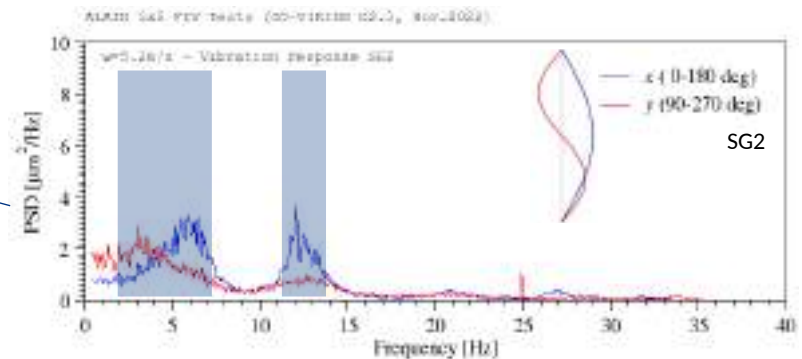
	1	2	3	4	5	6
Bundle 90°	6.0	14.5	23.9			



FIV tests results

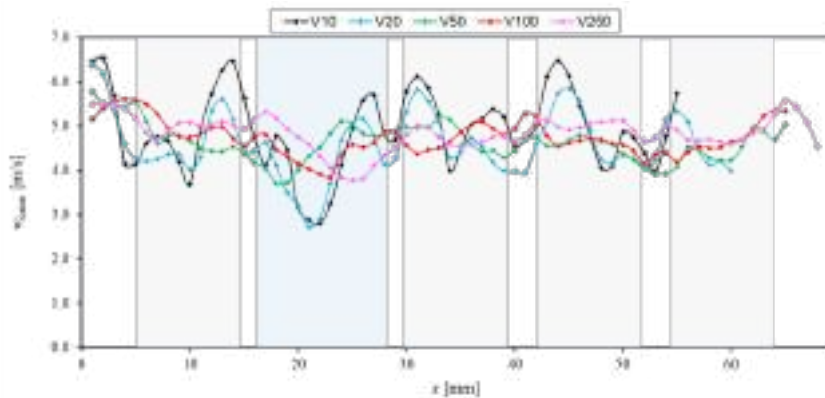
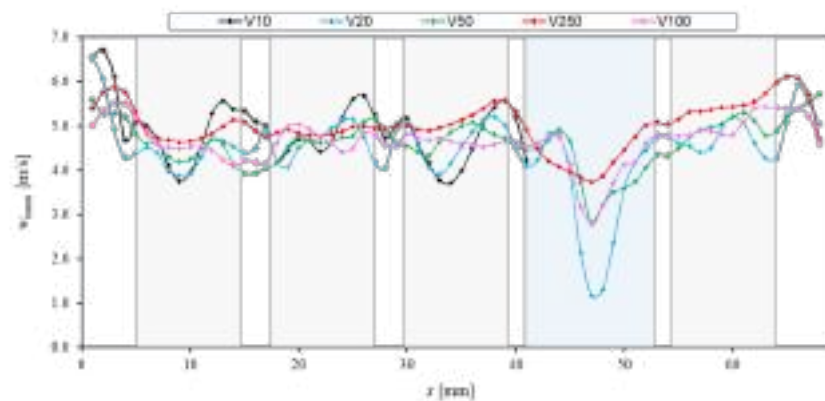
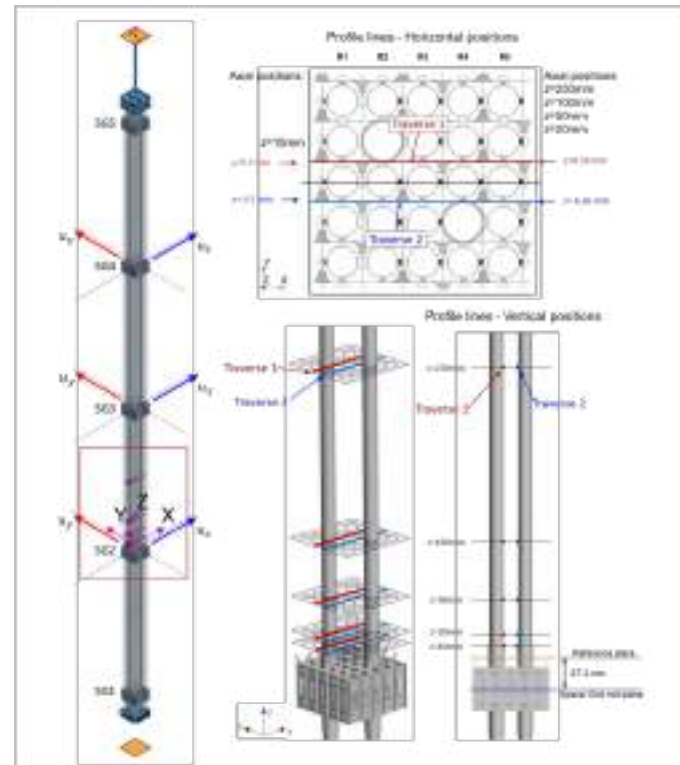
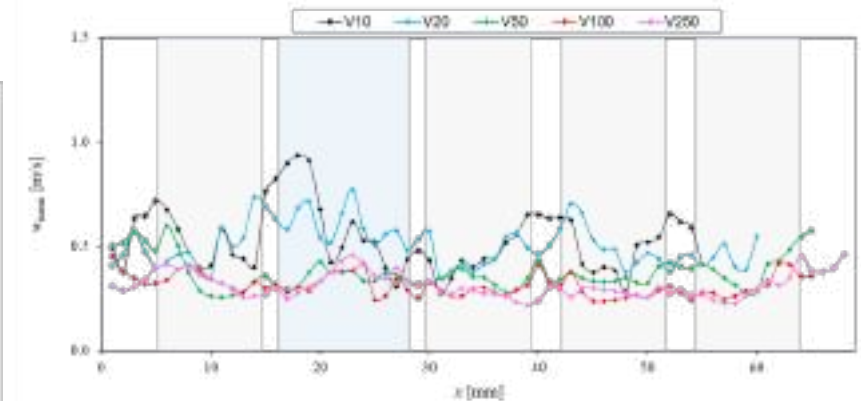
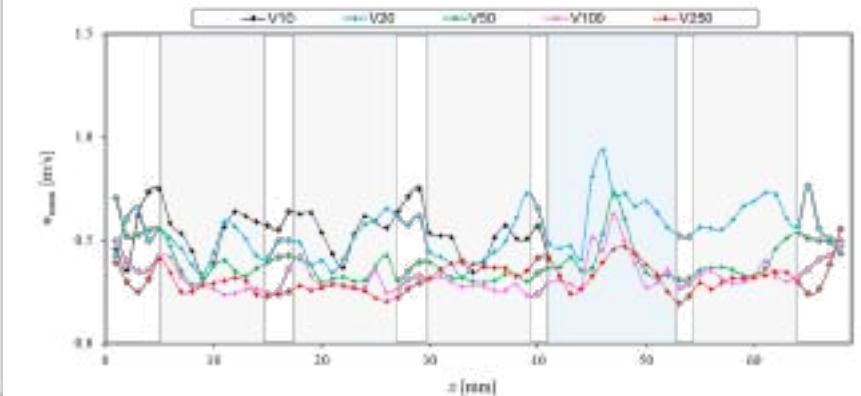


Vibration amplitude very small < 10 μ m RMS

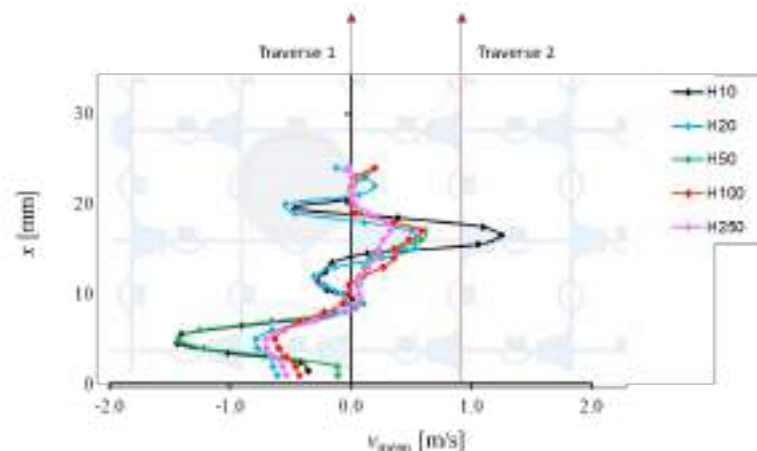
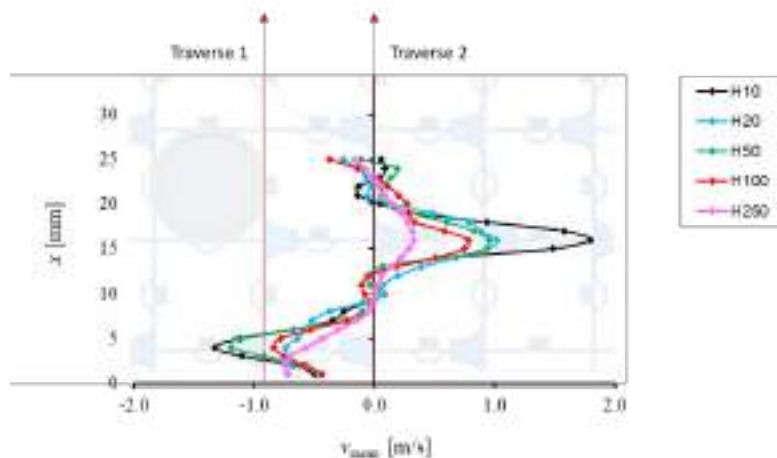
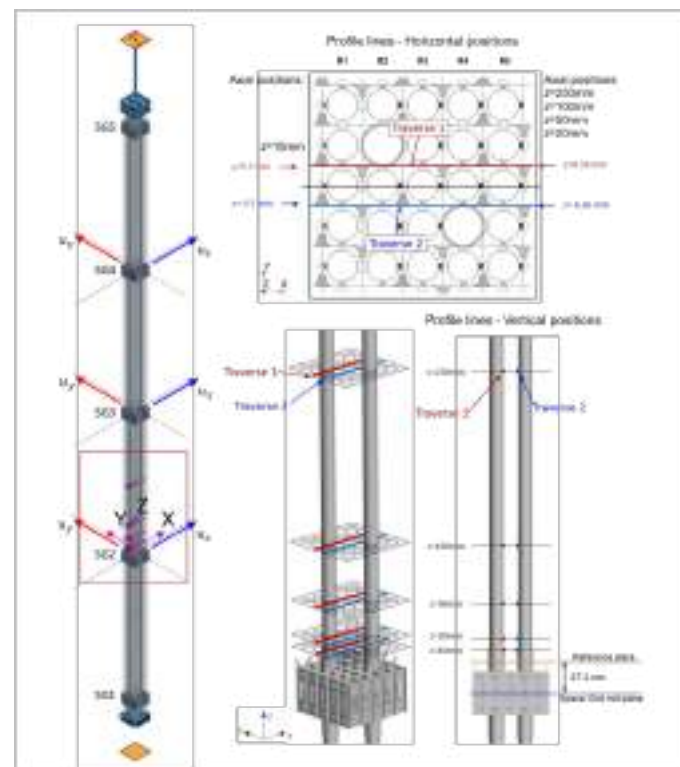
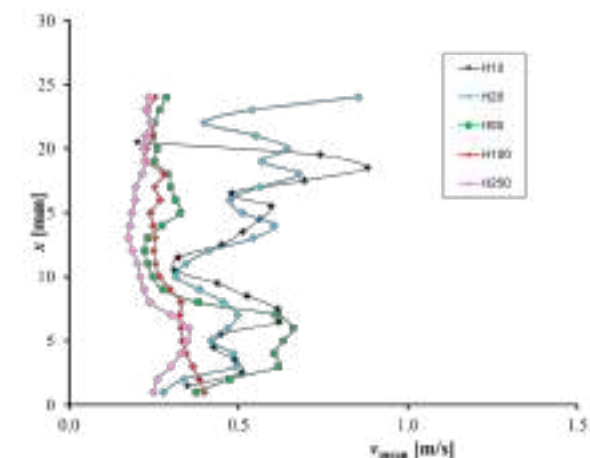
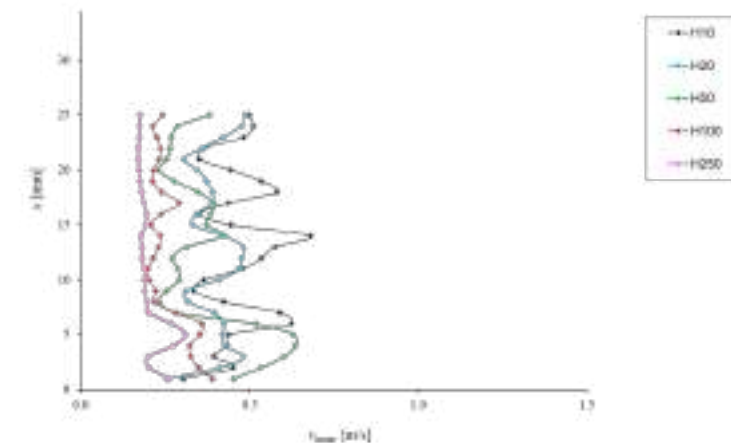


FIV test V326, w=5.2 m/s (24.08.2007)

LDA tests results – axial flow



LDA tests results – cross flow



Benchmark objectives

Objective is to perform numerical FIV simulations for given setup and flow rate corresponding to 5.2 m/s average axial velocity

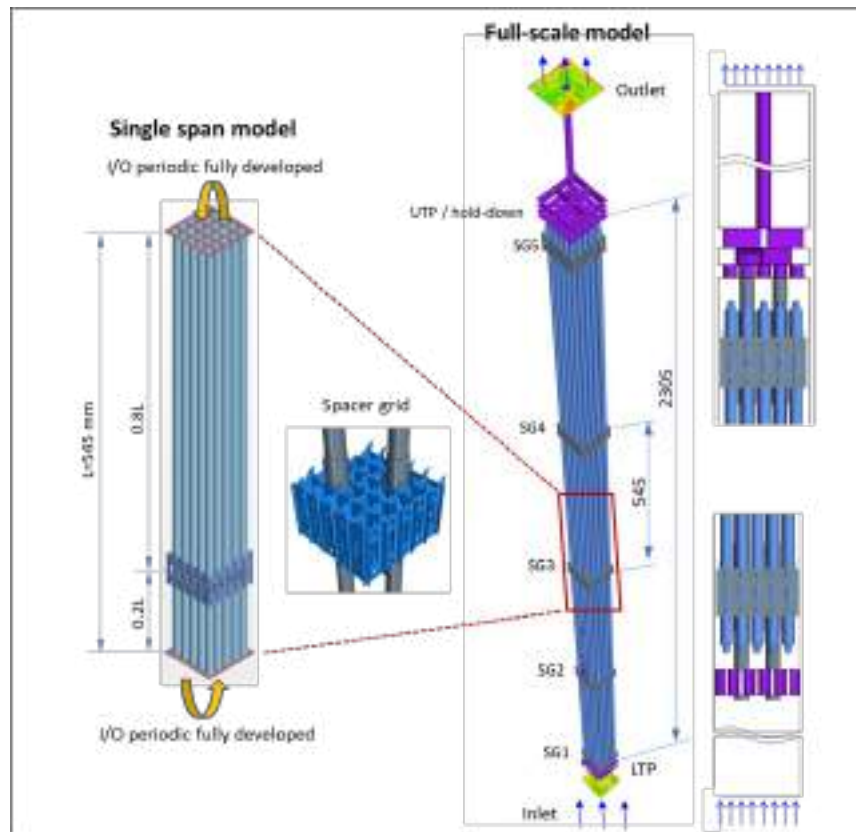
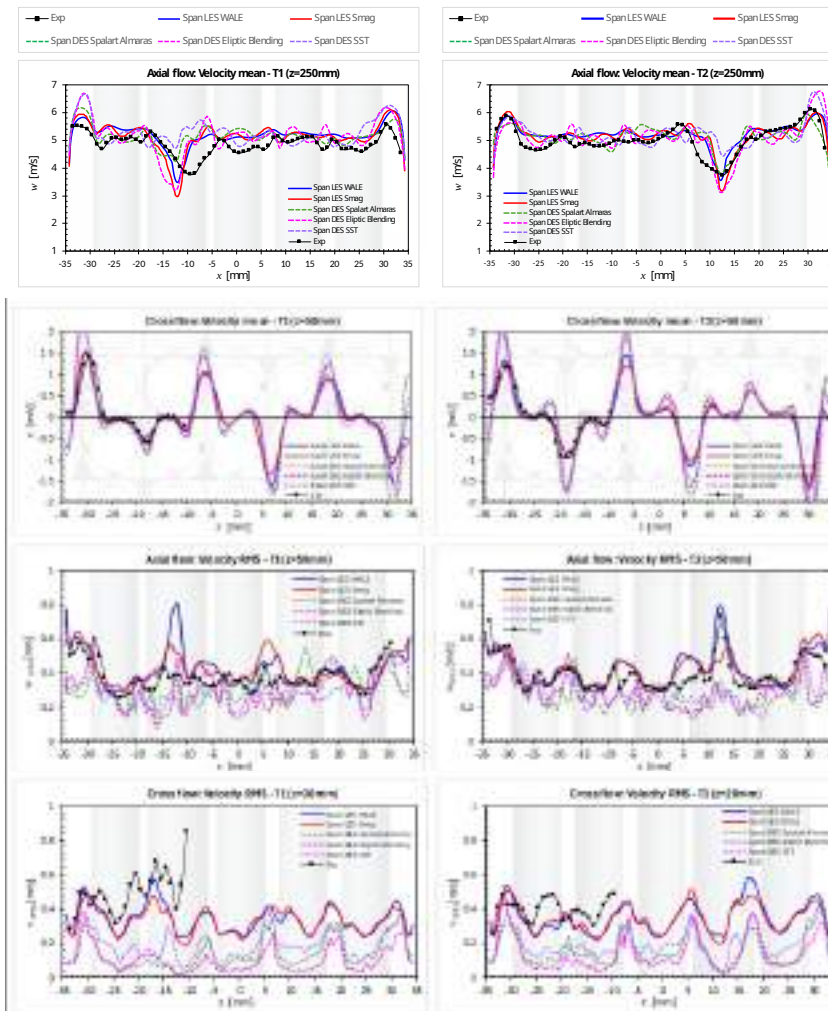
Numerical simulations should:

- Reproduce flow field
 - Mean velocity profiles
 - RMS velocity profiles
- Reproduce vibration behavior of the bundle measured experimentally in terms of
 - Vibration amplitude: RMS – displacement at SG 2, 3 and 4 (quantitative)
 - Vibration response frequency spectrum (qualitative)

Simulation approach by Framatome (1)

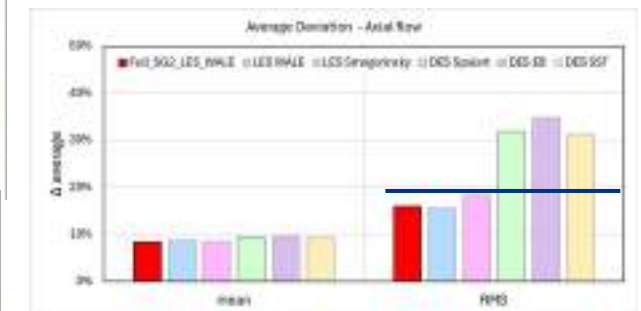
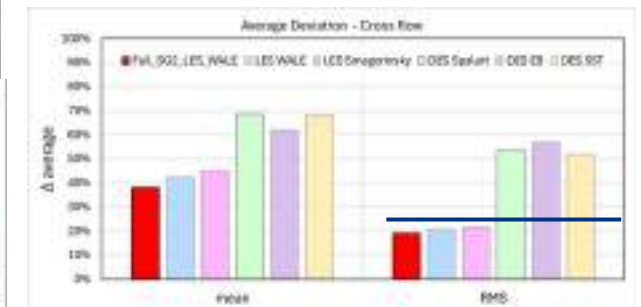


CFD simulation(s) on reduced (single span) and full-scale geometry using SRM (LES and DES) with wall models



CFD Simulation approaches and associated models used:

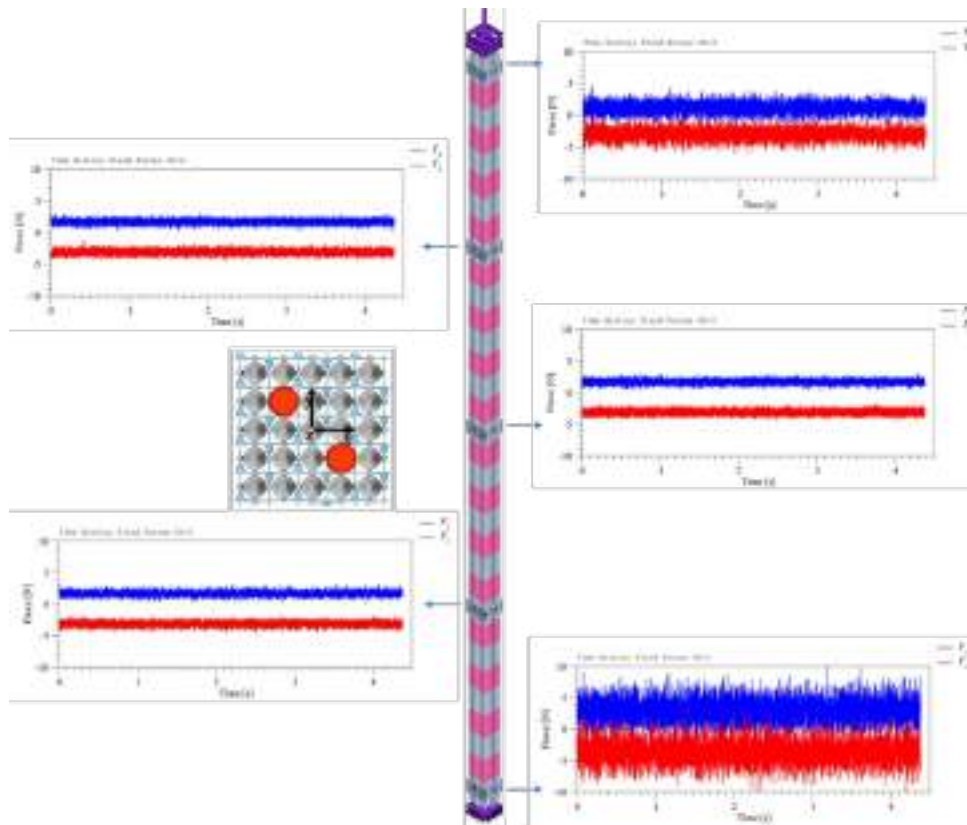
- LES with WALE SGS
- LES with Smagorinsky SGS
- DES Spalart Almaras
- DES Elliptic blending
- DES SST



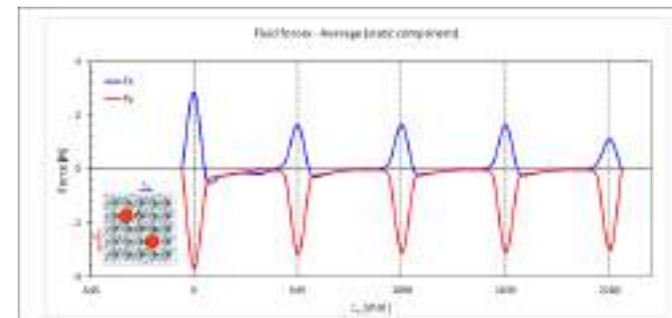
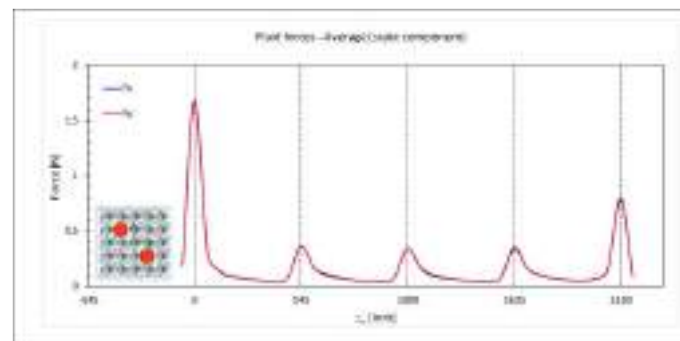
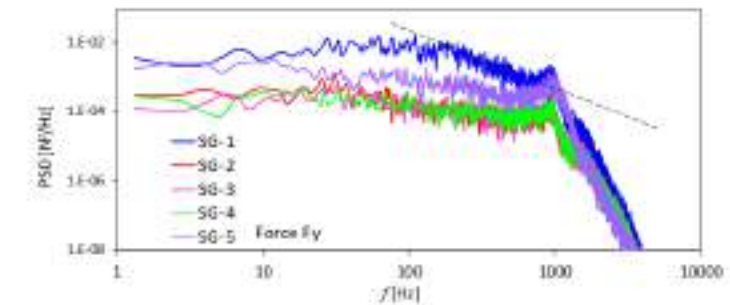
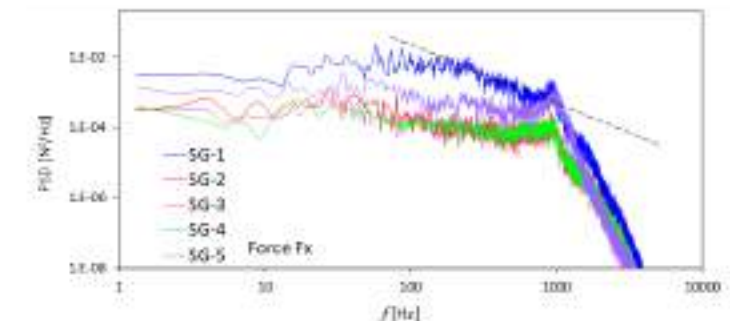
Simulation approach by Framatome (2)



Fluid forces - Time histories



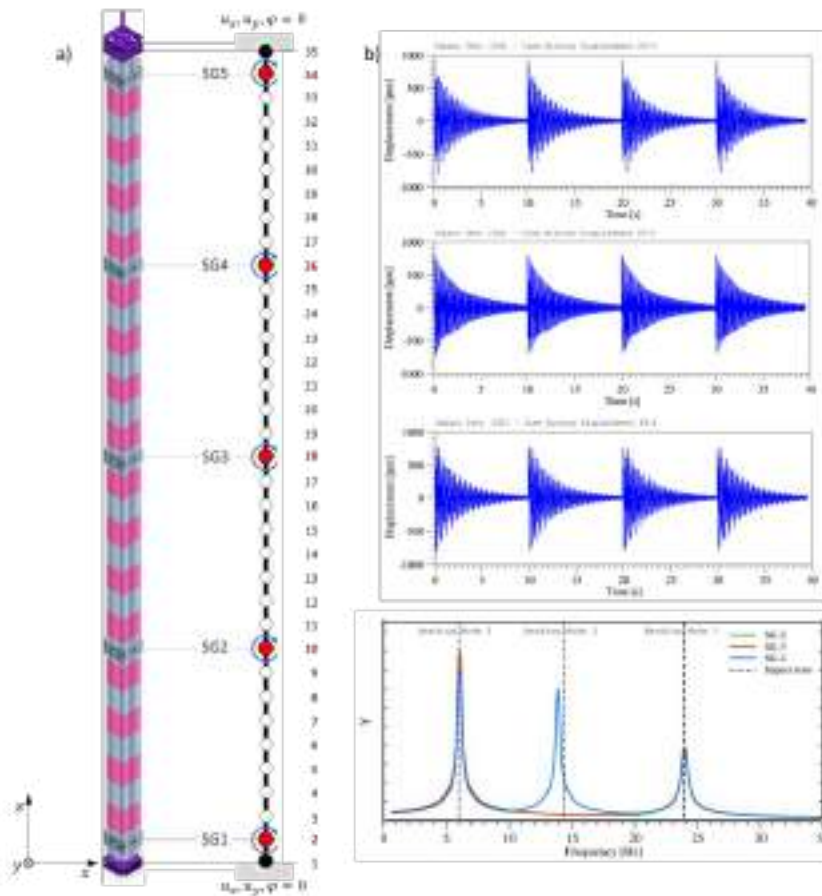
Fluid forces - Statistical analysis



Simulation approach by Framatome (3)

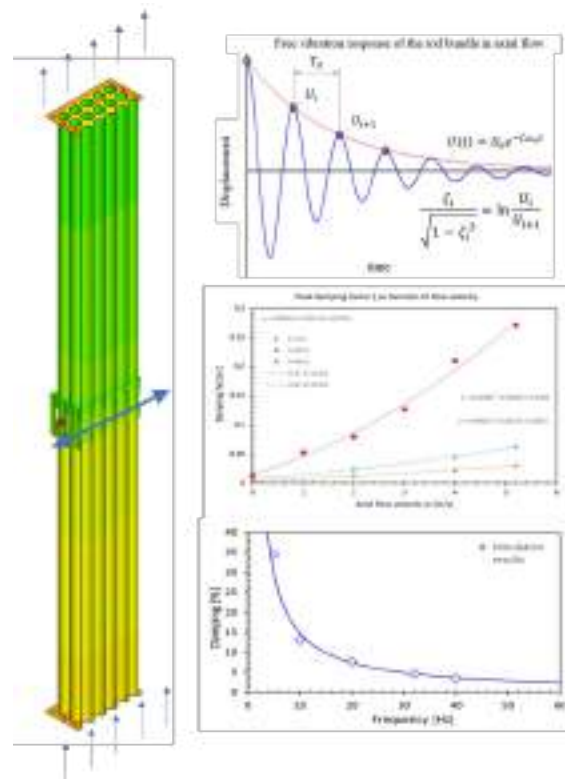


Mechanical model

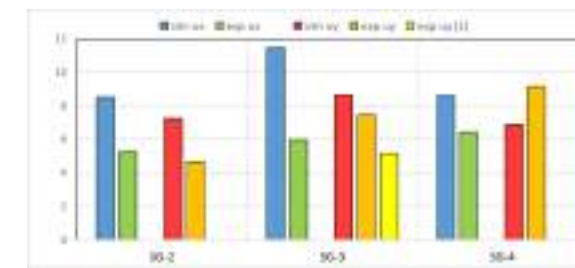
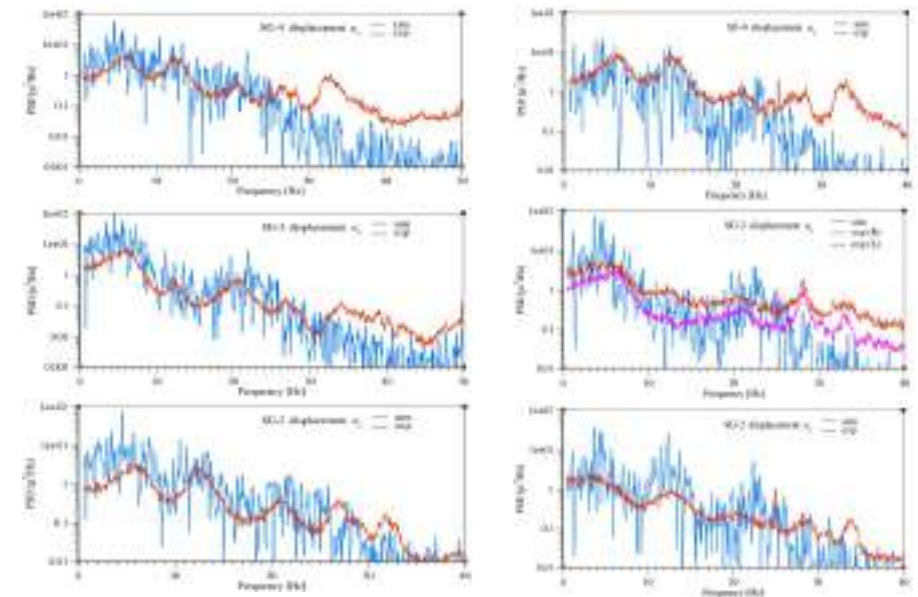


Effects of fluid

- Added mass
- Fluid damping



Vibration response



Partners



Associated Partners





Thank



Hidajet Hadžić
you!



Framatome GmbH



hidajet.hadzic@framatome.com



Funded by the
European Union



WP2: The ALAIN experiment and its analysis using GRS-FSI approach

Joachim Herb, Christoph Bläsius, GRS

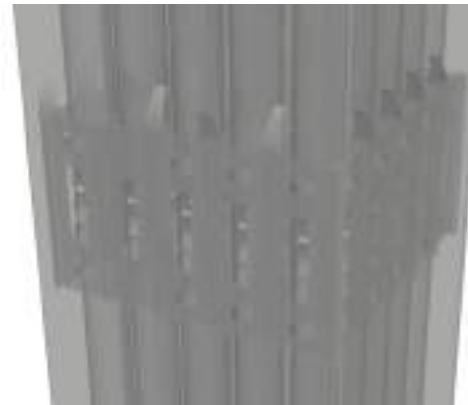
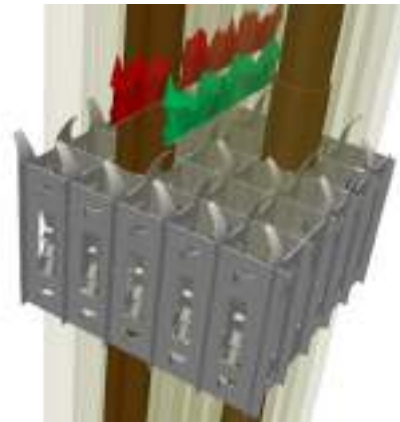
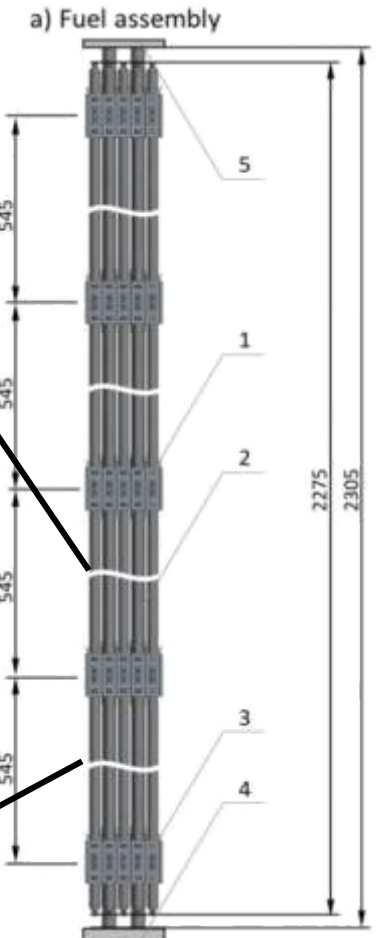
GO-VIKING FINAL WORKSHOP, 20 - 22.05.2026, GRS, Garching, Germany



Funded by
the European Union

One span geometry

- OpenFOAM model
- 5 x 5 rods
- Geometry from data deliverable D2.3
- 1 span



Mesh



- 1 span model (GRS)

cells: 5080663
Overall number of cells of each
type:

hexahedra:	4457567
prisms:	169392
wedges:	11
pyramids:	0
tet wedges:	667
tetrahedra:	75
polyhedra:	452951

Coarse Mesh LES

- Turbulence modes: LES, dynamicKEqn
- $\Delta t = 1e-6$ s (Courant Number max: ≈ 0.8)
- Results by averaging over last 0.25 s of 1 s long simulation
- Cyclic boundary conditions (bottom/top)
- Pressure gradient at baffle shortly above bottom with adaptive pressure jump to get mean velocity
- Incompressible fluid ($\nu = 6.5803 \cdot 10^{-7}$ m²/s)
- $y^+ < 670$
($\sim 1E6$ face elements, $\sim 8e3 > 100$, $\sim 37E3 > 50$, $\sim 750e3 > 10$)

Boundary conditions

	Walls	top/bottom	Baffle
p	zeroGradient	nonConformalCyclic	meanVelocityPressureJump
U	noSlip	nonConformalCyclic	cyclic
k	kLowReWallFunction	nonConformalCyclic	cyclic
omega	omegaWallFunction	nonConformalCyclic	cyclic
nut	nutUSpaldingWallFunction	nonConformalCyclic	Cyclic

LES-Simulations schemes



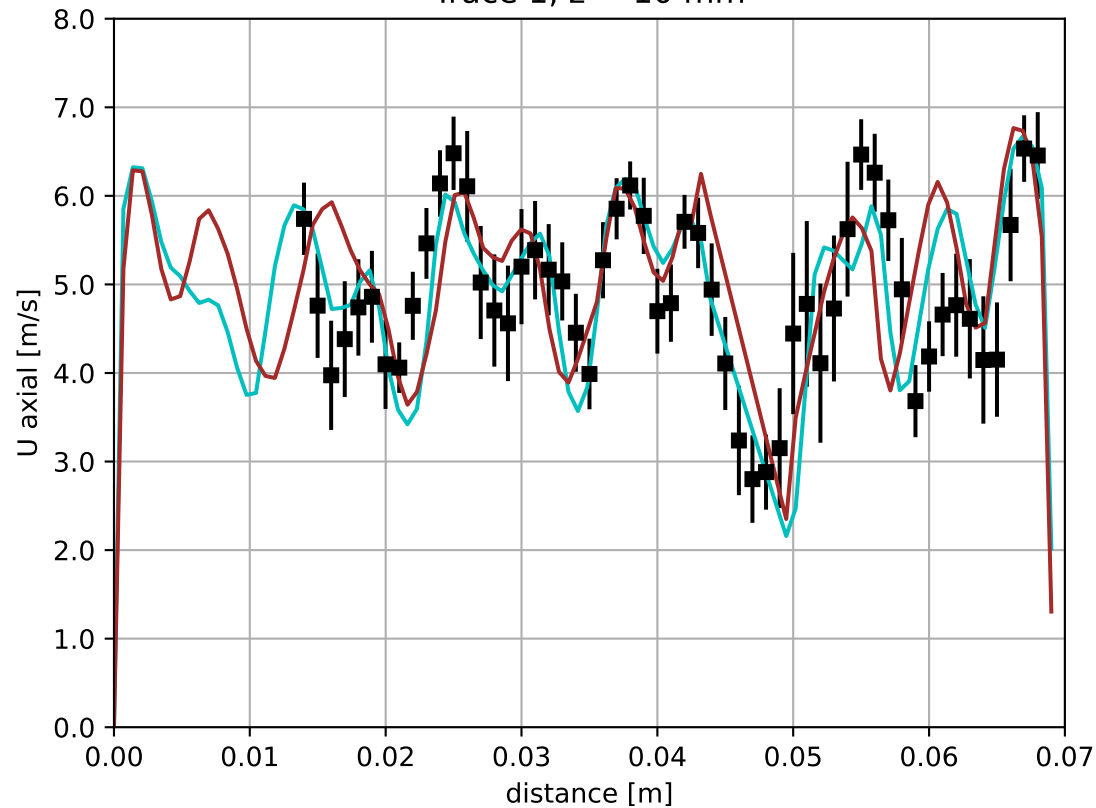
```
ddtSchemes
{
    default        backward;
}
gradSchemes
{
    default        Gauss linear;
}
divSchemes
{
    default        none;
    div(phi,U)      Gauss LUST grad(U);
    div(phi,k)       Gauss limitedLinear 1;
    div(phi,s)       bounded Gauss limitedLinear 1;
    div((nuEff*dev2(T(grad(U))))) Gauss linear;
}
```

```
laplacianSchemes
{
    default        Gauss linear corrected;
}
interpolationSchemes
{
    default        linear;
}
snGradSchemes
{
    default        corrected;
}
```

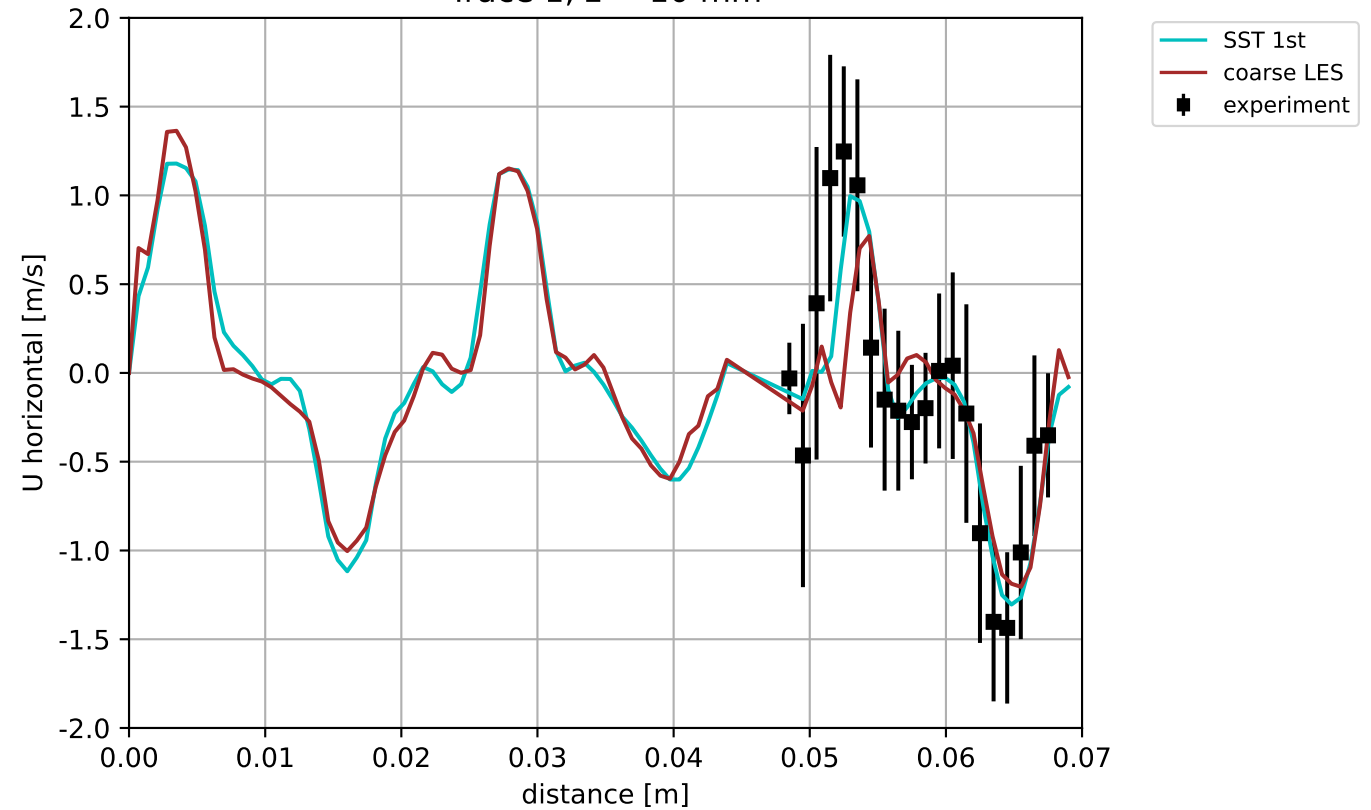
Mean velocities



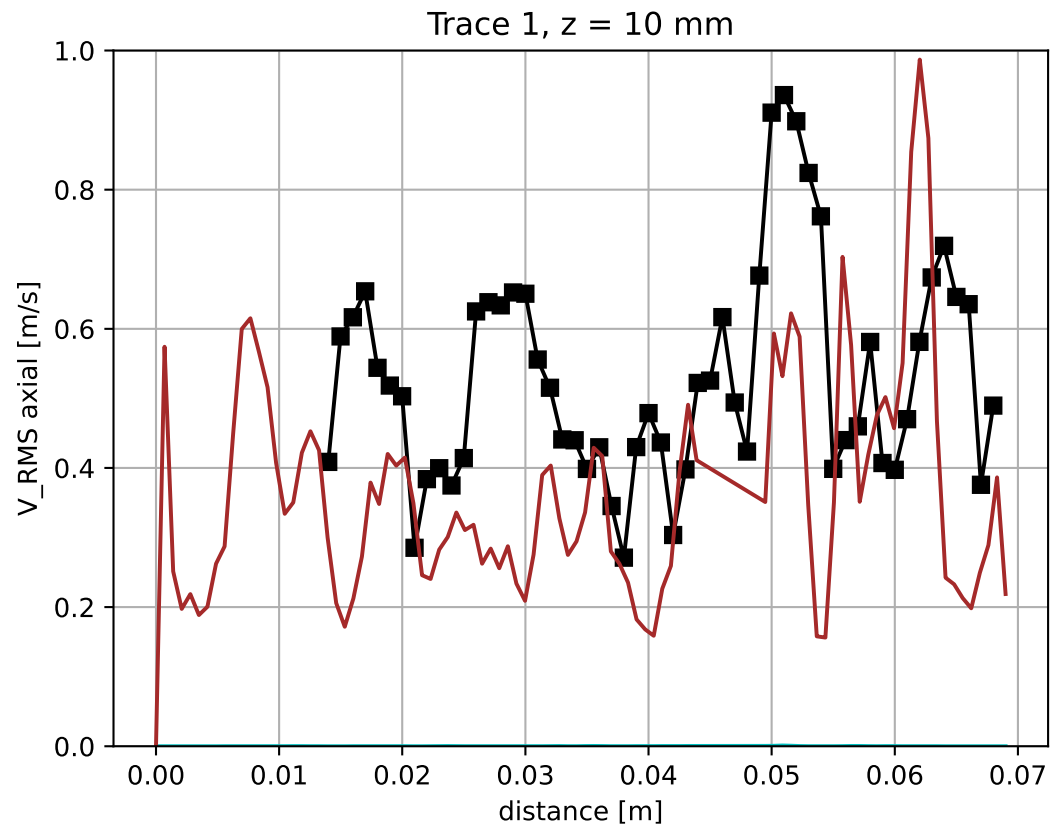
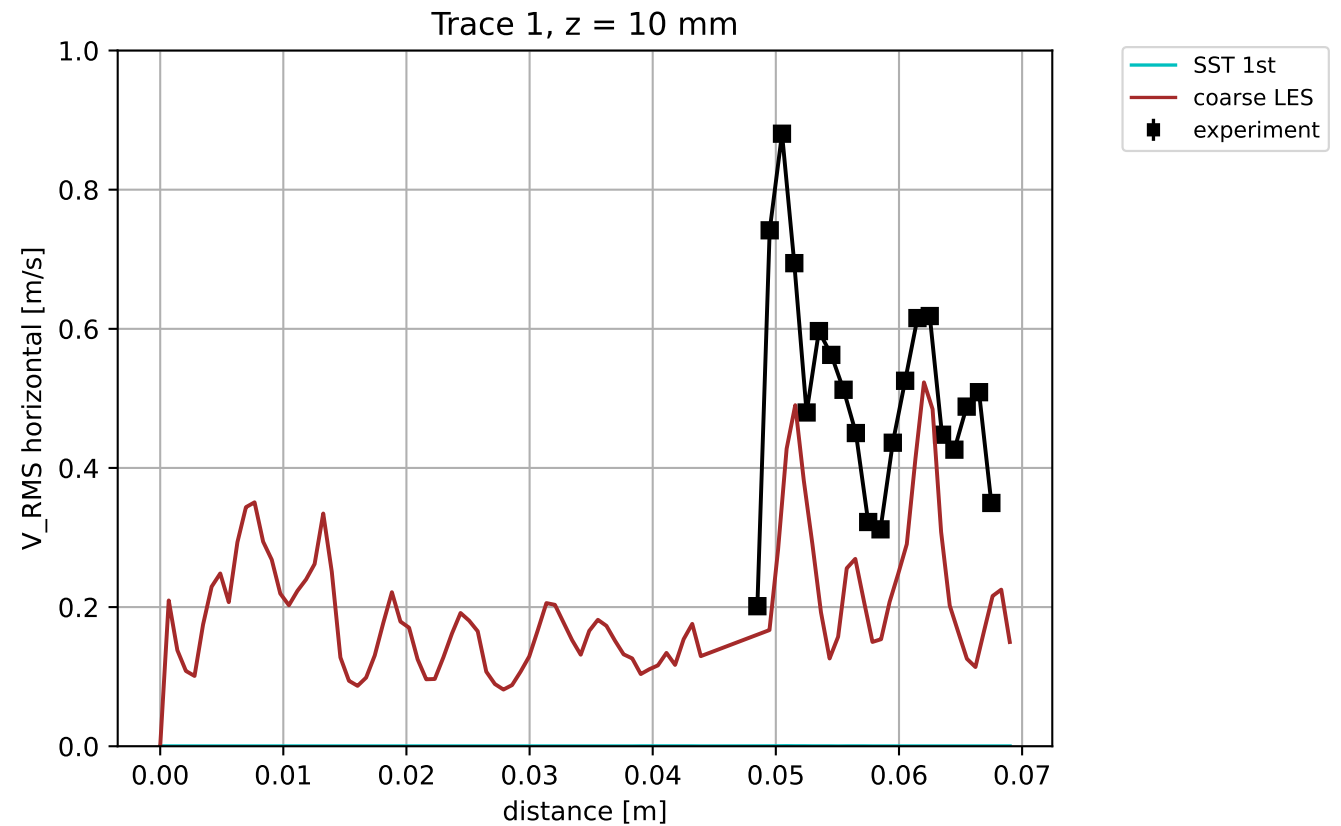
Trace 1, $z = 10$ mm



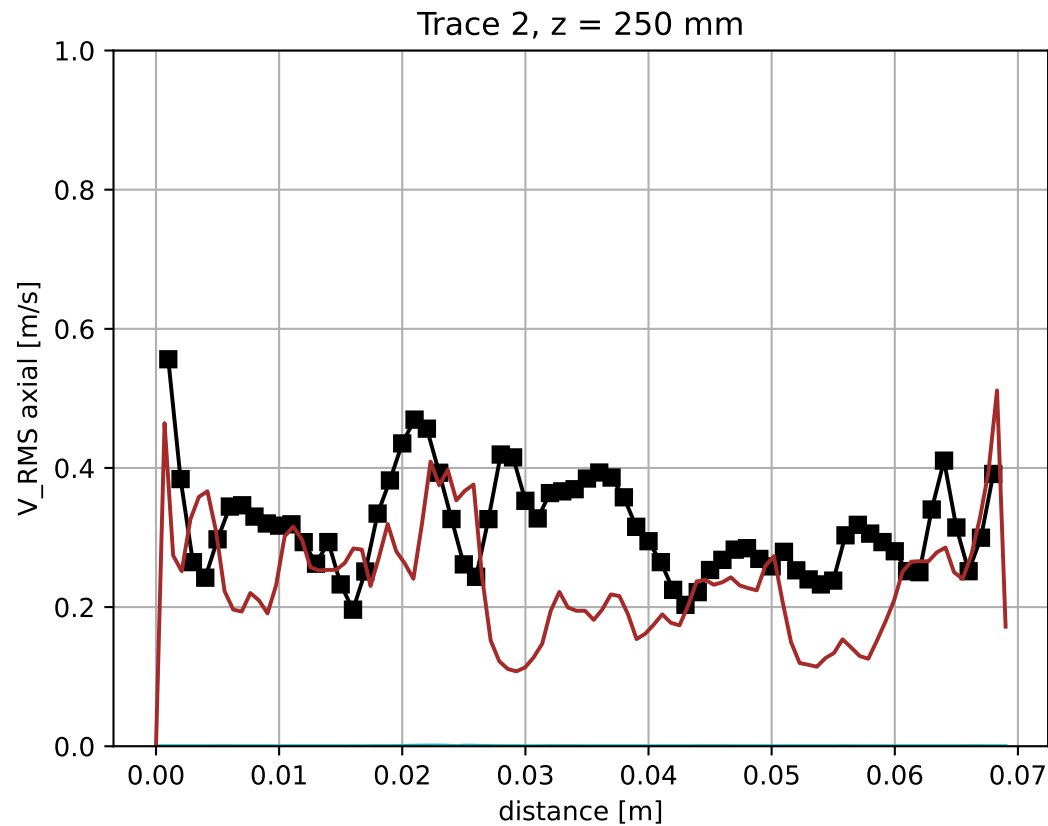
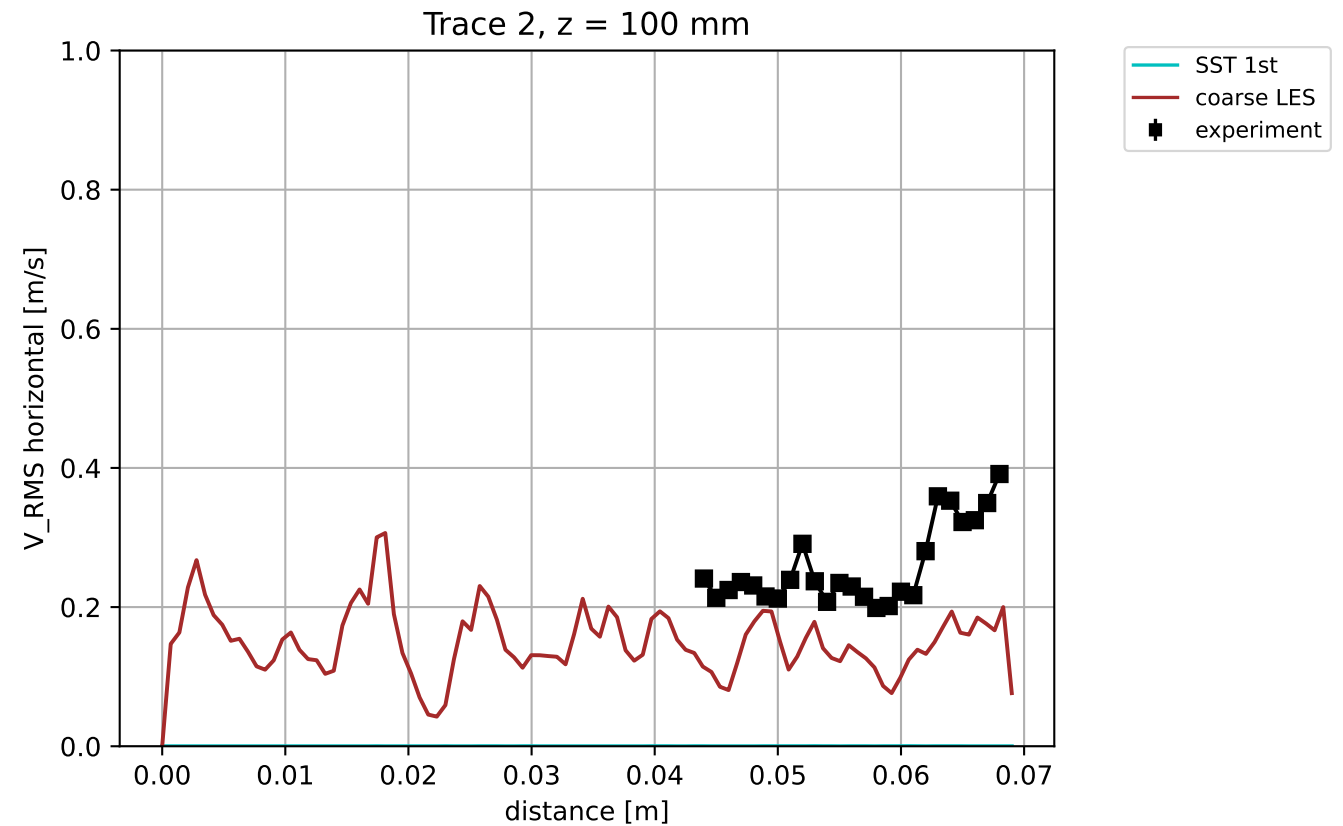
Trace 1, $z = 10$ mm



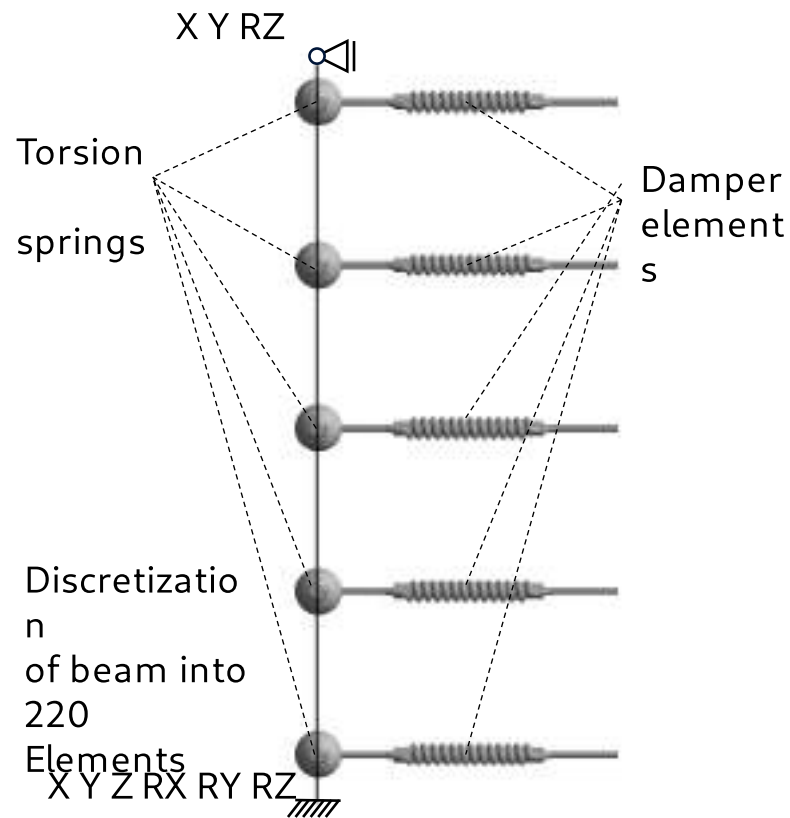
RMS velocities



RMS velocities



Mechanical Model



- Fuel assembly idealized as single beam
- Symmetry in X and Y-direction
- Torsion springs to mimic stiffening of spacer grids
- Damper elements to mimic fluidic damping
- Implemented in ANSYS Mechanical

Calibration process (stiffness)

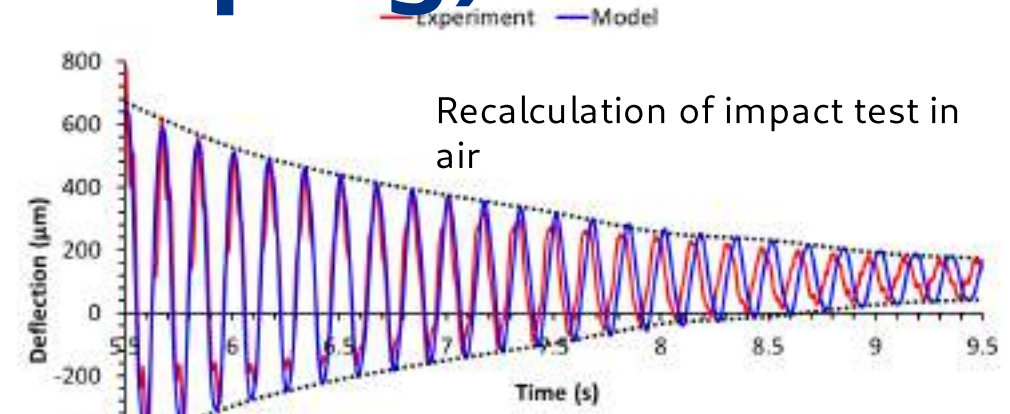
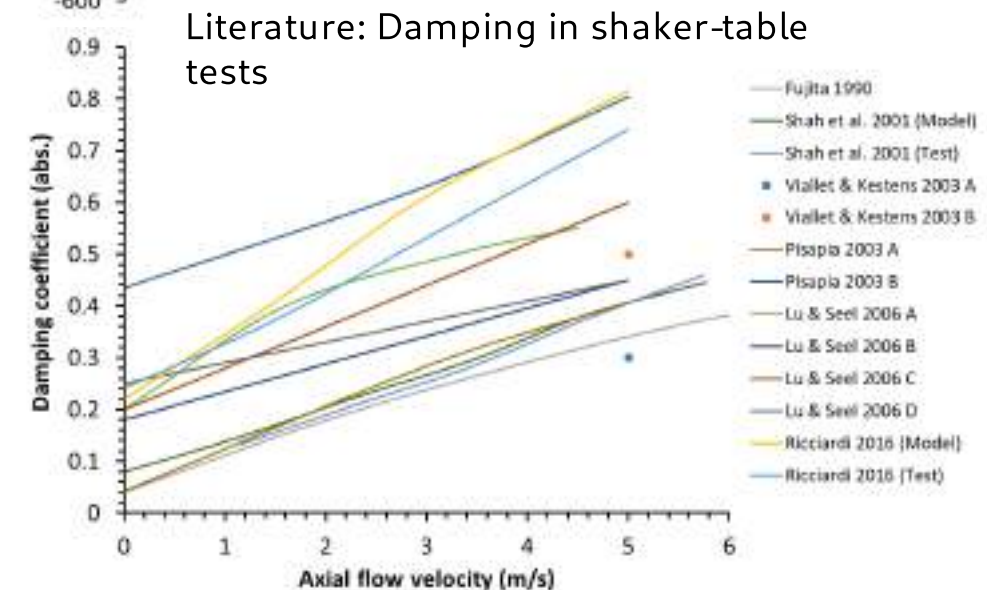
- Initial parameters based on geometry and material data
 - Adding cross section areas and moments of inertia
 - No parallel axis terms as fuel rods movable
 - Density by dividing total mass by product of beam length and cross section
 - Torsion stiffness arbitrarily high
- Final parameters by tuning moment of inertia and torsion springs to match natural frequencies and mode shapes

	1st	2nd	3rd
Experiment	6	14.5	23.9
FEM			2
Mode			

Calibration process (damping)



- Structural damping
 - Obtained by recalculating impact tests in air
 - Result: $D = 0.005$
- Fluidic damping
 - Literature on shaker-table tests on mock-up and real fuel assemblies under axial flow
 - Chosen: $D = 0.5$ (average), $D = 0.8$ (maximum)
- Iterative adaption of damper elements
- Final adjustment of beam stiffness
- No further added values for mass and stiffness



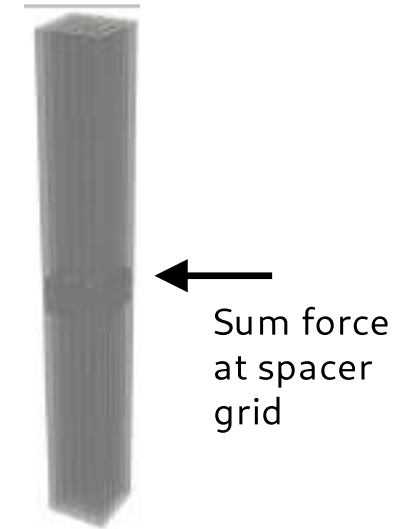
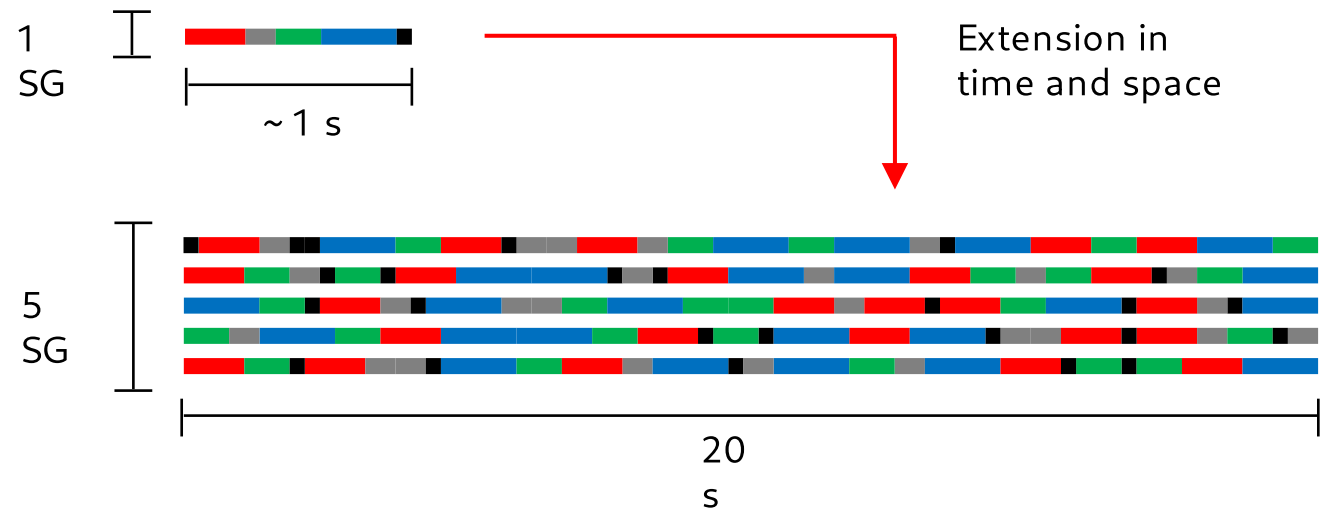
Data processing for mechanical model



- Mismatch between CFD and CSM

Available CFD results	Required CSM input
3D surfaces	1D beam
One span	Whole assembly
~ 1s	> 10 s for meaningful APSD

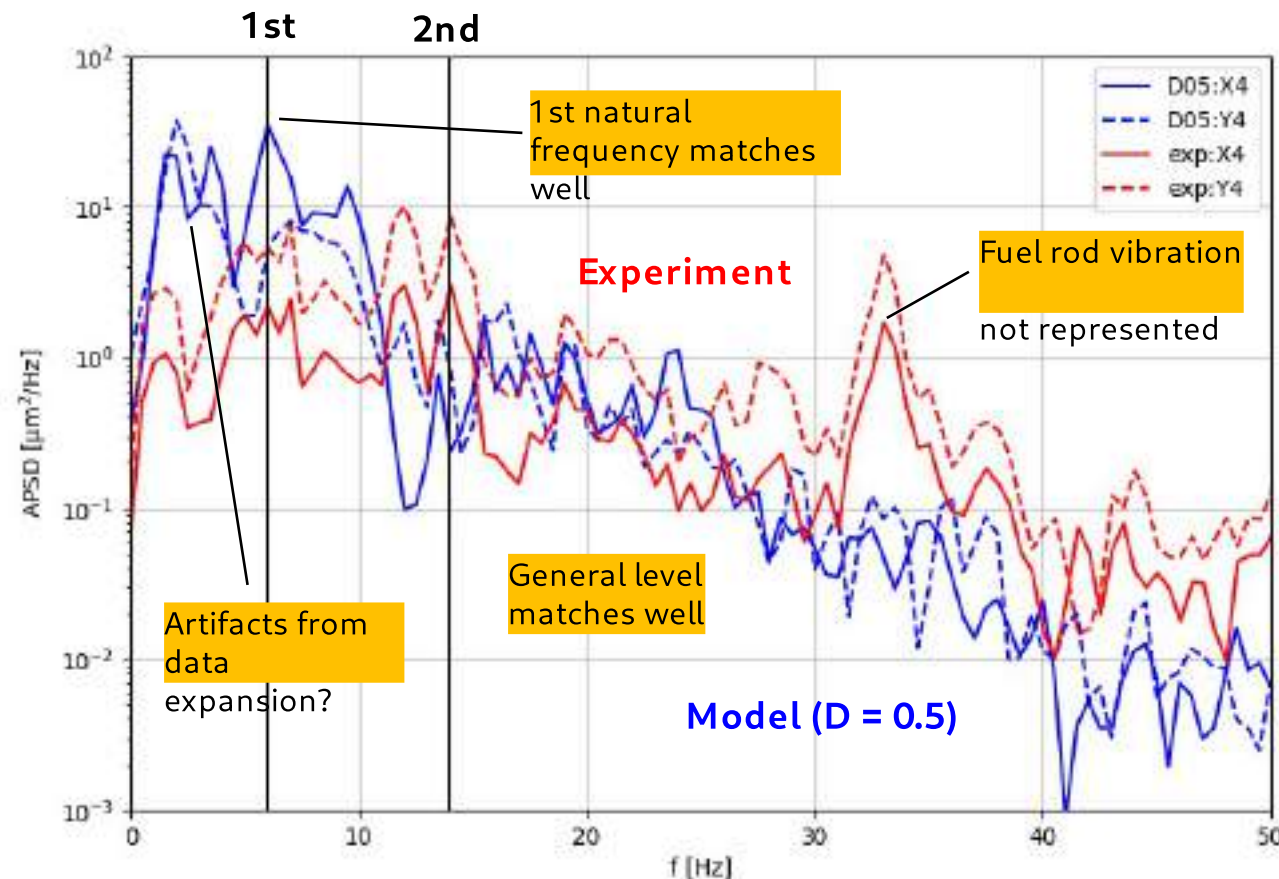
- 1) Summation of forces on structure to one single force acting on spacer
- 2) Subtraction of static part to minimize transient oscillation effects
- 3) Splitting of sum force into shorter intervals at zero crossings
- 4) Recomposed and distributed to full mock-up based on random patterns



Results



- Lateral deformations of SG2 in X/Y direction with damping $D = 0.5$

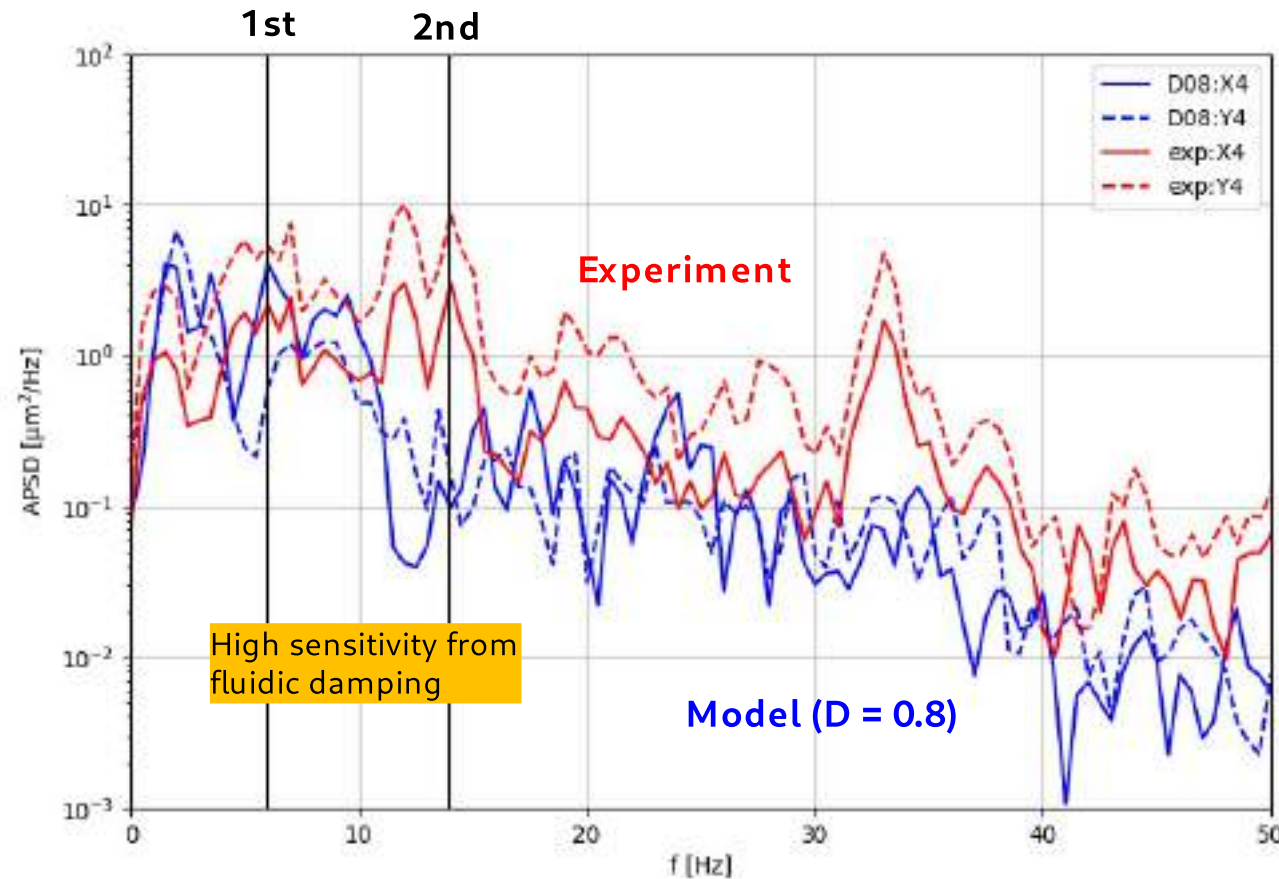


RMS	X	Y
Experiment	11.71 μm	9.69 μm
Model ($D = 0.5$)	6.66 μm	7.04 μm

Results



- Lateral deformations of SG2 in X/Y direction with damping $D = 0.8$



RMS	X	Y
Experiment	11.71 μm	9.69 μm
Model ($D = 0.5$)	6.66 μm	7.04 μm
Model ($D = 0.8$)	2.95 μm	3.1 μm

Conclusion and Outlook

- Simulation chain **reproduces main vibration phenomena** of experiment (mean displacement amplitude, APSD shape, lower natural frequencies)
- Low deflection amplitudes confirms applicability of unidirectional uncoupled analysis
- Potential improvements in mechanical domain: resolution of internal structure and vibration modes
- Potential improvements in fluid domain: finer, LES-grade mesh and larger models in time and space (at severely increased computational effort)

Partners



Associated Partners



* EDF R&D UK Center



Thank you!



Joachim Herb



GRS



Joachim.Herb@grs.de

www.go-viking.eu



Funded by the
European Union



WP2: ALAIN 5x5 Fuel Assembly

*Vladislav Filonov, Yuliia Filonova and Yaroslav Dubyk (IPP-
CENTRE)*

GO-VIKING Final Workshop 20 - 22.05.2026, Garching



Funded by
the European Union

Content



- Fluid model
- Mechanical model
- Coupling approach
- Conclusions

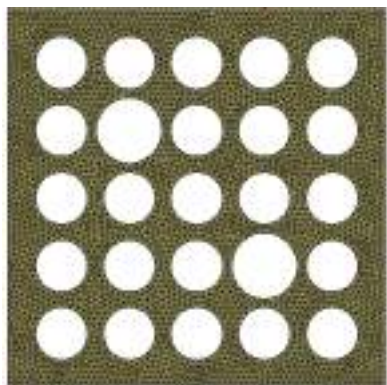
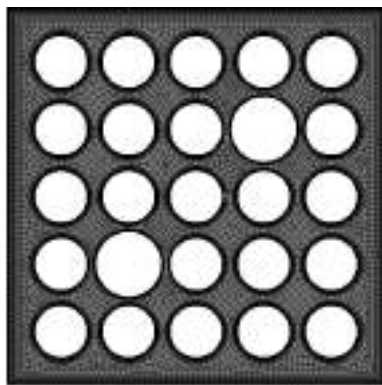
Fluid model. Meshes

#1. Fluid-Fluid Interface Coarse (FFI_L)

Nodes	≈ 3.9 million
Elements	≈ 13.0 million

Side 1

Side 2

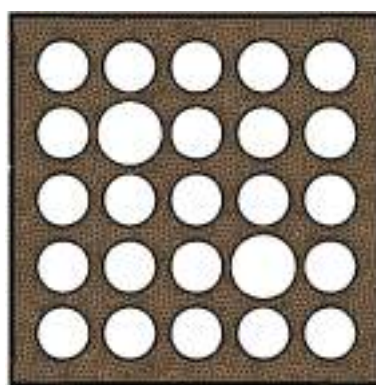
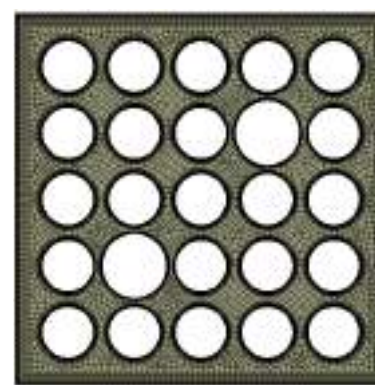


#2. Fluid-Fluid Interface Fine (FFI_F)

Nodes	≈ 8.0 million
Elements	≈ 22.6 million

Side 1

Side 2

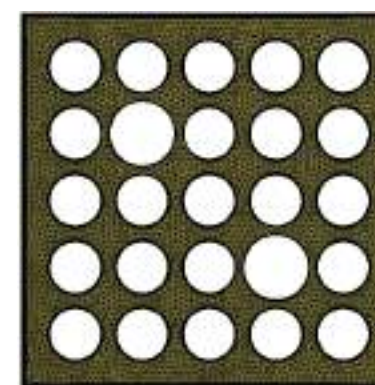
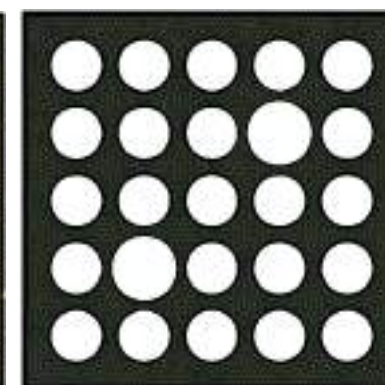


#3. Fully connected mesh (LNK)

Nodes	≈ 5.7 million
Elements	≈ 16.9 million

Side 1

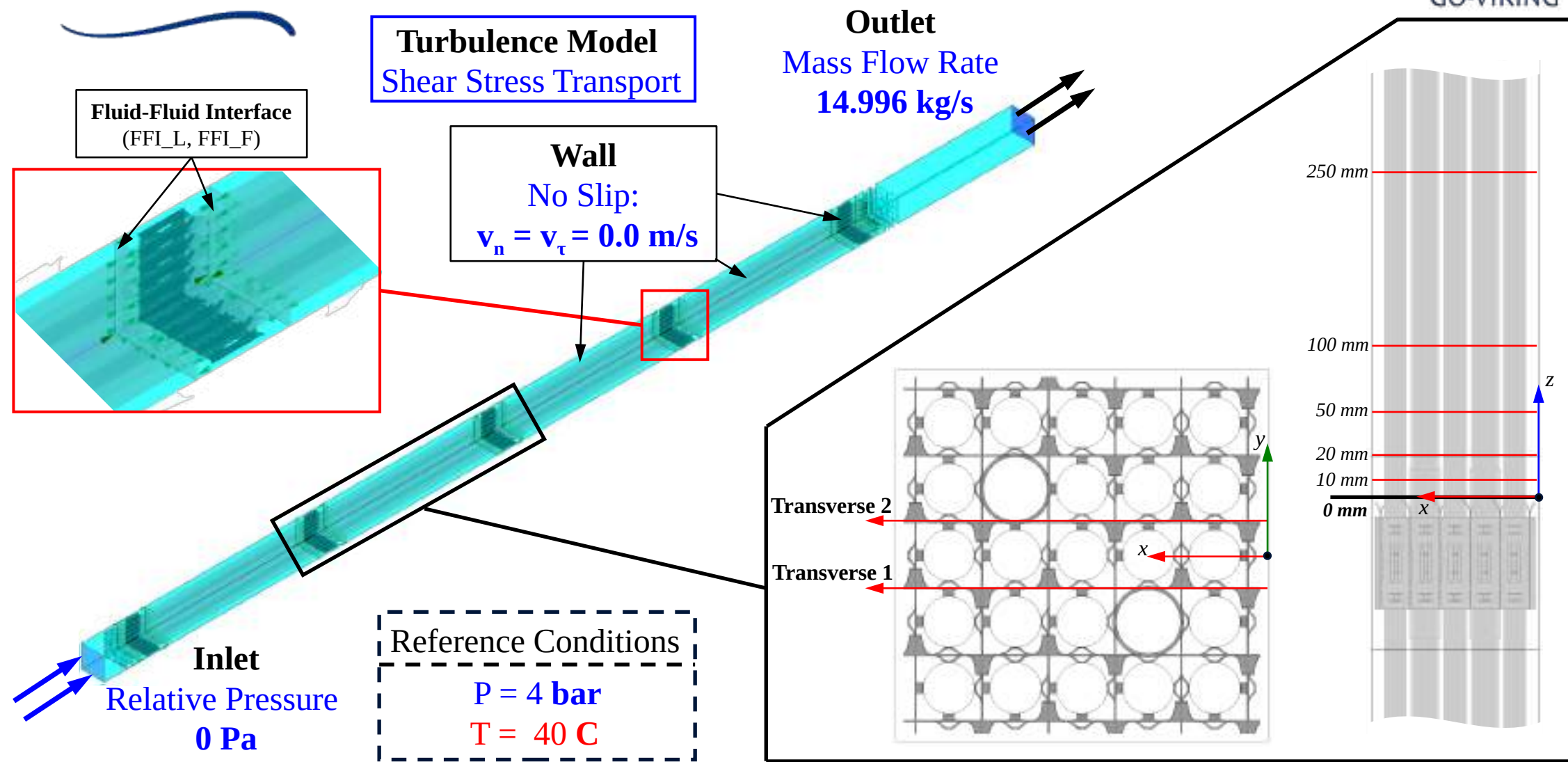
Side 2



General View of Model Meshes



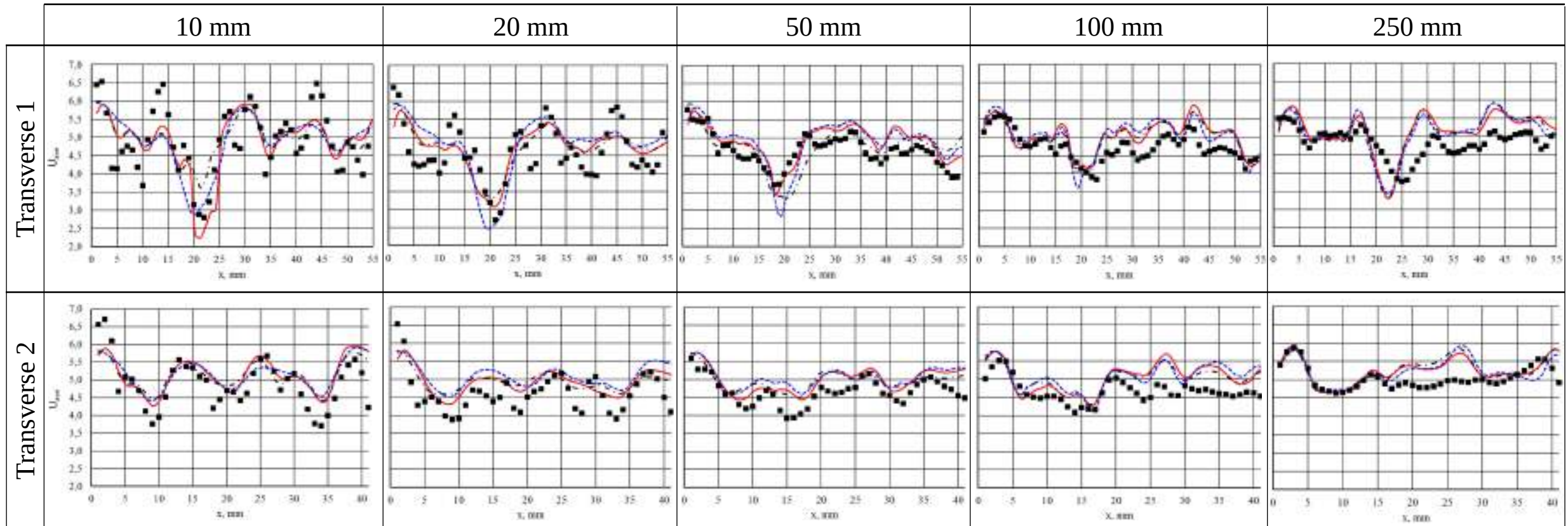
Fluid model. BCs and MPs



Fluid model. Results



Axial velocity

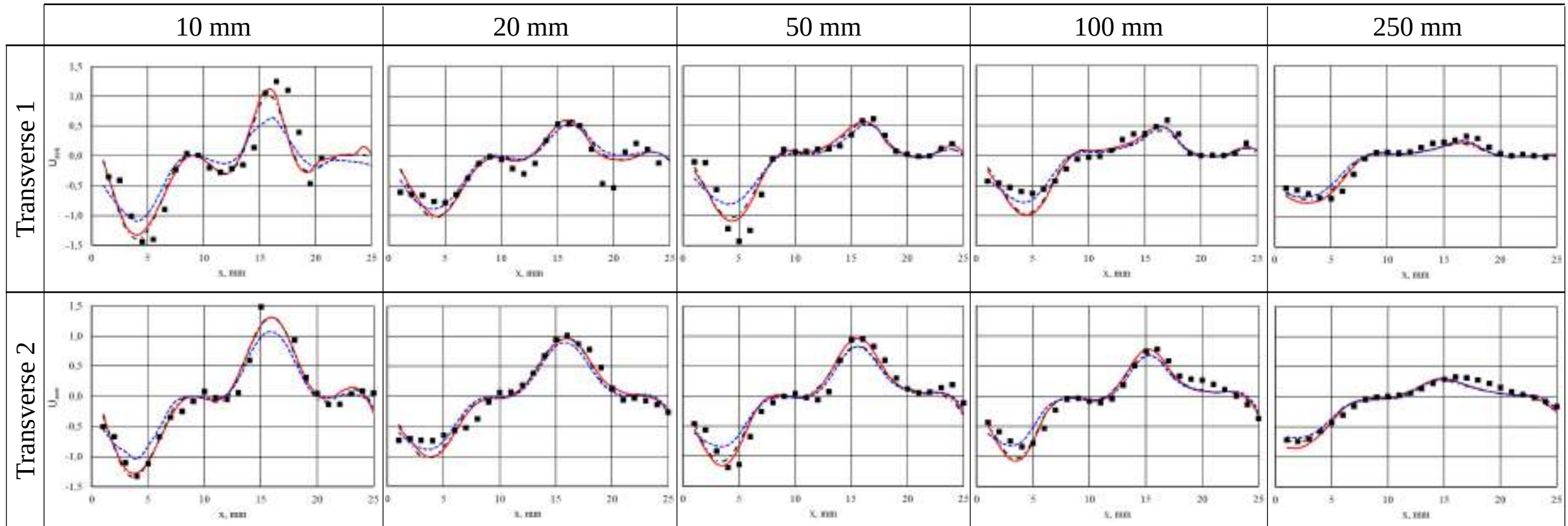


Legend: ■ Exp — FFI_F - - FFF_L - - - LNK

Fluid model. Results

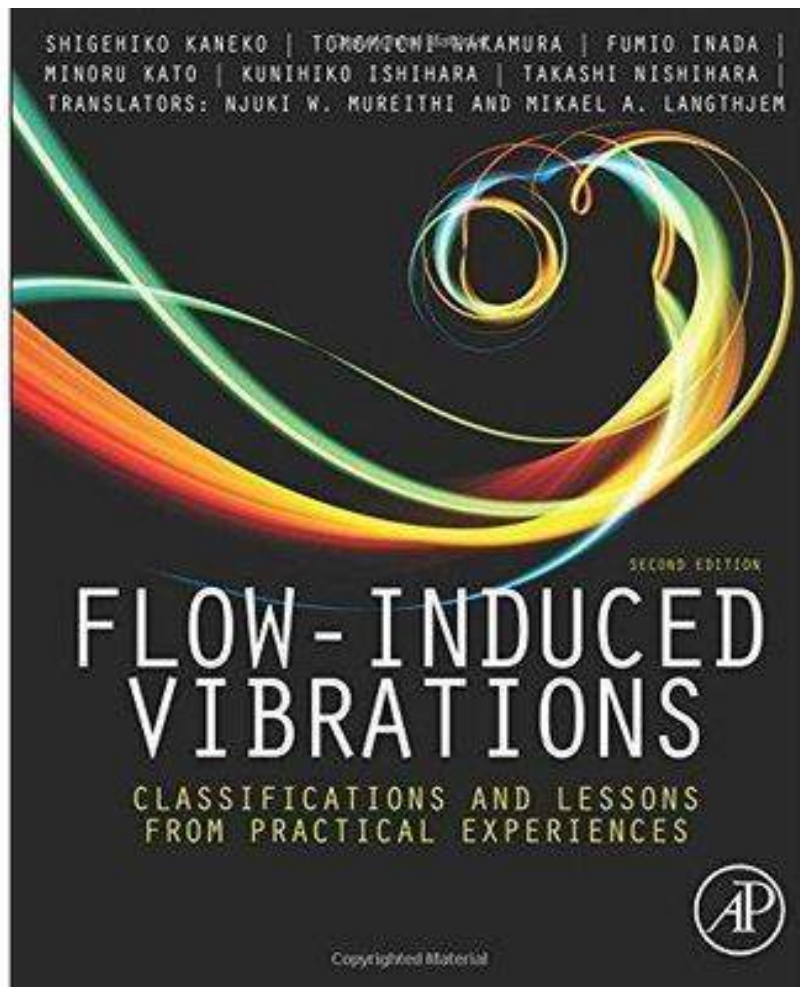
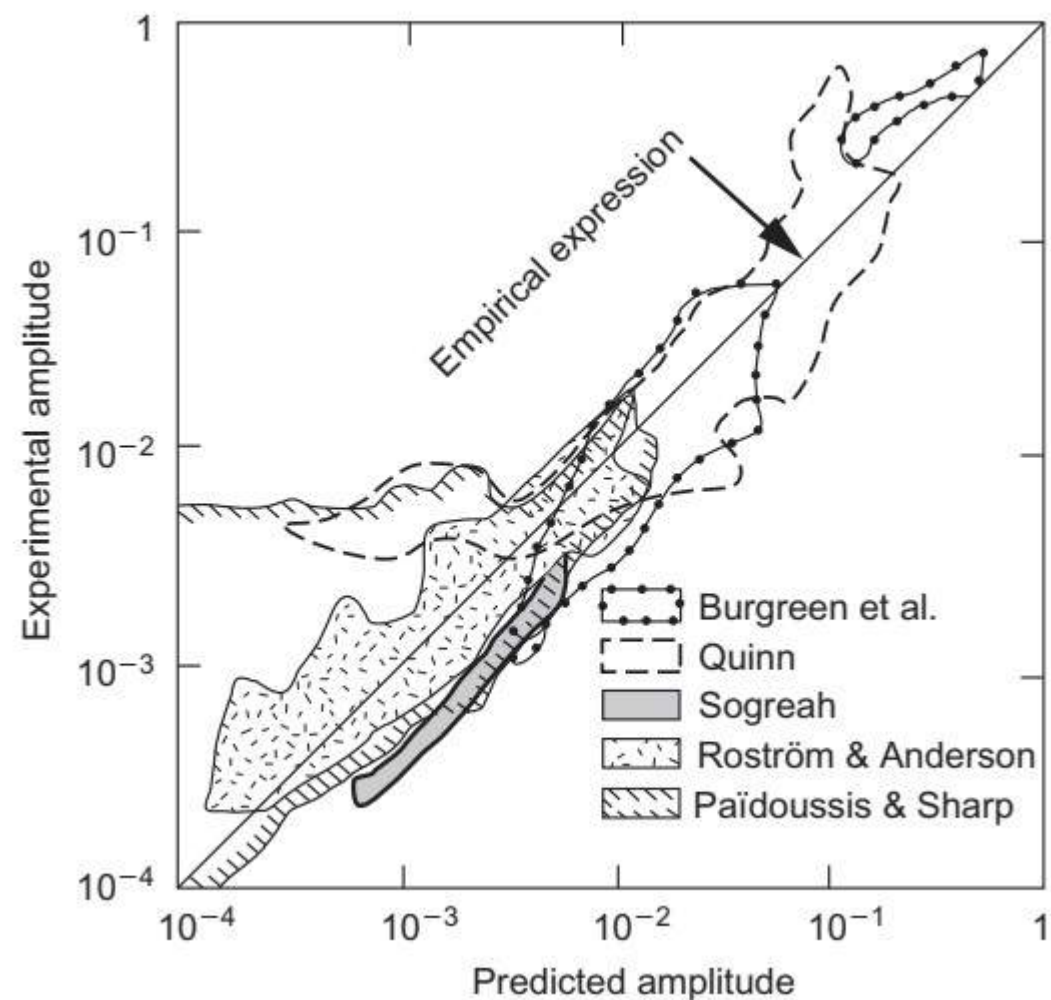


Cross velocity



Legend: ■ Exp — FFI_F - - FFF_L --- LNK

Vibration induced by axial flow



Païdoussis approach



$$\delta_v = 6.241e-05 \text{ [m]}$$

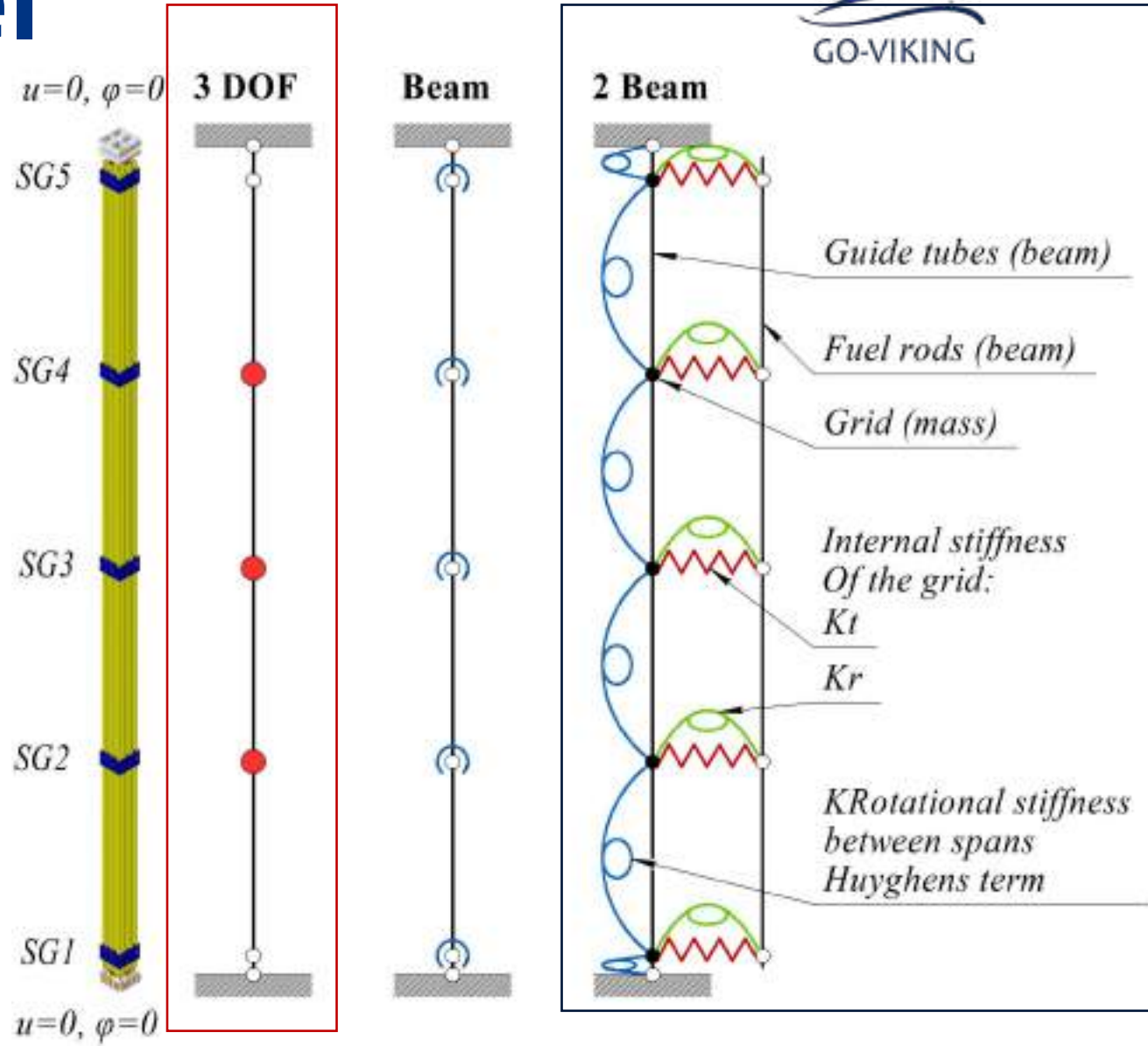
•

* M.P. Paidoussis, *An experimental study of vibration of flexible cylinders induced by nominally axially flow*, Nucl. Sci. Eng. 35 (1) (1969) 127138

Mechanical model

Possible ways in FA assembly representation:

- Rigid beams connected by torsion springs (3 DOF system);
- Flexible beams connected by torsion springs
- Two beams, one for guide tube another for fuel rods connected by torsion and tension springs



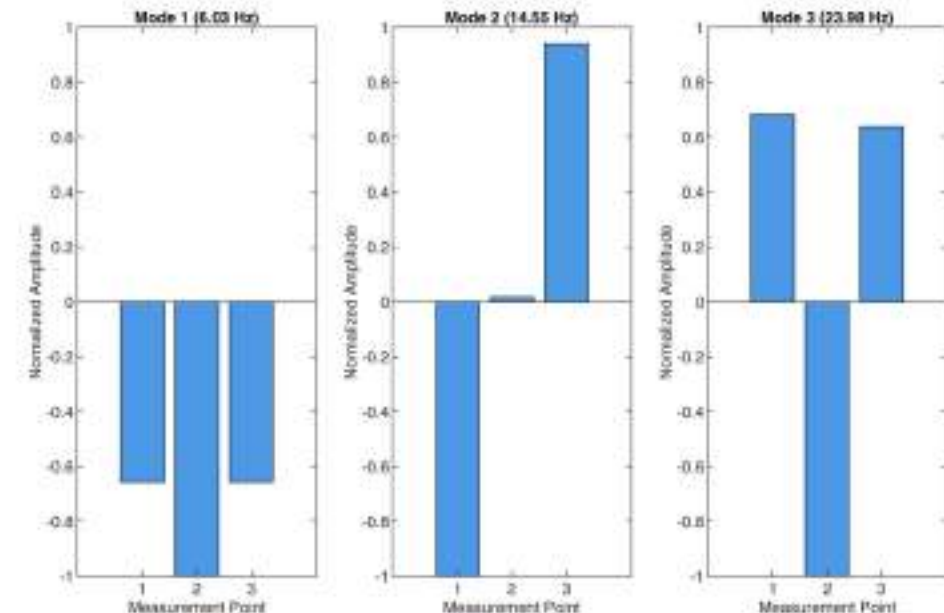
Restoring system characteristics from experimental data



- In practical structural dynamics, direct knowledge of system matrices (mass, damping, stiffness) is rarely available with complete accuracy. Instead, modal testing techniques allow us to extract observable modal properties: natural frequencies, damping ratios, and mode shapes. These can be inverted to reconstruct the underlying physical matrices of the system. This process is known as inverse modal identification.
- Inverse identification is inherently sensitive to the quality of experimental data. Challenges include noise contamination, incomplete mode shape sets, modal truncation, nonlinearity, and sensor/actuator limitations. Moreover, inverse problems are often ill-conditioned: small perturbations in the input (e.g., mode shapes) may lead to large variations in the output (e.g., reconstructed matrices).

Input data:

- Vector of natural frequencies {6.0 14.4 23.9} Hz
- Mode shapes
- Damping ratios





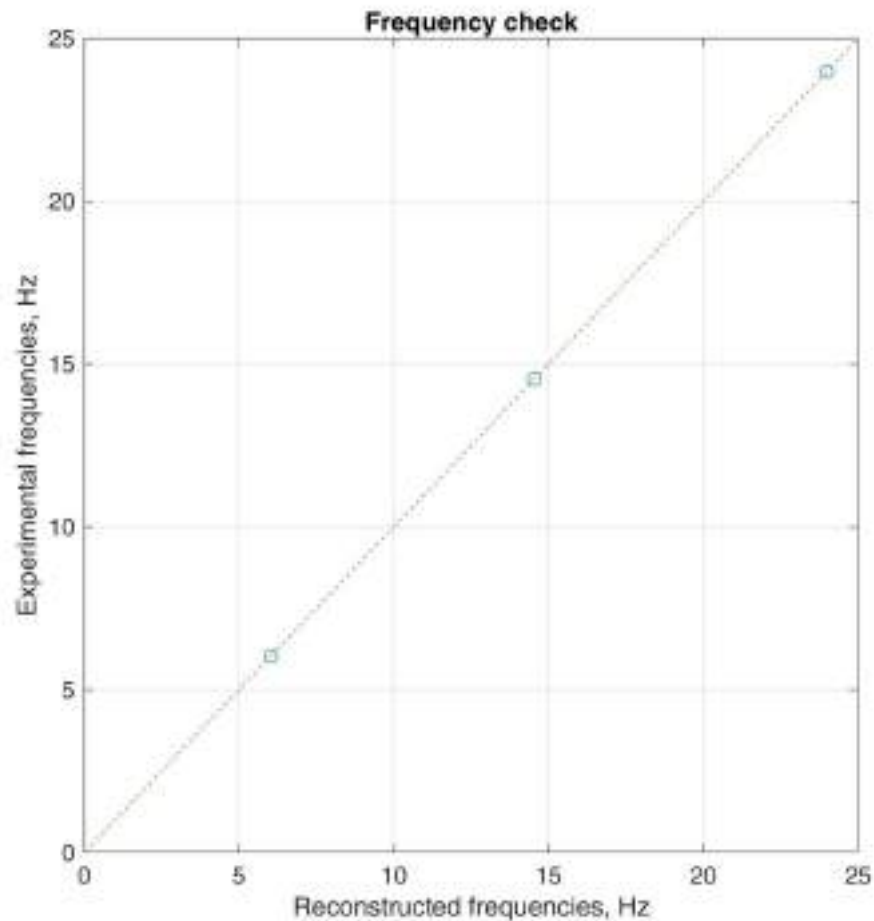
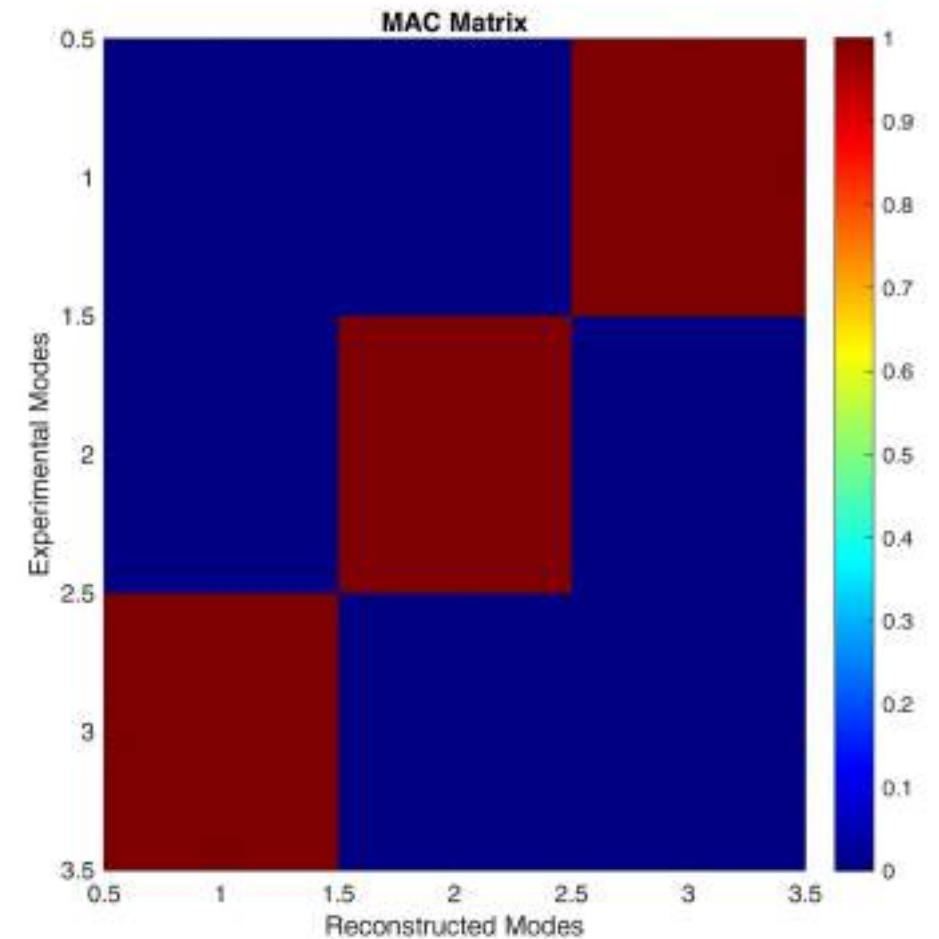
Matrix Reconstruction Methods

- To accurately capture the dynamic behavior of the assembly, three distinct mathematical approaches were utilized to reconstruct the Mass, Stiffness, and Damping matrices from experimental modal data:
- **Singular Value Decomposition (SVD) Method** * **Concept:** A foundational algebraic approach that extracts perfectly orthogonal mode shapes from raw experimental data.
 - **Application:** Reconstructs the system matrices by enforcing strict modal orthogonality. It is highly efficient and provides a mathematically pure baseline, though it assumes uniform mass distribution and idealized boundary conditions.
- **Direct Least Squares (Equation Error) Method**
 - **Concept:** Formulates the dynamic equations of motion as an overdetermined system and solves for the unknown matrix elements by minimizing the equation error.
 - **Application:** Directly calculates the symmetric Stiffness and Damping matrices based on a known diagonal Mass matrix. Unlike SVD, it does not force artificial orthogonality, providing a realistic physical compromise that better preserves the raw experimental mode shapes.
- **Direct Matrix Updating (Berman & Nagy / Baruch & Bar Itzhack)**
 - **Concept:** An advanced Finite Element Model Updating (FEMU) technique that uses an analytical baseline model (e.g., Hencky bar-chain) as a starting point.
 - **Application:** Solves a constrained optimization problem to make the smallest possible mathematical corrections to the baseline matrices. The Berman method updates the Mass matrix to satisfy orthogonality, while the Baruch method updates the Stiffness matrix to satisfy dynamic equilibrium, resulting in the most physically accurate system representation.

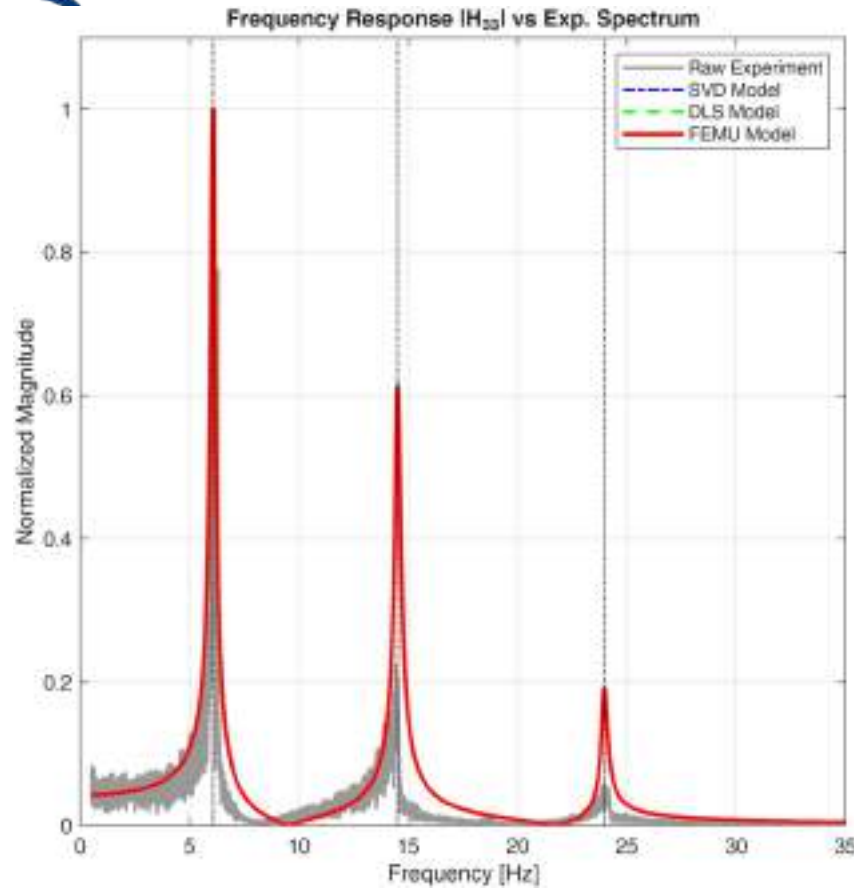
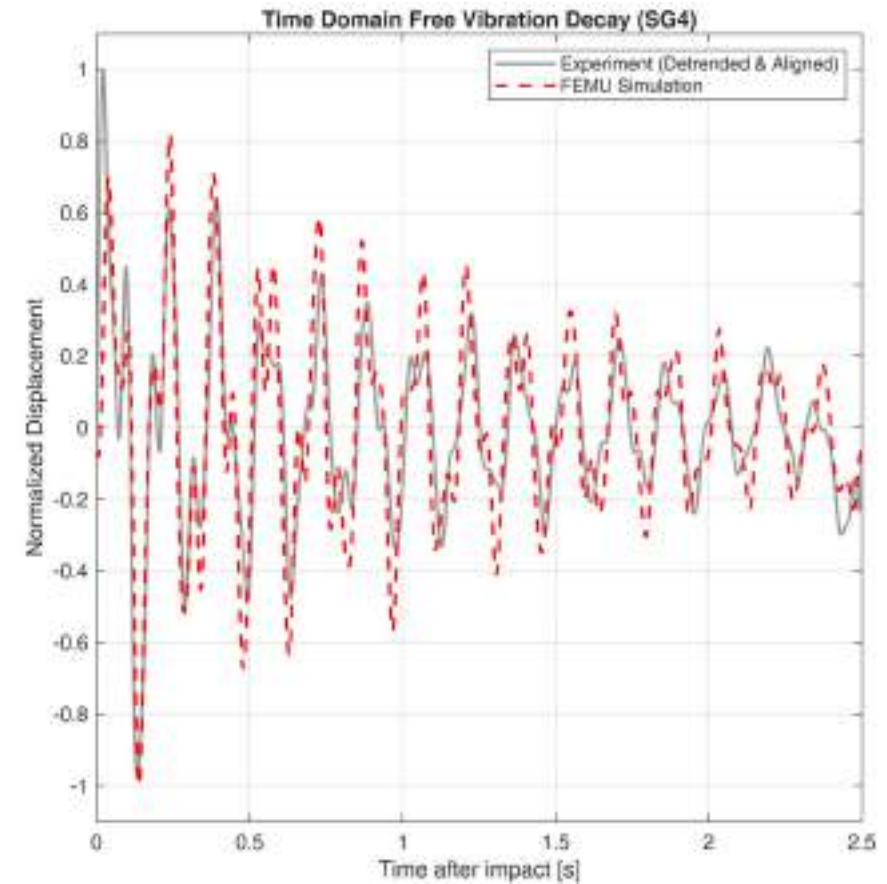
Checks



The Modal Assurance Criterion (MAC) is a quantitative measure of similarity between two mode shapes. It is defined as:

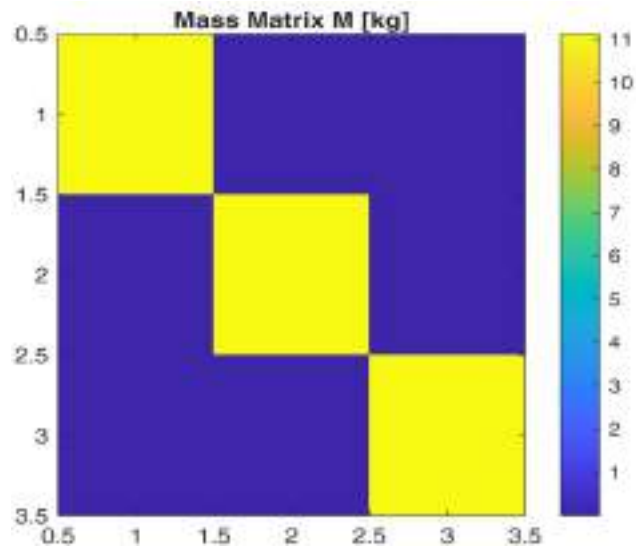
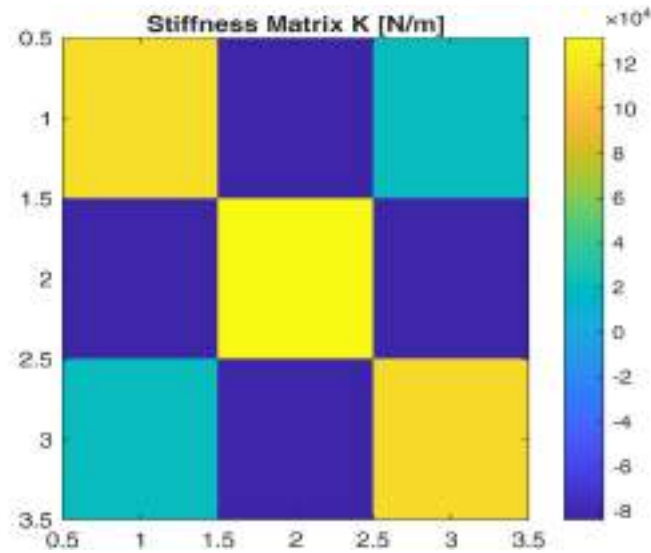
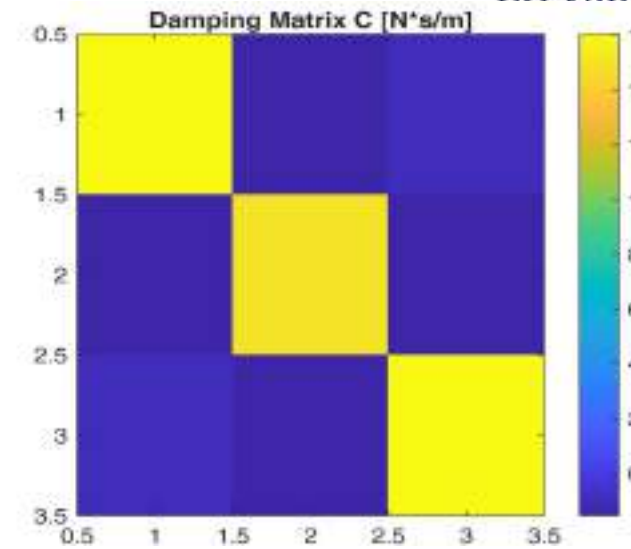
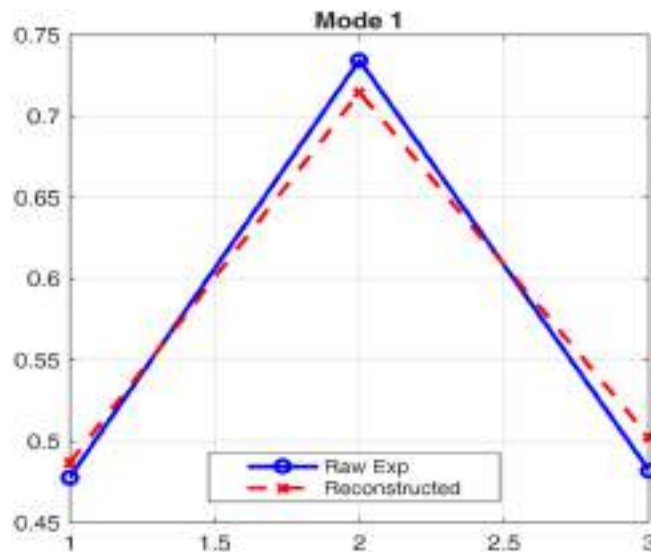
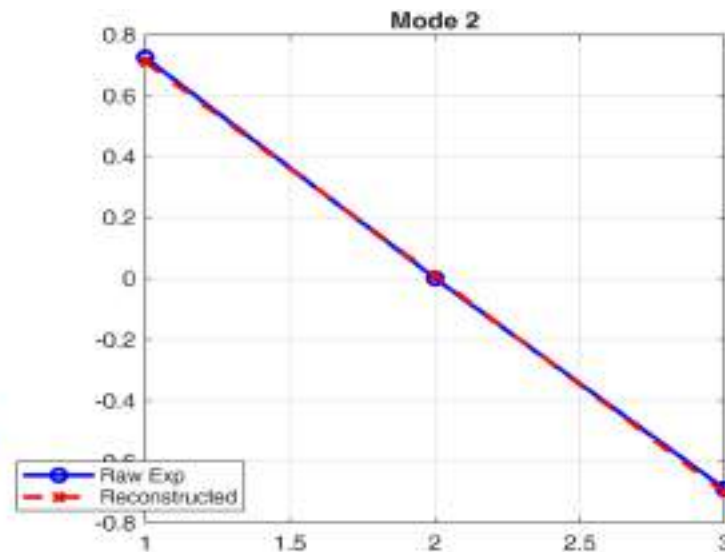
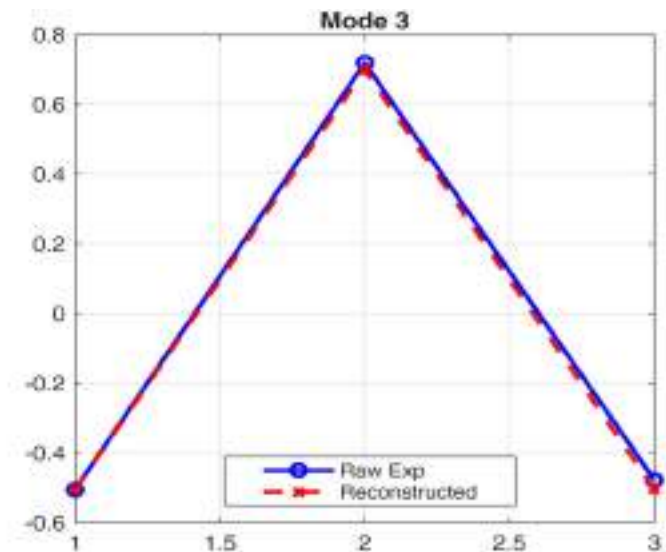


Checks



Singular Value Decomposition (SVD)
Direct Least Squares (Equation Error) method.
Berman & Nagy and Baruch & Bar Itzhack updating methods

Result



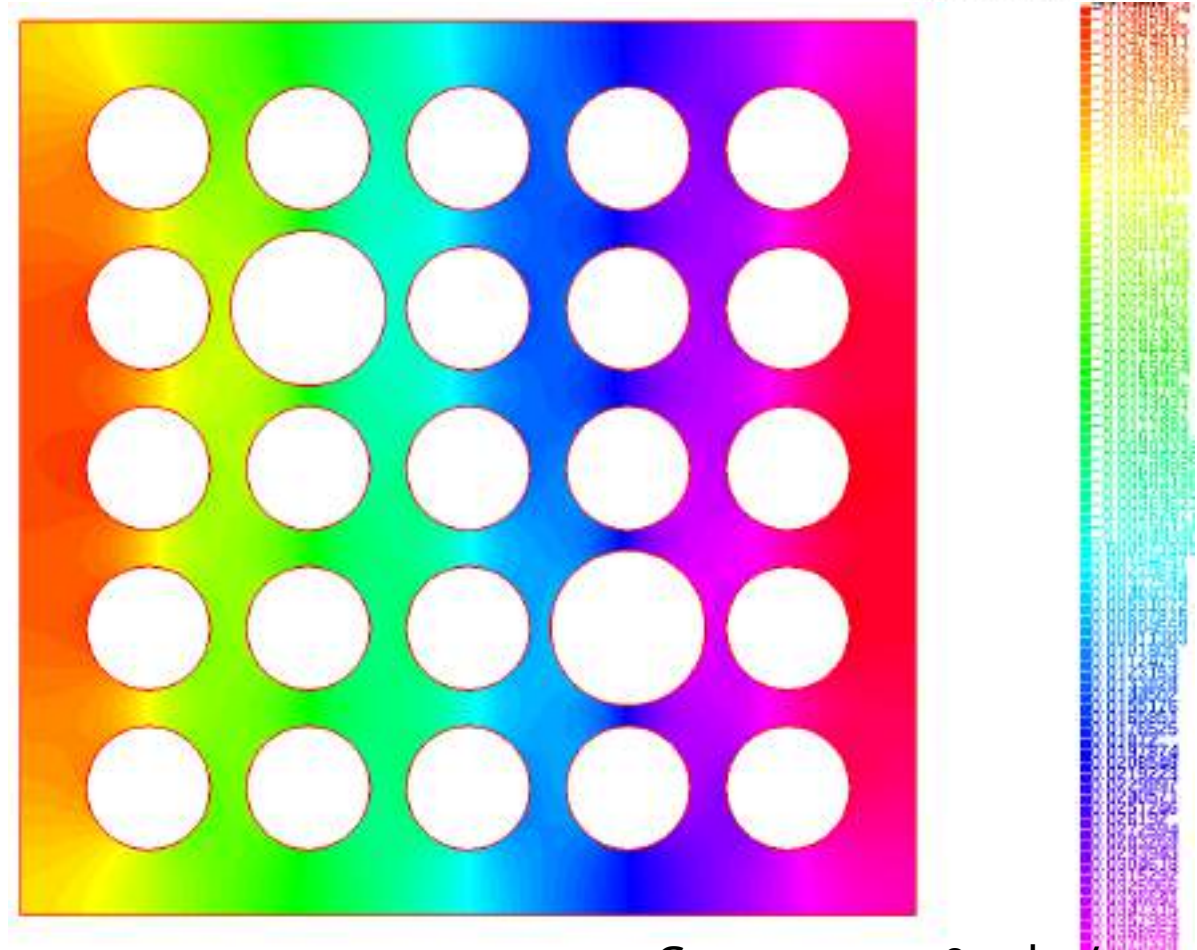
Added mass & damping

Still water, incompressible viscous fluid. Solution using FreeFEM++

```

91 problem Stokes([ux,uy,p],[vx,vy,q], solver=sparse) =
92   int2d(Th){
93     (in*omega*rho)*(ux*vx + uy*vy)
94     + mu*( dx(ux)*dx(vx) + dy(ux)*dy(vx)
95           +dx(uy)*dx(vy) + dy(uy)*dy(vy) )
96     - p*(dx(vx)+dy(vy))
97     - q*(dx(ux)+dy(uy))
98   }
99   + on(1, ux=in*omega, uy=0) // inner cylinder moves in x
100   + on(2, ux=0, uy=0); // outer wall at rest
101
102 // solve
103 Stokes;
104 // Adapt mesh for boundary layer
105 for (int step = 8; step < 2; step++) {
106   Th = adaptmesh(Th, real(ux), imag(ux), real(uy), imag(uy),
107                 err=0.05, nbvx=30000, hmin=1e-5);
108   Stokes;
109 }
110
111 // ----- compute complex hydrodynamic force on inner cylinder (label 1) -----
112 // Traction: t = (-p I + mu (grad u + grad u^T)) n
113 // x-component: t_x = -p*n.x + mu*( 2*d(ux)/dx * n.x + (d(ux)/dy + d(uy)/dx)*n.y )
114
115 complex Fx = int1d(Th,1){
116   -p*n.x
117   + mu*( 2.*dx(ux)*n.x + (dy(ux) + dx(uy))*n.y )
118 };
119
120 // Added mass and viscous damping (per unit thickness)
121 real mxx = - real(Fx)/(omega*2);
122 real cxx = imag(Fx)/omega;

```



$$M_{\text{added}_f1} = 5.10843 \text{ kg/m}$$

$$C_{\text{added}_f1} = 25.1841 \text{ kg/m}$$

$$M_{\text{added}_f2} = 4.90351 \text{ kg/m}$$

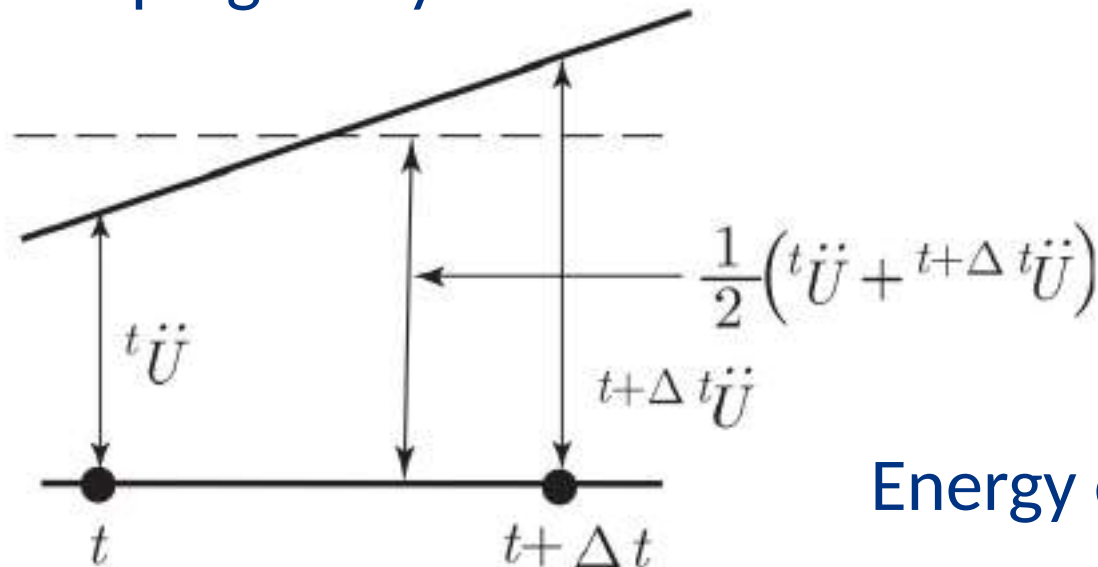
$$C_{\text{added}_f2} = 35.8753 \text{ kg/m}$$

$$M_{\text{added}_f3} = 4.82612 \text{ kg/m}$$

$$C_{\text{added}_f3} = 42.8861 \text{ kg/m}$$

Dynamic solution

The full **integration method** is a well-know traditional method, in our case we have implemented Newmark method with energy estimation to calculate the error. The benefit is that we can apply for the damping matrix of any type, i.e. obtained from experimental data and/or fluid damping analysis



$$\dot{U}_{t+\Delta t} = \dot{U}_t + [(1 - \delta)\ddot{U}_t + \delta\ddot{U}_{t+\Delta t}]\Delta t$$

$$U_{t+\Delta t} = U_t + \dot{U}_t\Delta t + [(1/2 - \alpha)\ddot{U}_t + \alpha\ddot{U}_{t+\Delta t}]\Delta t^2$$

Newmark $\delta = \frac{1}{2} \quad \alpha = \frac{1}{2}$

Energy error: $E_{ext} - [E_{kint} + E_{poten} + E_{damp}] < \varepsilon_{error}$



Dynamic solution

The **mode superposition method** is a dynamic analysis technique used to determine the response of a linear system to various excitations, with only some non-linearities allowed. It involves combining the natural frequencies and mode shapes obtained from a modal analysis to predict the structure's dynamic behavior. This method is computationally efficient, especially for linear dynamic analyses like harmonic and transient analysis. Especially it is efficient in combination with Duhamel integration technics

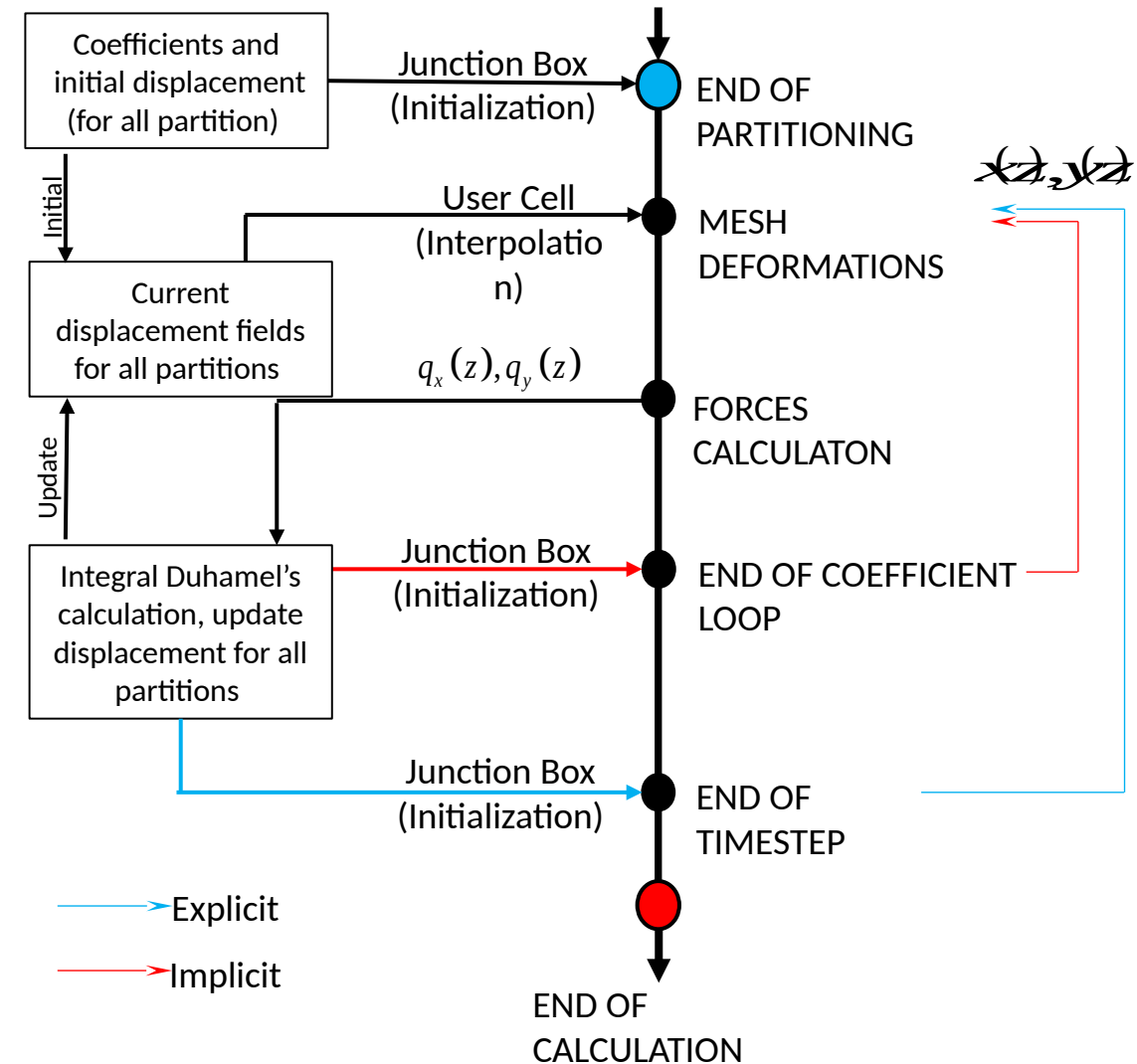
$$\ddot{\eta}_i(t) + \omega_i^2 \eta_i(t) = \int_0^l Y_i q(x, t) dx, \quad i = 1, 2, 3, \dots, \infty \quad \eta_i(t) = \frac{1}{\omega_i} \int_0^l Y_i \int_0^t q(x, \tau) \sin \omega_i(t - \tau) d\tau dx$$

The main problem is that the damping should be only of a special form, proportional damping. Based on comparison with full integration scheme, we have adjust the damping coefficient $\xi=0.22$.

Coupling. Fortran library



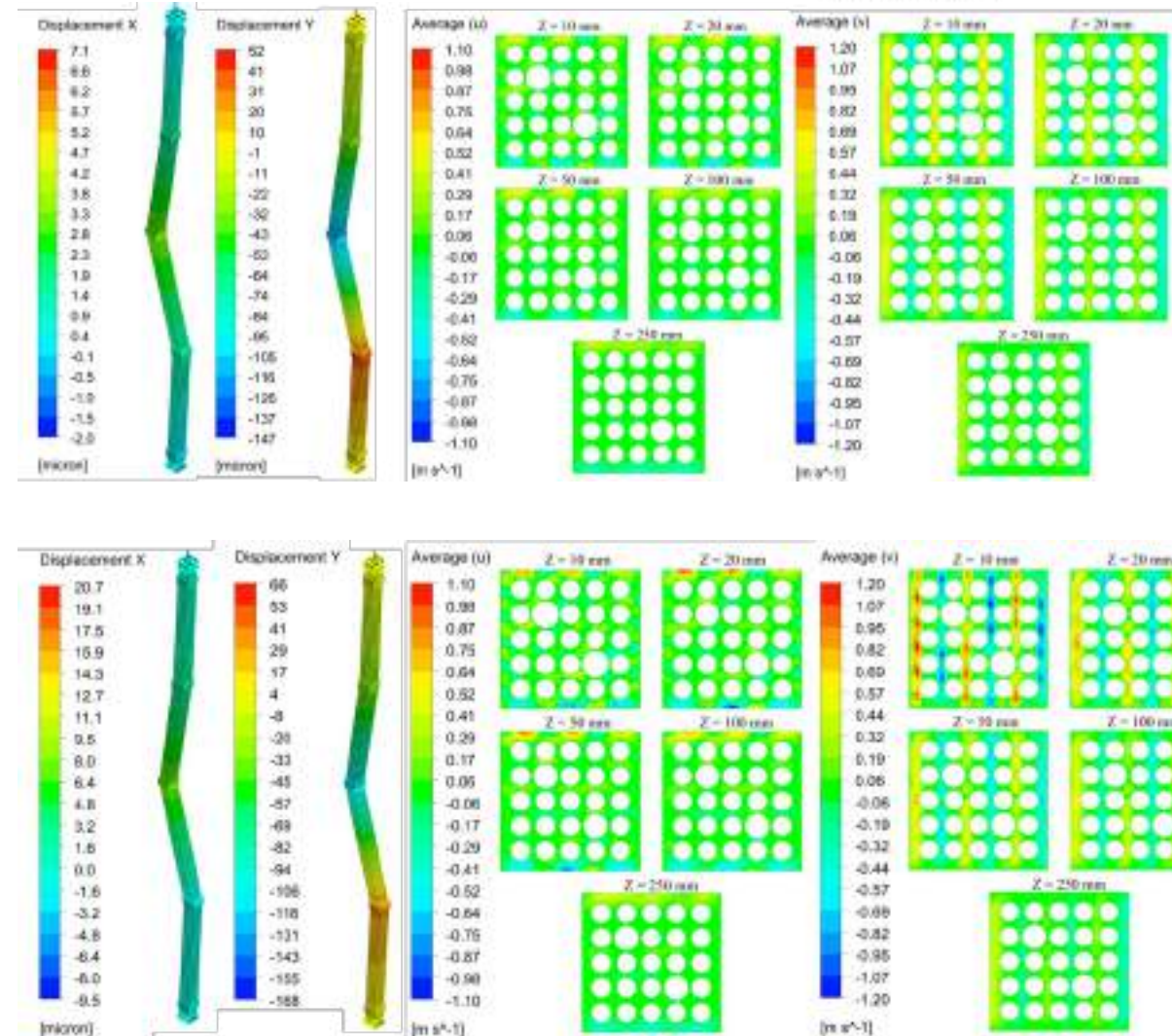
- The library is based on the following simplifying assumptions:
 - ✓ All elements of the assembly move as a homogenized body in the X and Y directions, which are considered independent.
 - ✓ The height displacement profile is formed as a linear function between the characteristic displacement points, which are selected at the centers of SG2, SG3 and SG4.
- For each reference point of the displacement calculation, the hydrodynamic force (the selected projection) is calculated as the sum on the SG walls, as well as on the halves of the lower and upper sections between the SGs.
- The displacement library implements two solvers, one of which is based on the Duhamel integral (DH) for an elastic dissipative system, and the second one is based on the Newmark method (NW). Both of these solvers are being tested further.
- Two options for determining the hydrodynamic force are also considered: total (SF) or exclusively the contribution of static pressure without taking into account the hydrostatic component (PF).
- The library is implemented in such a way that displacements can be calculated directly at the end of each coefficient loop or at the end of a time step, allowing for both implicit and explicit coupling. For further calculations, the explicit formulation is used.



Coupled FIV



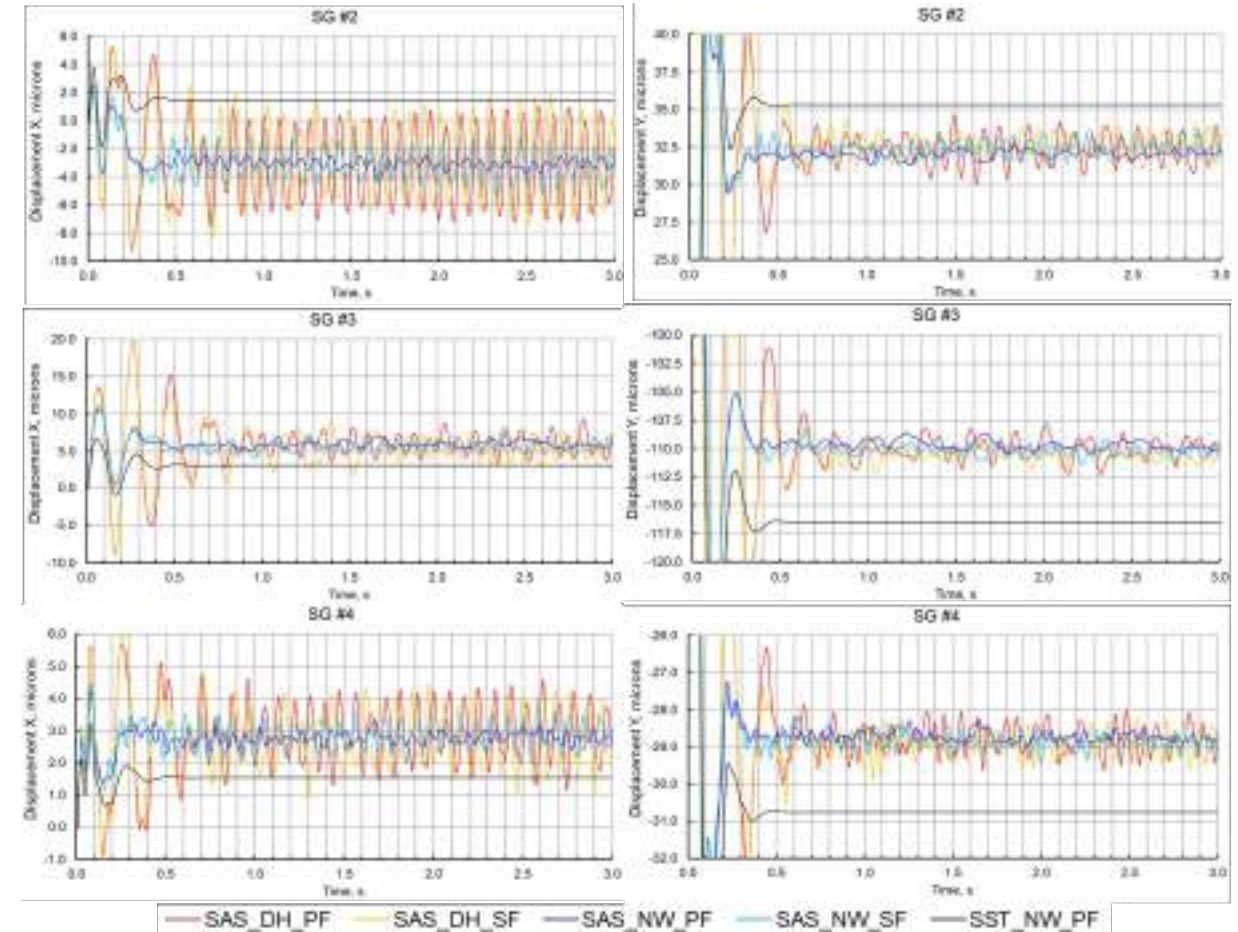
- SST model produces virtually no assembly oscillations — deviations from the initial position are fully damped even in the absence of structural damping
- SAS-SST model sustains oscillations throughout the transient — qualitatively consistent with experimental behavior
- Discretization parameters and time step size have negligible influence on displacements under SST
- Oscillation amplitude under SAS-SST is noticeably smaller than experiment — attributed to underestimation of turbulent pulsations by the model



Coupled FIV



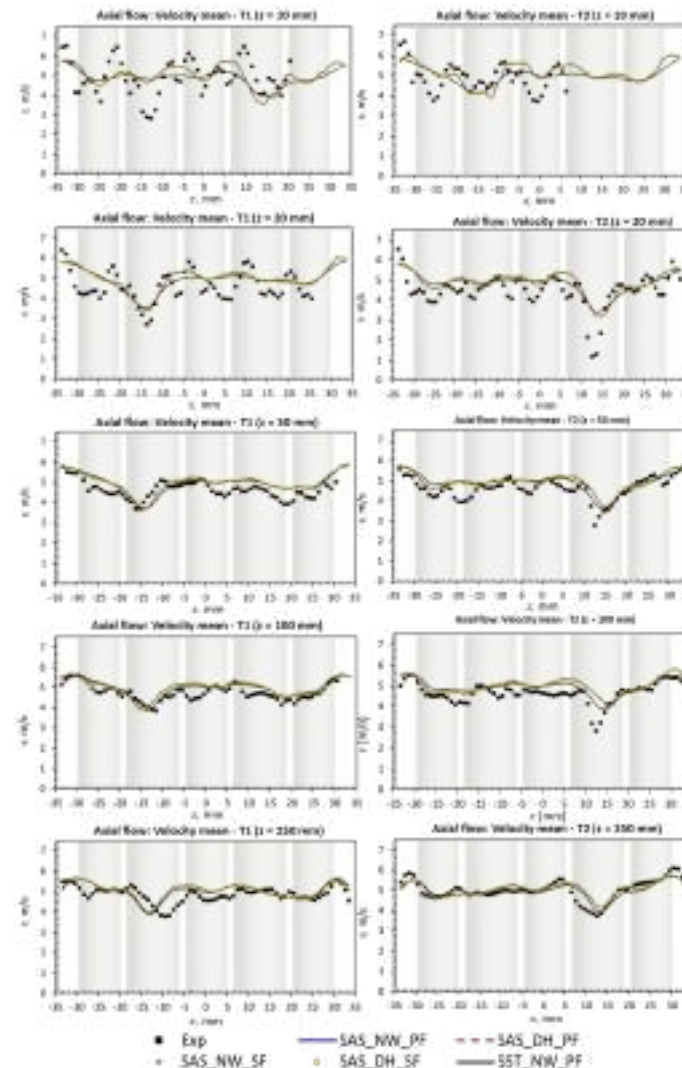
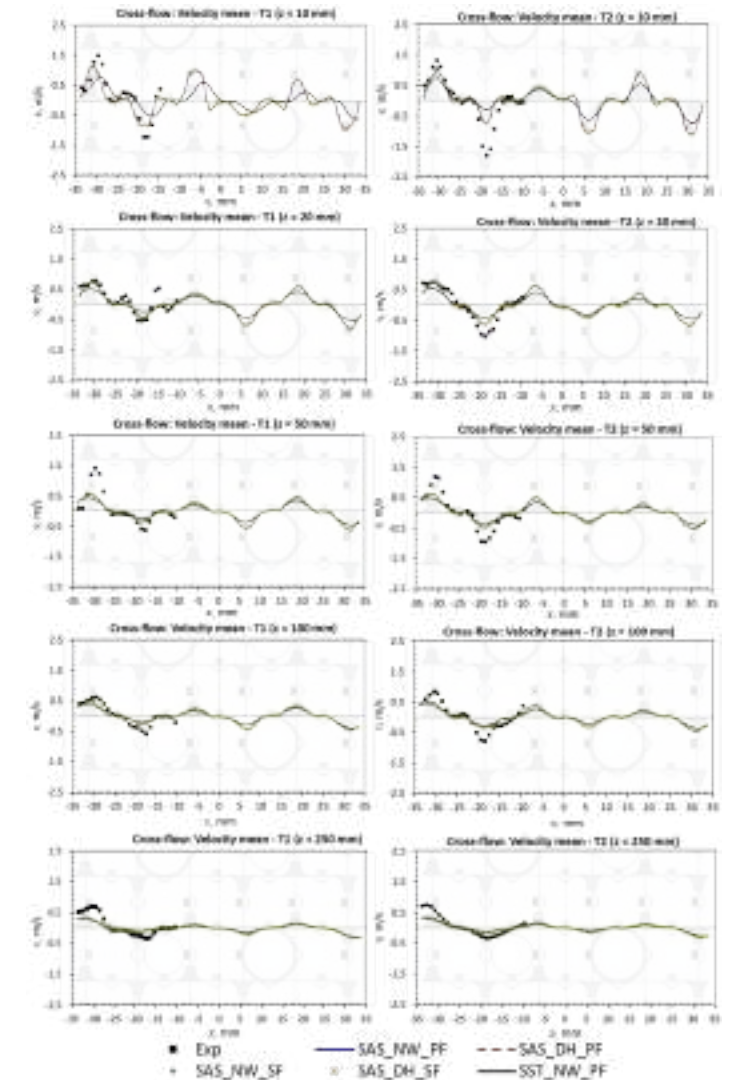
- Clear contrast in transient behavior: SST oscillations decay to near-zero within the first seconds; SAS-SST maintains persistent oscillations
- SAS-SST oscillation amplitude is significantly below experimental measurements
- Underestimation caused by the turbulence model's limited ability to resolve vortex structures that develop behind spacer grids
- RMS velocity values are 35–60% lower than experimental data across all calculation variants



Coupled FIV



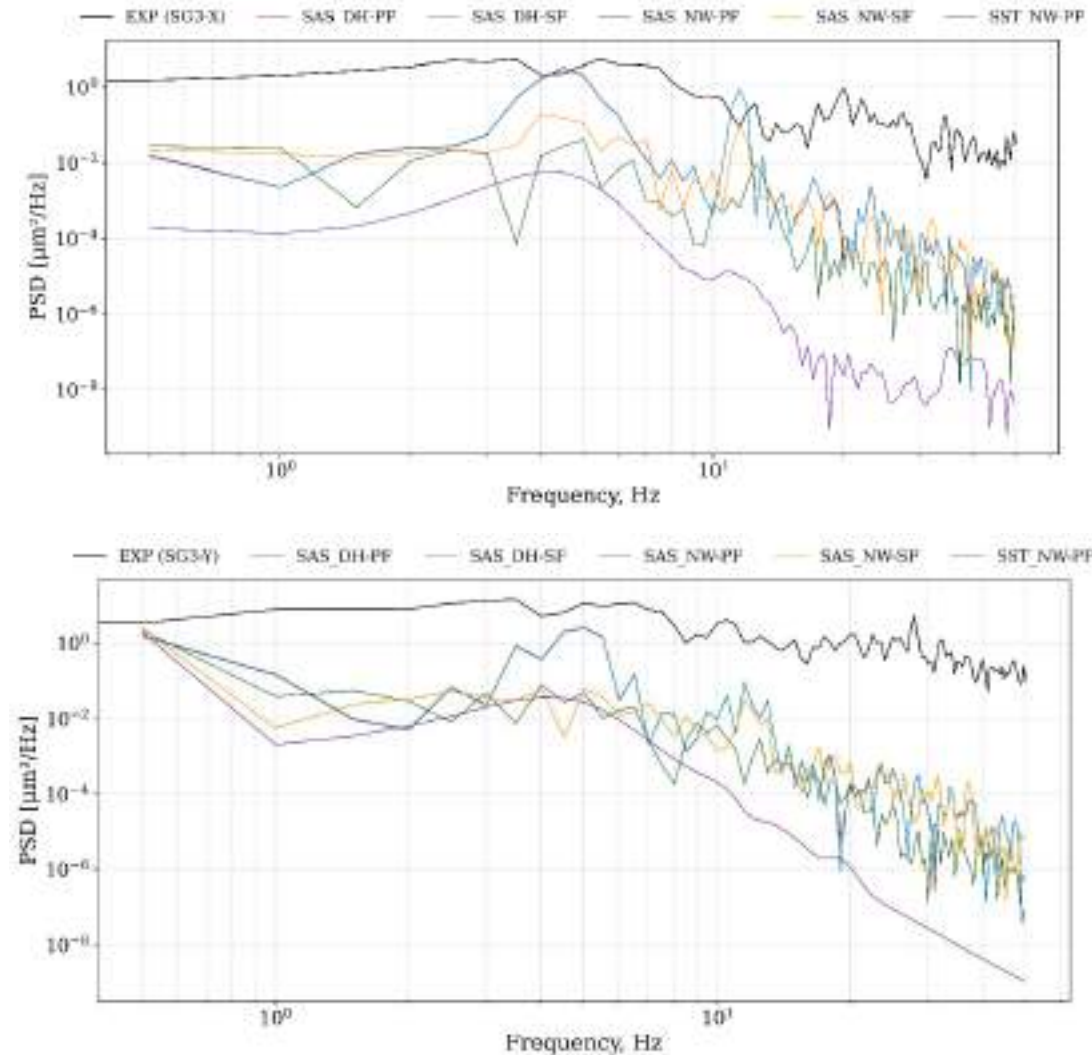
- Agreement with experiment improves with distance from spacer grids; significant discrepancies observed directly behind grids
- Axial velocity discrepancies near grids largely attributed to domain decomposition and fluid-fluid interface effects
- Transverse velocity profiles are more representative for hydrodynamic force estimation than axial profiles



Coupled FIV. Spectra comparisons



- Vibration spectra are qualitatively similar to experiment in the low-frequency range, but amplitudes are significantly underestimated in all calculation variants
- From a scientific perspective: resolving the turbulent vortex structure behind spacer grids requires SRS methods with $\sim 10^8$ cells and low-dissipation advection schemes
- For engineering use: URANS/SAS models supplemented with empirical or synthetic turbulence sources (e.g., Vortex Confinement) offer a practical path to better amplitude prediction
- SAS-SST acts as a deterministic eddy model — enabling oscillations that SST cannot reproduce, but still underestimates the hydrodynamic forcing level



Conclusions



- SAS-SST captures flow-induced assembly oscillations; SST model fully damps them — confirming that turbulent pulsation modelling is critical for vibration prediction
- All URANS/SAS variants underestimate turbulent velocity fluctuations — calculated RMS values are approximately 35–60% of experimental data
- Largest discrepancies occur directly behind spacer grids; prediction accuracy improves steadily with distance downstream
- Engineering solution: supplement computational models with empirical or synthetic turbulence inputs (e.g., Vortex Confinement, geometry-corrected swirl) to achieve acceptable vibration amplitude estimates within practical timeframes
- High-fidelity SRS calculations ($\sim 10^8$ mesh cells, low-dissipation schemes) can reproduce the turbulent structure more accurately but are computationally prohibitive for routine engineering analysis





Thank



Yaroslav DUBYK
you!



IPP-CENTRE



dubyk-yr@ipp-centre.com.ua

www.go-viking.eu



Funded by the
European Union



ALAIN Benchmark Outcome

GO-VIKING Final Workshop Garching 20-22 May 2026, Hidajet Hadžić, Framatome GmbH



Funded by
the European Union

Benchmark objectives

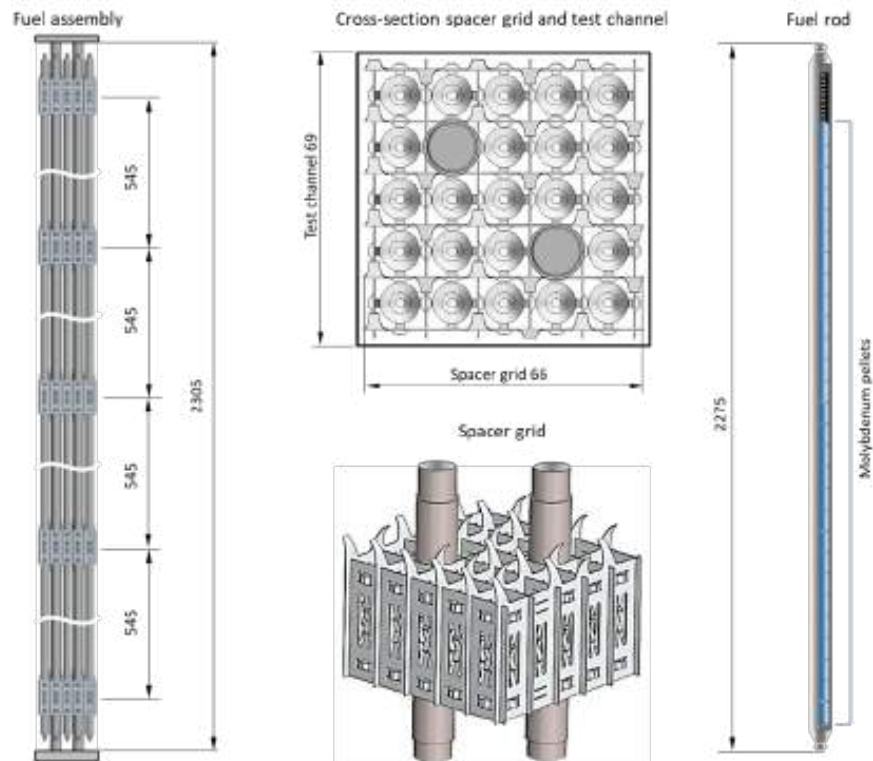
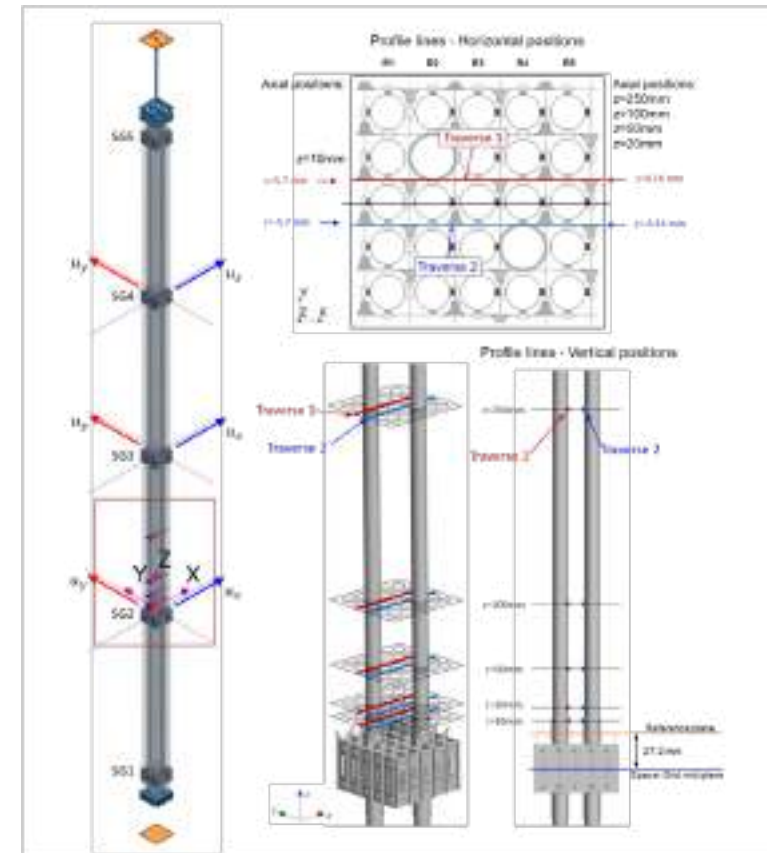
Objective is to perform numerical FIV simulations for given setup and flow rate corresponding to 5.2 m/s average axial velocity

Numerical simulations should:

- Reproduce flow field
 - Mean velocity profiles
 - RMS velocity profiles
- Reproduce vibration behavior of the bundle measured experimentally in terms of
 - Vibration amplitude: RMS – displacement at SG 2, 3 and 4 (quantitative)
 - Vibration response frequency spectrum (qualitative)

Problem definition

- Task: Simulation of FIV for the reduced 5x5 fuel assembly used in tests

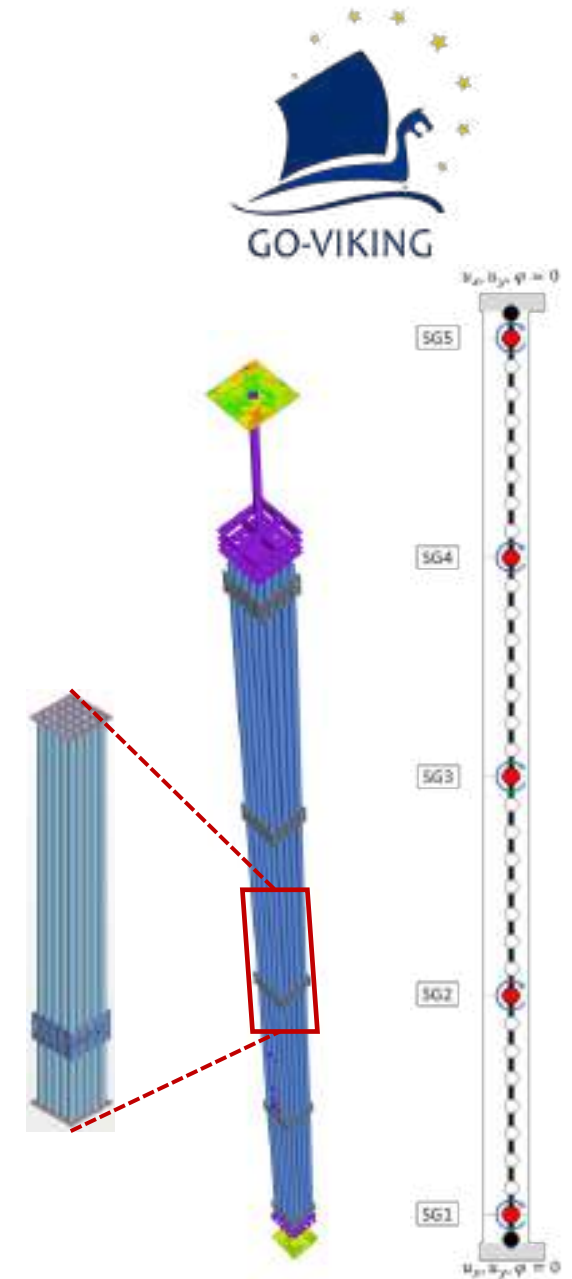


Benchmark participants

- FRA-G (Framatome GmbH)
- CEA (Commissariat à l'énergie atomique et aux énergies alternatives)
- GRS (Gesellschaft für Anlagen- und Reaktorsicherheit gGmbH)
- IPP (IPP Centre LLC)

Methods

- **General approach**
 - Use CFD for fluid flow analysis
 - Use mechanical model for structural dynamic analysis
- **CFD modeling – geometrical configurations**
 - Full-scale model (whole setup)
 - Single span model (only one span modeled)
- **Mechanical modeling**
 - Based on beam elements with additional features to match the mechanical properties of real structure
- **Analysis of FIV**
 - Decoupled → CFD and structural dynamic simulations performed separately
 - Coupled FSI → simultaneous simulations



Some general remarks



- **Configuration**

- From the “real-world” standpoint very small
- From the numerical simulation standpoint huge → thus, big impact on computational costs

- **Experimental results**

- Vibration amplitude measured in FIV tests a few μm
- Q1: what is FIV mechanism? Turbulence-induced?
- Q2: What are the consequences for numerical simulation?
→ need/necessity for coupled FSI or decoupled sufficient?
- Turbulence models to be used in CFD

- **Feasibility**

- What is feasible to do?



CFD simulation strategies

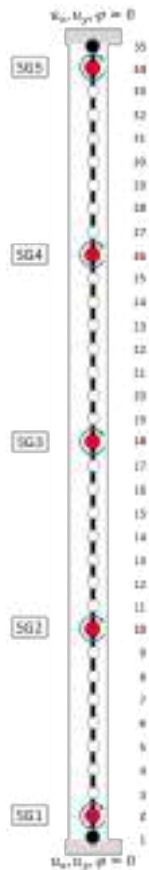


	FRA-G	CEA	GRS	IPP
CFD code(s)	STAR-CCM+	TrioCFD	OpenFoam	ANSYS CFX
CFD models geometry	single span full-scale	single-spn full-scale	single span	full-scale
Turbulence modeling approach	LES DES	LES URANS	LES URANS	SAS URANS
Turbulence / SGS models	Smagorinsky WALE SpalartAlmaras Eliptic Blending SST	WALE k-e	k-e (rlz) k-e cubic k-wSST RSMLLR	SAS-SST k-wSST

Mechanical model simulation strategies



FRA

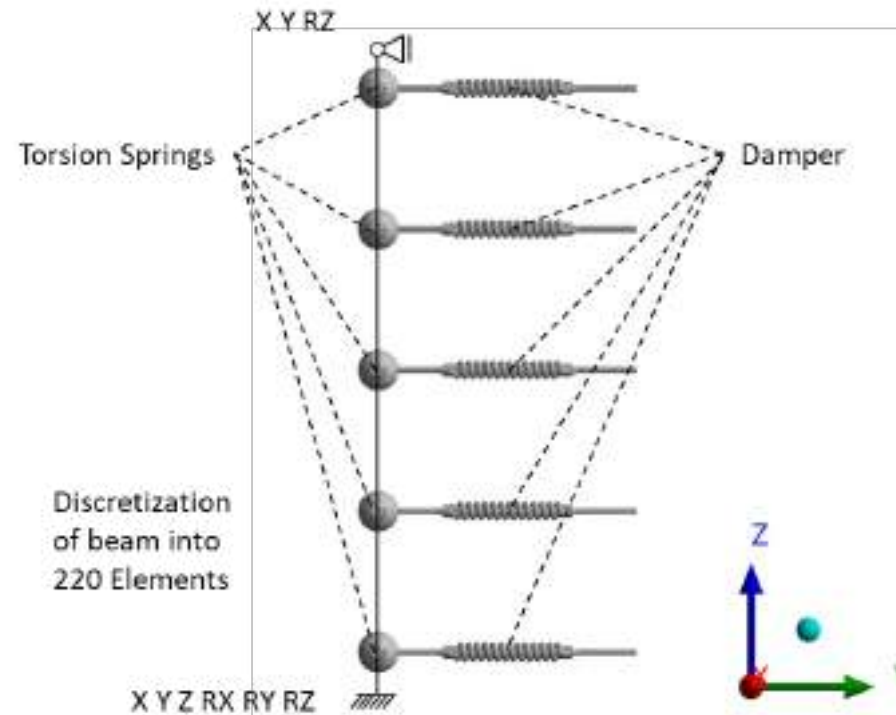


CEA



cast3m

GRS

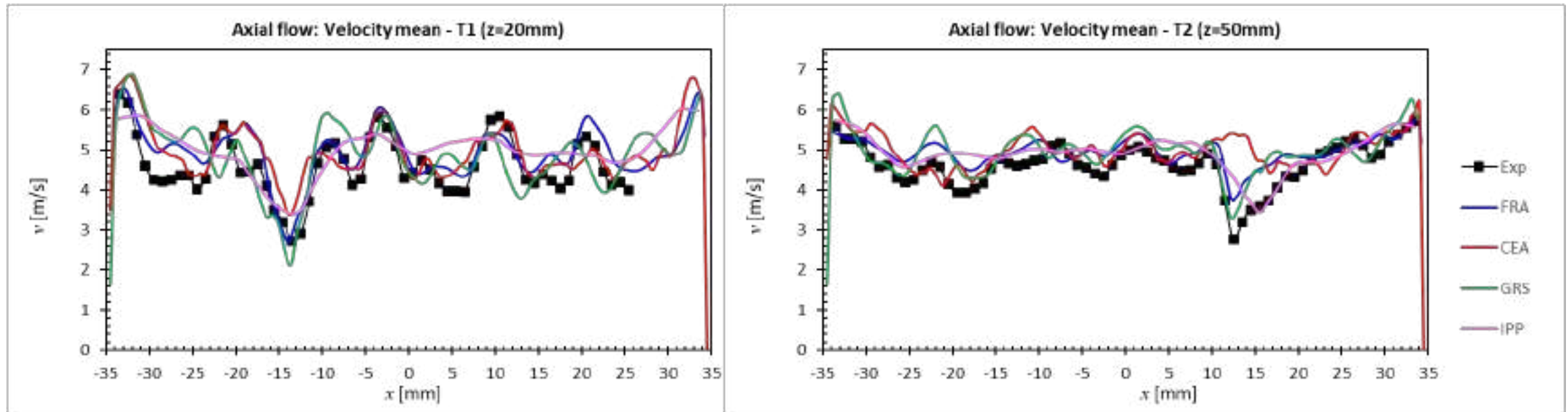


ANSYS Mechanical

IPP

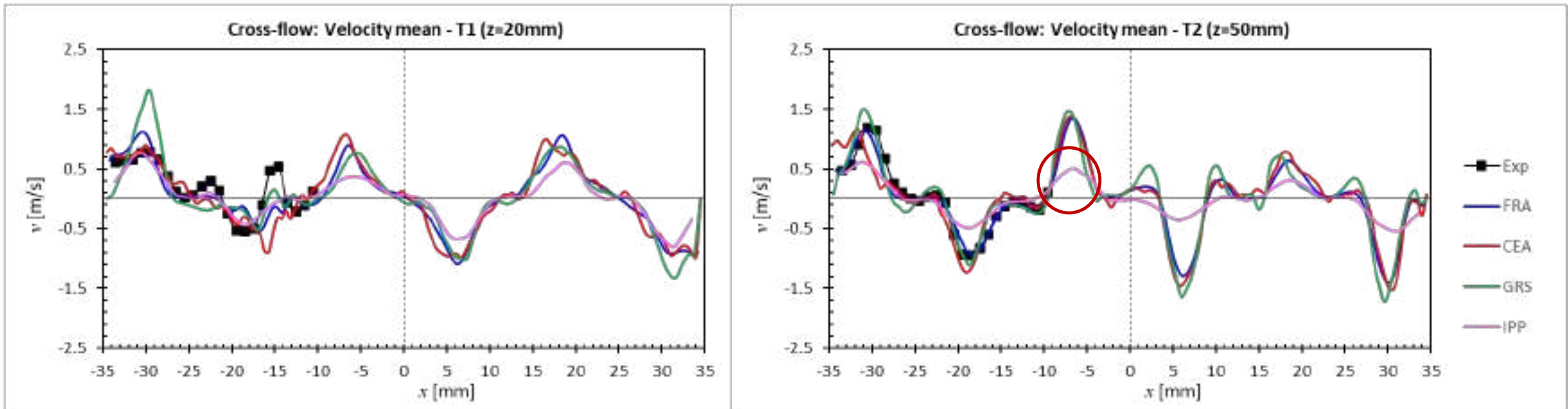


Comparison of selected results – flow profile



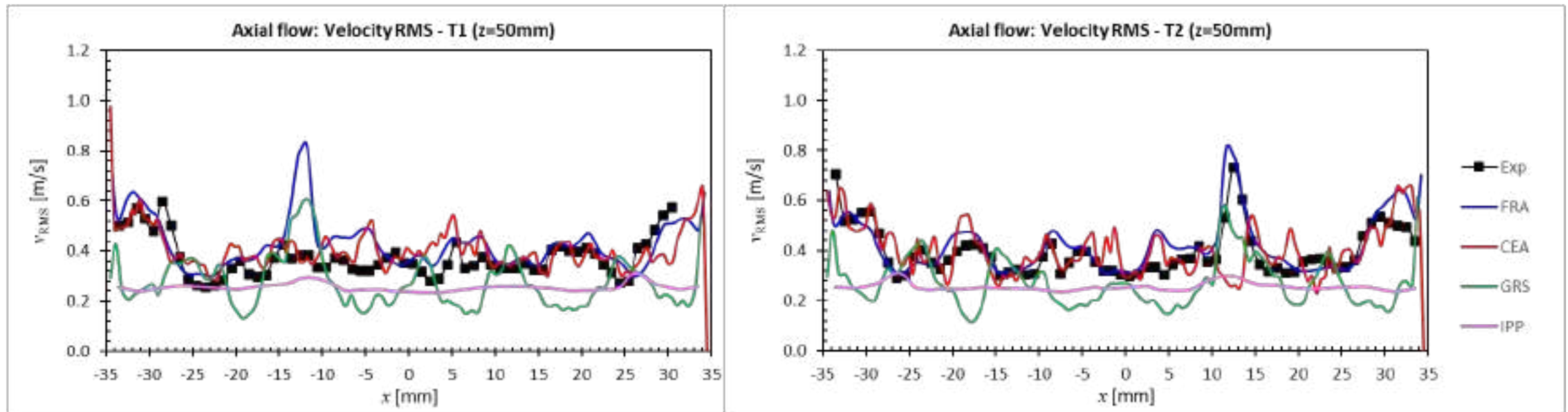
FRA: LES-WALE, CEA: LES_WALE, GRS: LES, IPP: SAS (NW, PF)

Comparison of selected results – flow profile



FRA: LES-WALE, CEA: LES_WALE, GRS: LES, IPP: SAS (NW, PF)

Comparison of selected results – flow profiles RMS

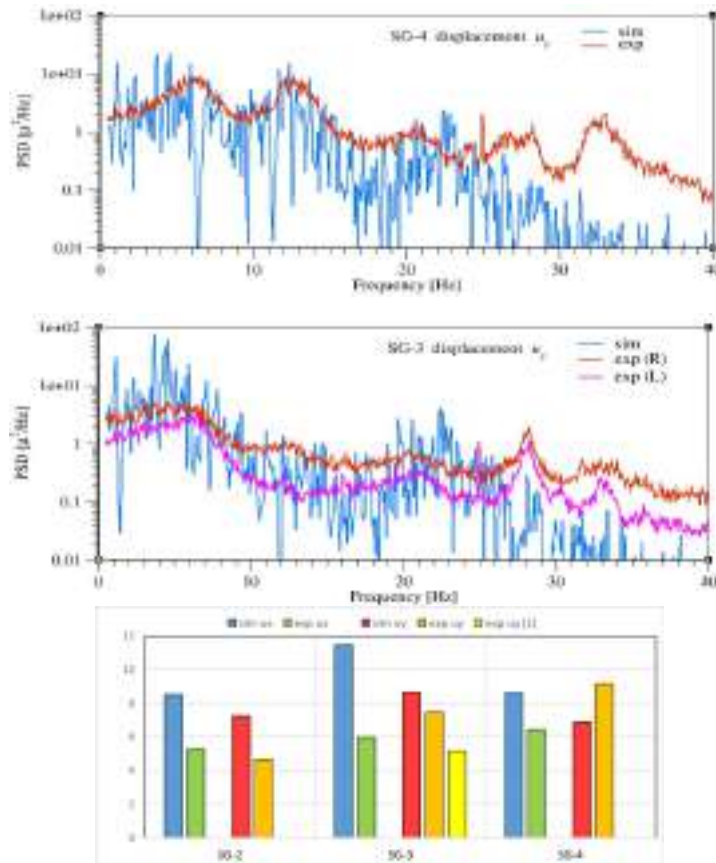


FRA: LES-WALE, CEA: LES_WALE, GRS: LES, IPP: SAS (NW, PF)

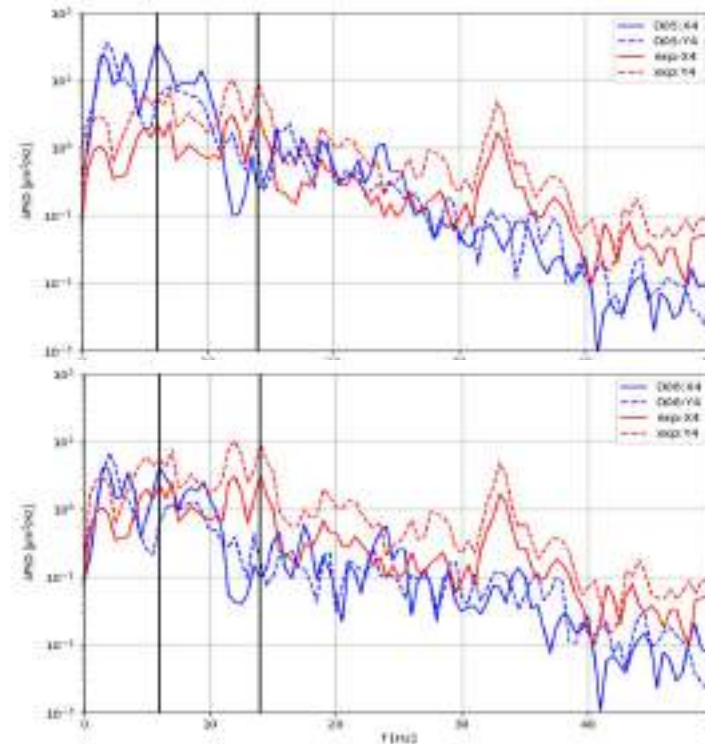
Comparison of selected results – displacements



FRA

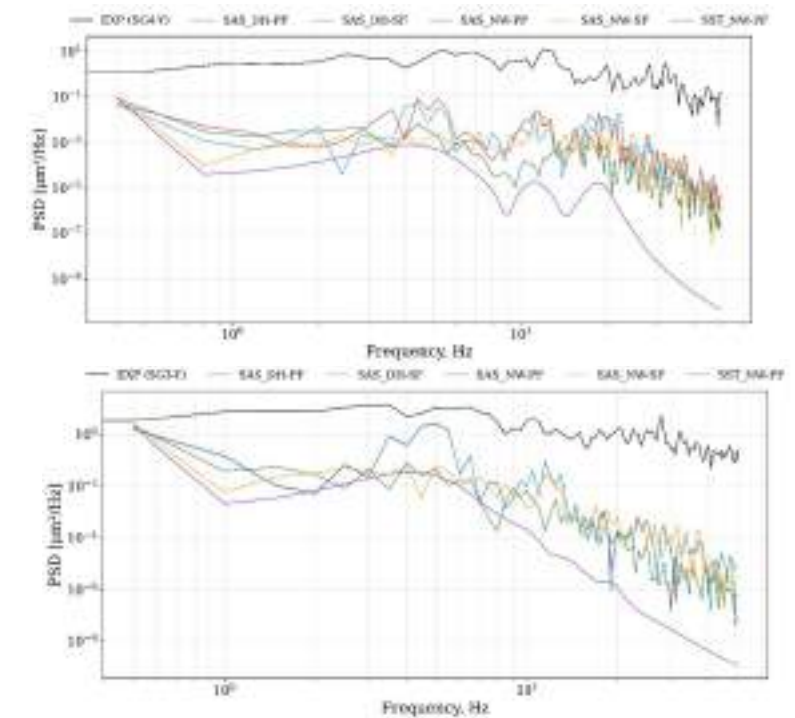


GRS



$U_x = 6.66 \mu\text{m}$ $u_y = 7.04 \mu\text{m}$ $D = 0.5$
 $U_x = 2.95 \mu\text{m}$ $u_y = 3.1 \mu\text{m}$ $D = 0.8$

IPP



Conclusions

- Case is very challenging for simulation – very high computational effort required
- Very low vibration amplitudes in the tests suggesting TIV as the main mechanism
- Turbulence-resolving methods required to capture turbulent fluctuations properly
- SAS approach seems not sufficient – too strong damping of fluctuations
- Need for coupled FSI not clear - due to computational effort and other problems related to moving mesh in combination with LES ; Only IPP performed coupled FSI
- Is decoupled simulation sufficient → for estimate of vibration amplitude in the frame of a TIV analysis may be sufficient, but one has to ensure that different errors/uncertainties cancel out resulting in a good result
- Important open point for this aspect is the evaluation of fluid effects on the dynamic behavior → added mass and fluid damping
- For unstable behavior coupled FSI is the only option
- ??

Partners



Associated Partners





Thank



Hidajet Hadžić
you!



Framatome GmbH



hidajet.hadzic@framatome.com



Funded by the
European Union



Single- and two-phase cantilever rod experiments on FIV in axial flow

Investigators: Mostafa R A Nabawy and Hector Iacovides

Other Contributors: Hao Li, Andrea Cioncolini, and Shanying Zhang



Funded by
the European Union

The University of Manchester

Scope of Current Research



- Generate high-resolution benchmark data that exposes new physical insights for axial-flow-induced vibrations in pressurized water reactors (PWRs) via measuring both structural responses and flow fields
- Use data to inform the development/validation of numerical models

Introduction

- There is a **paucity of experimental work** on axial-FIVs for nuclear reactor applications
- Usually, the **focus is solely on structural vibrations**, and not the associated flow fields
- We consider a **simple, yet informative experimental setup** of a cantilevered rod enclosed within a vertical tube while being subjected to axial water
- The rod is made of stainless steel and is packed with lead shot, yielding **dimensions and linear mass density akin to that of water-cooled nuclear reactor fuel rods**
- Two experimental configurations were considered: a **free-clamped configuration**, in which water flows towards the rod's free end, a configuration of huge relevance to FIVs within nuclear reactors; and a **clamped-free configuration**, in which water flows away from the rod's free end, a configuration that is very interesting from a fluid mechanics point of view
- **Different geometries for the rod free end** were also considered, and the **Reynolds number was changed over a broad range from the $O(10^3)$ to $O(10^4)$**
- Rod vibration motion and flow field behaviors were assessed using **non-contact optical methods**, including **rapid video imaging** to track the rod's motion and **particle image velocimetry (PIV)** to visualize the flow field

Experimental Rig

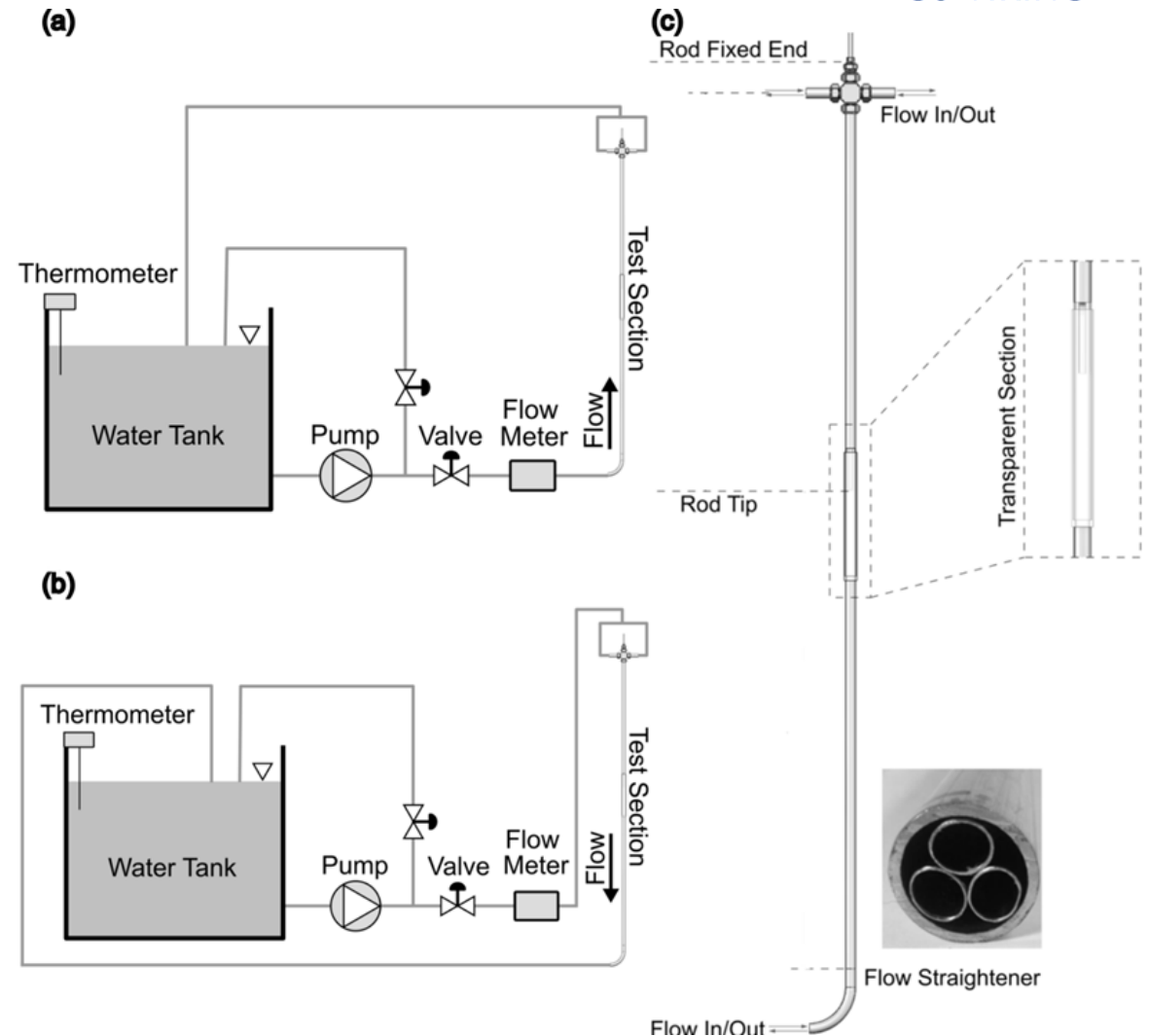


Experimental set-up is informative for PWRs; **simplified (single rod and no spacer grids) but not trivialized (tight confinement gives strong two-way fluid-structure coupling)** to combine rich physics with a relatively simple configuration (hence good for benchmarking)



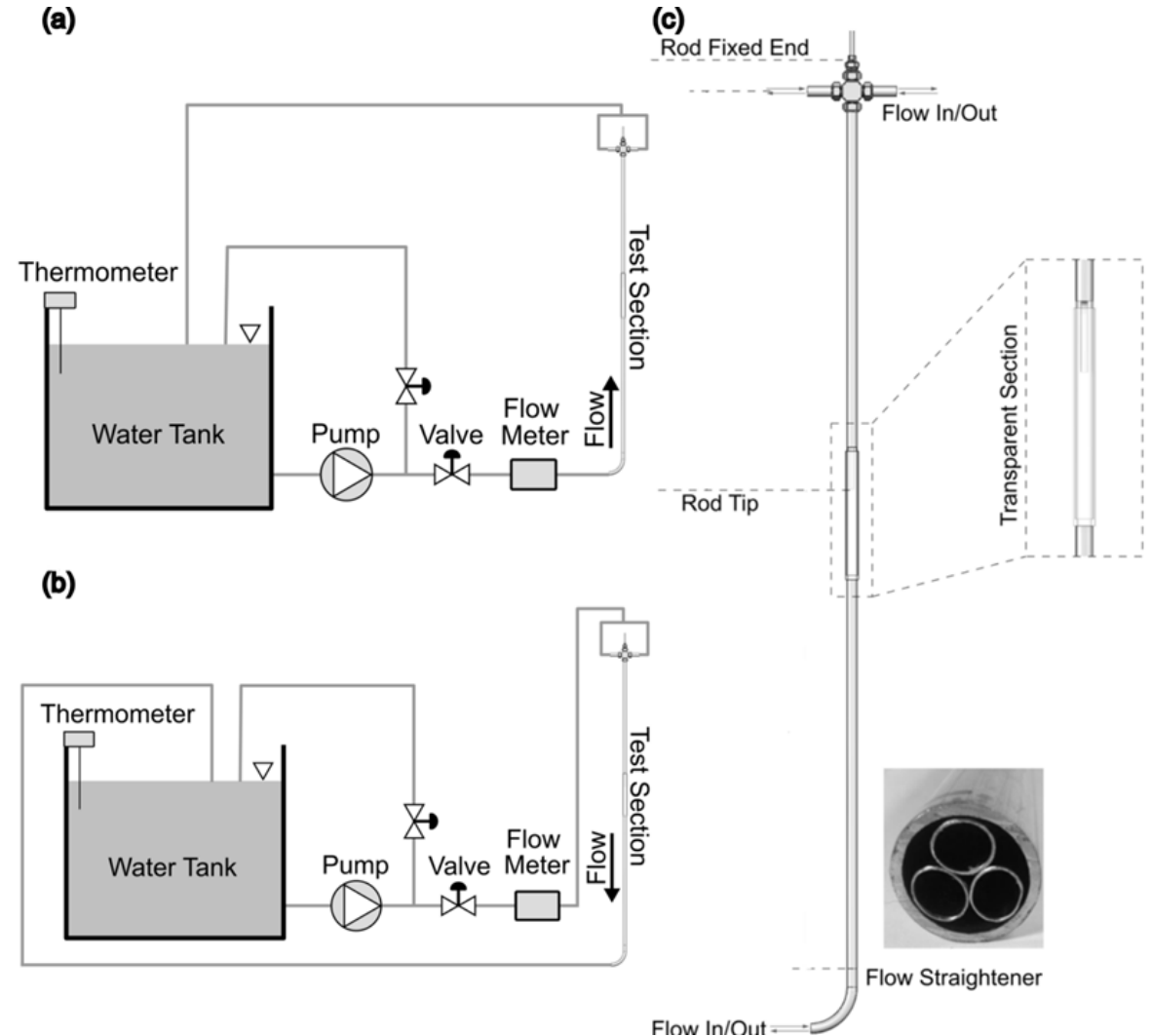
Experimental Rig

- A recirculating flow-loop system powered by a **10HP Leeson centrifugal pump**
- **Water was the test fluid**, with its density and viscosity characteristics being determined from its **temperature, recorded via a direct immersion digital thermometer** (ETI-Electronic Temperature Measurements Ltd; aquarium thermometer; temperature range: 248–343 K; accuracy within ± 1 K)
- Measurements were conducted in a temperature range of 17–23 °C
- The **mass flow rate was quantified using a robust turbine flow meter** (Cole-Parmer; flow rate range: 37.8–378 l/min; accuracy: 3.78 l/min) located downstream of the pump and upstream of the test section



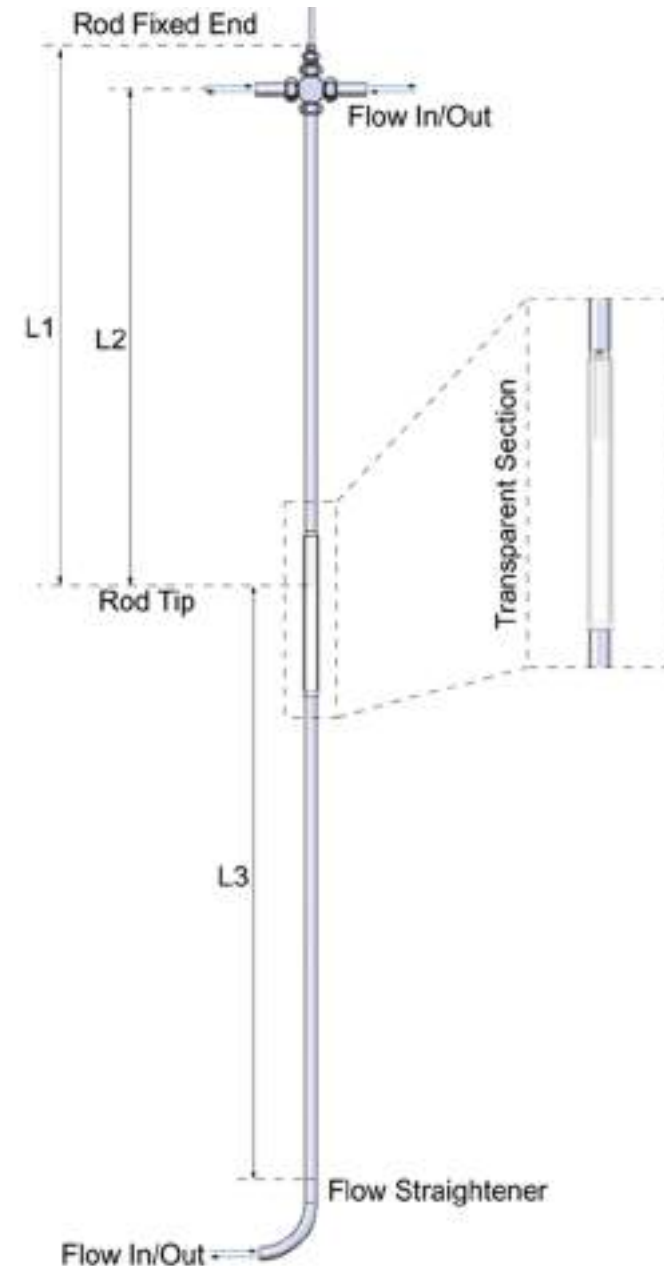
Experimental Rig

- A **flow straightener** was located at the **base** of the confining tube, to eliminate swirl and secondary flows, while maintaining minimal pressure drop
- The flow straightener was essentially made from three pipes with identical nominal dimensions to the vibrating rod piping and a length of 100 mm
- Sufficient distance from the top of the flow straightener to the free end of the vibrating rod was always ensured to **allow a flow recovery length no less than 50 times the pipe diameter**
- Conversely, in the clamped-free flow arrangement, **upstream disturbances exert minimal impact when the two incoming streams converge at the union cross**, so no flow straightener was incorporated in the feeding pipes



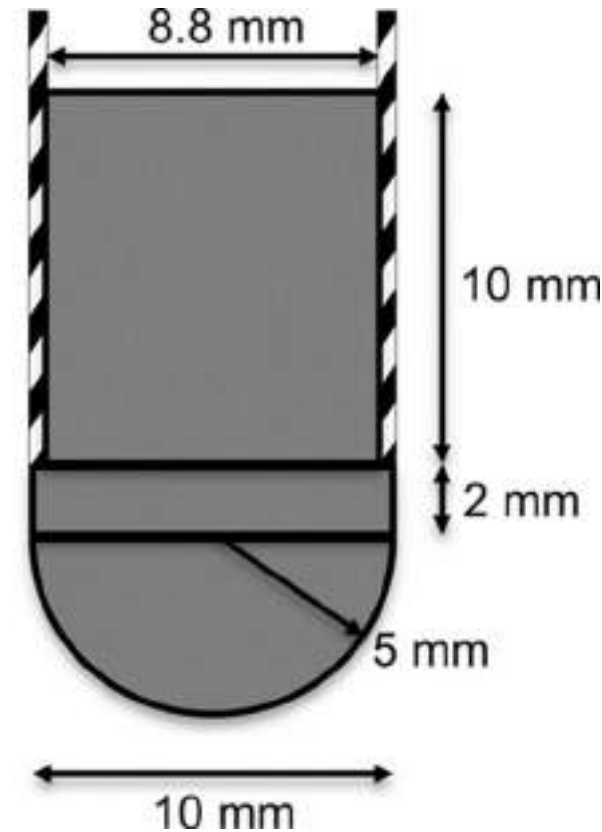
Test Section

- Flow direction: bottom-up (Free-Clamped) / up-bottom (Clamped-Free)
- Confining pipe has 21 ± 0.1 mm inner diameter
- Test piece is connected to two symmetrically routed pipes at the top via a union cross (Swagelok SS-25M0-4), which also holds the vibrating rod in place
- Free length of the rod (L1) is 1045 mm
- Distance between the centre of the symmetric pipes and the free end of the rod (L2) is 944 ± 1 mm
- Distance between the free-end of the rod tip the top section of the flow straightener (L3) is 1100 mm
- Flow straightener (length = 100 mm) is located at the bottom



Rod with Semi-spherical End-piece

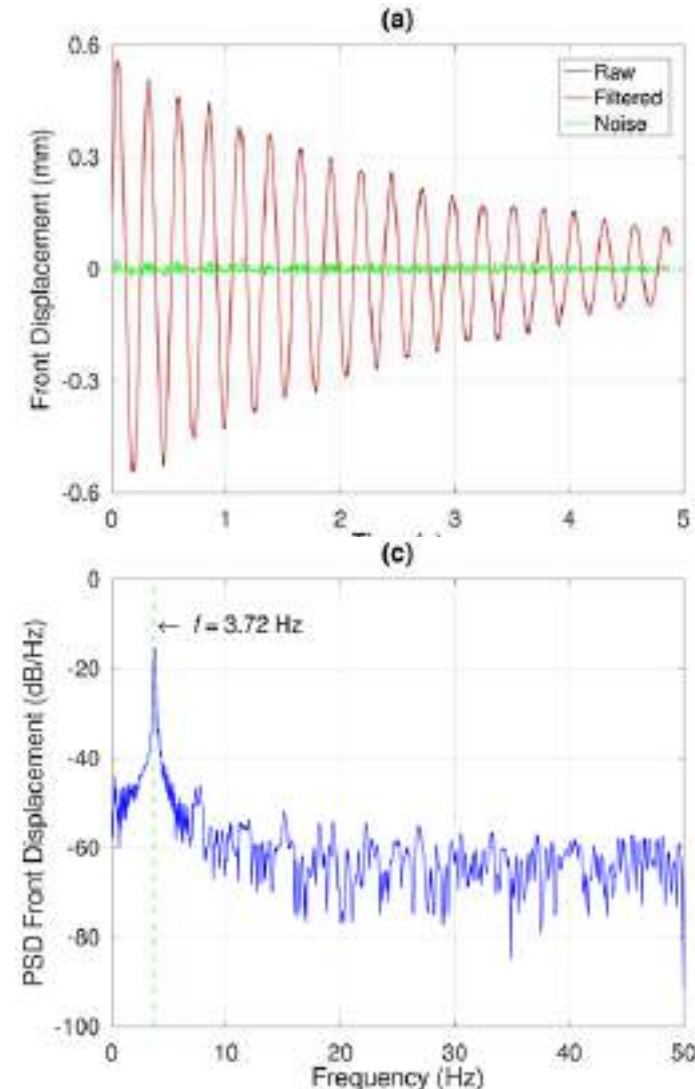
- Rod has 10.01 ± 0.06 mm diameter
- Hydraulic diameter is 11 mm
- Rod has a streamlined end-piece of semi-spherical shape
- Mass density of the rod is 588 g.m^{-1}
- Inside space of the vibrating rod included lead shots with a loose texture (density: 11.34 g/cm^3 ; diameters: 0.3-1.6 mm) to simulate the loading of fuel pellets in nuclear fuel assemblies
- Lead shot possessed a loose texture, **influencing the linear mass density of the rod**, while the other structural qualities remained unchanged
- Balancing test indicated a 0.5% displacement of the rod's center of mass towards the free end, which can be considered negligible



Free-vibration Tests



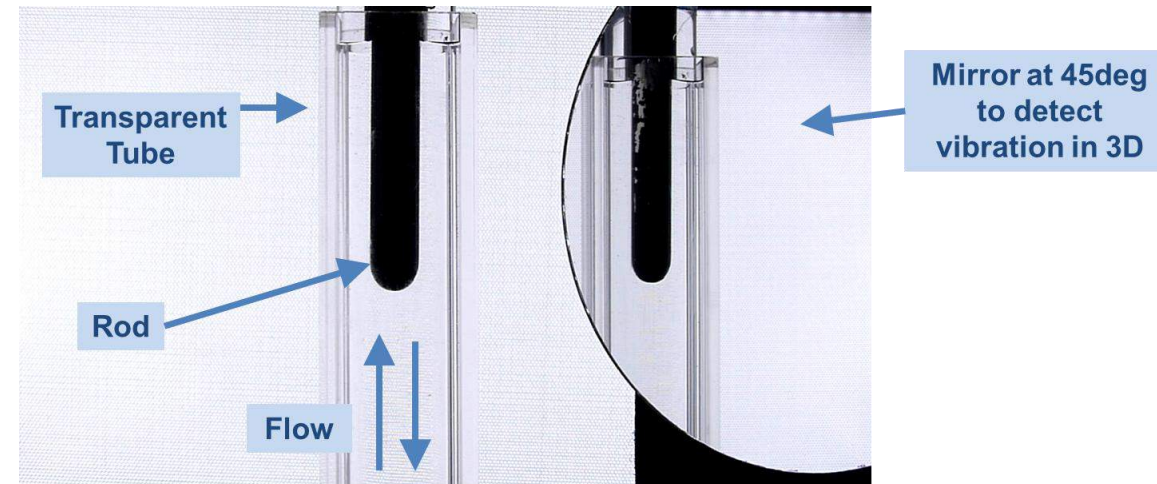
- Filling the test section with static water, while physically moving the rod from its equilibrium position, and thereafter recording and analyzing its resulting free vibrations
- Rod's **free vibration frequency was 3.72 Hz**, which was found to align with the theoretical value derived from traditional Euler-Bernoulli beam theory
- Additionally, experimentally measured **damping ratio was found to be 0.015 ± 0.001** , which again closely aligned (within 10%) with the theoretical model presented by Rogers et al. for a vibrating rod in a fluid-filled annulus



Measurements

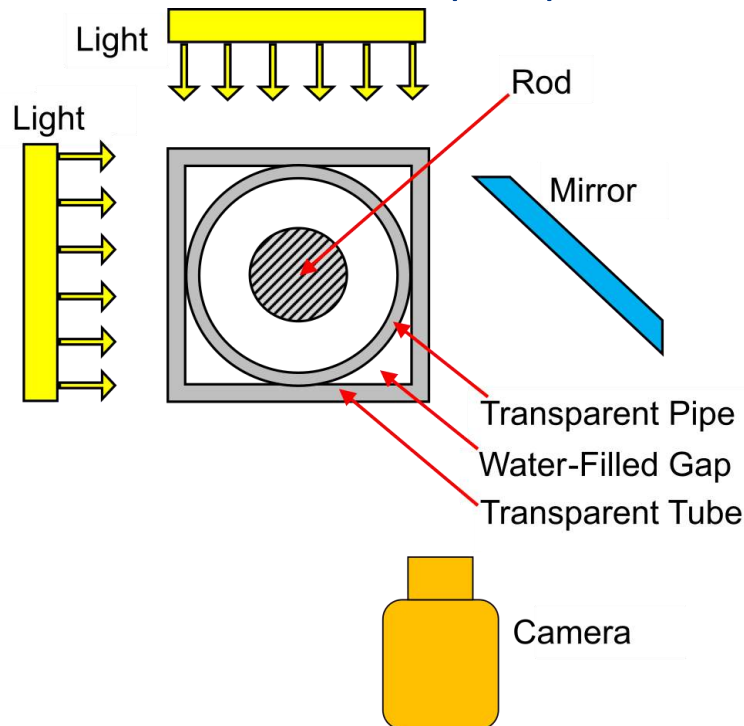
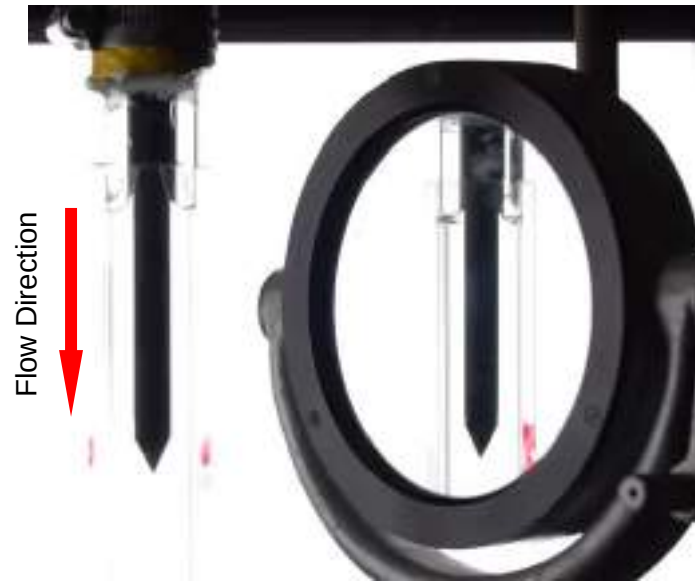


- A transparent rectangular box with flat surfaces was included around the transparent part of the tube and the space between it and the outer surface of the transparent segment of the confining tube was filled with water to minimize optical distortion by **index matching** (virtually eliminating the reflection losses associated with the solid-air interface)
- This effectively ensured the elimination of the induced lensing effect while visualizing the flow via the cylindrical pipe



Measurements-Rod Vibration

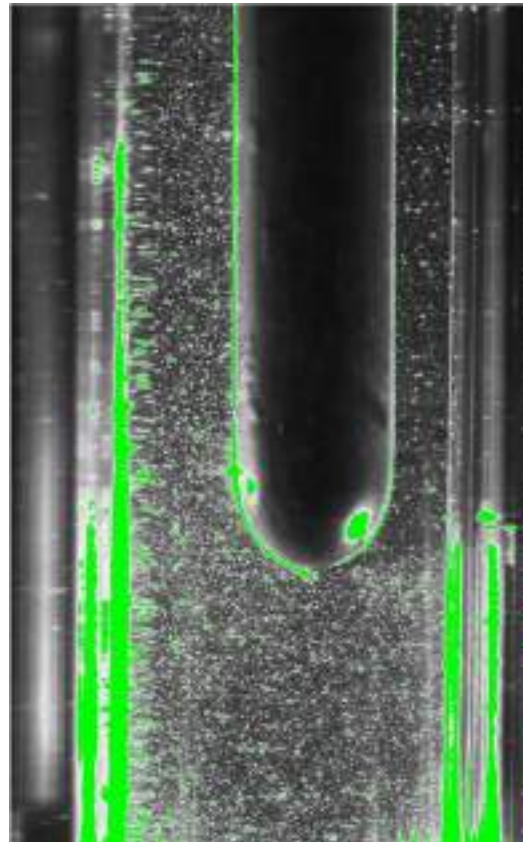
- Flow-induced motion recorded using a fast-imaging camera (Panasonic Lumix DMC-FZ200; image resolution: 1280 x 720 pixels; recording frequency: 100 fps; recording duration: 60 seconds)
- Improved optical tracking of the vibrating rod with 3D resolution capability (along with dedicated software for video postprocessing and time series analysis)



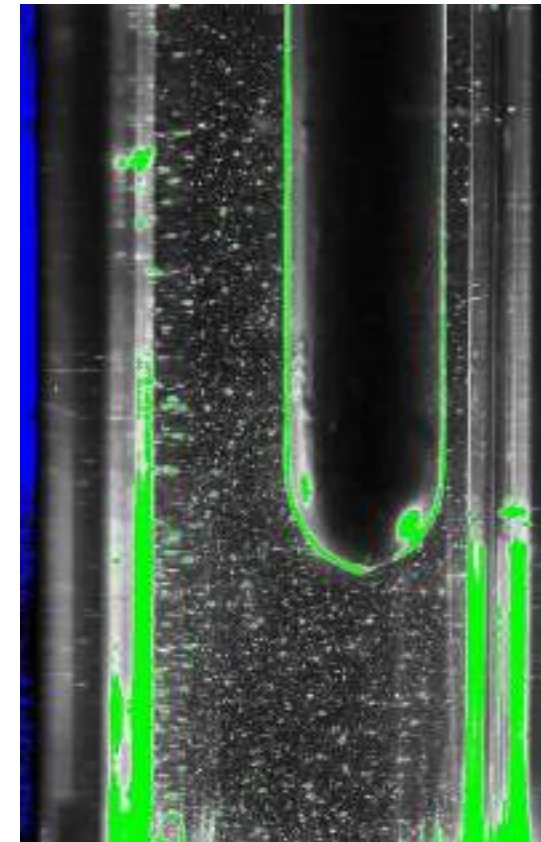
Example Results-Motion



Up-flow



Down-flow



Resulting Videos



Time Histories of
Rod Displacement



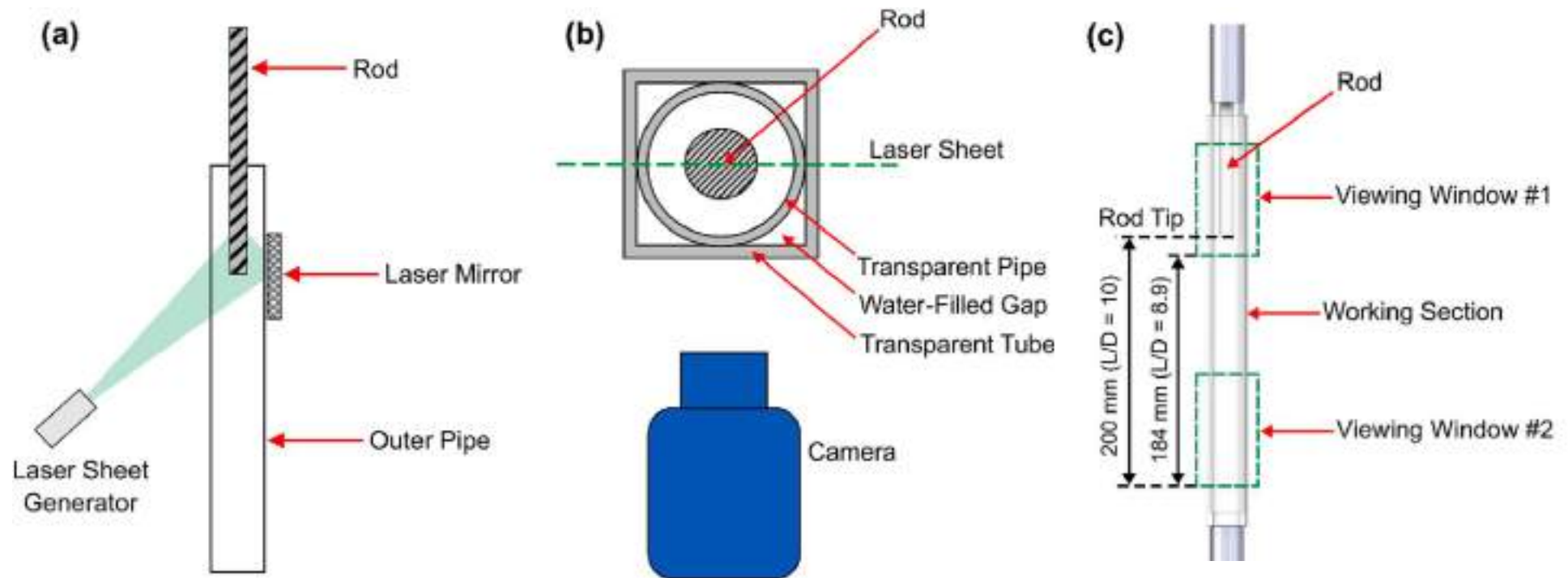
Displacement RMS



Frequency Spectrum

Measurements-Flow Field

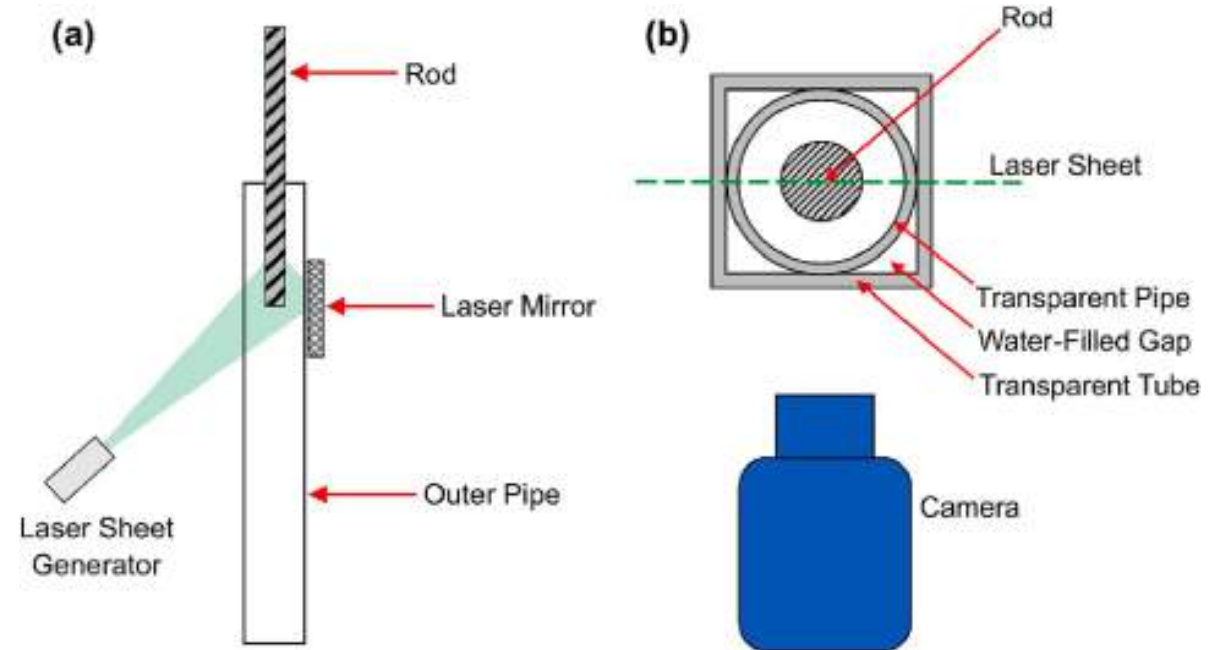
- Fluid flow velocity fields will be measured using a PIV setup for specific measurement cases of interest
- Flow fields were captured near the rod vibrating end as well as at $10D$ below the tip location



Measurements-Flow Field



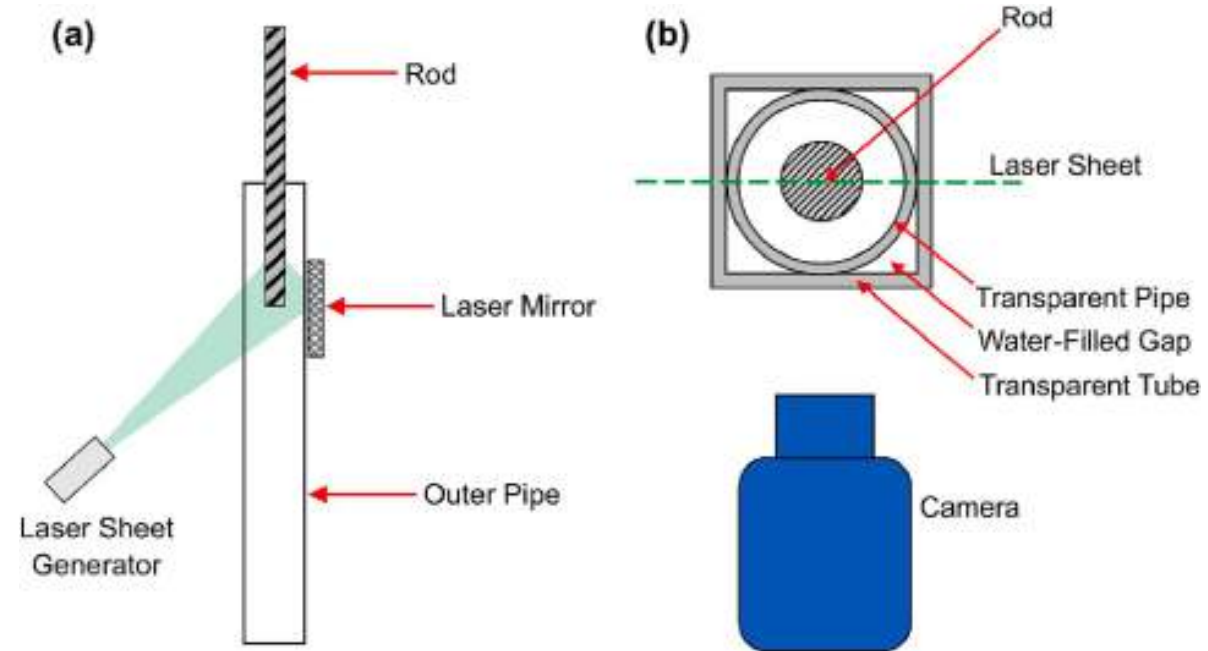
- A **Dantec Dynamics PIV system** was used, including a double cavity Nd:YLF laser that delivered a maximum output energy of 15 mJ at a wavelength of 527 nm
- The laser was connected to an articulated light arm with a beam-width-controlling optic and a diverging lens, which produced a **2D vertical laser sheet** in the central plane of the flow field
- The laser sheet **thickness was approximately 1 mm**
- The sheets were **positioned at an angle** to the flow direction to illuminate the flow field on both sides of the vibrating rod, so **preventing the rod's shadow from covering portions of the flow visualization area**



Measurements-Flow Field



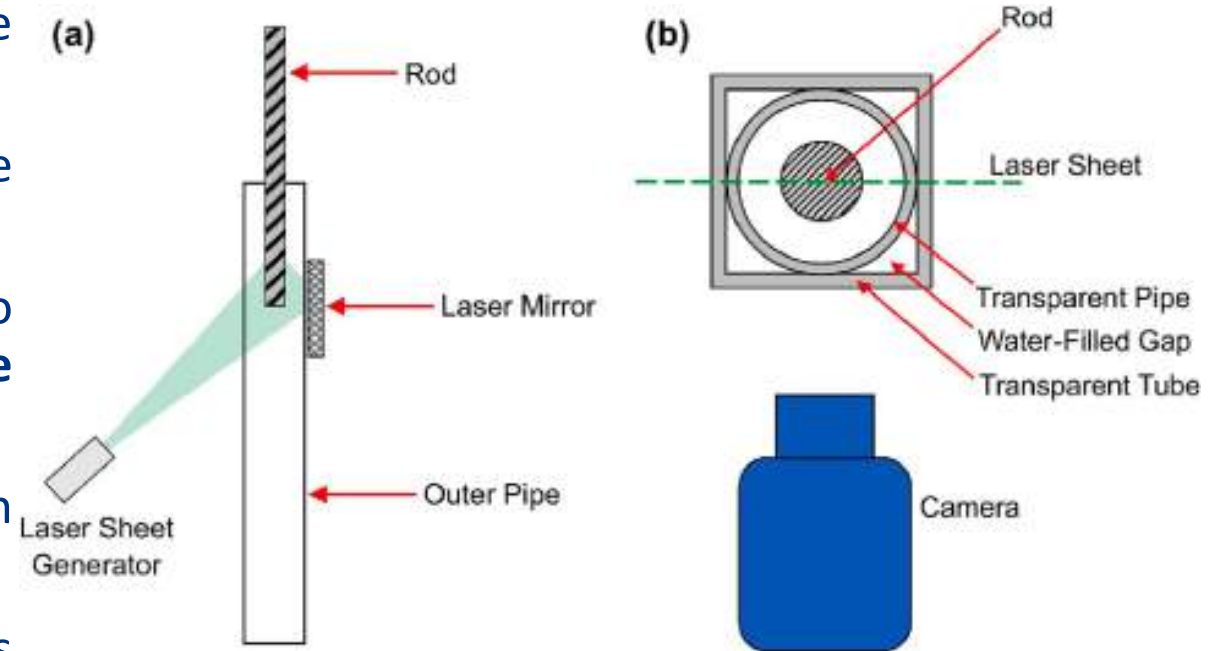
- Images were captured on double frames using a high-speed camera (**Phantom V310**) **operating at 200 frames per second** and 1280 x 800 full resolution, employing a Nikon AF Micro 60 f/2.8D lens
- **Hollow glass spherical particles** with a nominal diameter of 10 μm were used - The chosen particle size was sufficiently tiny to facilitate flow tracking while being big enough to provide intense light scattering for imaging, hence enabling precise recording within a compact interrogation window of the PIV pictures
- Following the addition of the particles, the rig was operated at its maximum flow rate for several minutes to **facilitate circulation and uniform distribution of the particles**



Measurements-Flow Field

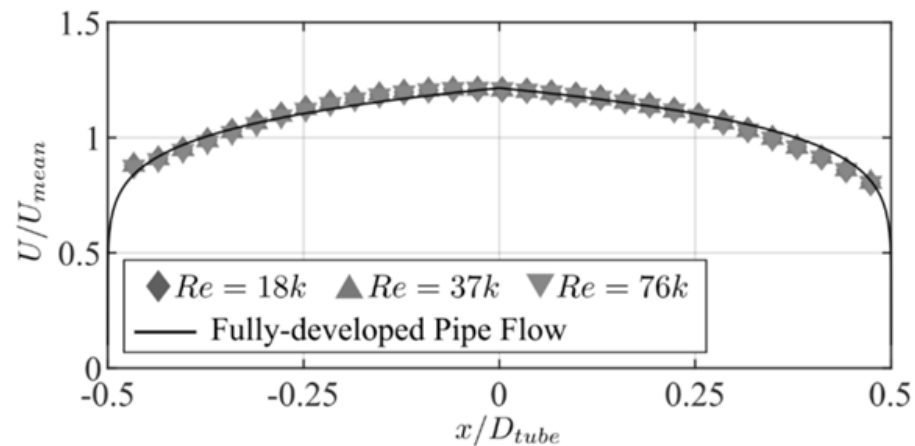
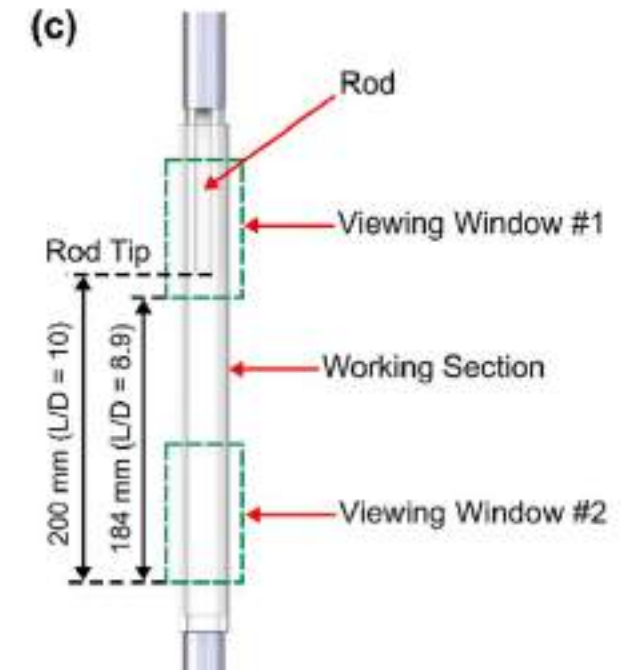


- The **laser pulses were synchronized with the camera** to ensure the pictures were acquired at the appropriate time
- A **Q-switch laser delay** was employed to regulate the emission of light pulses following each laser activation
- **Interval between pulses (50-200 μ s)** was established to **maximize mean displacement inside the target plane while minimizing particle loss from that plane**
- Picture pairs were analyzed using the PIVLab package in MATLAB to deduce the velocity fields
- Each picture pair was segmented into discrete regions termed **interrogation windows**
- A **correlation peak at the most probable displacement position** of the particles within each interrogation window was identified



Measurements-Flow Field

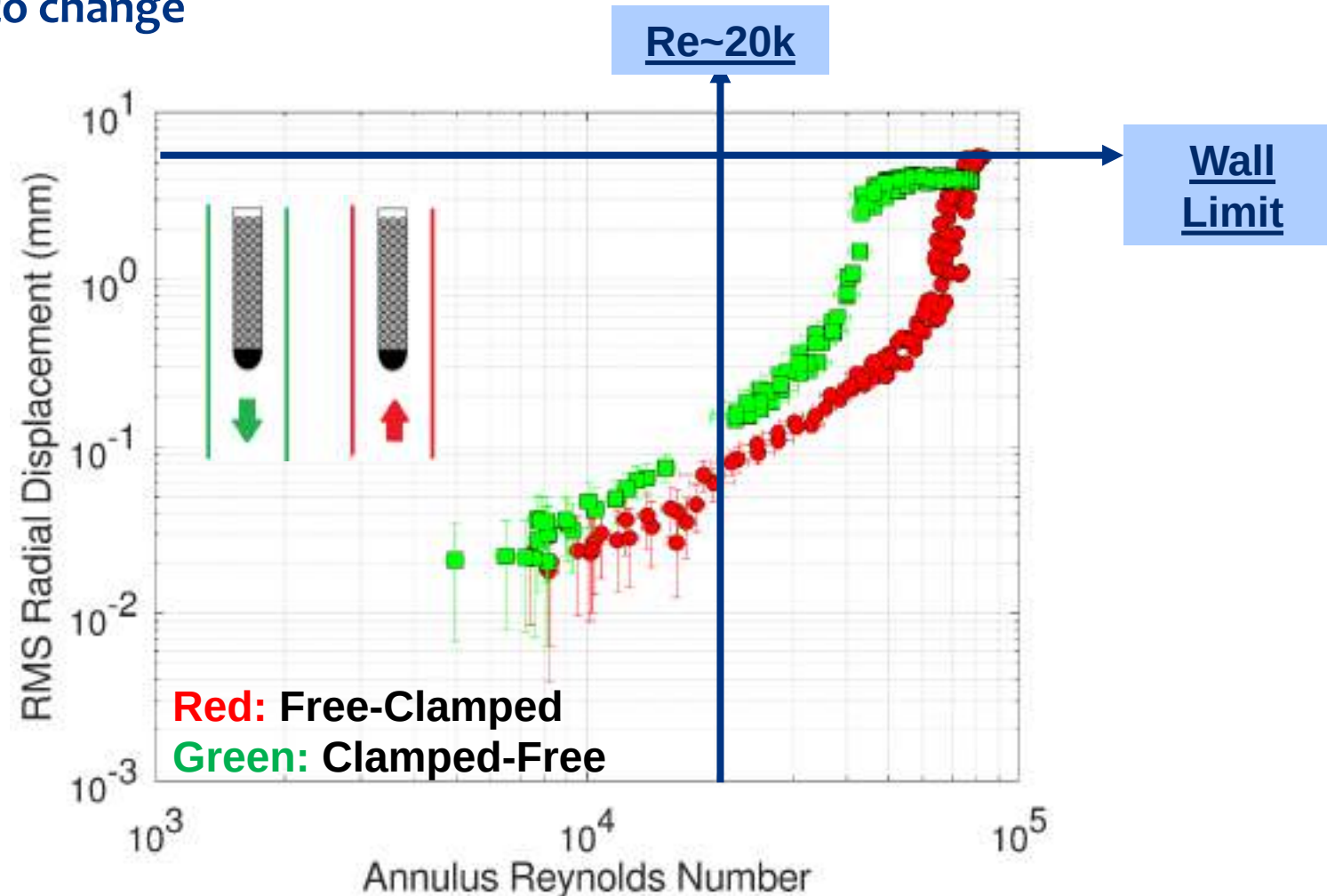
- In the free-clamped flow setup, further PIV measurements were conducted at approximately 10 diameters upstream of the rod tip (viewing window #2) to confirm the establishment of **fully developed pipe flow conditions upstream of the rod**



Example Results-Motion

Results are preliminary and subject to change

- **Free-Clamped:** Flutter at low Reynolds numbers & Buckling at high enough Reynolds numbers
- **Clamped-Free:** Flutter only and no buckling
- As displacement reaches around 0.2-0.3 mm, the slope of the RMS displacement vs Reynolds number curve increases sharply till:
 - **Free-Clamped:** the rod buckles touching the confining tube and remaining still in a deflected configuration
 - **Clamped-Free:** The rod continues to vibrate and does not buckle



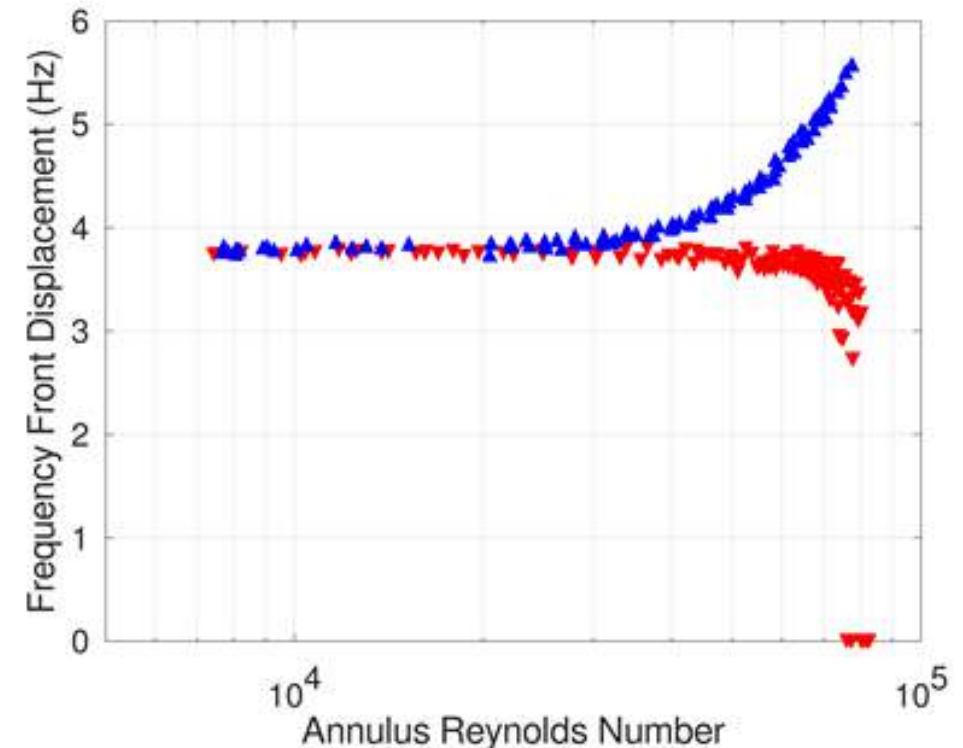
Example Results-Motion



Free-Clamped: Vibration frequency diminishes until it attains zero when the rod buckles, resulting in the absence of periodicity in the rod's vibration (i.e., no predominant peaks in the power spectrum)

Clamped-Free: Vibration frequency persists in rising with the further increases in the Reynolds number

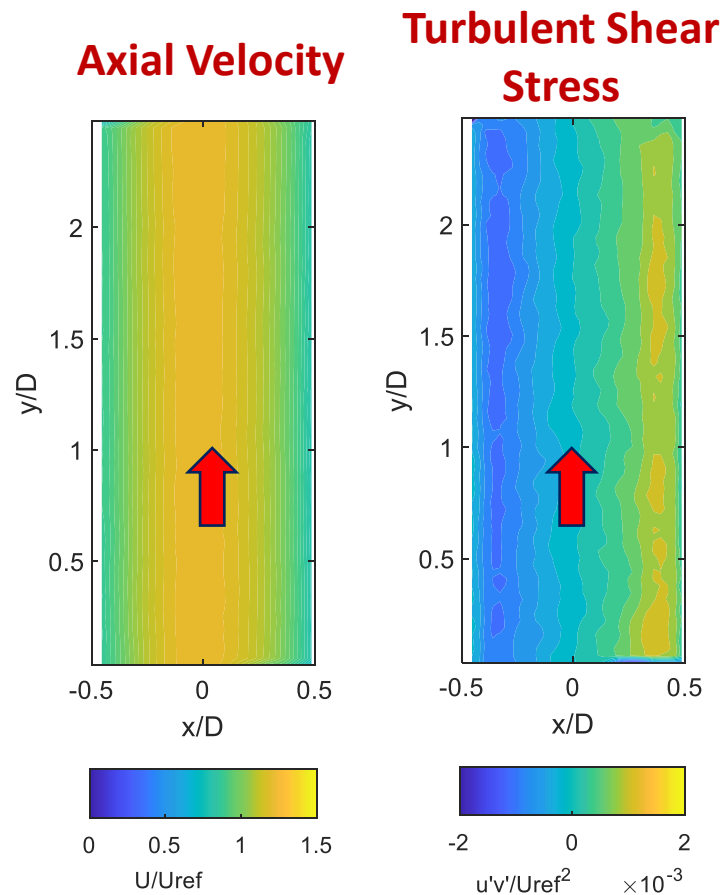
Vibration frequencies correspond to the dominant peak in the power spectra of the rod displacement time series



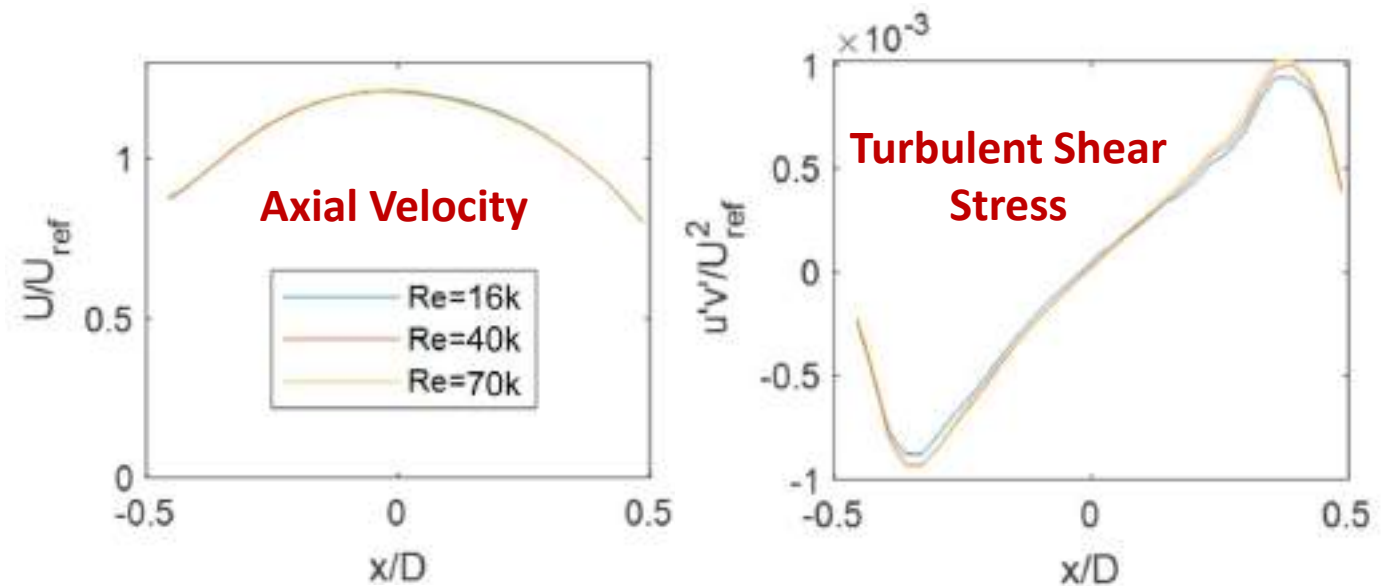
PIV Results



Time-averaged
Contours upstream of
the rod, at $Re=70k$

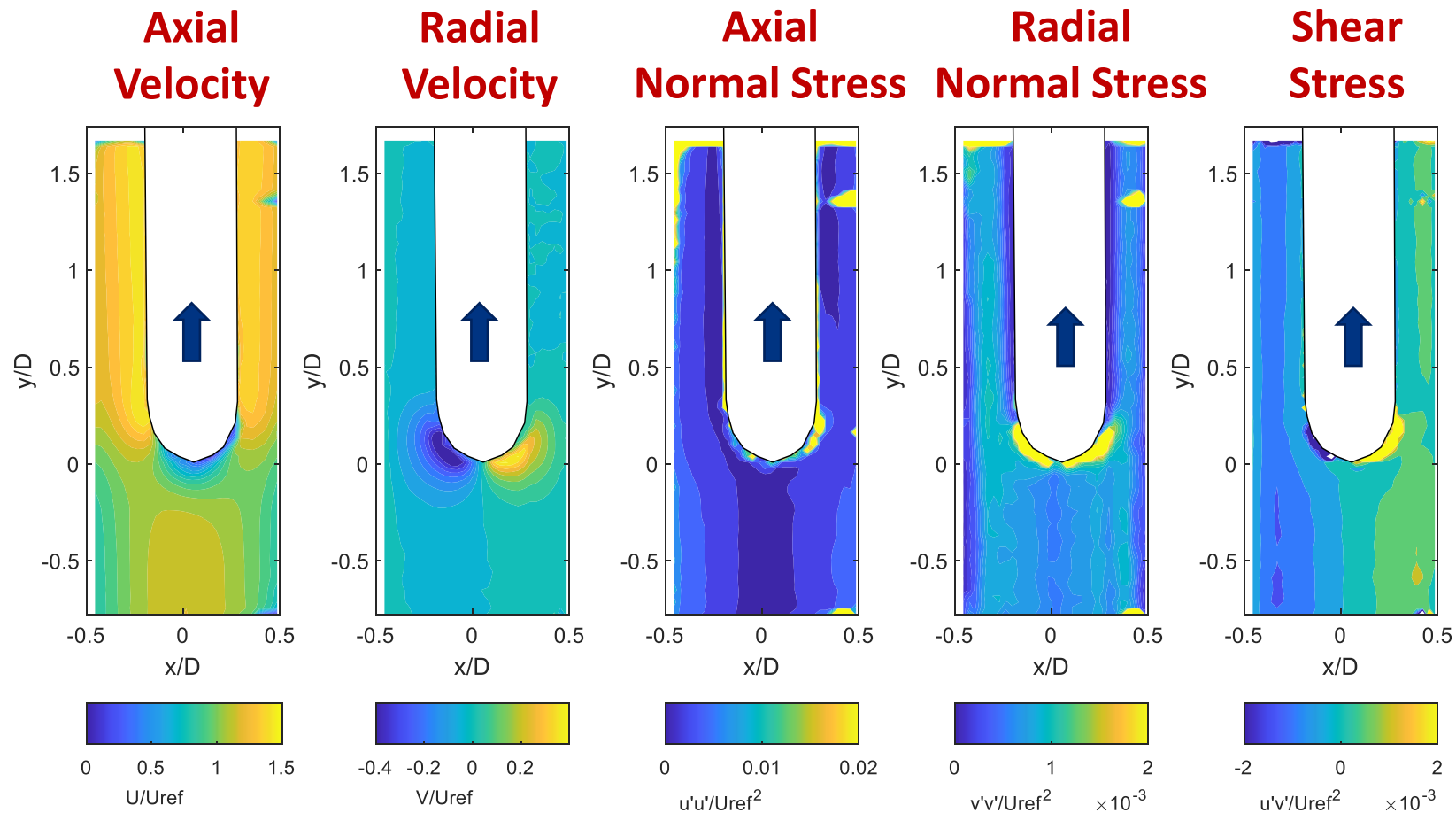


Time-averaged profiles at $10D$
upstream of the rod



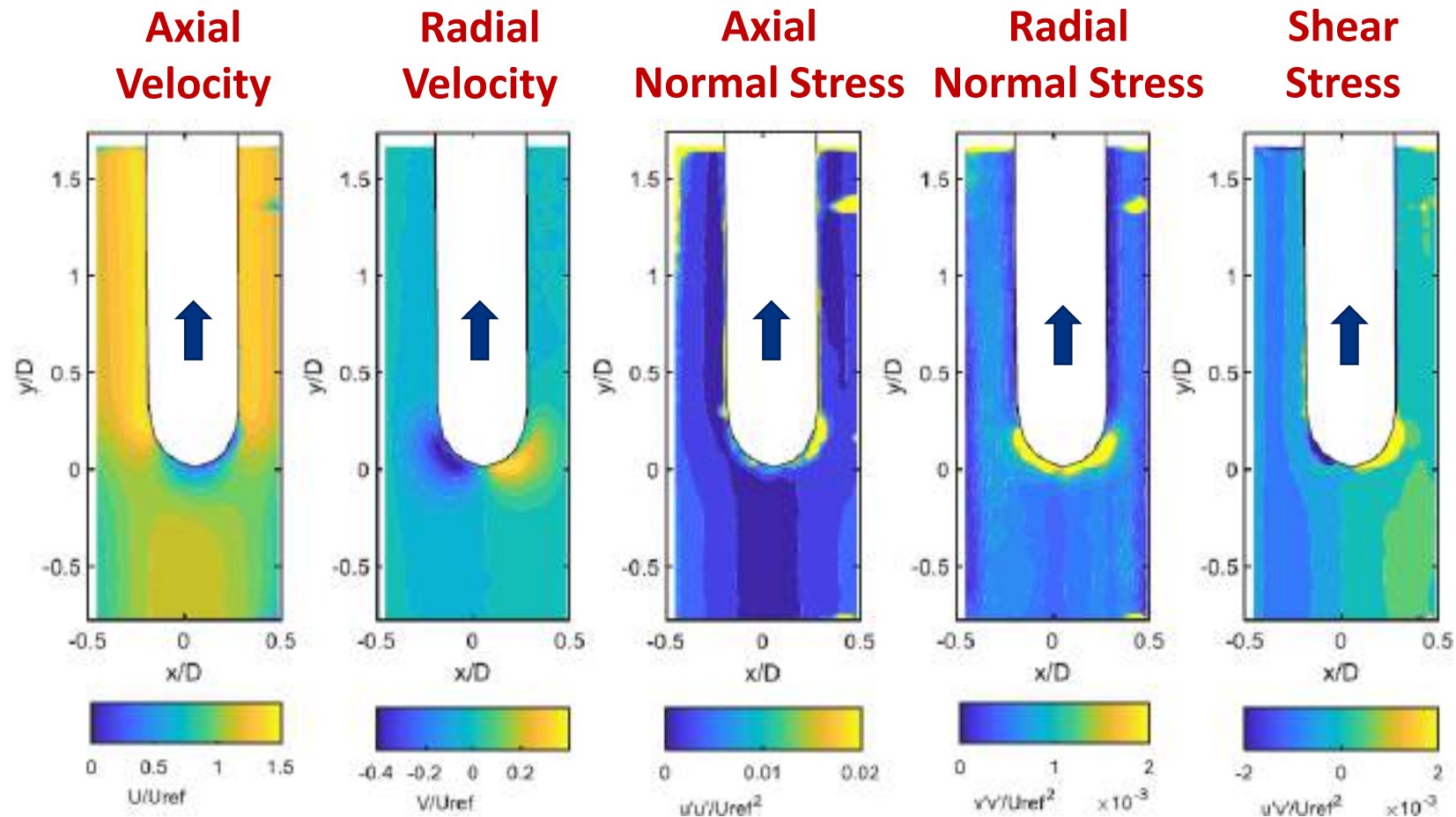
PIV Results

- **Free-Clamped:** Mean and turbulent flow fields at $Re=20k$



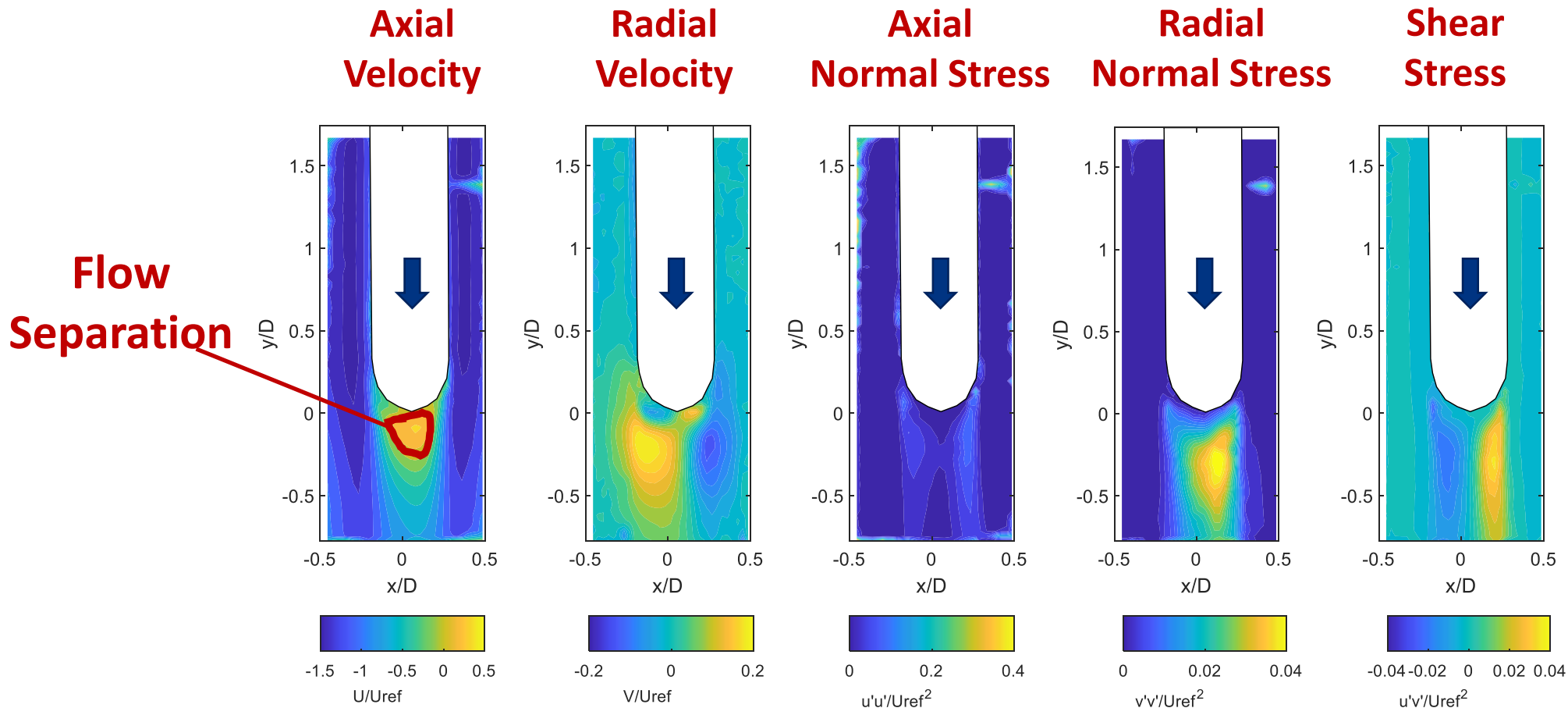
PIV Results

- **Free-Clamped:** Mean and turbulent flow fields at $Re=40k$



Results-PIV

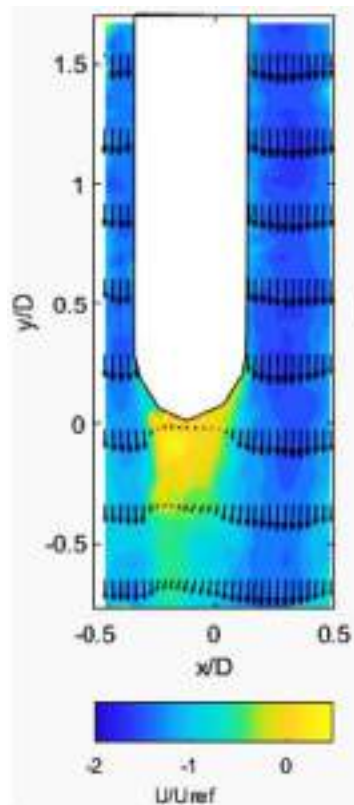
- Clamped-Free: Mean and turbulent flow fields at $Re=20k$



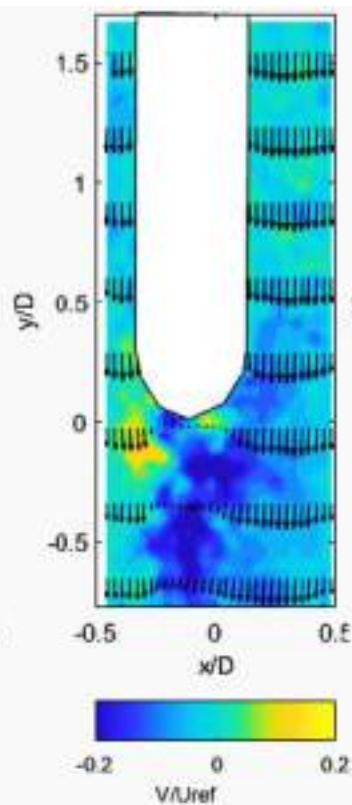
Results-PIV

- **Clamped Free:** instantaneous flow fields at $Re=56k$

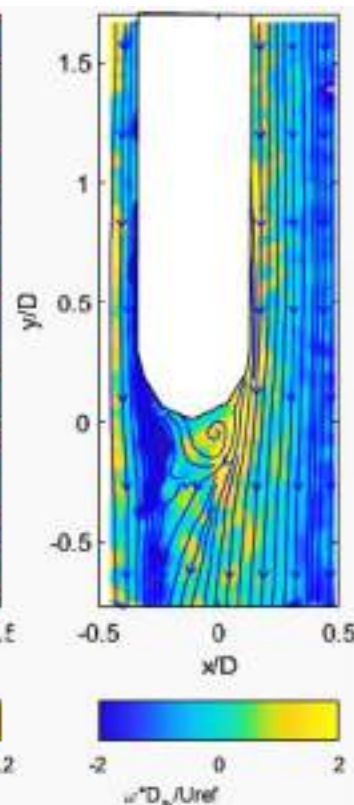
**Axial
Velocity**



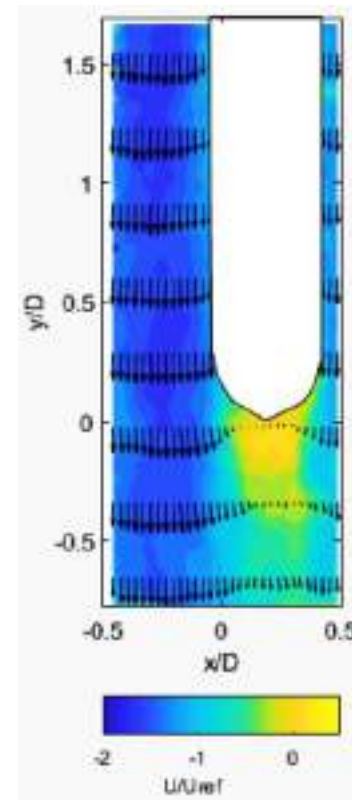
**Radial
Velocity**



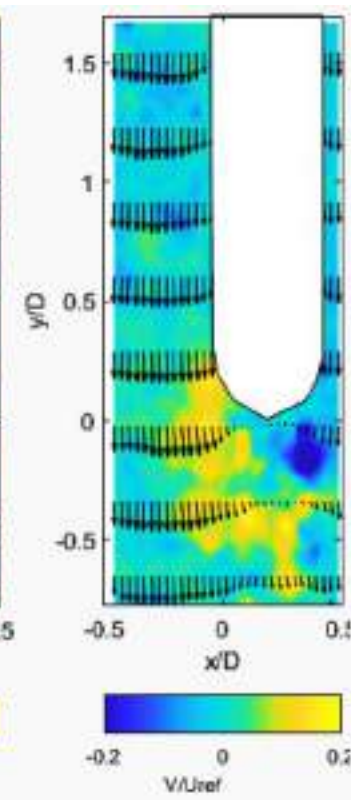
Vorticity



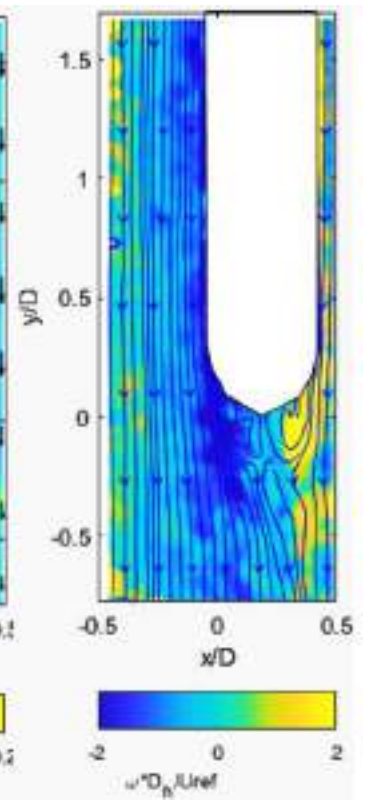
**Axial
Velocity**



**Radial
Velocity**



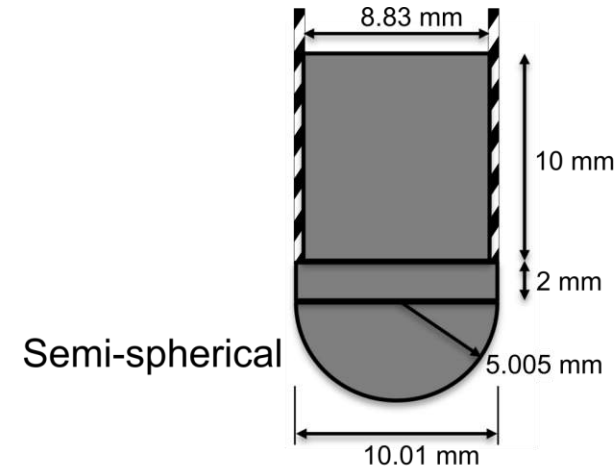
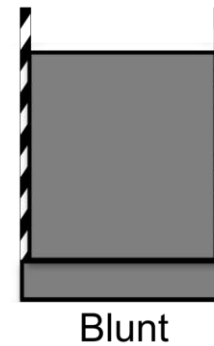
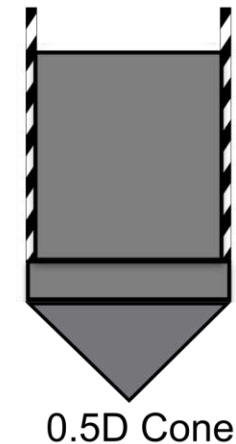
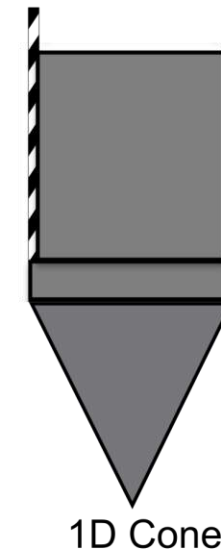
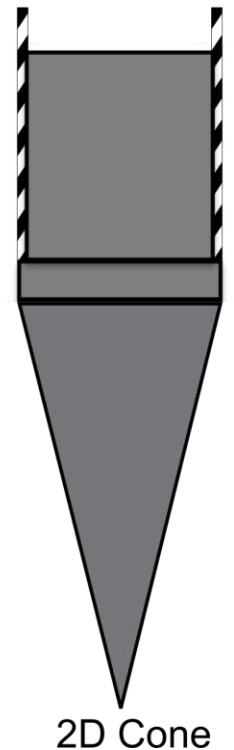
Vorticity



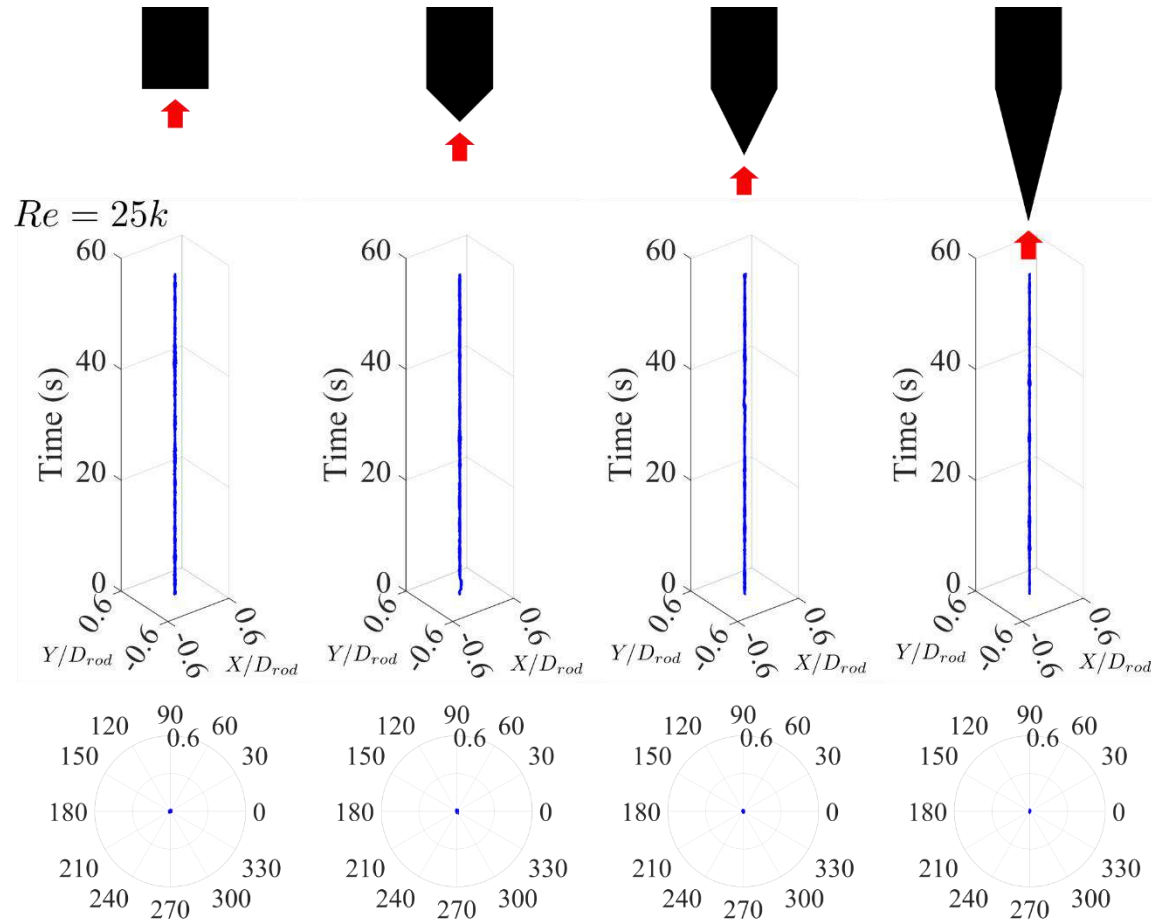
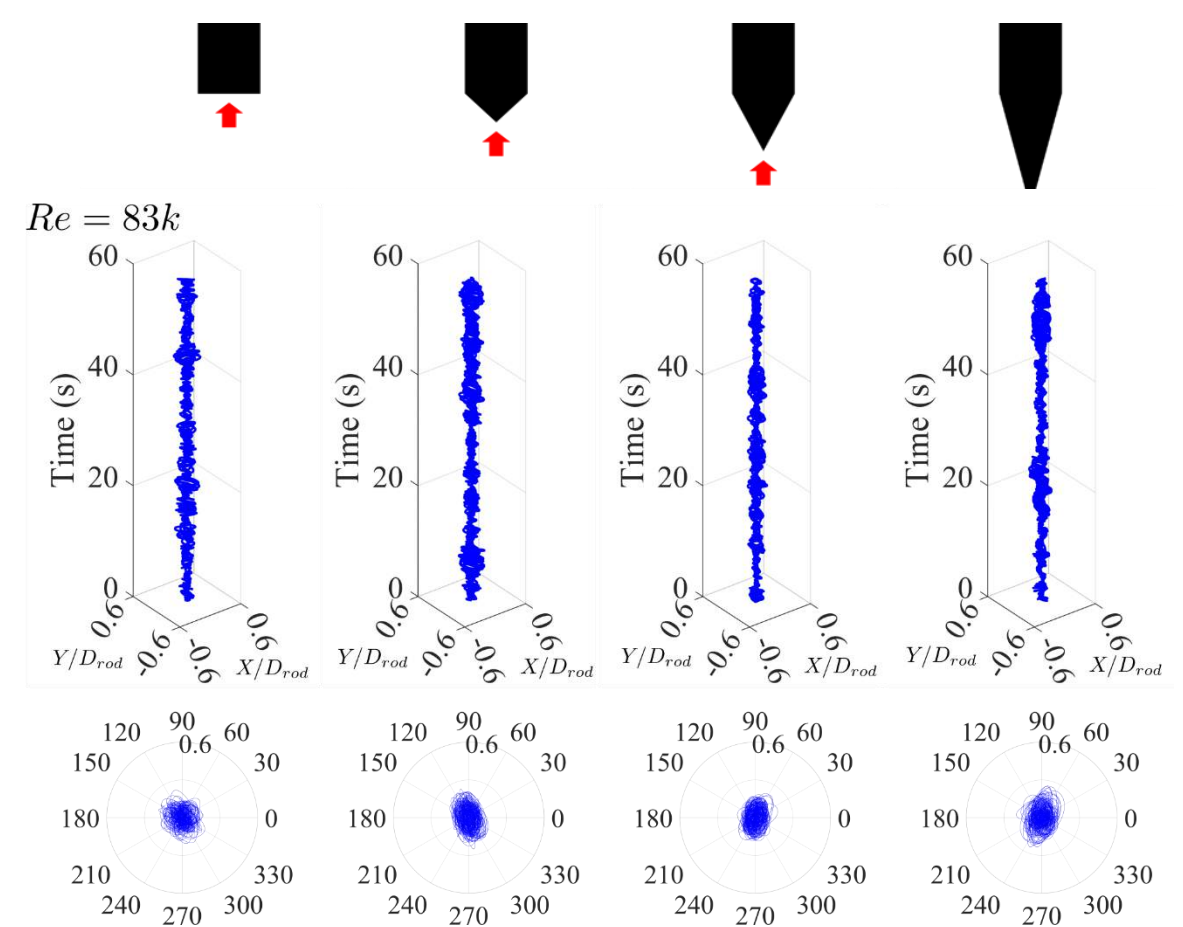
Other Rod End-Pieces



- Rod has diameter (D) of 10.01 ± 0.06 mm
- Hydraulic diameter is 11 mm
- Mass density of the rod is 588 g.m^{-1}
- Baseline Rod has a streamlined end-piece of semi-spherical shape
- Additional end shape including Blunt end and Cone end with height of 0.5D, 1D and 2D

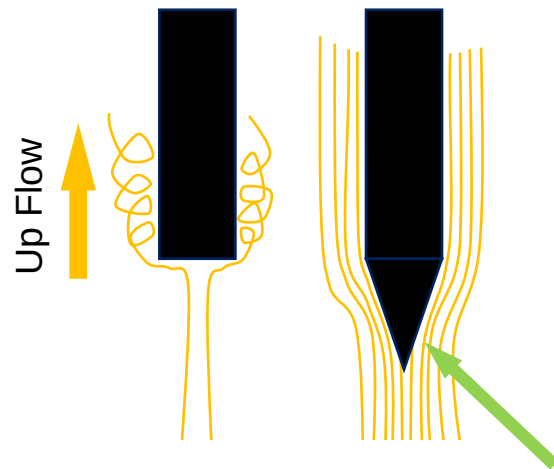


Example Results-End Shape Effect

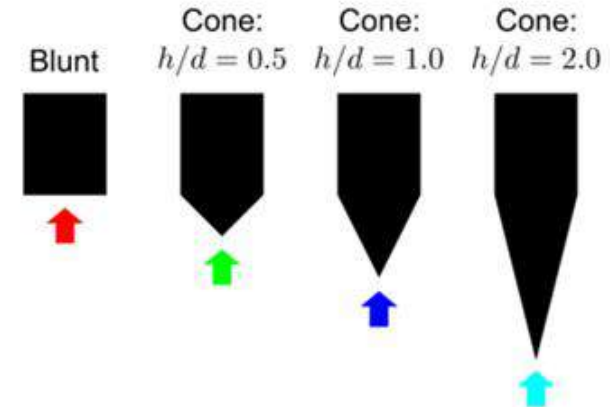
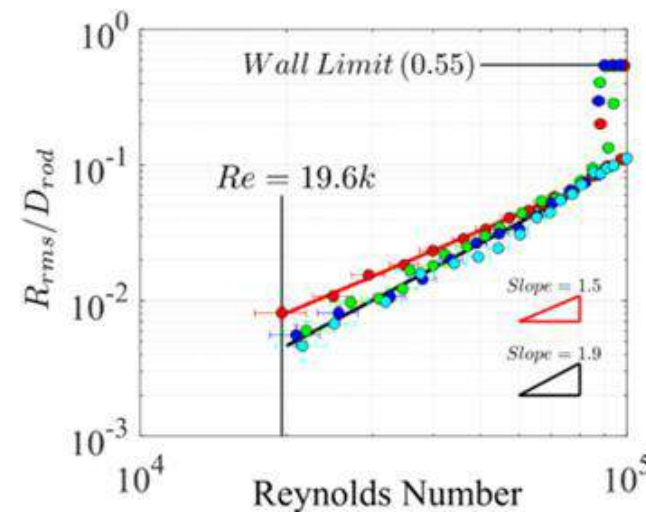
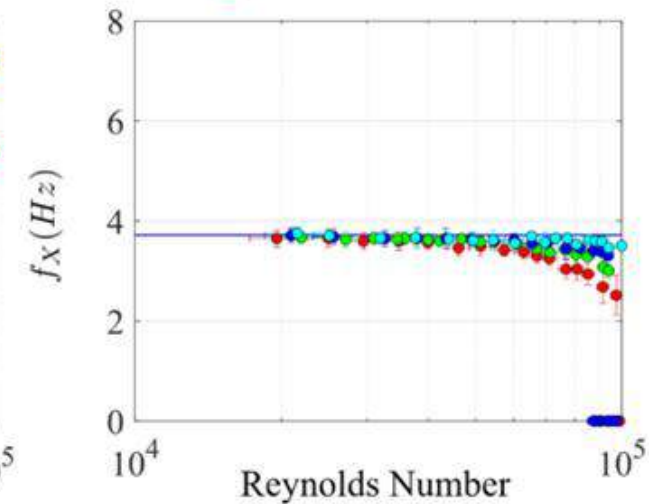


Example Results-End Shape Effect

- Cone end shapes resulted in smaller rod vibrational (RMS) displacement
- Buckling was generally found at Re above 70k~90k
- no consistent trend of end shape effect on rod buckling was observed



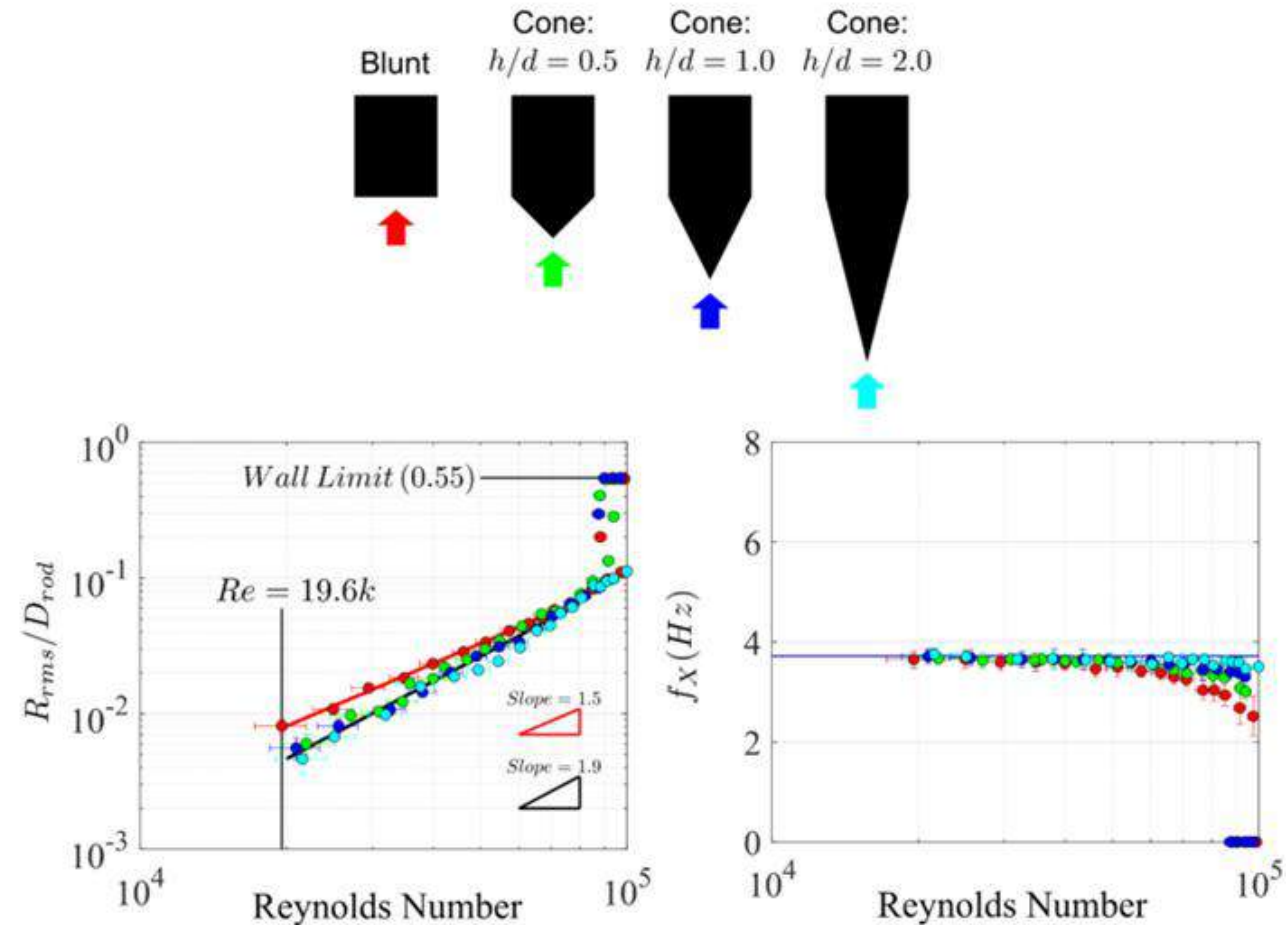
Cone end shapes results in more streamlined flow leading to reduced RMS amplitude



Example Results-End Shape Effect



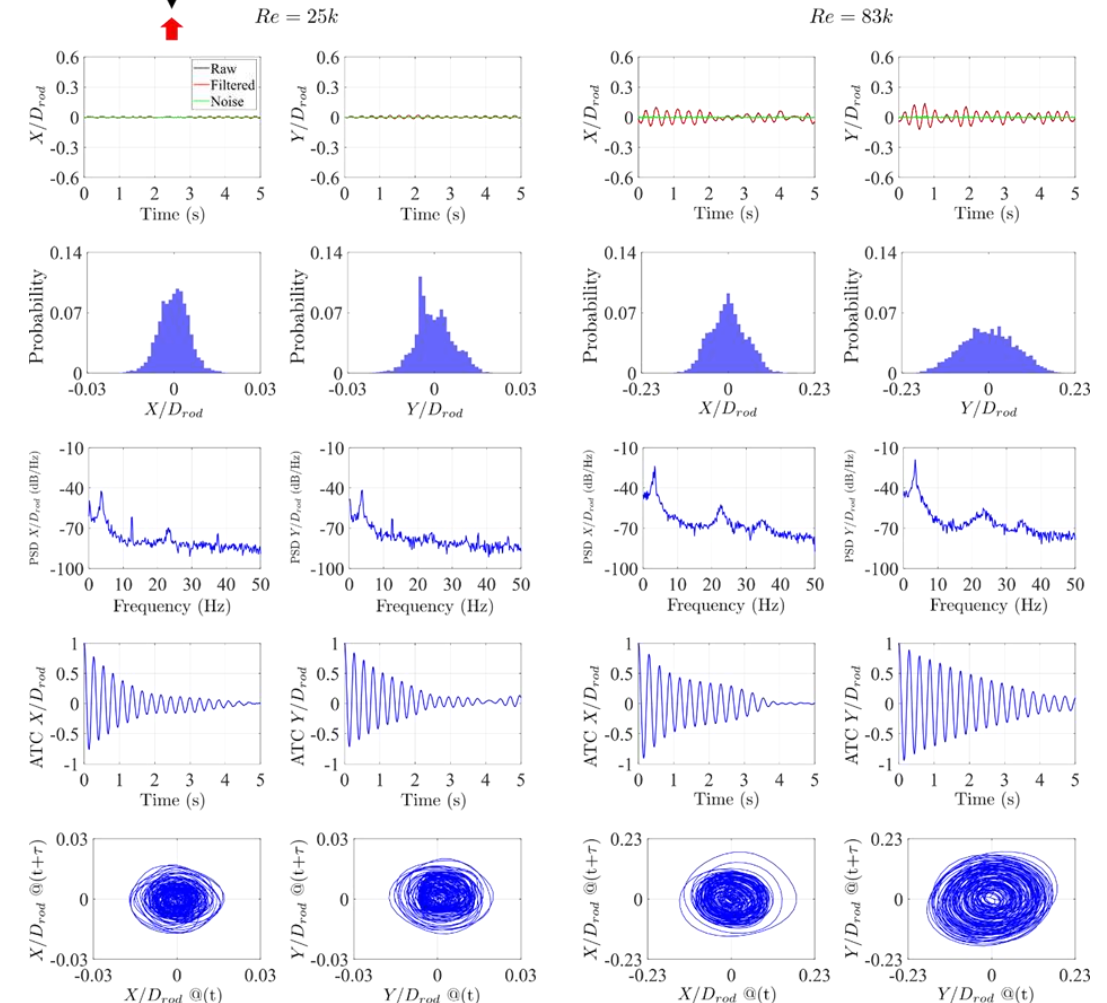
- Regardless of the tip arrangement, the RMS displacements of the rods demonstrate a **near linear increase** with the increase of the Reynolds number before buckling, but the vibration **frequencies decrease** as the Reynolds number increases
- At high Reynolds numbers, buckling was evident for the blunt and cone tips with h/d ratios of 0.5 and 1, but **not in the case of the cone tip with a h/d ratio of 2**
- It is important to note that the buckling Reynolds number is sensitive to the initial conditions of the testing apparatus
- Prior to buckling, the blunt tip rod often exhibits more RMS displacement than the cone tip designs at low Reynolds numbers. However, as the Reynolds number increases, the difference in RMS displacement between the blunt tip and the cone tips diminishes



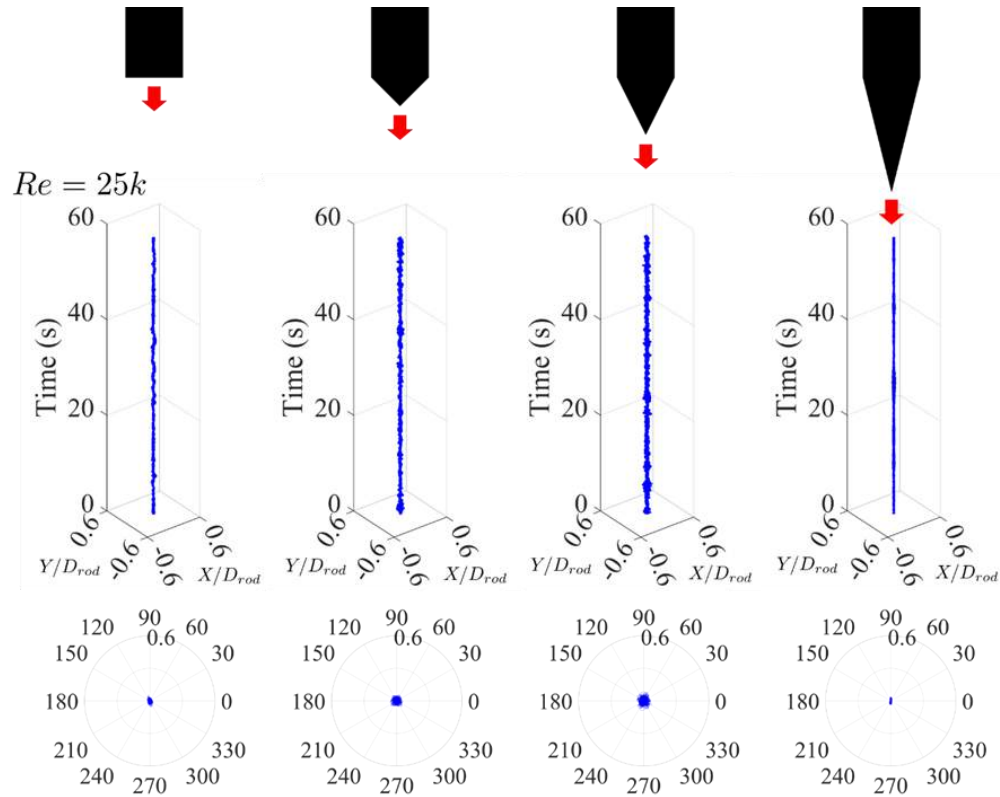
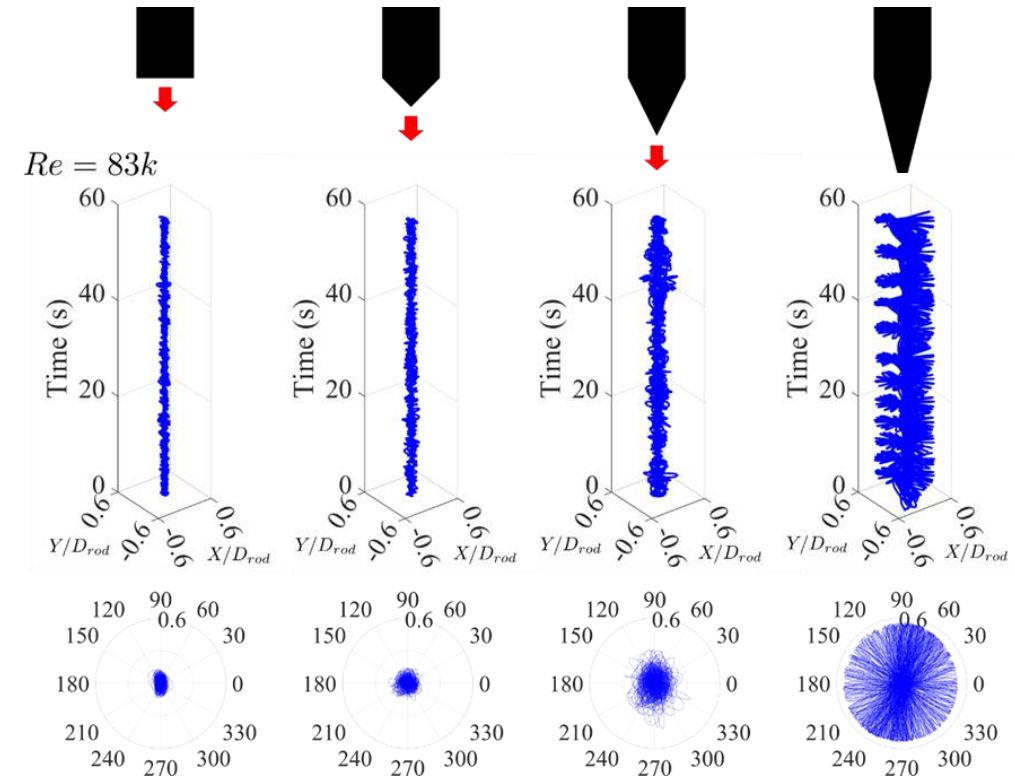
Example Results-End Shape Effect



- Histograms that depict a "bell-shaped" distribution, the autocorrelation functions exhibited periodicity and quick decay, and the reconstructed attractors had a "blob-like" shape
- All these trends indicated that the rods display **fuzzy period-1 motion**
- As the **Reynolds number increases**, the **periodic aspect of rod vibration becomes more pronounced**, resulting in a **wider displacement histogram** and an **extended decline** in the autocorrelation function compared to lower Reynolds number conditions

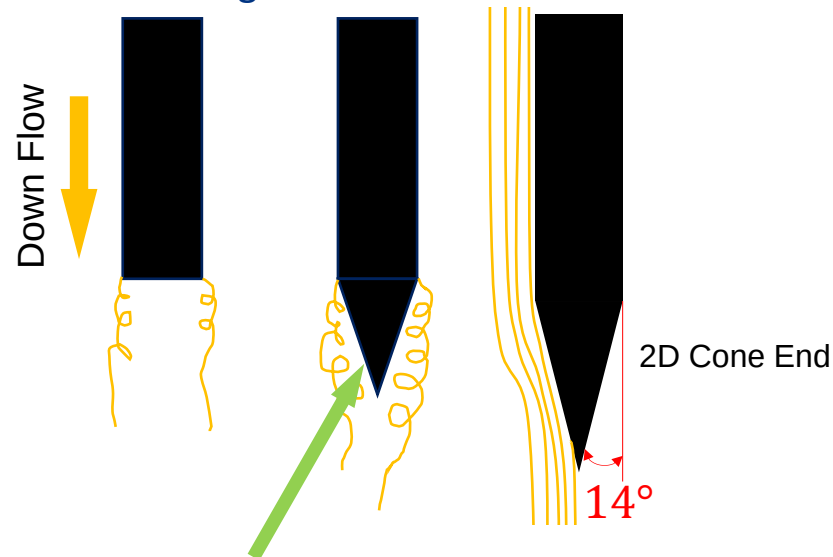


Example Results-End Shape Effect

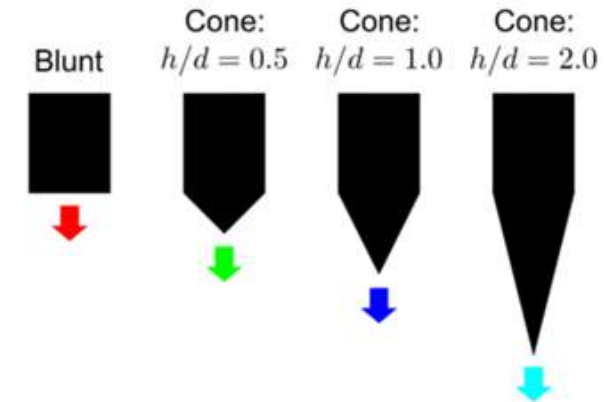
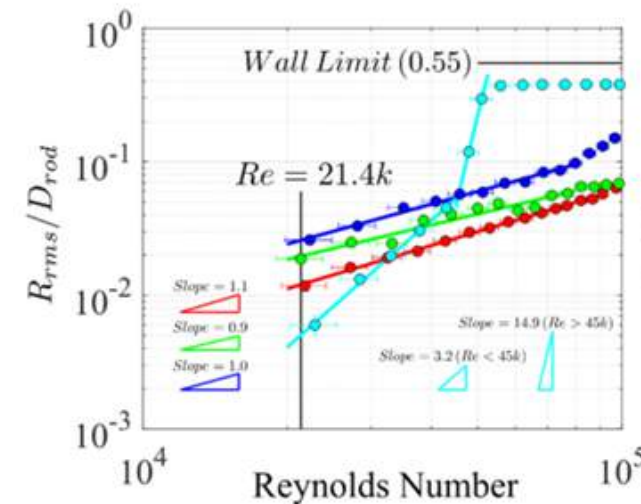
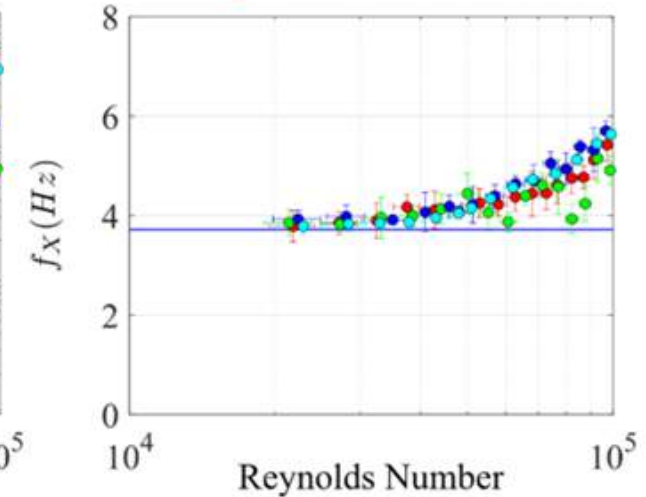


Example Results-End Shape Effect

- Increasing 'sharpness' of rod free end leads to increased rod vibrational (RMS) displacement
- 2D Cone end show small vibration amplitude at low Re (between 20k and ~40k): flow remain attached due to the small transition angle



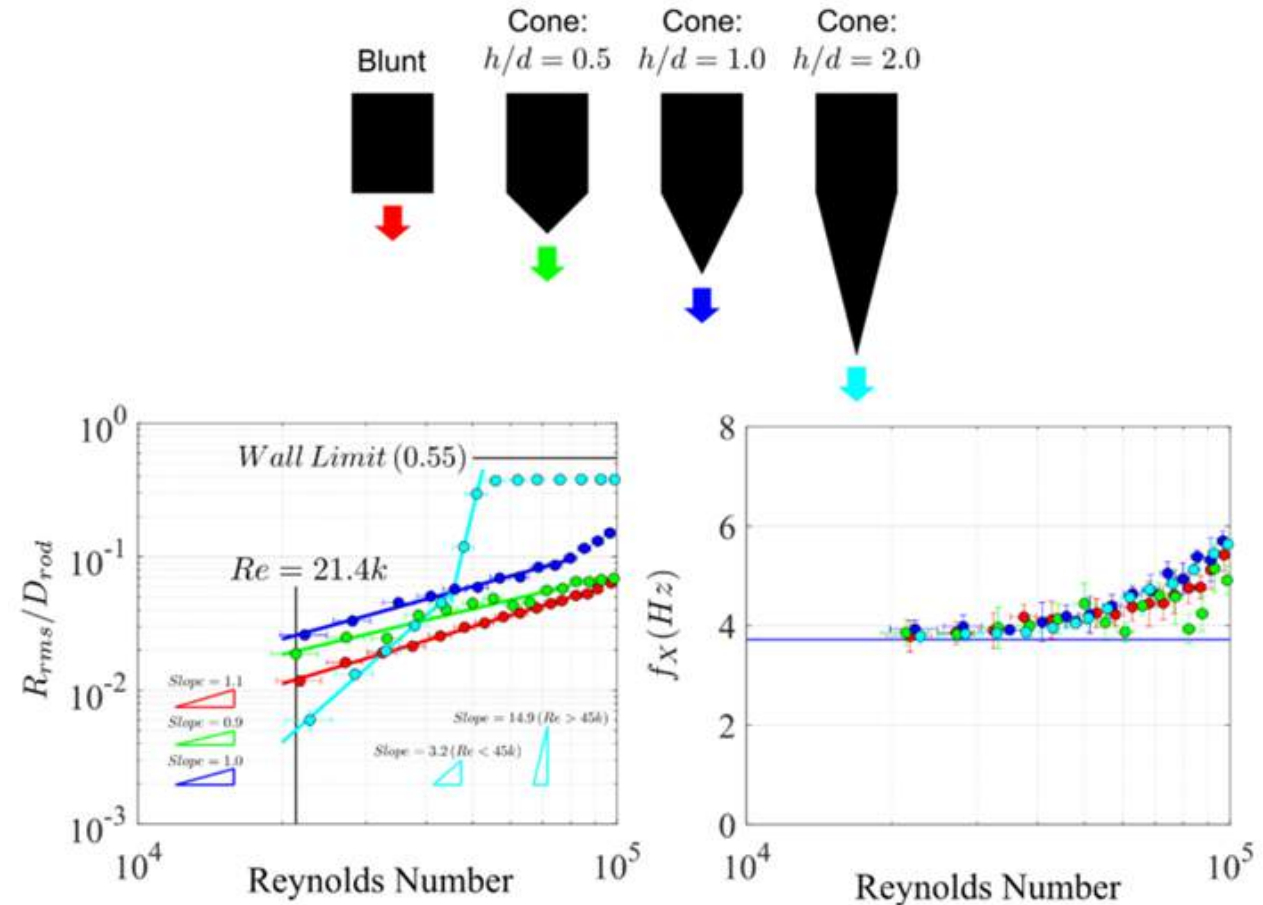
Sharper end shape has larger acting surface for flow disturbances leading to increased RMS amplitude



Example Results-End Shape Effect



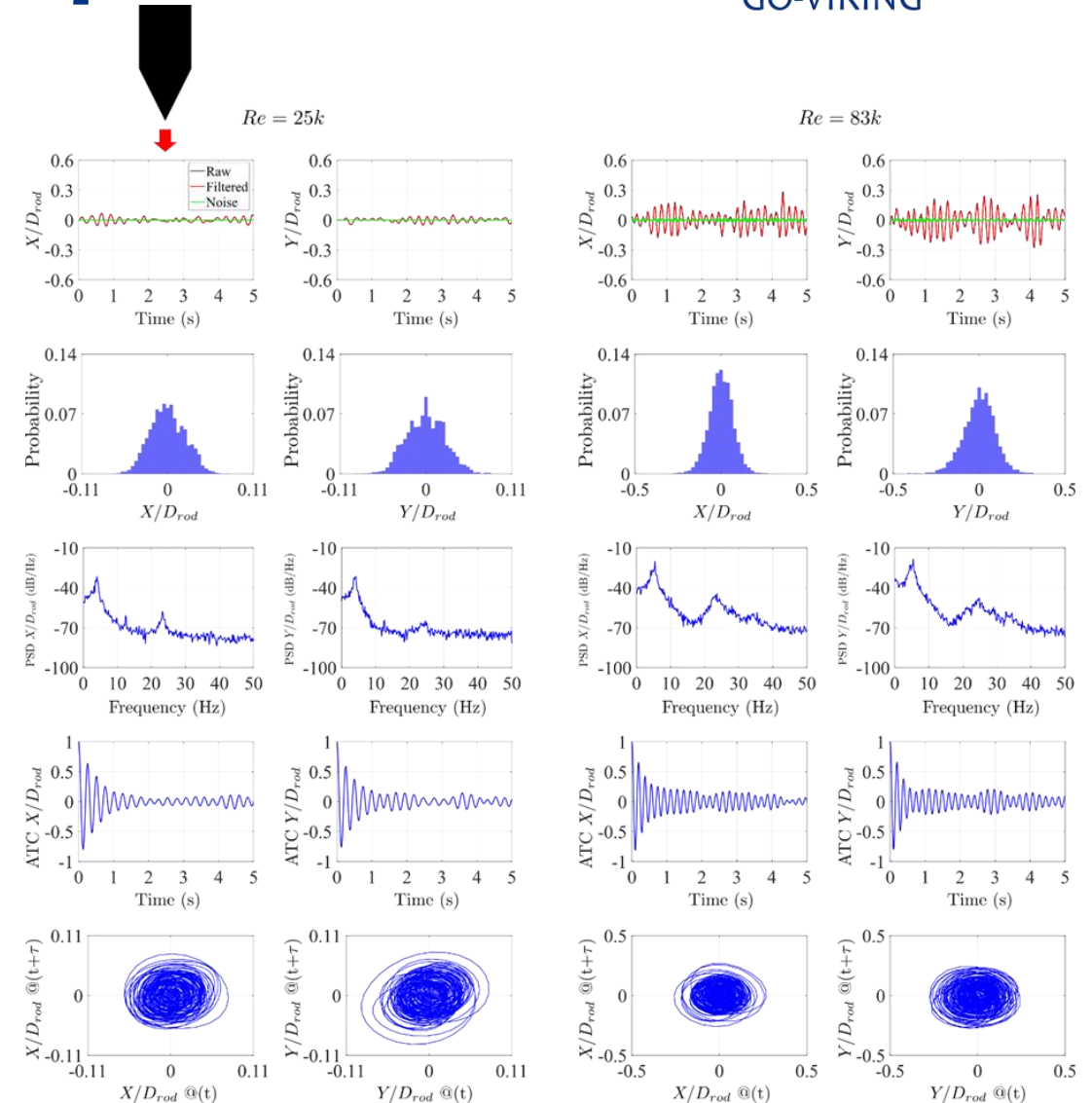
- The vibrational behavior of the rod with the cone ratio of $h/d = 2$ is significantly different: At the lower end of the Reynolds number range (about 20k), the RMS displacement levels are lower than those of the rods with the other three tips; nevertheless, the increase with Reynolds number is much steeper
- A change in rod vibration dynamics transpires at a Reynolds number of around 45k, marked by a substantial increase in the steepness of the rod displacement gradient
- As a result of this significant rise in RMS displacement, at Reynolds numbers over 50k, the rod makes contact with the tube's confining wall, leading to a plateau in displacement
- Notably, for all other tip geometries, the rod does not contact the confining tube, even at the highest Reynolds number (about 100k)



Example Results-End Shape Effect



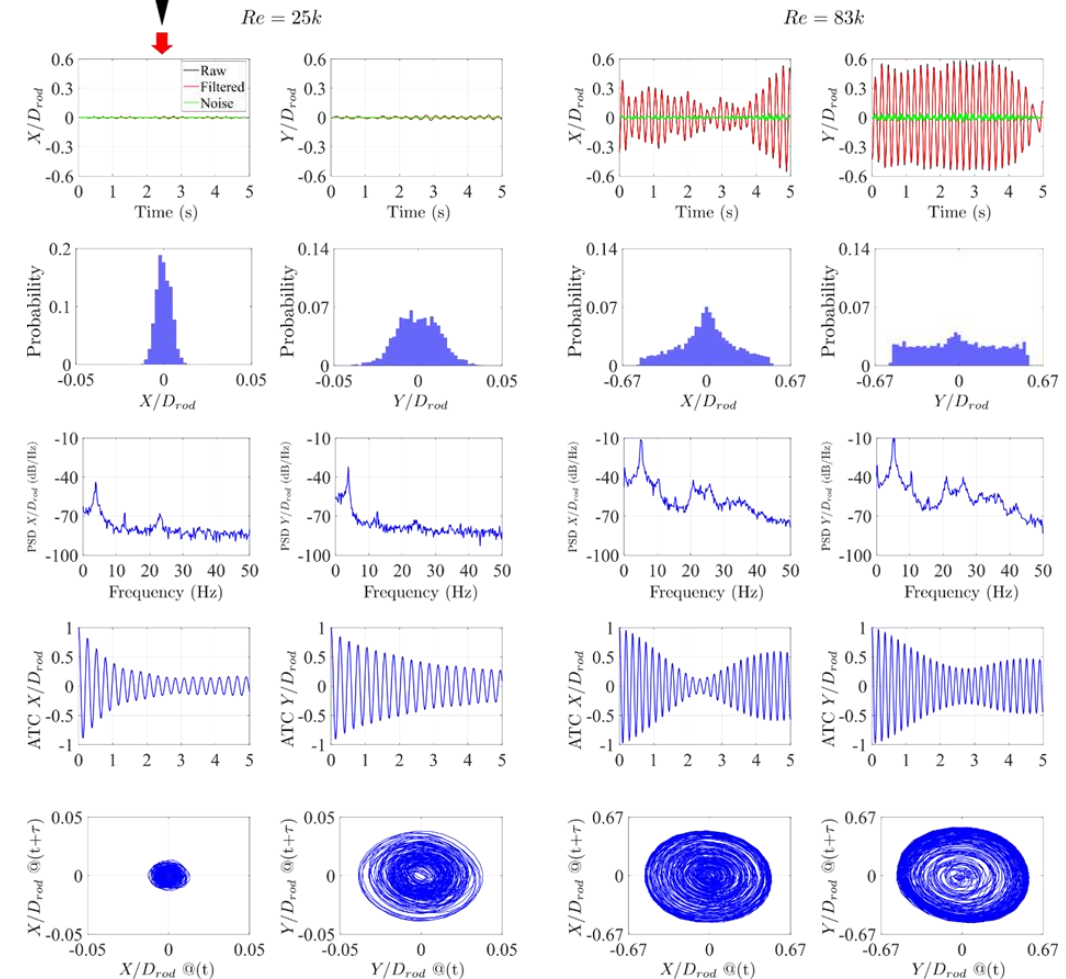
- Except for the cone tip with $h/d = 2$, the displacement histograms have a "bell-shaped" distribution, the autocorrelation functions are periodic and demonstrate fast decay, and the reconstructed attractors are "blob-like"
- This signifies that the rod vibrations encompass both periodic and chaotic components, i.e., **fuzzy period-1 motion**



Example Results-End Shape Effect

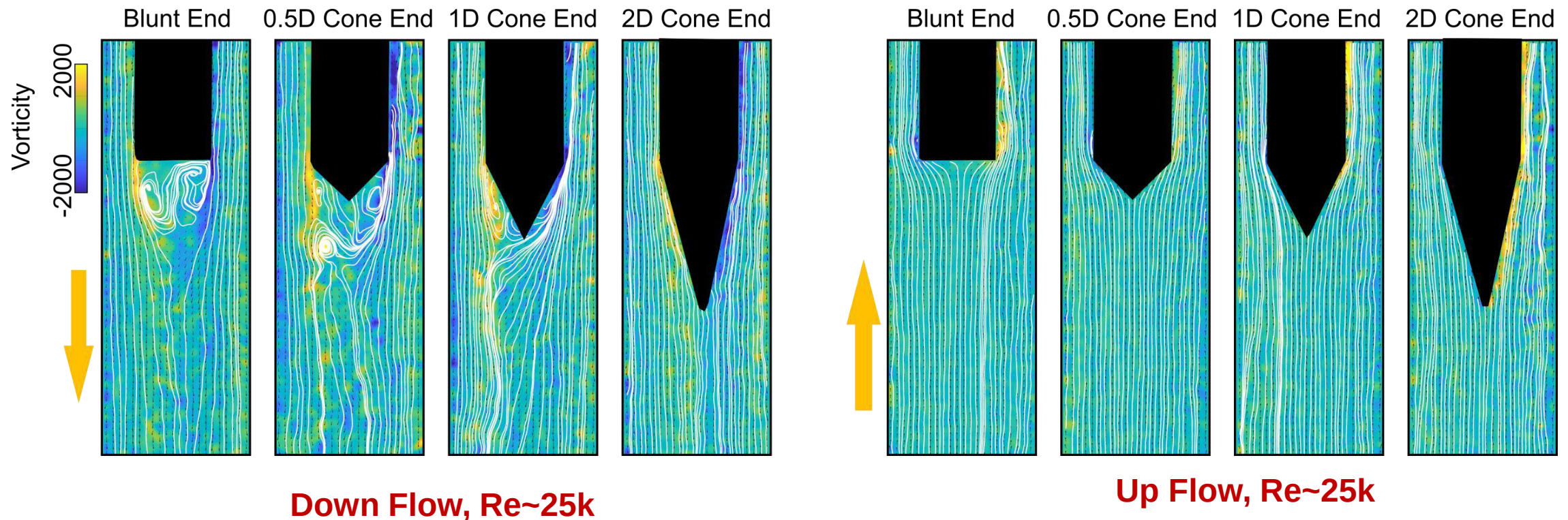


- The cone tip, with a height-to-diameter ratio of 2, has an equally fuzzy period-1 motion at low Reynolds numbers. However, at the higher Reynolds number of 83,000, the displacement **histograms start to display greater breadth and tend to create a "bridge-shaped" distribution**
- The **decay of the autocorrelation functions diminishes**, indicating that the **periodic component becomes increasingly prominent in respect to the chaotic component of the rod's motion**
- The **reconstructed attractors are primarily "circular"**
- These indicate that the **fuzzy period-1 motion observed at low Reynolds numbers evolves into a flutter-like, large-amplitude oscillation at higher Reynolds numbers**



Example Results-End Shape Effect

- Instantaneous* flow fields and vorticity contours near the tip of the rod for low Re cases



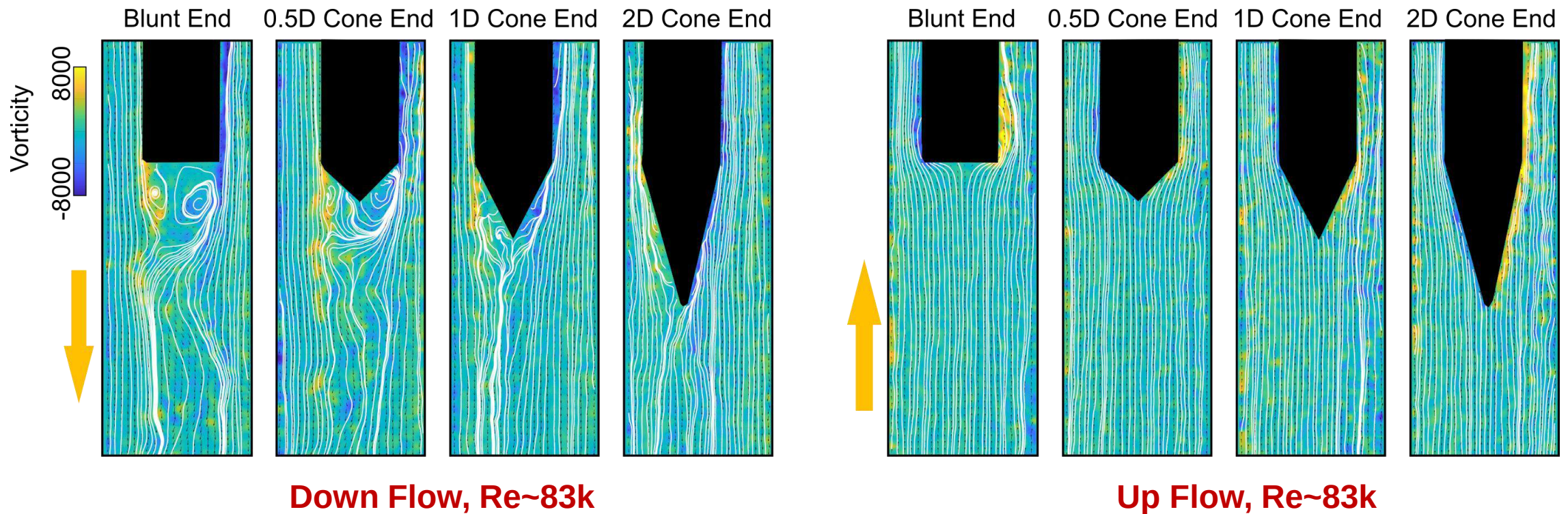
Unsteady flow separation considerably affects rod vibrations for blunt and cone tips with h/d ratios of 0.5 and 1, but for the cone tip with a h/d ratio of 2, turbulent buffeting is the primary source of excitation

Blunt tip configuration exhibits a pronounced stagnation zone with significant separation bubbles with considerable vorticity magnitudes emerge on each side of the rod

Example Results-End Shape Effect

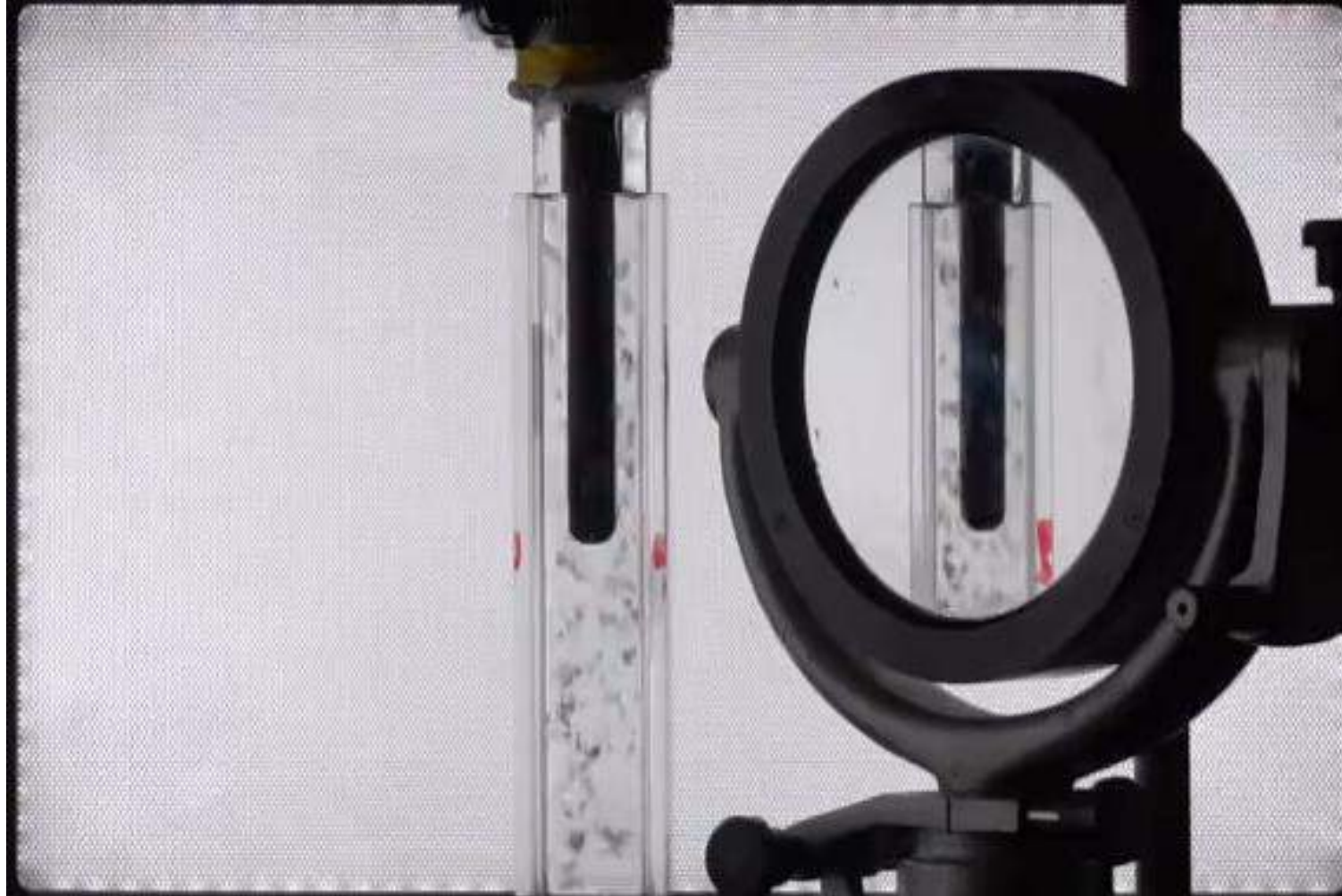


- *Instantaneous* flow fields and vorticity contours near the tip of the rod for high Re cases



An increase in the cone height-to-diameter ratio exacerbates the effects of unsteady flow separation due to the increased surface area available for flow disturbances

Measurements-Two Phase Flow

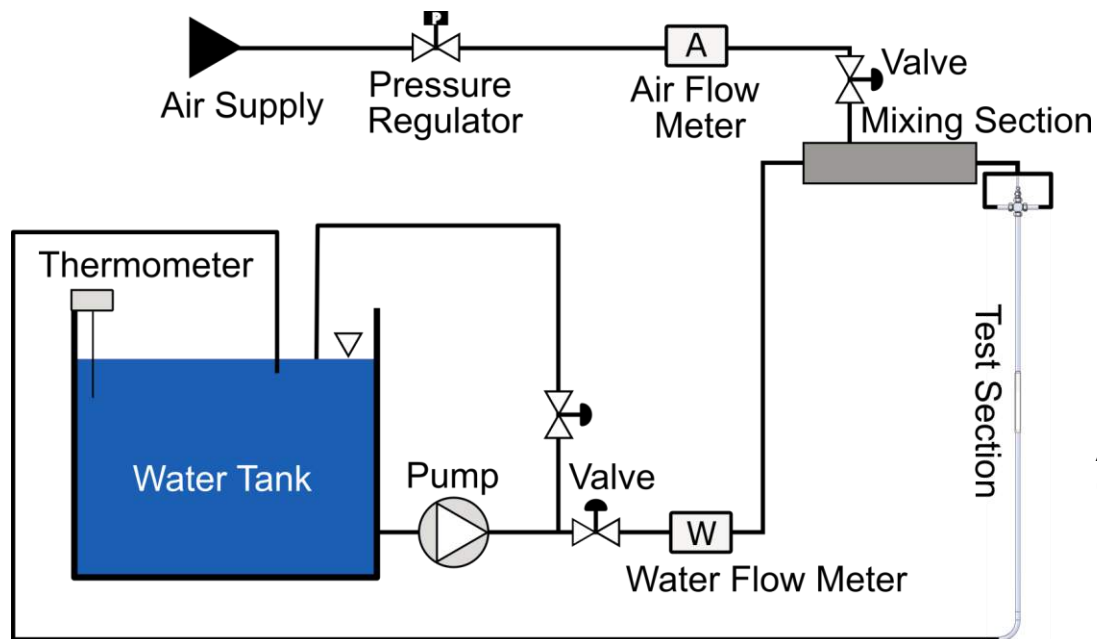
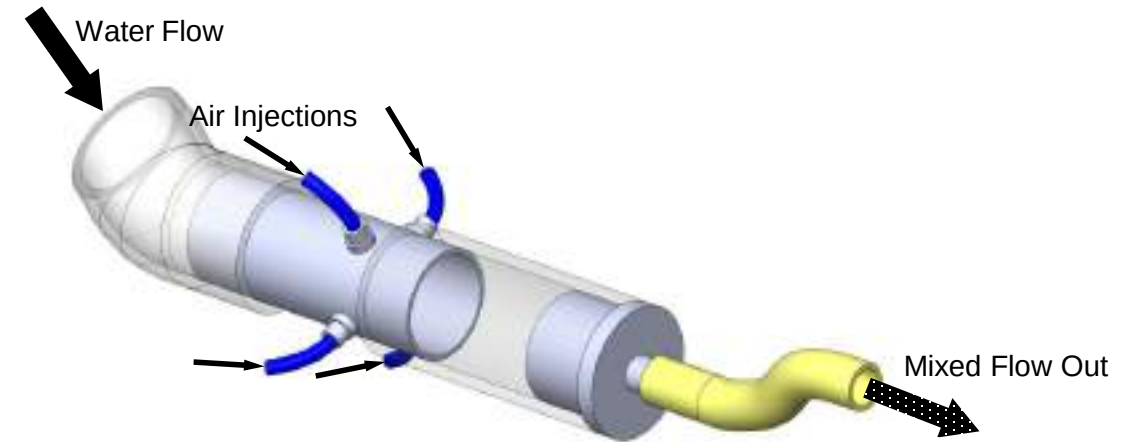


Trial run at flow rate~18 GPM

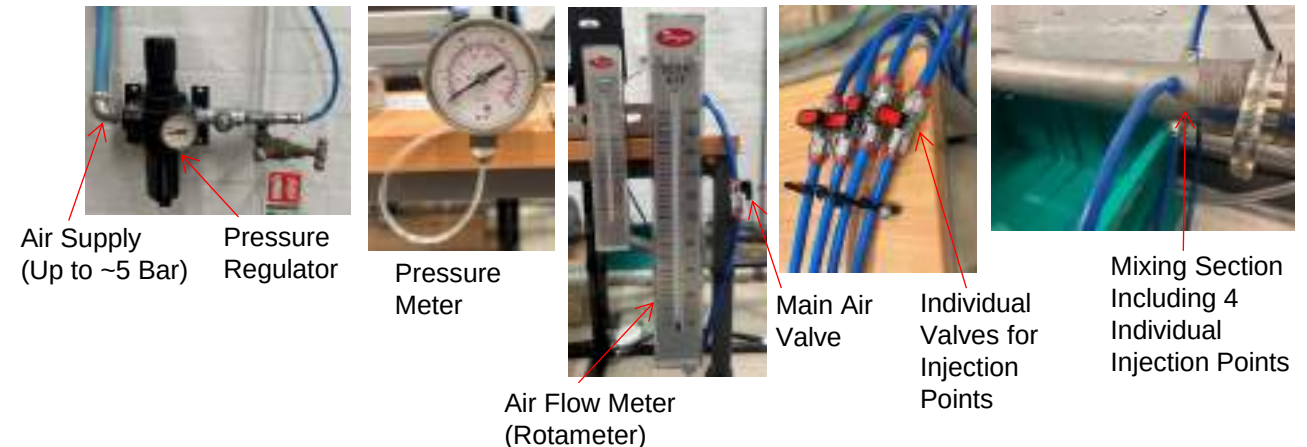
Experimental Rig



- Four air injection points located 360° around the mixing section
- Valves can be open/closed individually to modulate the mixing of air and water

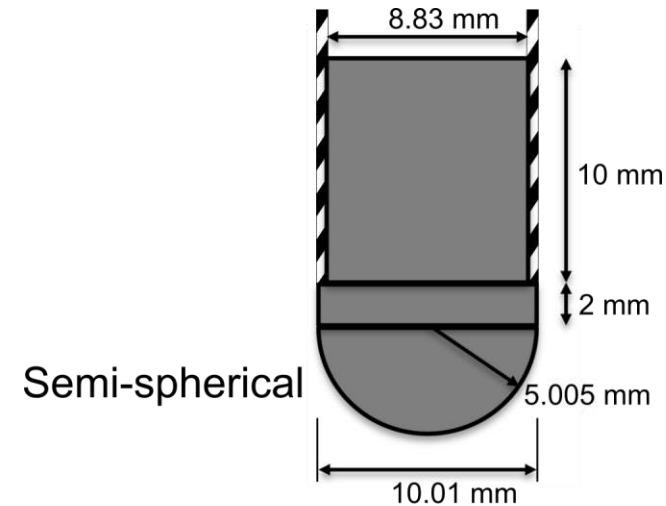
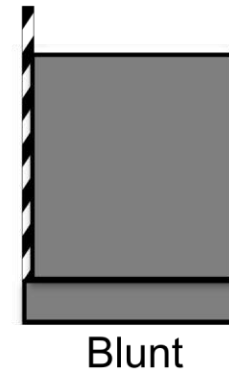
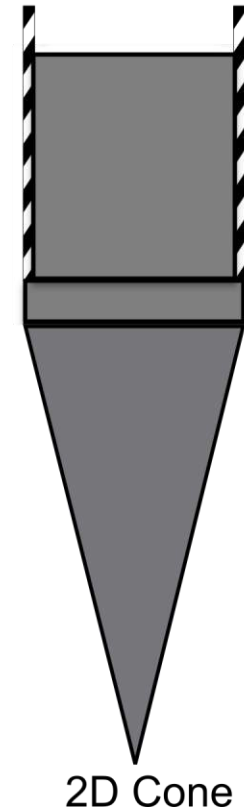


Air Injection Components:



Rod

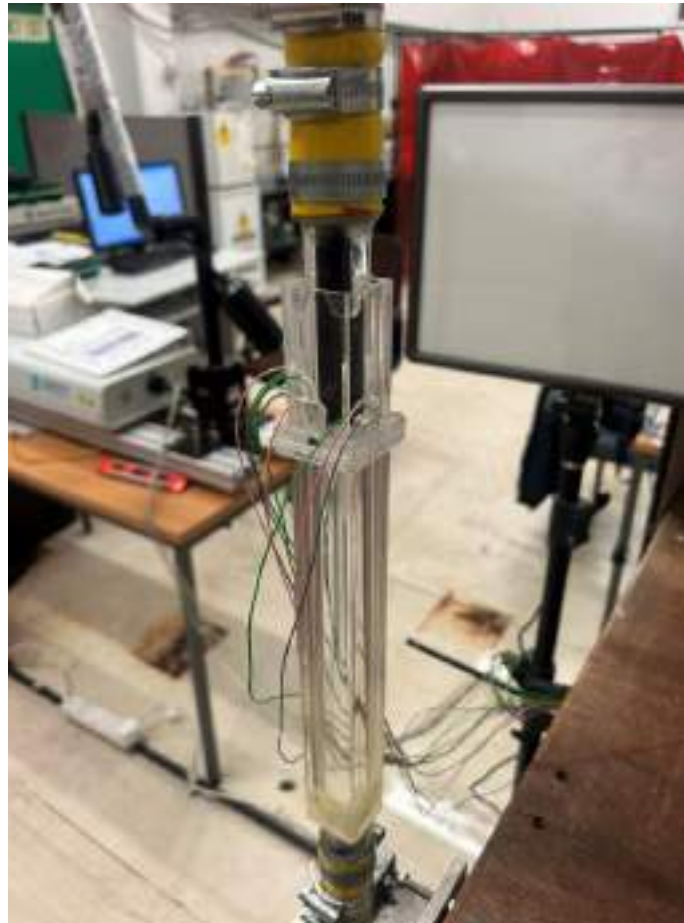
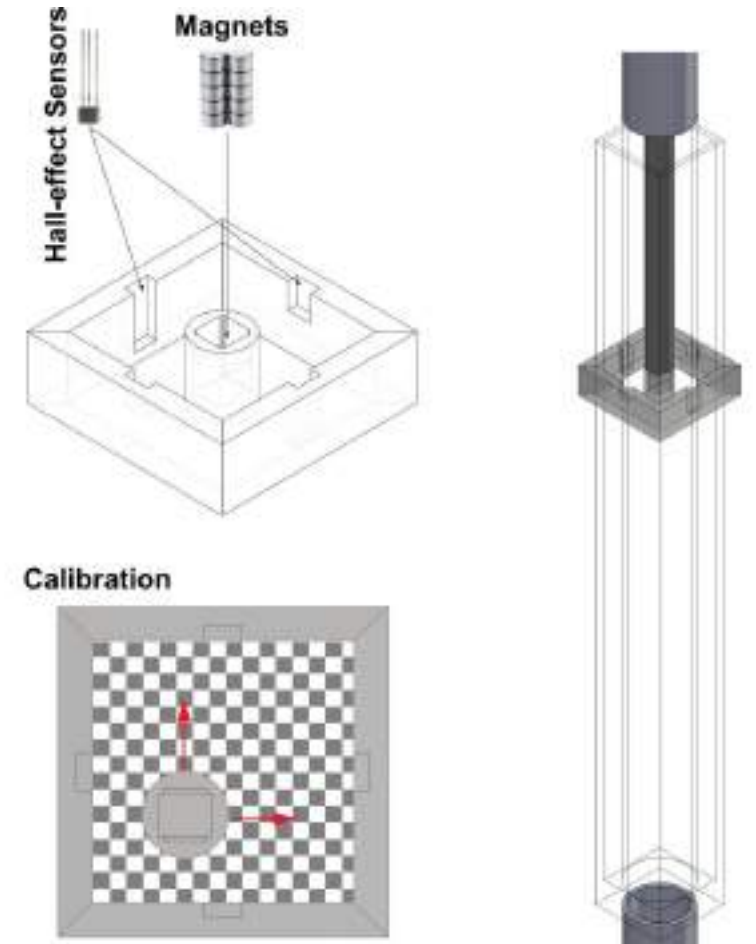
- Rod has diameter (D) of 10.01 ± 0.06 mm
- Hydraulic diameter is 11 mm
- Mass density of the rod is $\sim 588 \text{ g.m}^{-1}$
- Baseline Rod has a streamlined end-piece of semi-spherical shape
- Additional end shapes will be measured, including Blunt end and Cone end with 2D height



Measurements-Rod Vibration



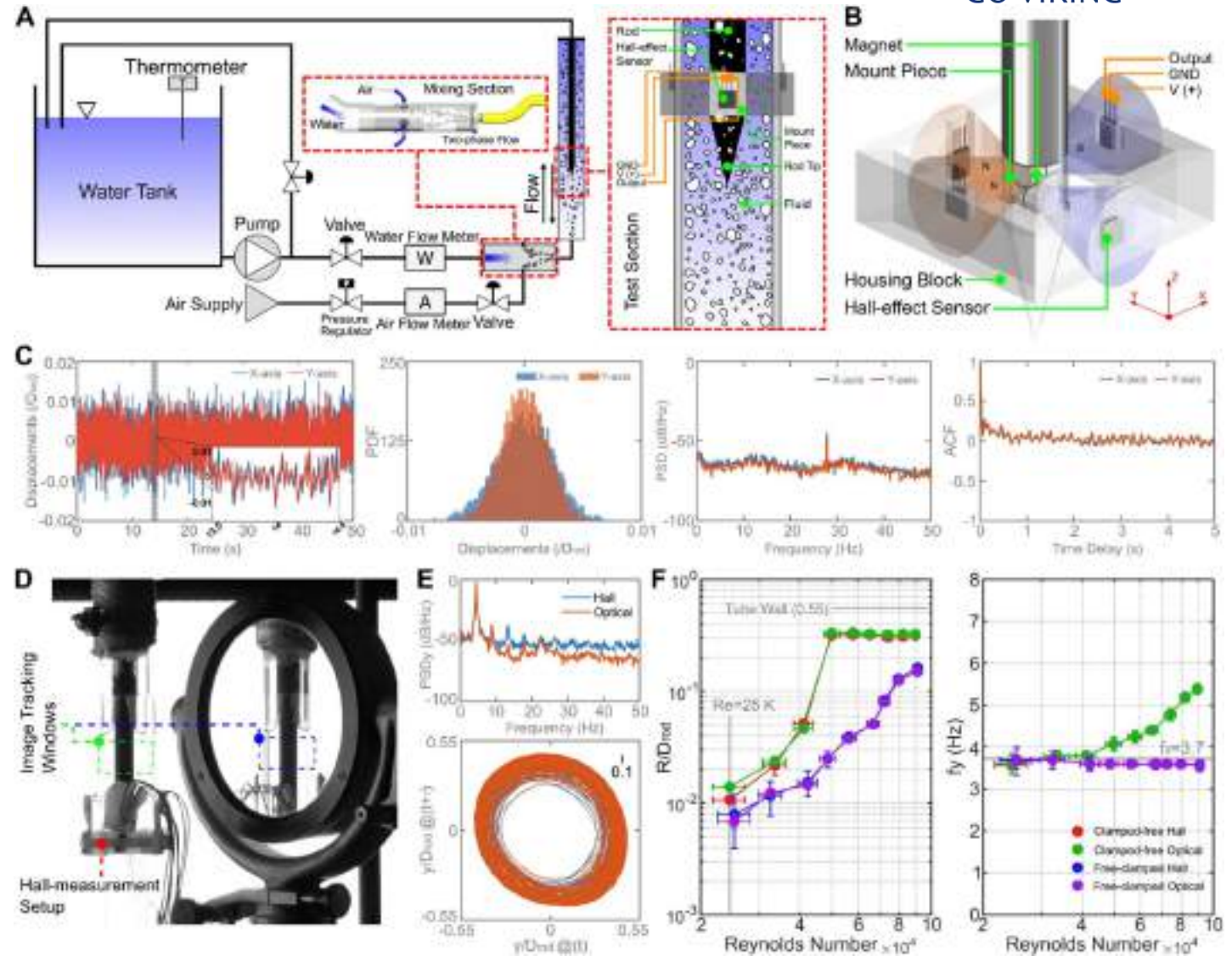
- Optical tracking fails in two-phase flow due to the bubbles blocking the field of view!
- Flow-induced motion measured using an in-house developed technique that utilises Hall effect sensors
- Capable of tracking rod vibration with 3D motion along with dedicated software for data postprocessing and time series analysis



Measurements-Rod Vibration



- Proposed method shows RMS displacement error values $< 40 \mu\text{m}$
- Validation via simultaneous measurements using optical tracking and the Hall effect technique shows excellent agreement



Measurements-Two Phase Flow Examples

Up Flows

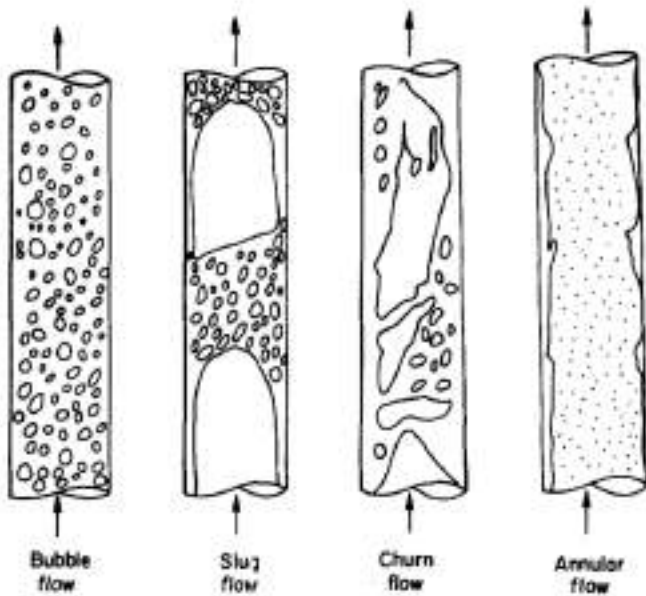
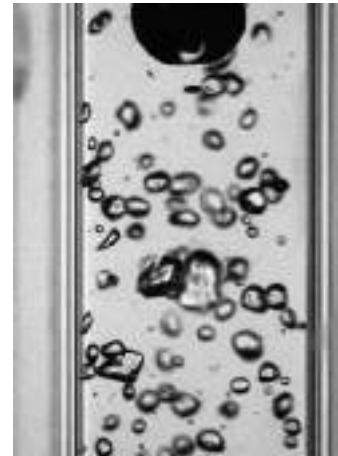


Figure 1. Flow patterns in vertical flow.

Taitel et al. 1980.



Bubble Flow



Slug Flow



Slug/Churn Flow



Churn/Annular Flow

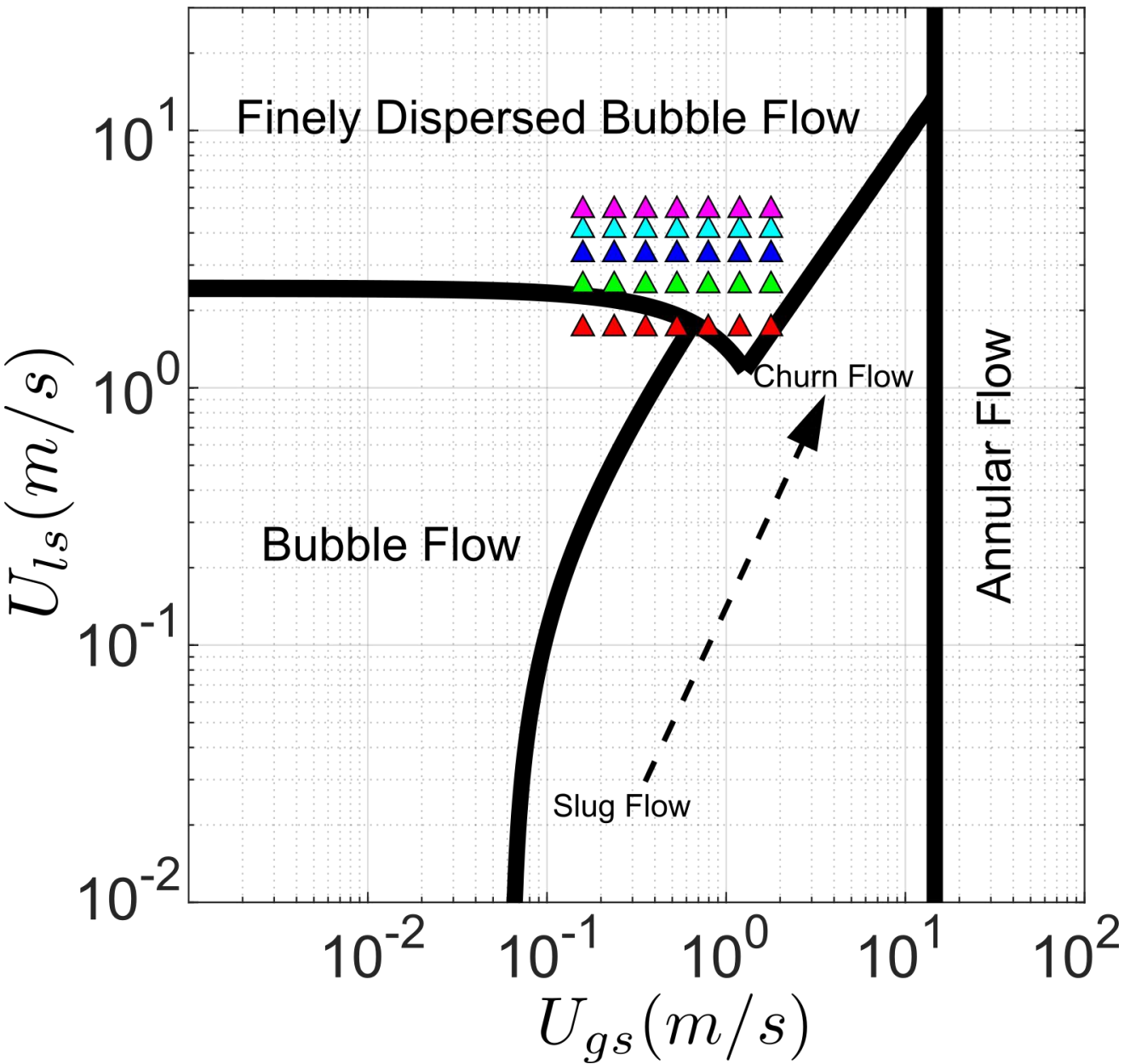
Two Phase FIV Experimental Cases:

Patch	Case	Q water (GMP)	Q air (l/min)	Uls (m/s)	Ugs (m/s)	alpha	Flow Regime
1	0	9.21	0.00	1.70	0.00	0.00	Single Phase
	1	9.21	3.25	1.70	0.16	0.09	Bubble
	2	9.21	4.86	1.70	0.24	0.12	Bubble
	3	9.21	7.28	1.70	0.35	0.17	Bubble
	4	9.21	10.89	1.70	0.53	0.24	Bubble
	5	9.21	16.29	1.70	0.79	0.32	Bubble
	6	9.21	24.37	1.70	1.19	0.41	Bubble
	7	9.21	36.46	1.70	1.78	0.51	Bubble
2	0	13.54	0.00	2.50	0.00	0.00	Single Phase
	1	13.54	3.25	2.50	0.16	0.06	Bubble
	2	13.54	4.86	2.50	0.24	0.09	Bubble
	3	13.54	7.28	2.50	0.35	0.12	Bubble
	4	13.54	10.89	2.50	0.53	0.18	Bubble
	5	13.54	16.29	2.50	0.79	0.24	Bubble
	6	13.54	24.37	2.50	1.19	0.32	Bubble
	7	13.54	36.46	2.50	1.78	0.42	Bubble
3	0	17.88	0.00	3.30	0.00	0.00	Single Phase
	1	17.88	3.25	3.30	0.16	0.05	Bubble
	2	17.88	4.86	3.30	0.24	0.07	Bubble
	3	17.88	7.28	3.30	0.35	0.10	Bubble
	4	17.88	10.89	3.30	0.53	0.14	Bubble
	5	17.88	16.29	3.30	0.79	0.19	Bubble
	6	17.88	24.37	3.30	1.19	0.26	Bubble
	7	7.88	36.46	3.30	1.78	0.35	Bubble
4	0	22.21	0.00	4.10	0.00	0.00	Single Phase
	1	22.21	3.25	4.10	0.16	0.04	Bubble
	2	22.21	4.86	4.10	0.24	0.05	Bubble
	3	22.21	7.28	4.10	0.35	0.08	Bubble
	4	22.21	10.89	4.10	0.53	0.11	Bubble
	5	22.21	16.29	4.10	0.79	0.16	Bubble
	6	22.21	24.37	4.10	1.19	0.22	Bubble
	7	22.21	36.46	4.10	1.78	0.30	Bubble
5	0	26.54	0.00	4.90	0.00	0.00	Single Phase
	1	26.54	1.5	4.90	0.07	0.01	Bubble
	2	26.54	3.25	4.90	0.16	0.03	Bubble
	3	26.54	4.86	4.90	0.24	0.05	Bubble
	4	26.54	7.28	4.90	0.35	0.07	Bubble
	5	26.54	10.89	4.90	0.53	0.10	Bubble
	6	26.54	16.29	4.90	0.79	0.14	Bubble
	7	26.54	24.37	4.90	1.19	0.20	Bubble
	8	26.54	36.46	4.90	1.78	0.27	Bubble

Uls: Water Superficial Velocity; Ugs: Air Superficial Velocity; alpha: Void Fraction

Experimental Cases:

▲ Patch 1 ▲ Patch 2 ▲ Patch 3 ▲ Patch 4 ▲ Patch 5



Definitions

Superficial Velocities

$$U_{gs} = \frac{Q_{gs}}{S_{tube}}$$

$$U_{ls} = \frac{Q_{ls}}{S_{tube}}$$

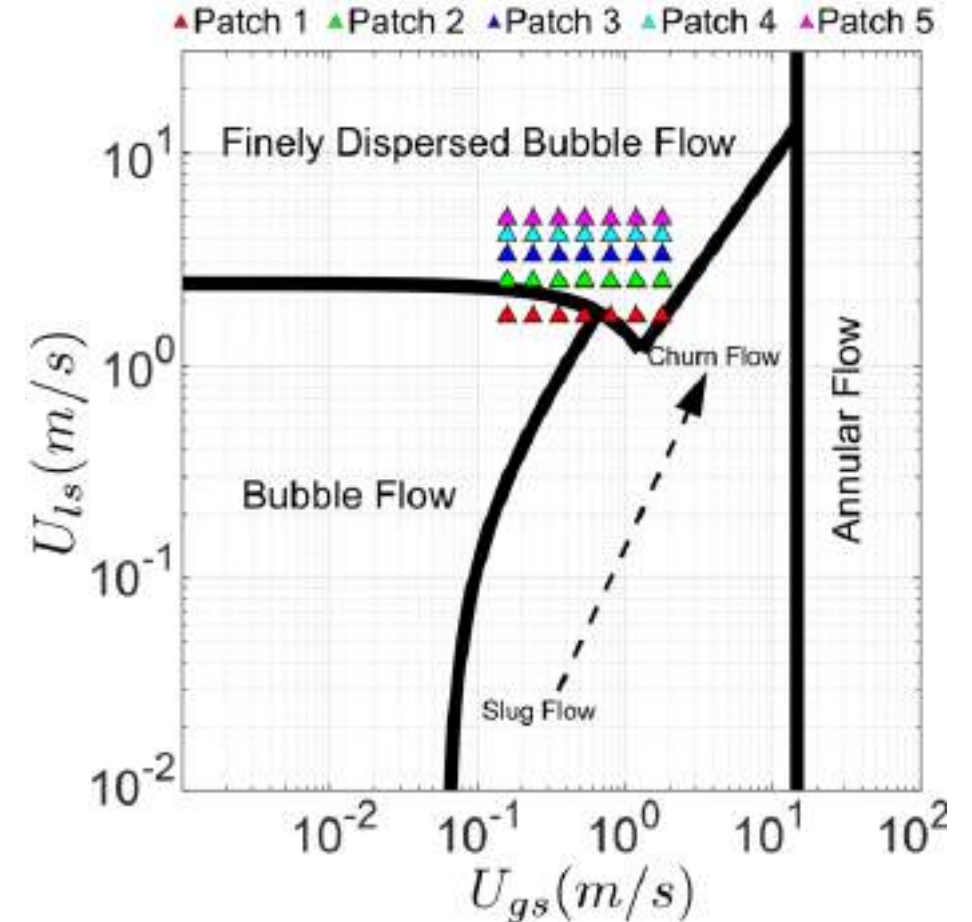
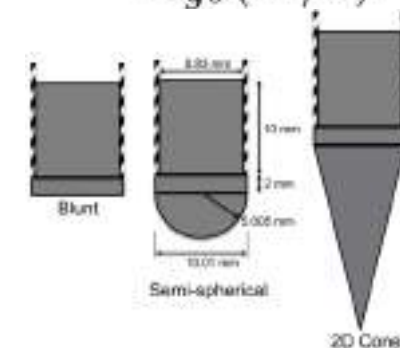
Superficial Annulus Reynolds Number

$$Re_{als} = \frac{\rho U_{als} D_{hyd}}{\mu}$$

Volumetric Void Fraction

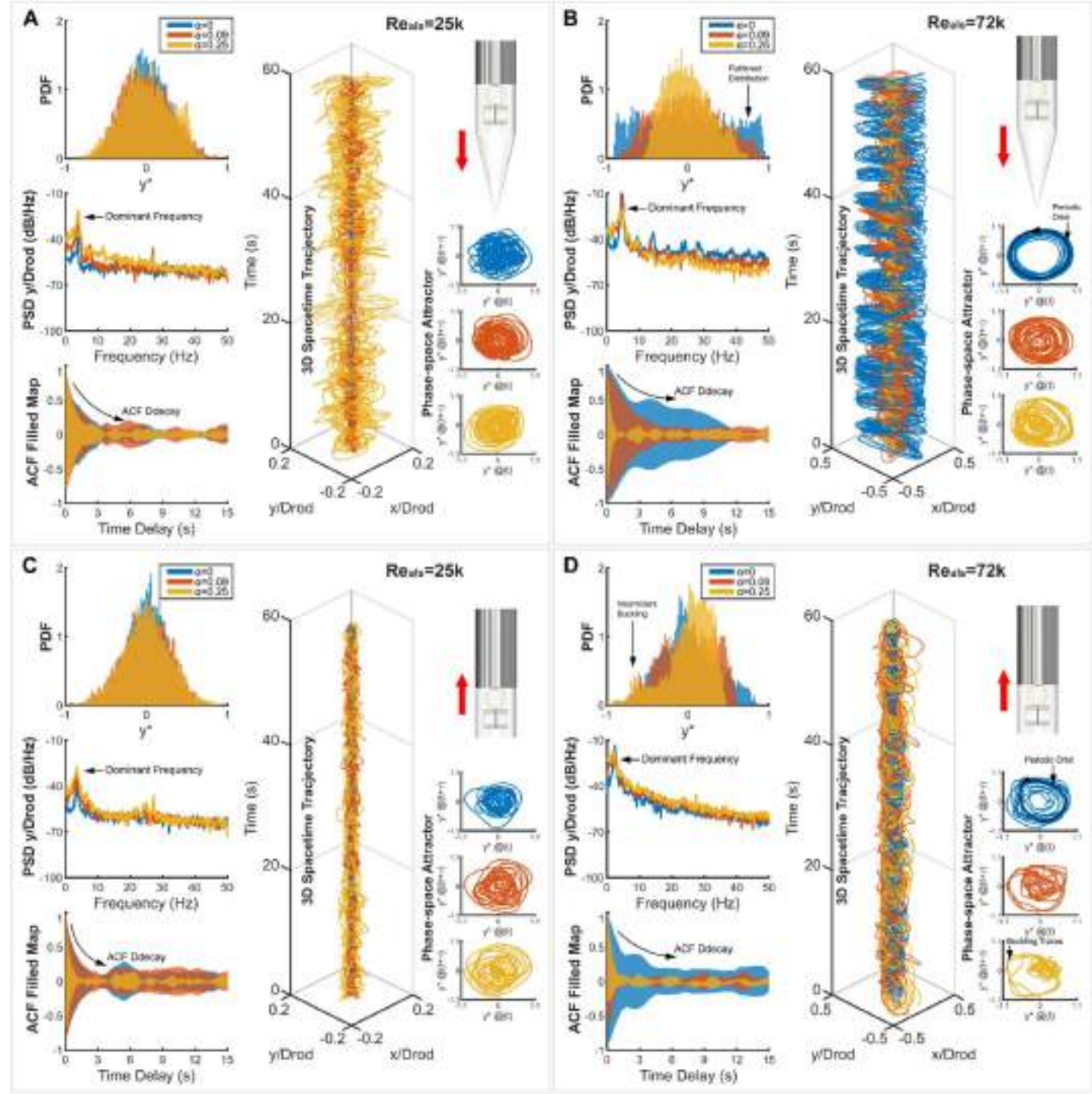
$$\alpha = \frac{U_{gs}}{U_{gs} + U_{ls}}$$

Cases Investigated



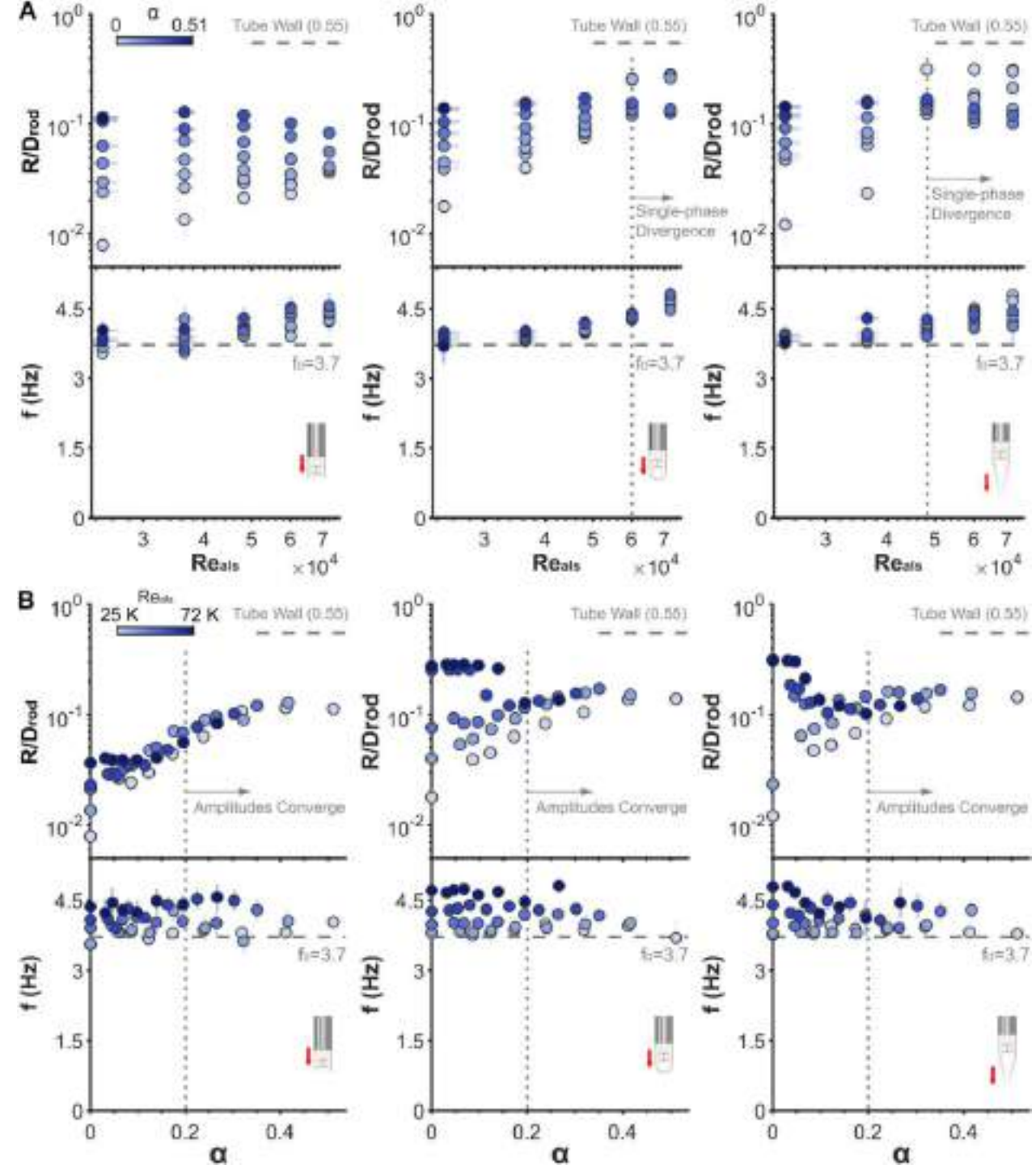
Two-Phase Displacement Results

- Competing stochastic and periodic rod vibration dynamics across two-phase flow regimes and flow/geometry configurations.
- (A, B) Clamped-free conical-tip dynamics: Large-amplitude oscillation (flutter) occurs for single-phase flow ($\alpha=0$) at $Re=72k$: flattened PDF, ring-shaped attractor indicating periodic motion, slow-decaying ACF.
- (C, D) Free-clamped blunt-tip dynamics: Intermittent buckling emerges at high $\alpha=0.25$ and $Re=72k$: localised PDF peak region; attractor show traces away from centroid.
- Key observations:
 - (i) Single-peaked PSD and amorphous attractor topology, signature of fuzzy-period 1 dynamics, except single-phase flutter divergence at $Re=72k$ (clamped-free conical tip).
 - (ii) Higher Re , i.e., (B), (D), leads to enhanced periodic vibrations (flattened/off-centred PDF distribution, slower ACF decay at low α).
 - (iii) Elevated α universally reduces periodicity and shifts dynamics to more stochastic regimes (Flattened converted to Gaussian PDF profile, more rapid ACF decay, wider PSD peak energy).



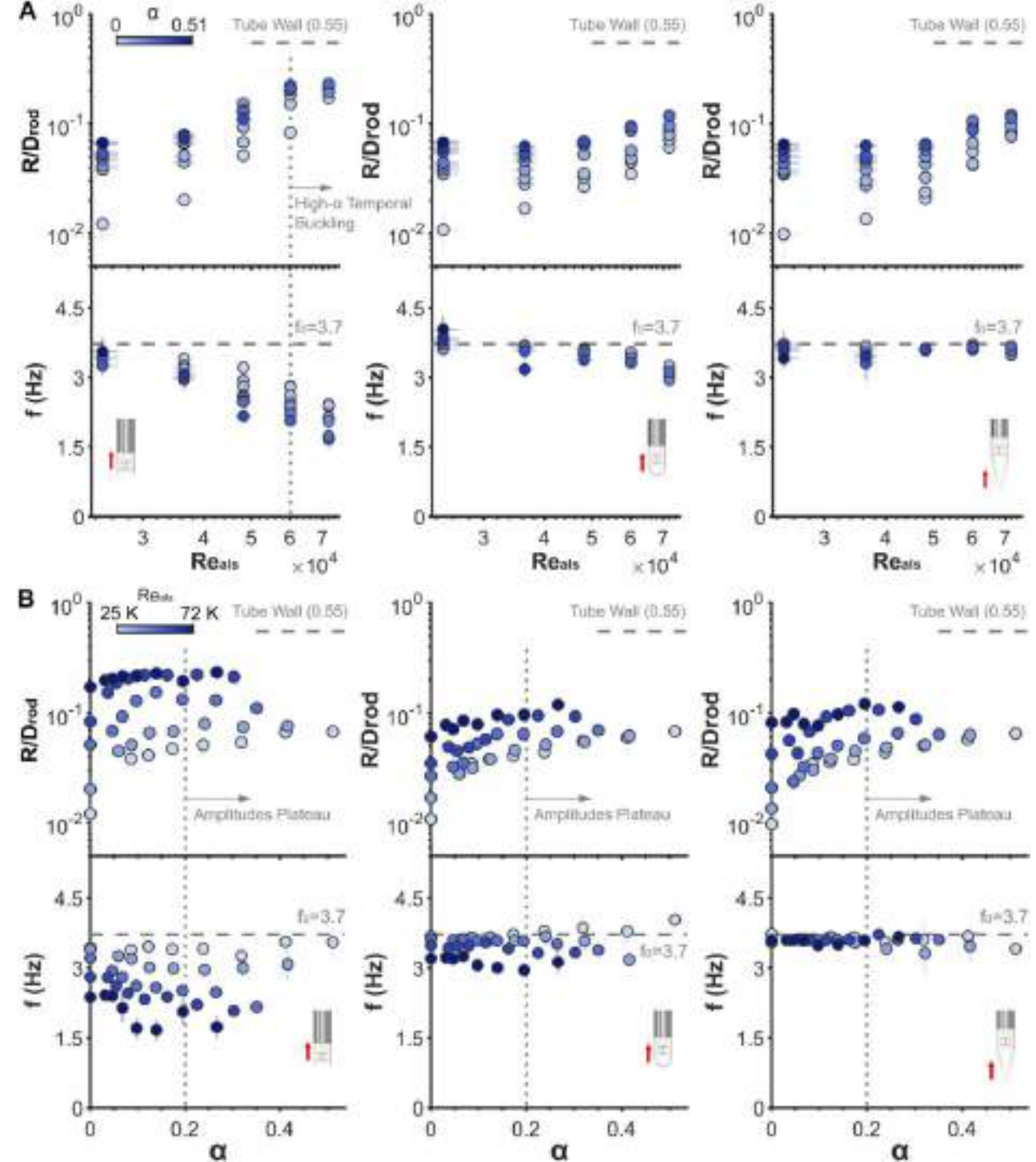
Two-Phase Displacement Results - CF

- Single-phase flows display linear amplitude growth with Re , culminating in flutter divergence for hemispherical ($Re=60k$) and conical ($Re=48k$) tips.
- Two-phase flows suppress this instability, yielding amplitudes plateau/decline despite rising
- Amplitudes show Re -dependent trends for $\alpha < 0.2$, increasing at low Re (stochastic forcing dominance) but saturating/decreasing at high Re (periodic forcing suppression)
- Above $\alpha = 0.2$, amplitude values converge across tip geometries and Re



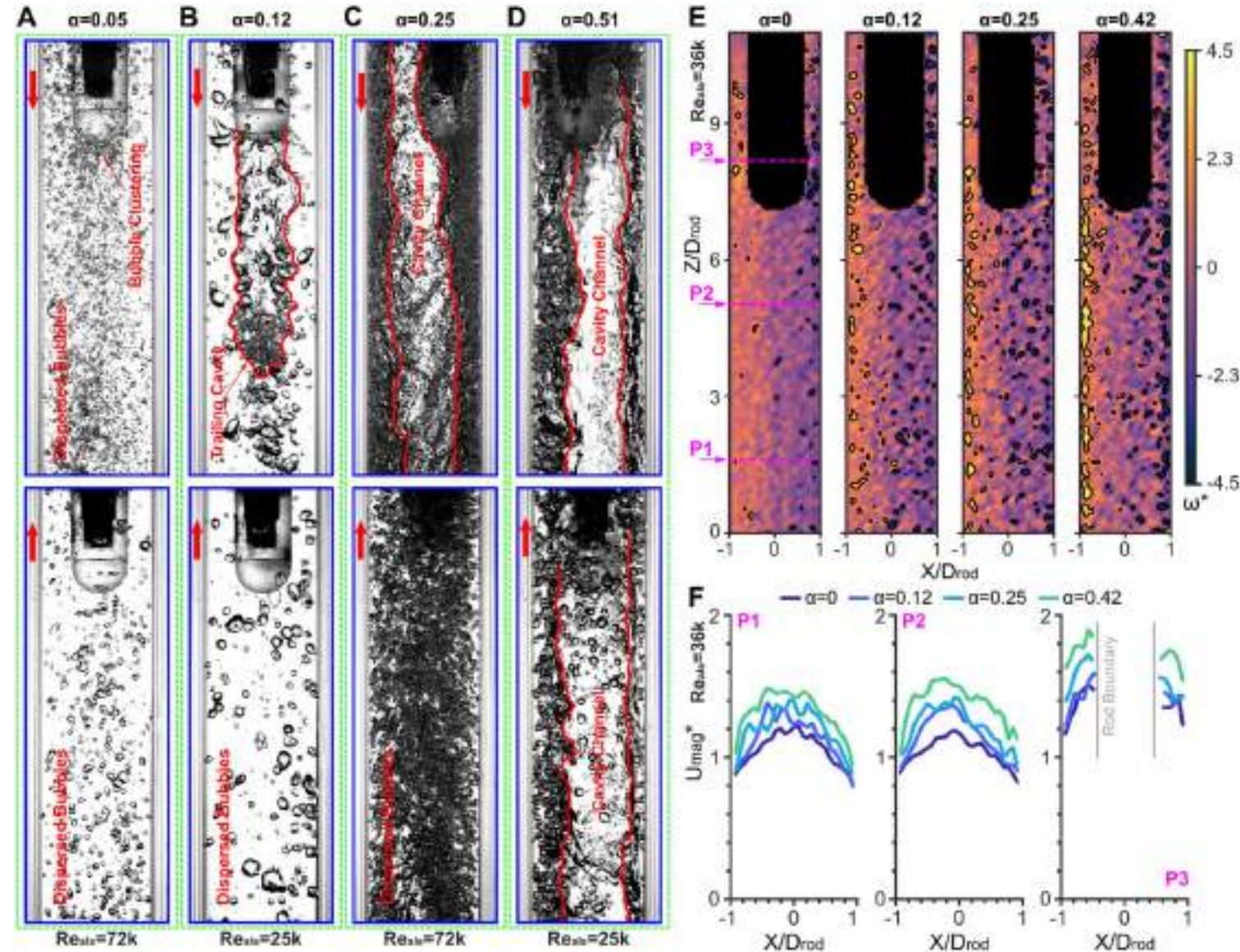
Two-Phase Displacement Results F-C

- Amplitude growth weakens with rising α as two-phase driven stochastic effects reduce Re dependence
- Intermittent buckling induced by two-phase flows ($\alpha > 0.2$) for blunt tip at $\text{Re} = 60\text{k}$ due to two-phase stochastic amplification
- Frequencies decline with Re, contrasting clamped-free rods - Tapering of tip geometry leads to less frequency reduction (e.g., conical tips)
- Amplitudes increase gradually for $\alpha < 0.2$ but saturate for $\alpha > 0.2$ at high Re, consistent with clamped-free rods, signaling universal stochastic dominance regime above $\alpha = 0.2$



Two-Phase PIV Results

- Bubble clustering/coalescence into cavity channels at elevated α , while higher fragments larger air packets into smaller dispersed bubbles.
- Normalised PIV-LIF vorticity field contours indicate vortex boundaries identified using the Q-criterion, highlighting turbulence and flow unsteadiness escalates with rising α .
- Extracted flow velocity profiles at 3 axially-spaced positions (-6.5, -2.5 and +1 from the rod tip), demonstrate increased stochastic fluctuations of flow velocity as α increases.



Main Outcomes



- *An experimental setup has been developed to generate high-resolution benchmark data that exposes new physical insights for axial-flow-induced vibration of a rod in a confining tube, including both structural response and flow field measurements*
- *Interesting dynamics have been identified using the tracking experiments with clear differences between the free-clamped and clamped-free cases*
- *PIV measurements (via both instantaneous and time-averaged measurements) provided clear characterisation of the flow field in the region near the rod tip (and at 10D below it) allowing this data to inform the development/validation of numerical models*

Main Outcomes



- *For the rod with the semi-spherical tip shape:*
 - *The dynamics of the rod in the free-clamped configuration mostly demonstrate flutter at low Reynolds numbers*
 - *The flow field measurements revealed increased turbulence intensity around the rod's tip, indicating that turbulent buffeting is the principal source of excitation*
 - *The rod eventually buckled, making contact with the surrounding tube, and remained almost stationary as the Reynolds number increased further*
- *In the clamped-free mode, the rod's dynamics were only affected by flutter, with no indications of buckling observed*
- *Flow field measurements demonstrate that the downflow remains largely symmetric up to a Reynolds number of 29k; beyond this limit, the flow around the rod tip becomes asymmetric, and the flow separation region enlarges due to the substantial amplitude and three-dimensional characteristics of the rod vibrations*

Main Outcomes



- *For the rod with the blunt and cone end-pieces:*
 - *At low Reynolds numbers, the primary motion of the rod across all tip geometries was characterized as fuzzy period-1, signifying a periodic motion with chaotic characteristics*
 - *This behavior was observed in both the free-clamped and clamped-free configurations*
 - *An increase in the Reynolds number results in phenomena such as flutter-like oscillation in the clamped-free configuration and buckling in the free-clamped configuration.*

Main Outcomes



- *Unsteady flow separation considerably influenced the rod vibration dynamics, with its significance fluctuating depending on the rod tip configuration and flow orientation:*
 - *In the **free-clamped configuration**, unsteady flow separation was seen to enhance flow stimulation of rod vibrations*
 - *The **blunt tip, marked by enhanced flow separation**, led to larger vibration displacement compared to the cone tips*
 - ***As the Reynolds number escalates, movement-induced excitations intensify**, resulting in a convergence of the displacements between the blunt tip and the cone tips*
- *In the **clamped-free configuration**, at low Reynolds numbers, **unsteady flow separation acts as an extra source of flow excitation at the rod's tip**. In this scenario, cone tip geometries with increased height-to-diameter ratios produce a larger surface area interacting with the wake vortices, hence leading to enhanced rod vibration displacements*
- *The **blunt tip with separated flow demonstrates less susceptibility to movement-induced excitations**, whereas the cone tip with minor flow separation is more prone to these excitations*
- *This results in heightened vibration displacements, **ultimately causing flutter divergence of the rod***

Main Outcomes



- For the two-phase flow experiments:
 - Two-phase flows impose **dual competing effects** on axial-FIVs: (i) **stochastic excitations** escalate with α due to gas cavity impacts and two-phase driven turbulence, amplifying vibration amplitudes at low Re , and (ii) **motion-induced periodic forces** diminish as two-phase flows disrupt coherent structures, suppressing flutter divergence at high Re
 - This trade-off defines **a critical threshold of $\alpha=0.2$, beyond which stochastic effects dominate, overriding flow configuration, Reynolds number, and tip geometry—a shift from single-phase FIV case**, where periodic instabilities govern large amplitude vibrations and consequently failure mechanisms.
 - The dominant stochastic regime above $\alpha=0.2$ indicates that design optimizations effective with single-phase flows become less effective in high- α environments, necessitating new strategies, e.g., turbulence baffling/damping and/or gas cavity fragmentation
 - The dual role of two-phase flows—amplifying stochastic vibrations while suppressing periodic instabilities—highlights the need for condition-specific strategies to manage flow-induced vibrations in two-phase flow environments

Publications

Journal Publications:

Li, H., Cioncolini, A., Iacovides, H., Benguigui, W., & Nabawy, M. R. A. (2026). Unveiling the fundamentals of two-phase axial-flow-induced vibrations of cantilever rods. *Scientific Reports*, 16: 5102. (doi: [10.1038/s41598-026-35337-4](https://doi.org/10.1038/s41598-026-35337-4))

Pauzi, A. M., Iacovides, H., Cioncolini, A., Li, H. & Nabawy, M. R. A. (2025). Application of URANS simulation and experimental validation of axial flow-induced vibrations on a blunt-end cantilever rod for nuclear applications. *Arabian Journal for Science and Engineering*, 50 (5), 3435-3455. (doi: [10.1007/s13369-024-09505-5](https://doi.org/10.1007/s13369-024-09505-5))

Mao, W., Iacovides, H., Cioncolini, A., Li, H. & Nabawy, M. R. A. (2024). A validated numerical methodology for flow-induced vibration of a semi-spherical end cantilever rod in axial flow. *Physics of Fluids*, 36(7): 075155. (doi: [10.1063/5.0211524](https://doi.org/10.1063/5.0211524))

Li, H., Cioncolini, A., Zhang, S., Iacovides, H. & Nabawy, M. R. A. (2024). Tip shape effects on the axial-flow-induced vibration of a cantilever rod. *Journal of Fluids and Structures*, 127: 104132. (doi: [10.1016/j.jfluidstructs.2024.104132](https://doi.org/10.1016/j.jfluidstructs.2024.104132))

Cioncolini, A., Zhang, S., Nabawy, M. R. A., Li, H., Cooper, D. & Iacovides, H. (2023). Experiments on axial-flow-induced vibration of a free-clamped/clamped-free rod for light-water nuclear reactor applications. *Annals of Nuclear Energy*, 190: 109900. (doi: [10.1016/j.anucene.2023.109900](https://doi.org/10.1016/j.anucene.2023.109900))

Conference Publications:

Li, H., Nabawy, M. R. A., Iacovides, H., Benguigui, W. & Cioncolini, A. (2025). Experiments on axial-flow-induced vibration of a cantilever rod in two-phase flow. *AIAA Scitech 2025 Forum*, Orlando, FL, USA, 6-10 January 2025. (doi: [10.2514/6.2025-1687](https://doi.org/10.2514/6.2025-1687))

Cioncolini, A., Li, H., Zhang, S., Nabawy, M. R. A. & Iacovidis, H. (2023). Experiments on axial-flow-induced vibration of a free-clamped rod for light water nuclear reactor applications. In *20th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-20)*, Washington, DC, USA, 20-25 August 2023.

Partners



Associated Partners



*EDF R&D UK Center



Thank you!



Mostafa Nabawy



University of Manchester



mostafa.ahmednabawy@manchester.ac.uk

www.go-viking.eu



Funded by the
European Union



Cantilever Rod: Benchmark Results

GO-VIKING Workshop

Mohammed Muaaz M. D. Hussain,

GRS, Garching, Germany, 20 May 2026



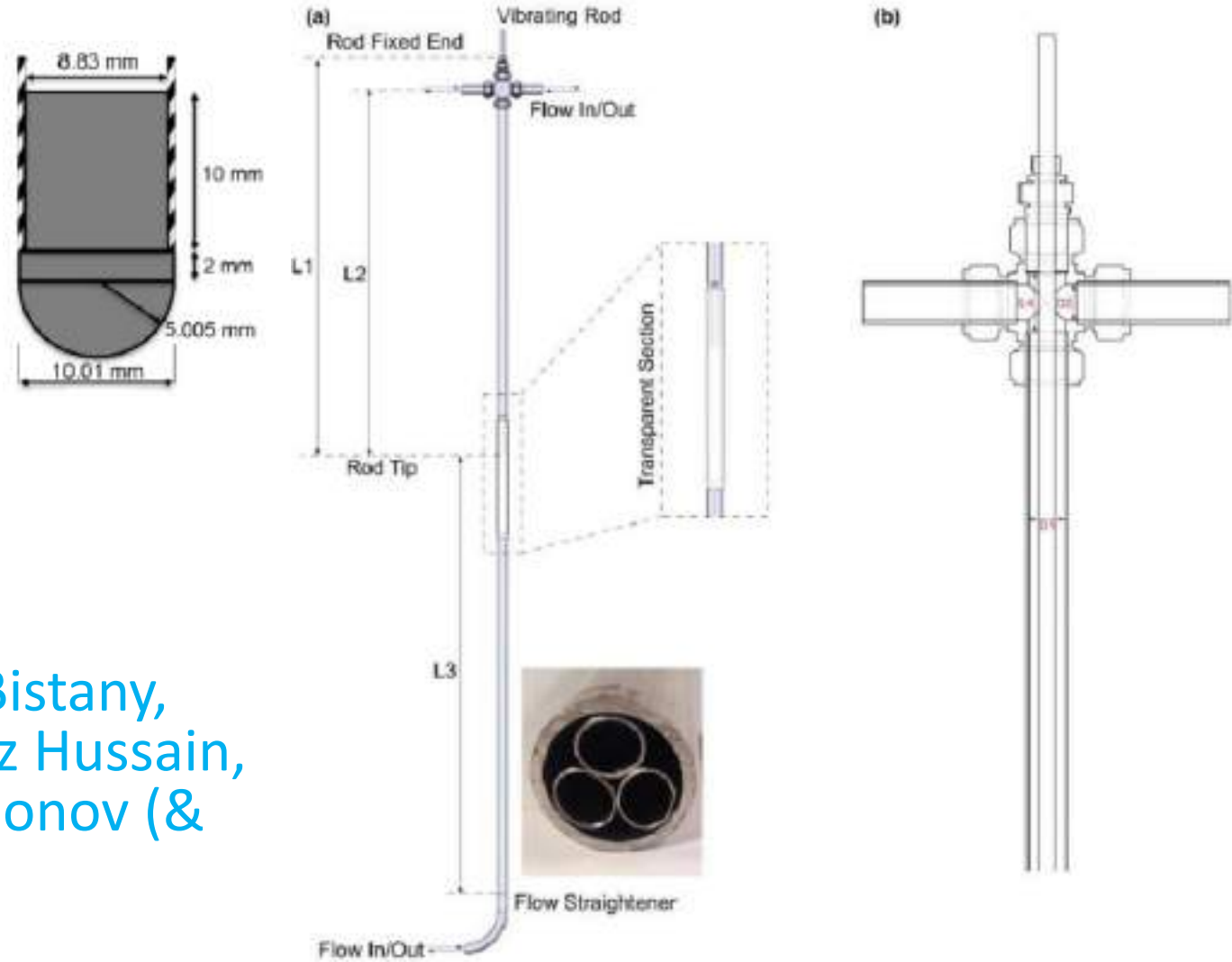
Funded by
the European Union

NRG ref. 2.6206.20/26.347195 C&S/MH/ES
EU DuC = 0E001

Introduction



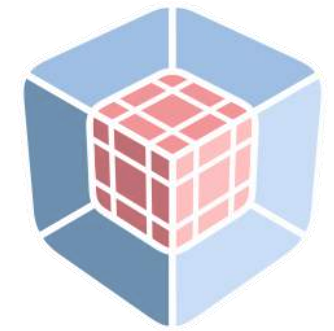
- Cantilever Rod Experiment
- Free-Clamped / Upflow
- Hemispherical Tip
- $Re_{ann} = 21.2k$
- Working Fluid → Water
- Temperature Range → 17-23 C
- 5 Participants: PSU, EDF, NRG PALLAS, EDF-Energy, IPP
- Involved Principle Researchers: Sean Bistany, Thomas Norddine, Mohammed Muaaz Hussain, Ewan Harris, Yaroslav Dubyk, Yuliaa Filonov (& others)



Different Approaches



Institution	Approach	Turbulence Modelling	Solid Modelling	Software(s)
PSU	CFD	DNS	NA	NekRS V23
EDF	Two-way FSI	WR-LES - Smagorinsky	Euler-Bernoulli Beam Model	Code_Saturne V8 + Beam Model implementation
NRG PALLAS	Two-way FSI	URANS - Standard K-Epsilon Low Re (cubic)	Finite Element CSM	Simcenter STAR-CCM+ V2306
EDF-Energy	Two-way FSI	URANS - SSG RSM	Finite Element CSM	Simcenter STAR-CCM+ V2206
IPP	Two-way FSI	URANS - K-Omega SST	Euler-Bernoulli Beam Model	ANSYS CFX + User Library (FORTRAN) implementation of Beam Model



Solid Domain

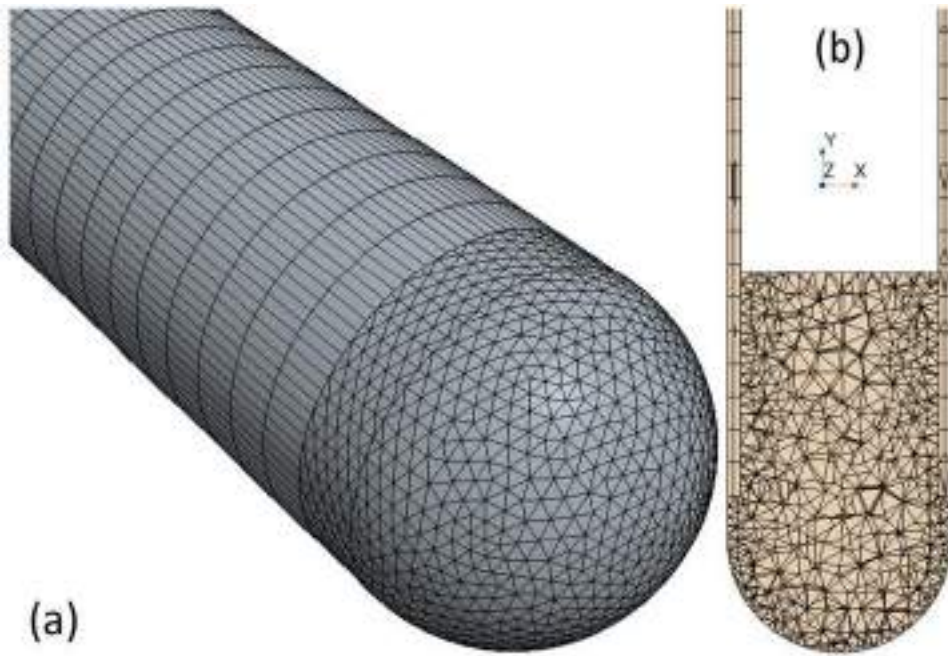


- In the experiment, we have a hollow stainless steel pipe filled with lead shots and plugged with an aluminum end cap

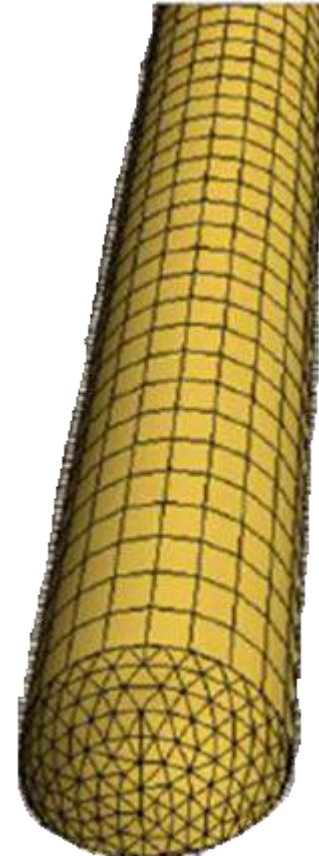
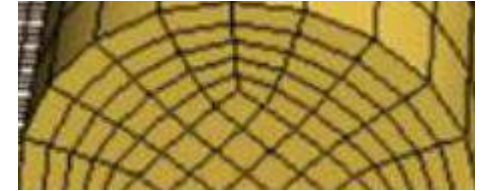
Institution	Geometry Model	Mesh Type	Mesh Count
EDF	Homogenous Rod with one Material	1D Elements	3800 Elements
NRG PALLAS	Composite Rod with modified SS Hollow Cylinder + Aluminum Plug	Structured Hexahedral for the Hollow Cylinders, Tetra- hedral for the Aluminum Plug	178.445k Cells
EDF-En- ergy	Homogenous Rod with one Material	Structured Hexahedral for the Cylinders, Tetrahedral for the Hemispherical End	62.5k Cells
IPP	Homogenous Rod with one Material	1D Elements	100 Points

Solid Domain - Meshes

- NRG PALLAS



- EDF-Energy



Solid Material Properties



Institution	Density (kg/m ³)	Youngs Modulus (GPa)	Poissons Ratio (-)	Other Provided Data
EDF	33676.3131	193	NA	Quadratic moment of Area = 1.94E-10 m ⁴ Mechanical Damping Coefficient = 0.08 kg/(ms)
NRG PALLAS	33676.313 (SS) 2740 (Aluminum)	193 (SS) 69 (Aluminum)	0.26 (SS) 0.33 (Aluminum)	Rayleigh Damping Model Constant = 7.701 s
EDF-Energy	7490	76.4	0.25	
IPP	33676.31312	193	0.3	Quadratic moment of Area = 1.9443E-10 m ⁴

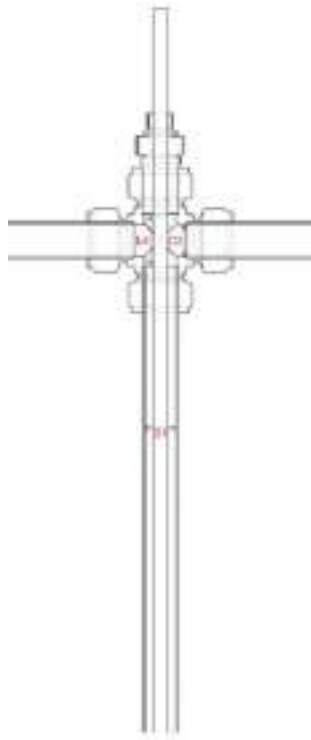
Solid Boundary Conditions



Institution	Top Surface/Node	Other Surfaces/Nodes	Remarks
EDF	Wall + Constrained	FSI Interface	
NRG PALLAS	Wall + Constrained		Surface Roughness applied to wetted surfaces = 2.5 μm (SS), 0.45 μm (Aluminum)
EDF-Energy	Symmetry + Constrained		Symmetry is applied because the full length of the rod is not present. Only the length from the union cross height to the tip is considered.
IPP	Wall + Constrained		

Fluid Domain

- In the experiment, we have a 1 inlet and 2 outlets as well as a union cross

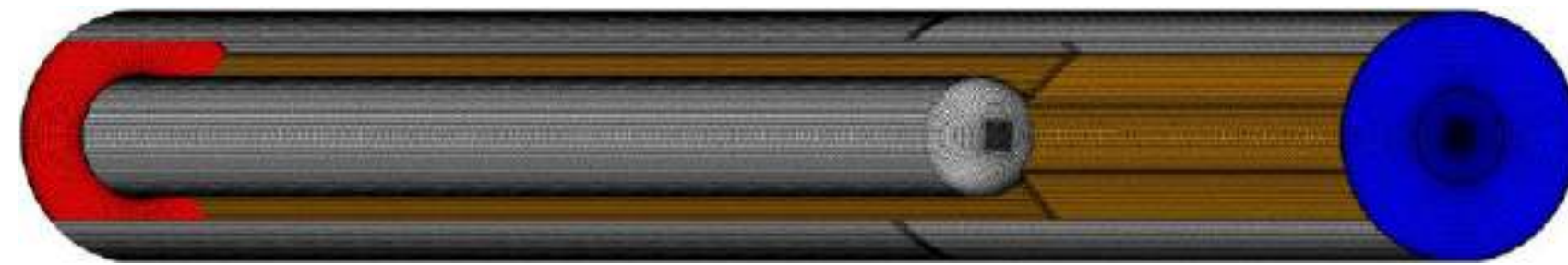


Institution	Geometry Model	Mesh Cell Type	Mesh Cell Count (M Cells)	Wall Y+	Courant Number
PSU	With respect to the experiment, 346 mm of Confining Pipe in Test Section considered of which 220 mm of the Rod Length is present. The scaled dimensions are correspondingly 16.476 m and 10.476 m respectively. 1 Inlet – 1 Outlet Configuration.	Structured Hexahedral	2.65 (Polynomial Order=7)	<1	Max. = 2.23
EDF	Entire Rod length + $6D_{\text{pipe}}$ considered. 1 Inlet – 1 Outlet Configuration.	Structured Hexahedral	144	Max. = 1.4 Surf. Avg. = 0.577	Max. = 1 Vol. Avg. = 0.126
NRG PALLAS	Entire Rod Length + full Vertical Confining Pipe in Test Section + Union Cross considered. 1 Inlet-2 Outlet Configuration	Hexahedral	8.49	Max. = 0.5132 Surf. Avg. = 0.1196	Max. = 29.4672 Vol. Avg. = 0.2278
EDF-Energy	Rod length from Union Cross + $5D_{\text{pipe}}$ considered. 1 Inlet – 1 Outlet Configuration.	Hexahedral	0.2	Max. = 39.4 Surf. Avg. = 31.6	Max. = 0.5 Vol. Avg. = 0.1
IPP	Entire Rod Length + full Vertical Confining Pipe in Test Section + Union Cross considered. 1 Inlet-2 Outlet Configuration	Structured Hexahedral	2.03	Max. = 324.47 Surf. Avg. = 0.353	Max. = 999.9 Vol. Avg. = 33.2

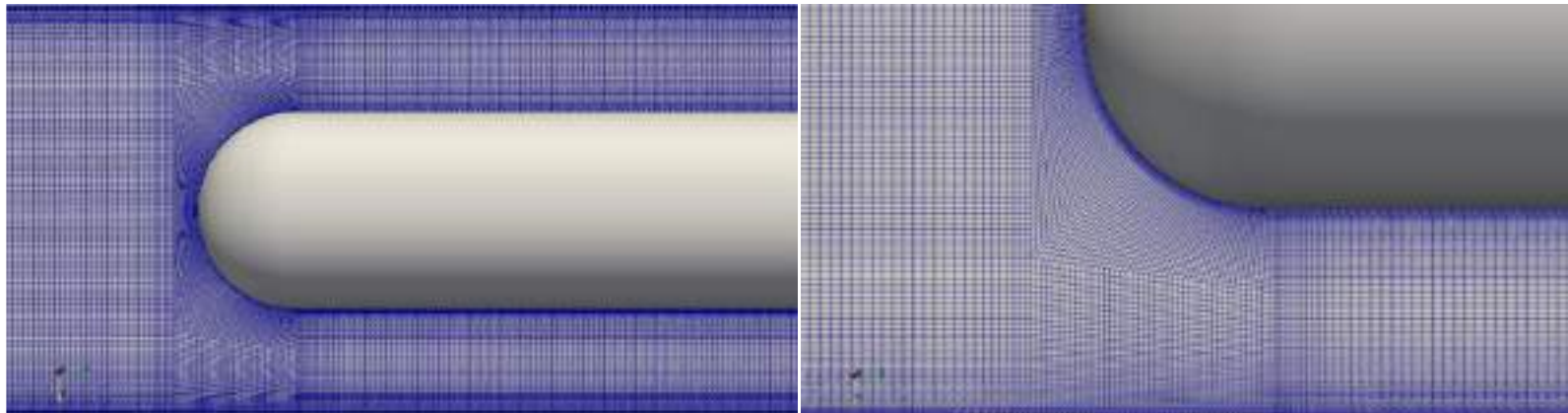
Fluid Domain - Meshes



- PSU

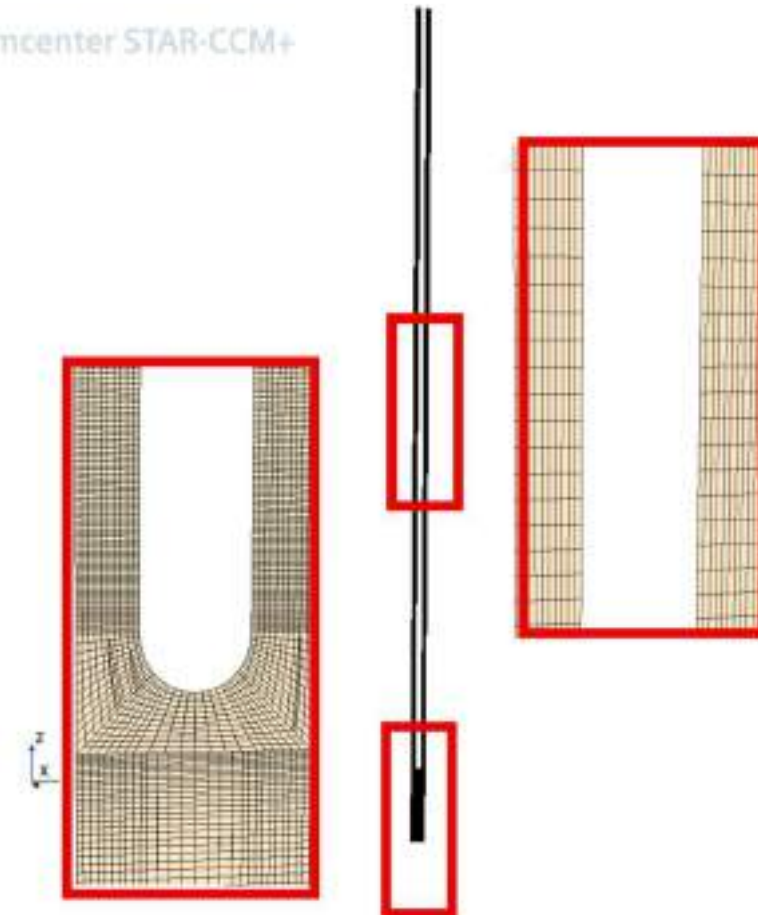


- EDF



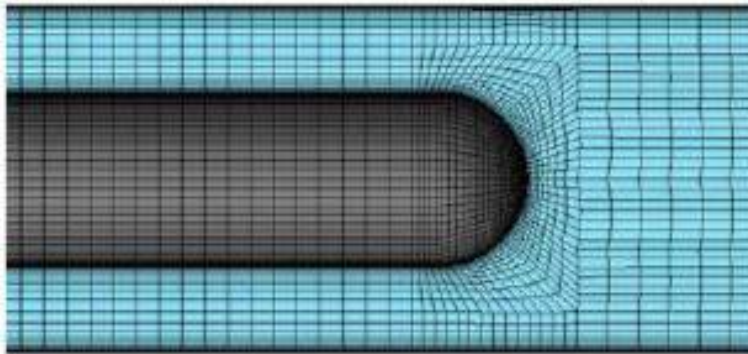
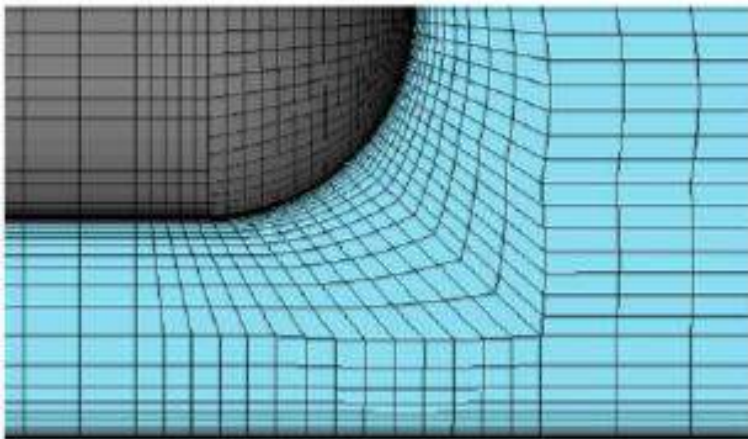
- EDF-Energy

Simcenter STAR-CCM+

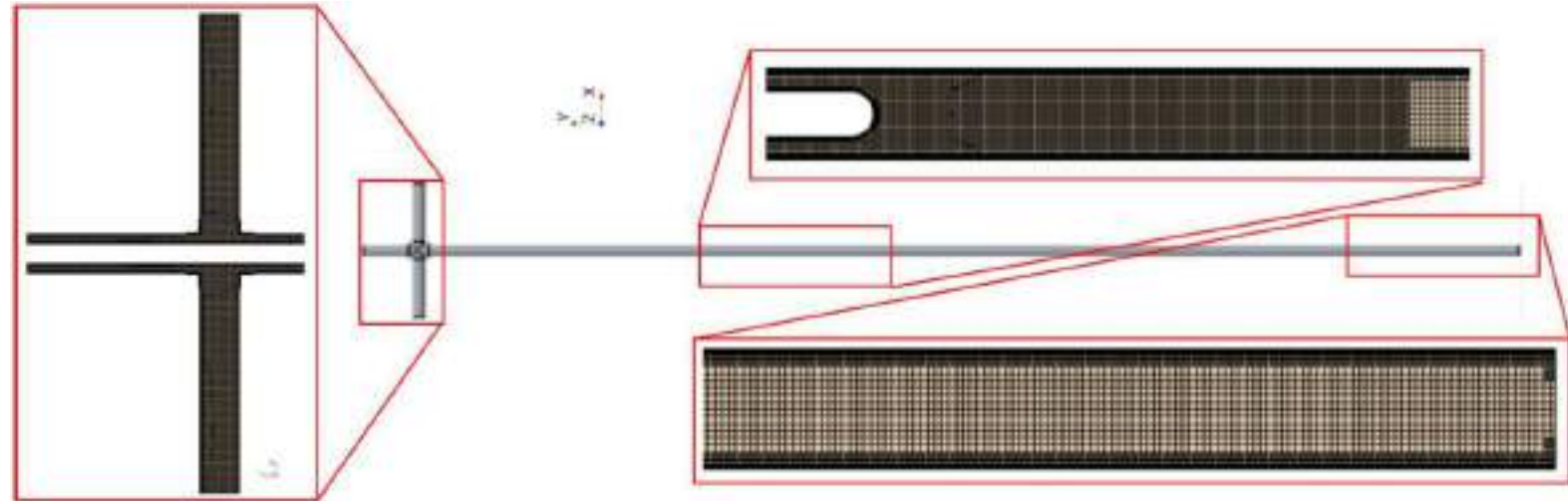


Fluid Domain - Meshes

- IPP



- NRG PALLAS



Fluid Material Properties



Institution	Density (kg/m ³)	Dynamic Viscosity (mPa.s)	Remarks
PSU	1	0.03195374325	Properties modified to achieve low compute cost while realizing same Re
EDF	997.567	1.047133	Slight difference arises from the choice of temperature of water for which material properties are extracted
NRG PALLAS	998.23	1.002	
EDF-Energy	997.57	1.047	
IPP	998.573	1.04737	

Fluid Boundary Conditions



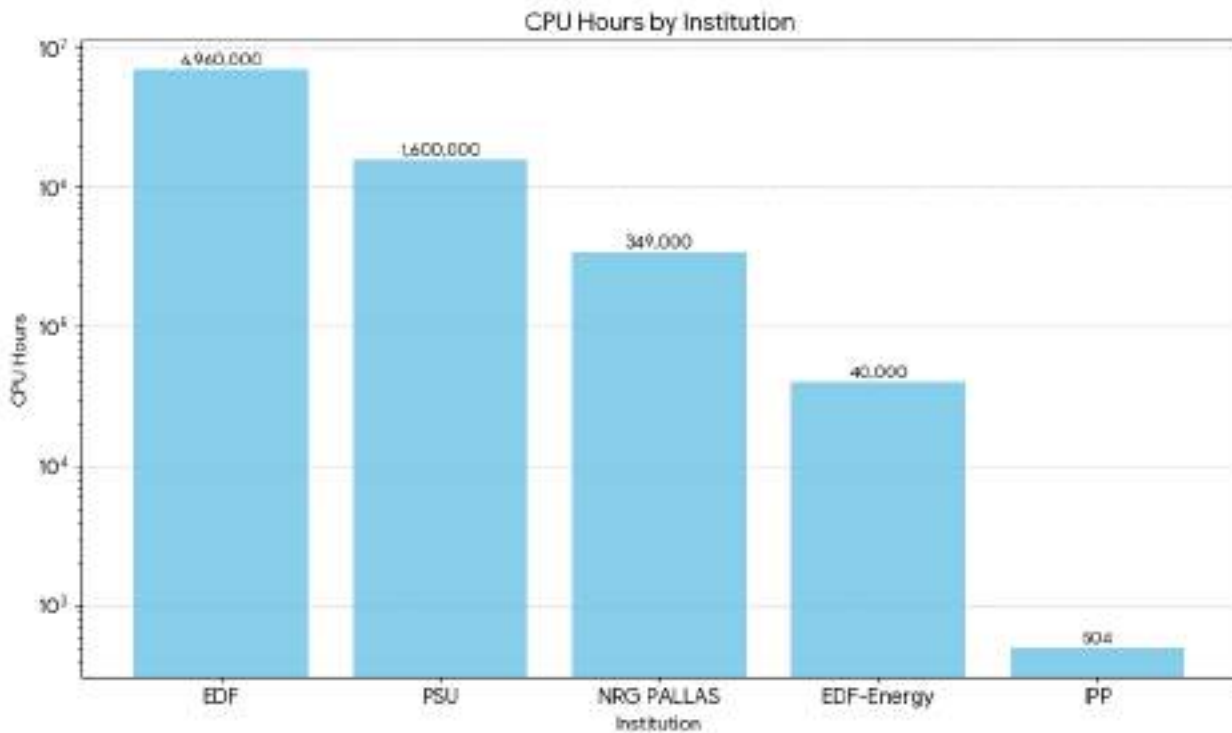
Institution	Inlet BC	Outlet(s) BC	Remarks
PSU	3D velocity profile with average 1 m/s	0Pa Gauge Pressure Outlet	Inflow velocity profile obtained by precursor simulation of flow through a pipe of length $4D_{\text{pipe}}$ with repeating/cyclic boundary conditions. Viscosity is modified to obtain required Re.
EDF	3D velocity profile with average 1.57 m/s		Inflow velocity profile obtained by precursor simulation of flow through a pipe of length $6D_{\text{pipe}}$ with repeating boundary conditions
NRG PALLAS	Single value of 1.51 m/s applied to whole patch		Given that the whole confining pipe length is present, inflow is expected to develop well enough before rod tip.
EDF-Energy	Single value of 1.59 m/s applied to whole patch		Given that confining pipe length of $4.76D_{\text{pipe}}$ is present upstream of rod tip, inflow is expected to develop well enough before rod tip.
IPP	MFR of 0.5149 kg/s applied to Inlet patch		Given that the whole confining pipe length is present, inflow is expected to develop well enough before rod tip.

Numerical Settings



Institution	Time Step Size (ms)	Total Simulation Time (s)	Order of Discretization	Other Provided Data	Institution	Time Step Size (ms)	Total Simulation Time (s)	Order of Discretization	Other Provided Data
PSU	0.0013422	8.59014	Temporal = 2 nd (BDF2/EXT2) Spatial = 8 th (N=7)	Average number of inner velocity iterations per time step = 1.4 Average number of inner pressure iterations per time step = 2 Number of subcycling steps per time step = 2	NRG PALLAS	0.1	19.643	Temporal = 2 nd (BDF2) Spatial = 2 nd	Number of inner iterations per time step = 25 SIMPLEC pressure-velocity coupling scheme
EDF	0.015	82	Temporal = 2 nd Spatial = 2 nd	Fractional step method similar to SIMPLEC Tolerance for velocity and pressure resolution = 1.0×10^{-5} Average number of inner iterations = 3.0 for velocity and 14.7 for pressure Max number of RHS reconstruction sweeps = 5 for pressure and 10 for velocity	EDF-Energy	0.15	11.0001	Temporal = 2 nd (BDF2) Spatial = MUSCL 3 rd (momentum transport), 2 nd (RST transport)	Number of inner iterations per time step = 25 SIMPLE pressure-velocity coupling scheme
					IPP	20	40	Temporal = 2 nd (Backward Euler) Spatial = 2 nd (High Resolution)	Number of inner iterations per time step = 10 Coupled pressure-velocity coupling scheme

Computational Resources

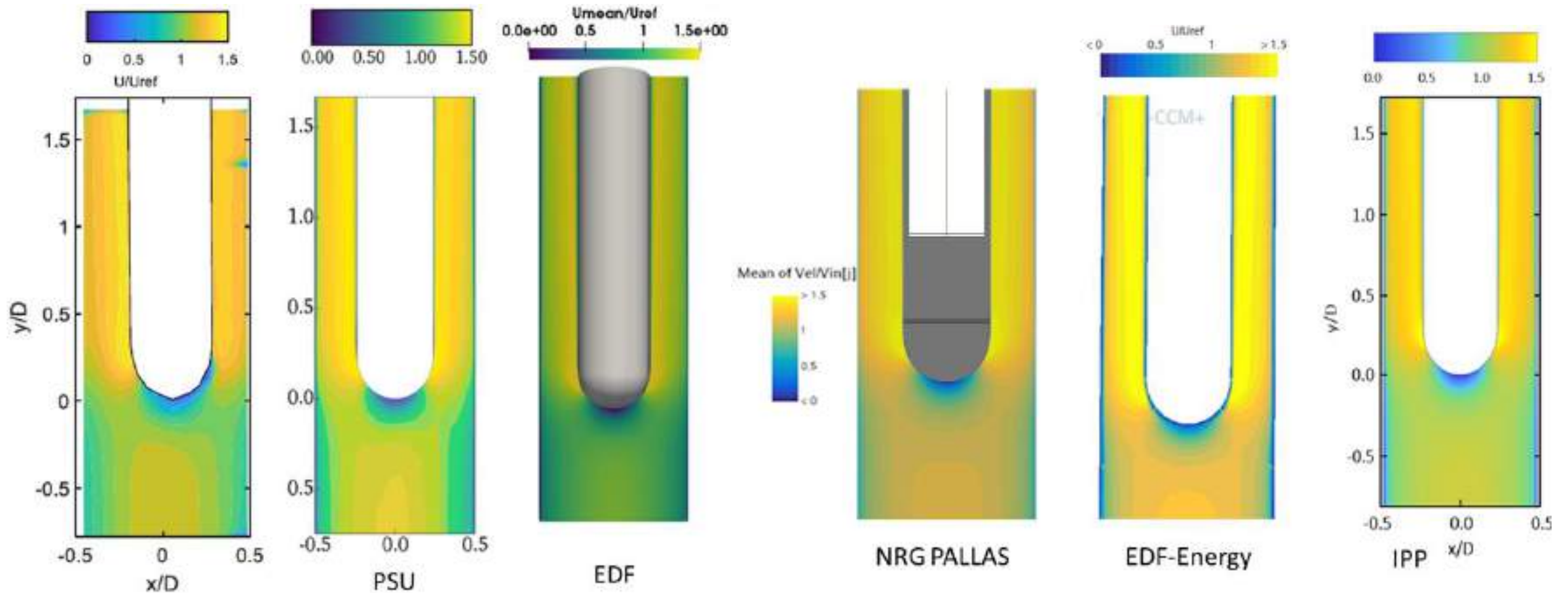


Institution	Wall Clock Time	Equivalent CPU/Node hours	RAM	No. of (CPU/GPU) Cores	Other Provided Data
PSU	≈15d	≈25E3 node h	70 TB	560 GPUs	Averaging runs storage space = 234 TB Q-criterion video storage space = 124 TB Number of nodes used = 70 Each node has one 64-core AMD EPYC CPU and four AMD MI250X GPUs, which are treated as eight GPUs (or GCDs) by the system Approximate costs per time step: RHS assembly (makef) = 34% Velocity Solve = 9% Pressure Solve 40% Execute step (averaging)/TKE budgets = 5-10%
EDF	92d	6.96E6 CPU h	16.875 TB	3150 CPU cores	Number of nodes used = 90
NRG PALLAS	145d9h20m	3.49E5 CPU h	256 GB	100 CPU cores	Number of nodes used = 1
EDF-Energy	14d	4E4 CPU h	480 GB	120 CPU cores	Number of nodes used = 3
IPP	25h9m	5.037E2 CPU h	11.08 GB	20 CPU cores	Number of nodes used = 1

Results: Qualitative



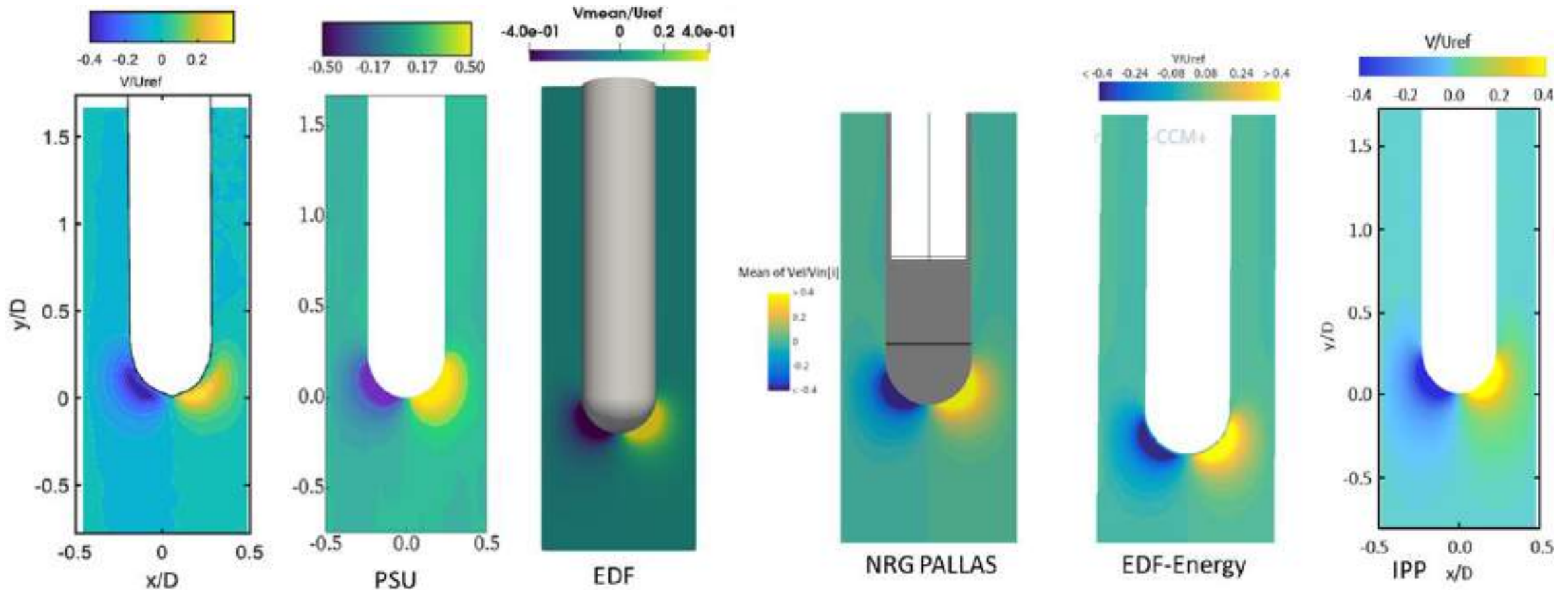
- Axial Component of Velocity



Results: Qualitative



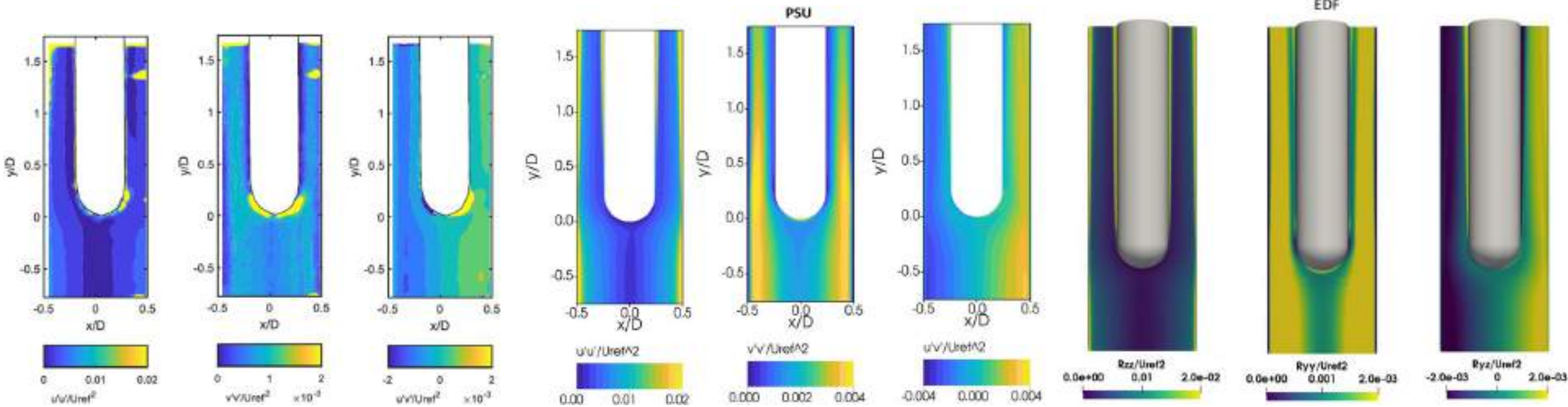
- Radial Component of Velocity



Results: Qualitative



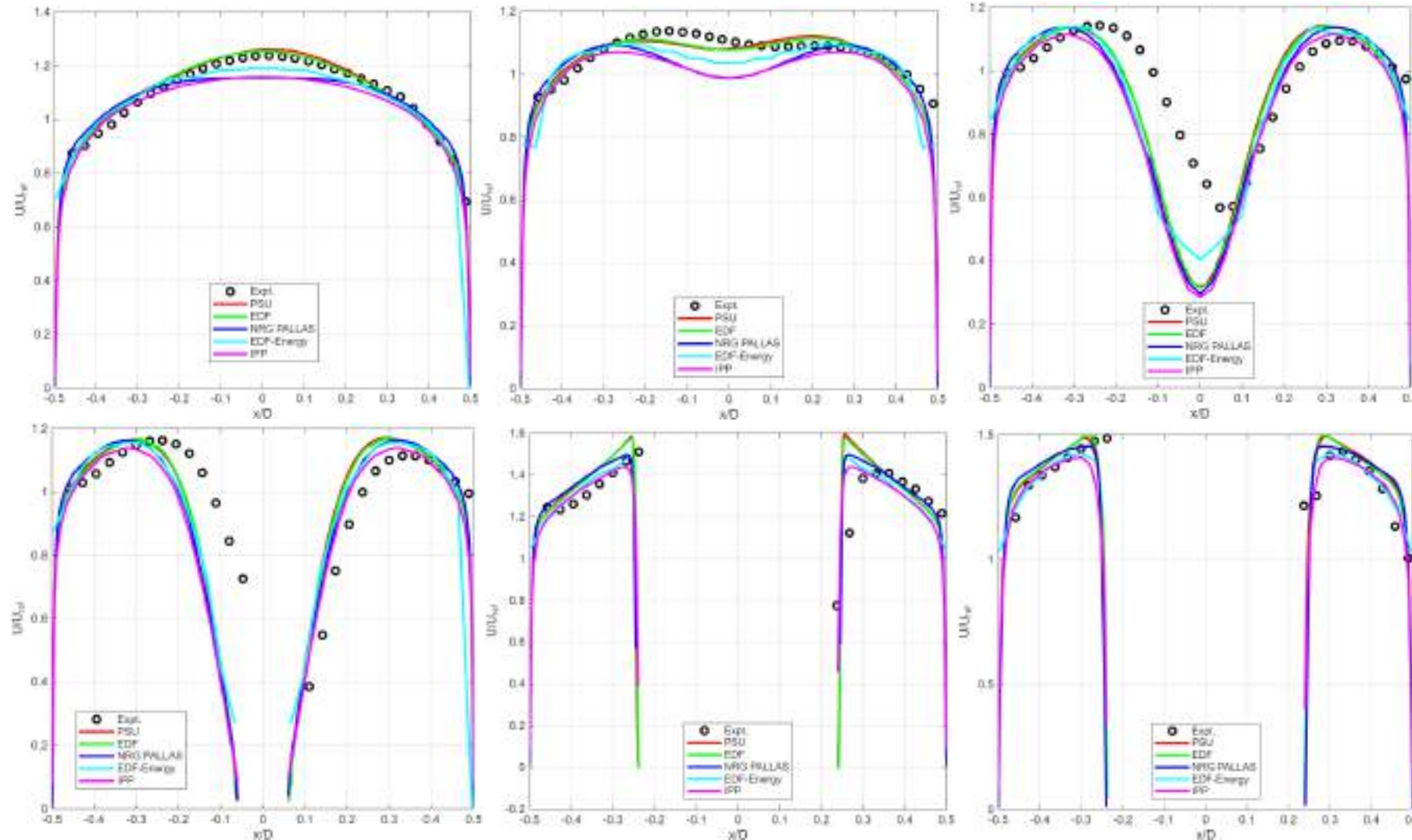
- Stress Components



Results: Quantitative – Horizontal Profiles



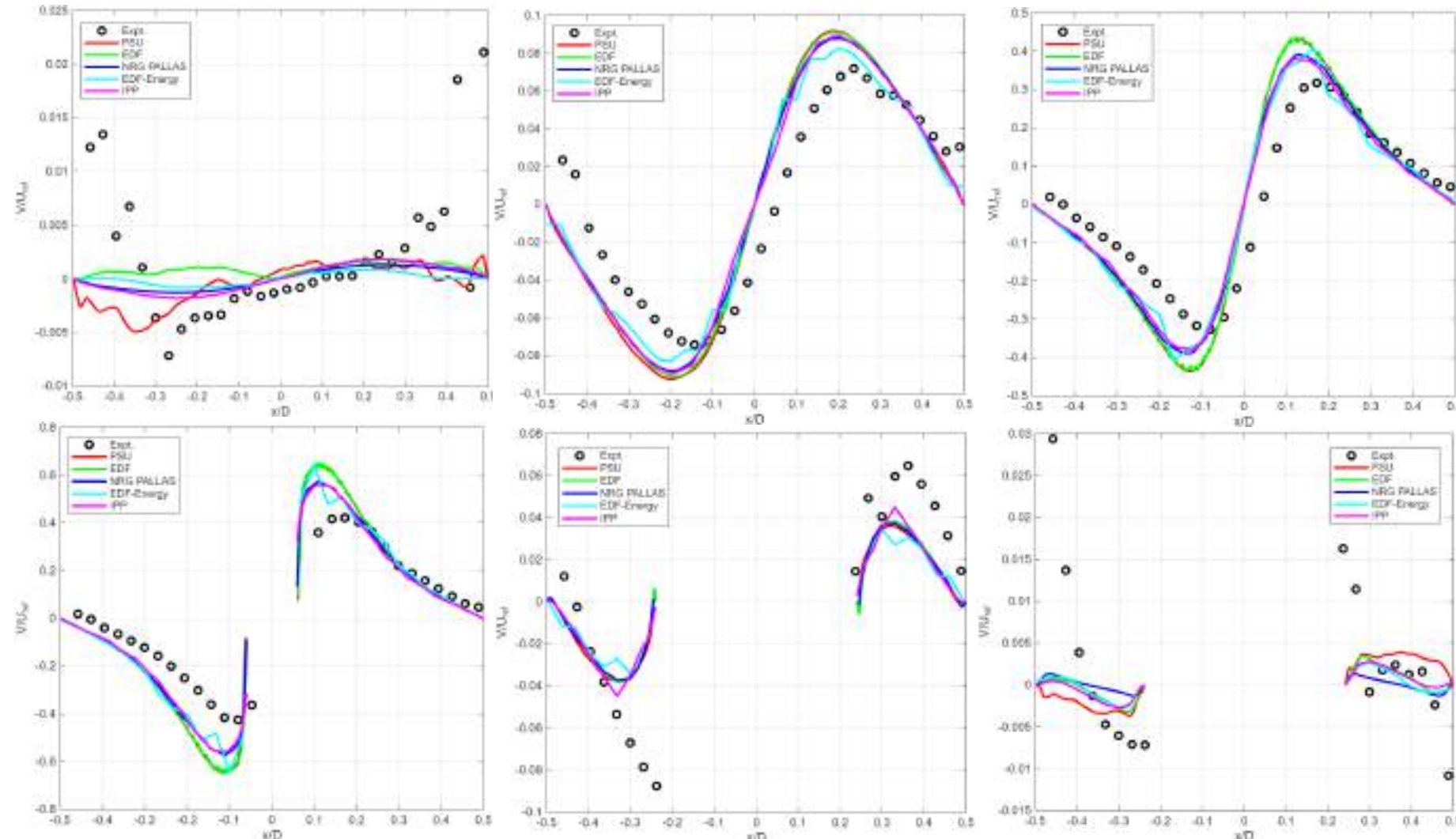
- Axial Component of velocity
- Locations:
 - -15.6 mm
 - -3.8 mm
 - -0.49 mm
 - +0.16 mm
 - +6.7 mm
 - +29.1 mm



Results: Quantitative – Horizontal Profiles



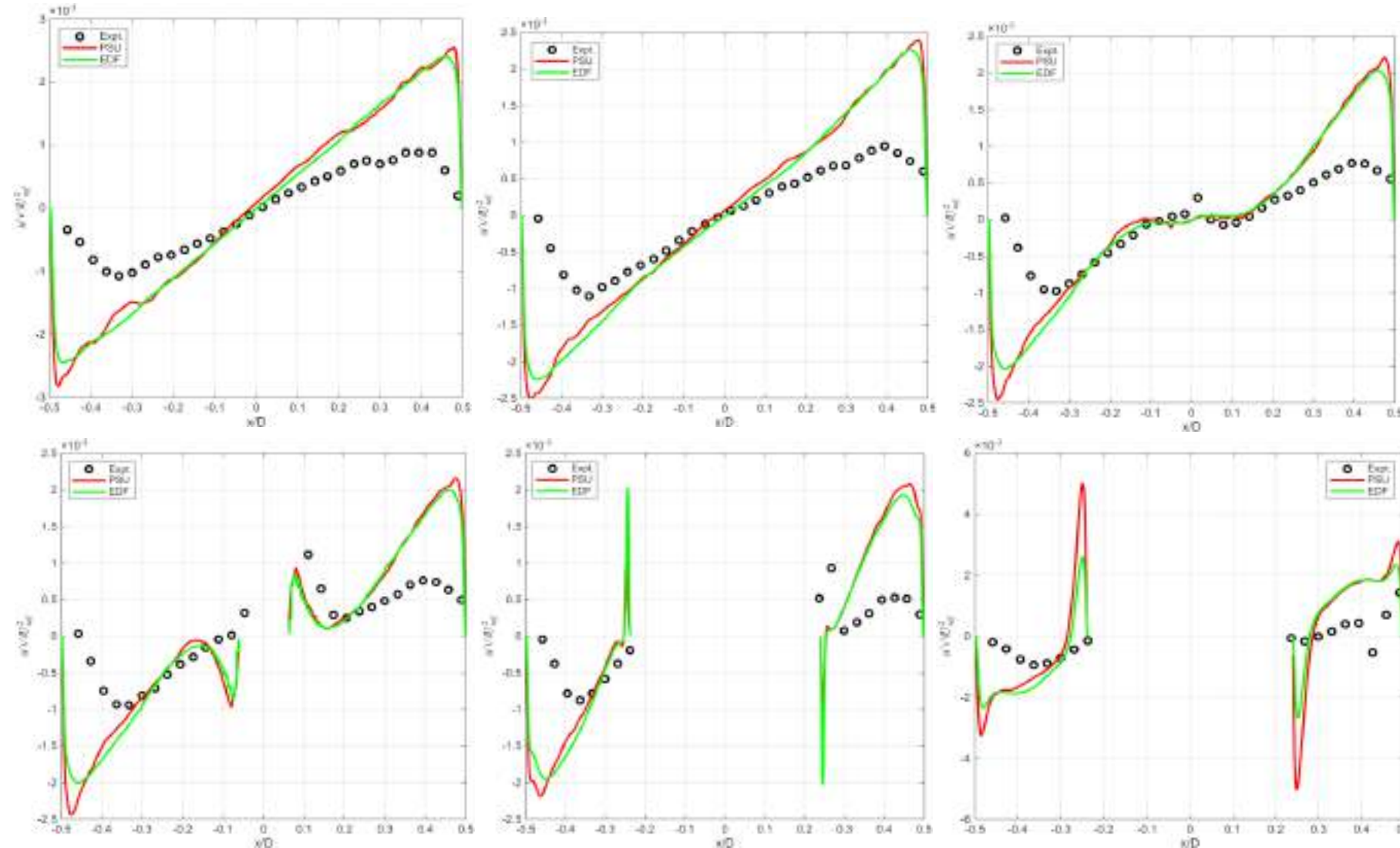
- Radial Component of velocity
- Locations:
 - -15.6 mm
 - -3.8 mm
 - -0.49 mm
 - +0.16 mm
 - +6.7 mm
 - +29.1 mm



Results: Quantitative – Horizontal Profiles



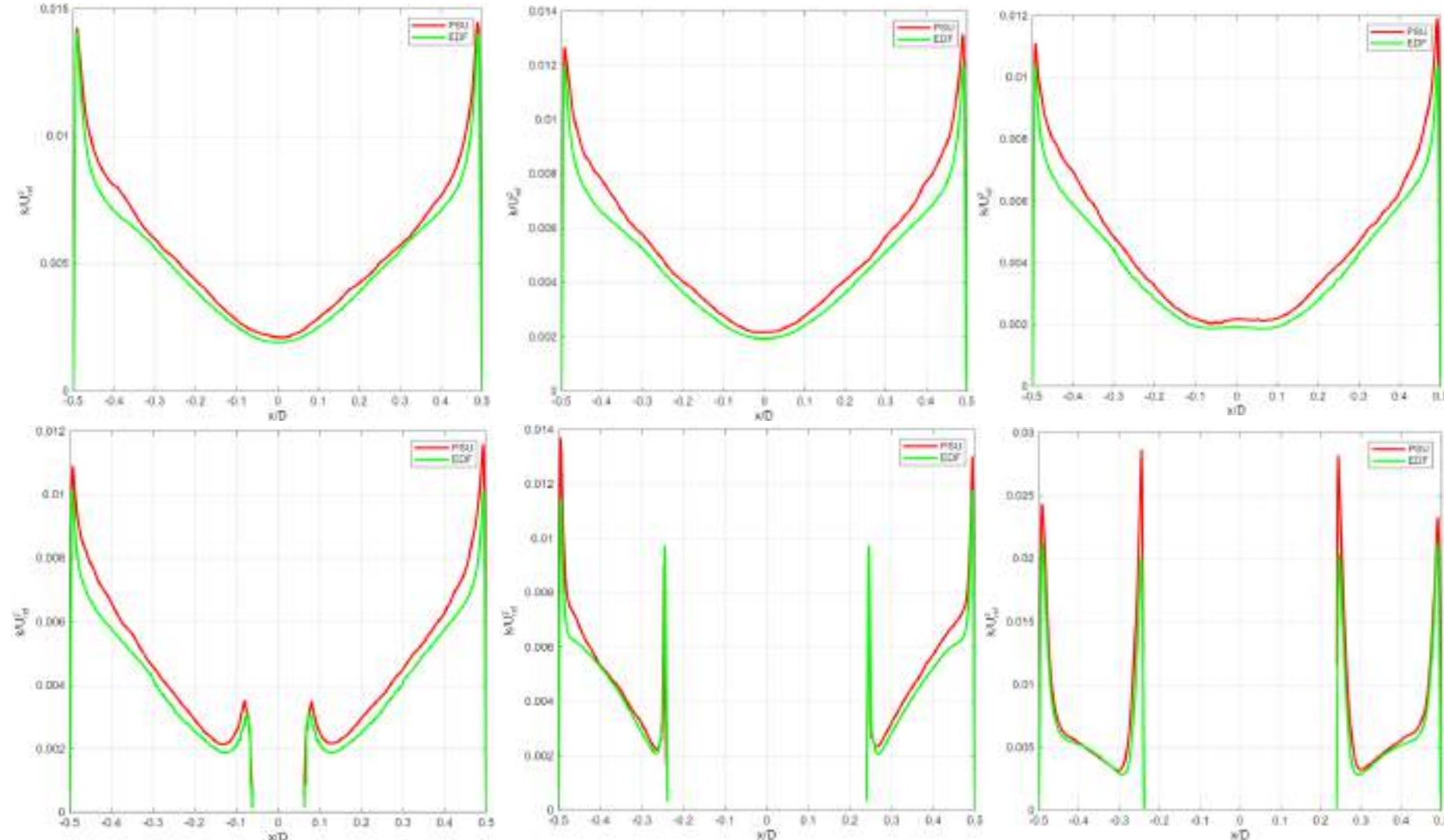
- $U'V'$
- Locations:
 - -15.6 mm
 - -3.8 mm
 - -0.49 mm
 - +0.16 mm
 - +6.7 mm
 - +29.1 mm



Results: Quantitative – Horizontal Profiles



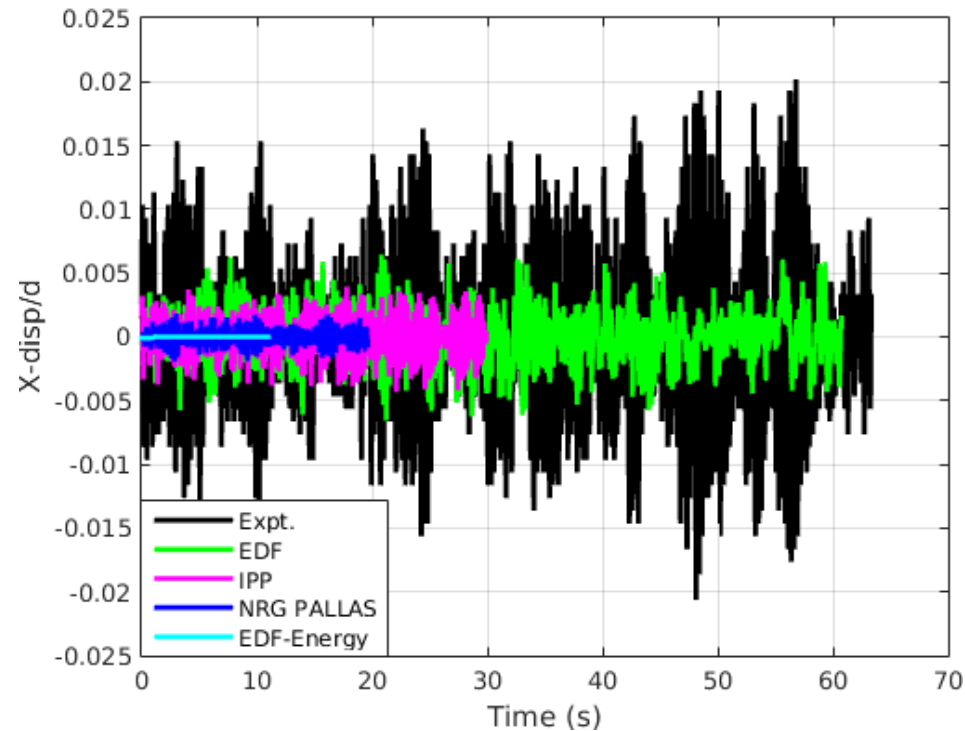
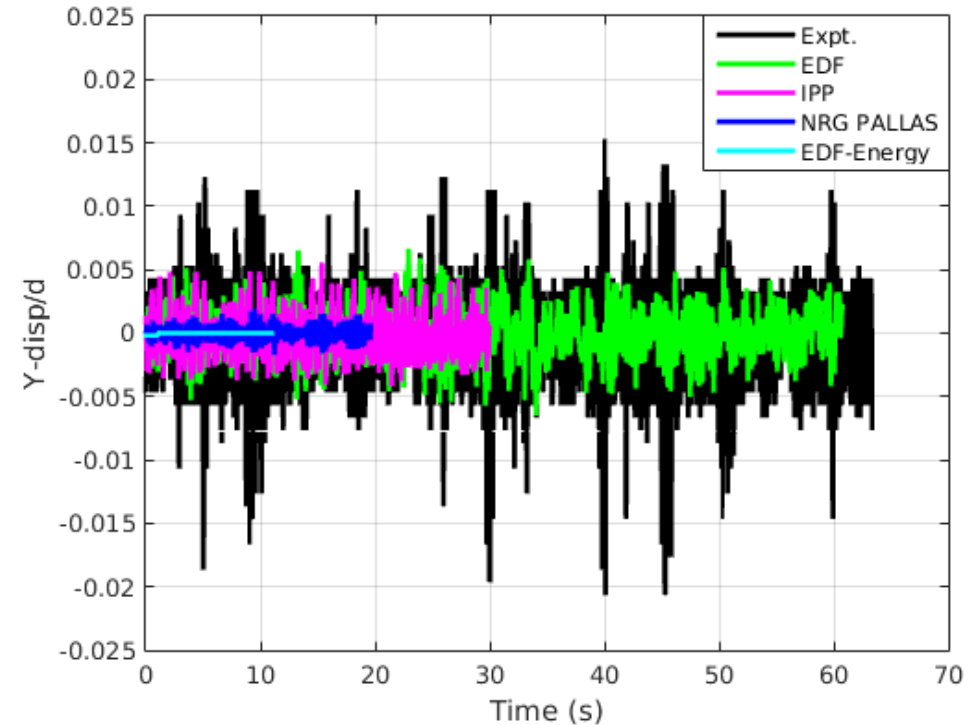
- TKE
- Locations:
 - -15.6 mm
 - -3.8 mm
 - -0.49 mm
 - +0.16 mm
 - +6.7 mm
 - +29.1 mm



Results: Vibration

- Vibration Amplitudes

Institution	RMS X-Disp. (μm)	RMS Y-Disp. (μm)	RMS Polar Disp. (μm)
EDF	20.4478	19.2663	28.0945
NRG PALLAS	6.1213	5.4553	8.1994
EDF-Energy	0.0073	0.0044	0.0085
IPP	16.1641	17.9912	24.1860
Experiment	64.7558	48.138	80.6882

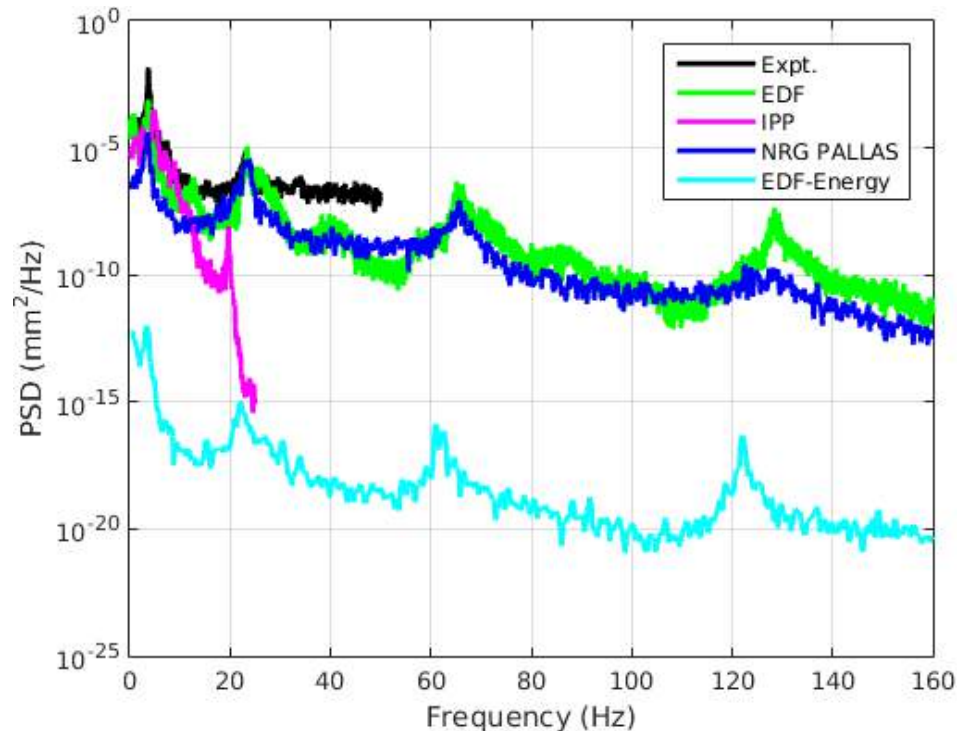
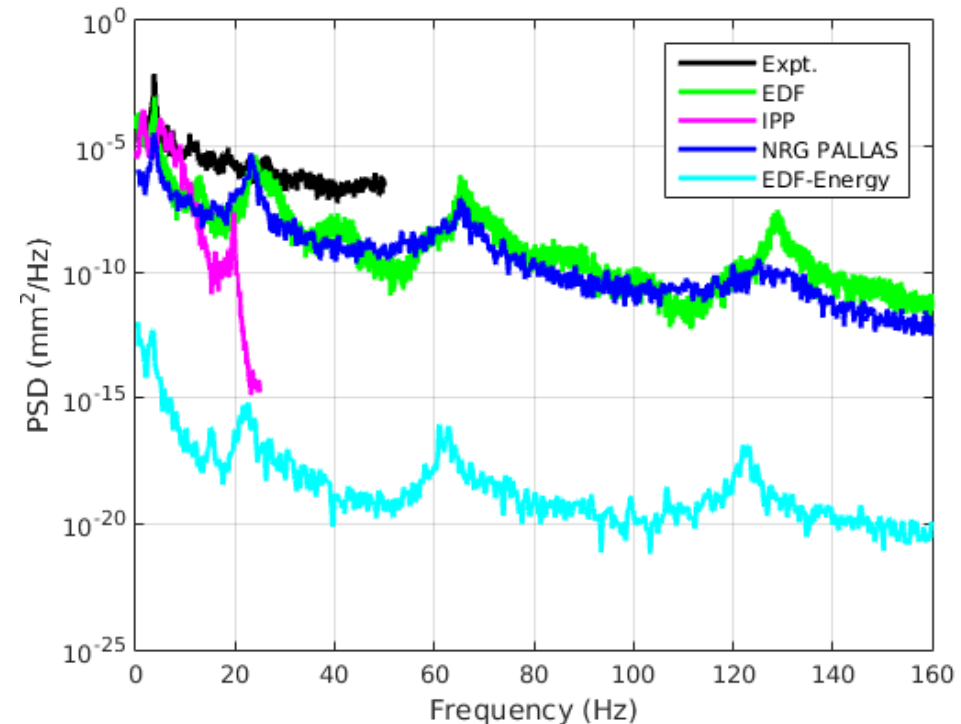


Results: Vibration



- Vibration Frequency

Approach	Front (Hz)	Side (Hz)	Average (Hz)
EDF	3.70	3.80	3.75
NRG PALLAS	3.64	3.71	3.68
EDF-Energy	3.50	3.50	3.50
IPP	5.00	1.50	3.25
Experiment	3.76	3.71	3.73
Analytical	3.71	3.71	3.71



Conclusions

- Most influential choice on the results is the turbulence model
- Above overshadows any gains (if any) obtained through
 - Using Full CSM over Beam Model
 - Including roughness of Rod
 - 2-outlet configuration over 1-outlet configuration
- Concerning meshing strategy – wall resolved required for URANS to predict any vibrations
- A good practice followed by all FSI participants is to validate involved material properties and meshes using the vibration in air and water tests

Partners



Associated Partners





GO-VIKING

Thank you!



Mohammed Muaaz M. D. Hussain



NRG PALLAS



mohammed-muaaz.hussain@nrgpallas.com

www.go-viking.eu



Funded by the
European Union



High-Fidelity Simulations of Cantilever Beam Experiment

Dr. Elia Merzari, Sean Bistany– PSU



Funded by
the European Union

Simulation Setup



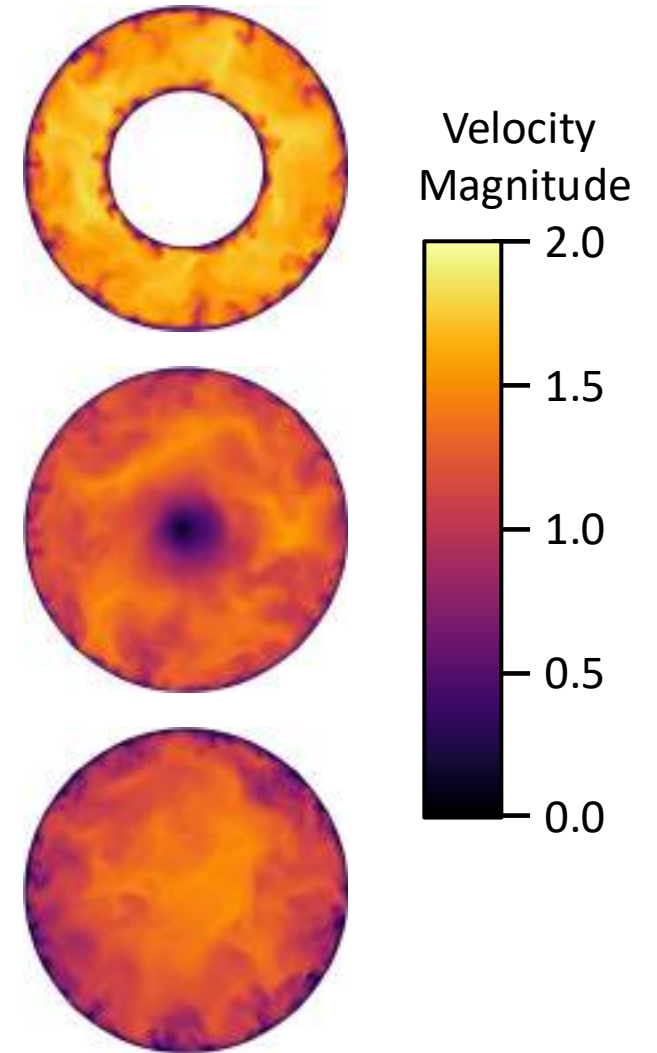
- Annulus Reynolds Number

$Re_a = 21,200$ (also computed at $\sim 11,000$)

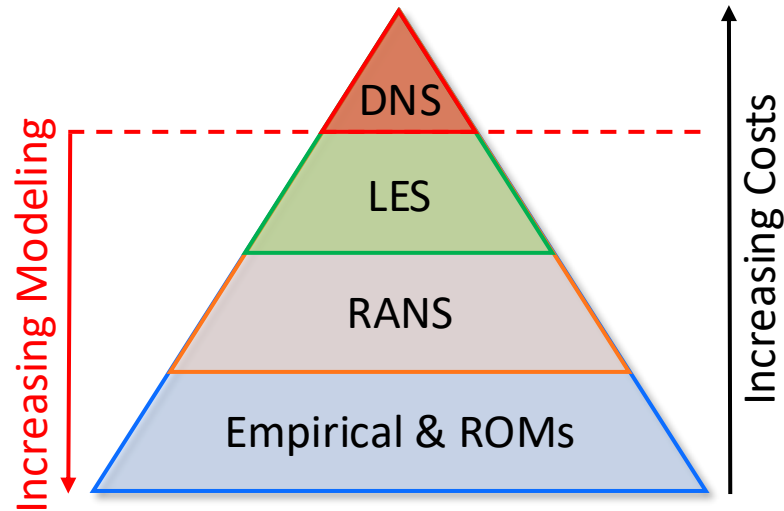
- Nondimensional formulation
- Boundary Conditions
 - Recycling inlet, Dong outflow, & no-slip walls
- Grid resolution characterized by,

$$\eta = \left(\frac{v^3}{\varepsilon} \right)^{1/4}$$

Flow ↑



Computational Costs



Frontier supercomputer at ORNL.

Table 1. Frontier energy consumption – 9,400 compute nodes.

Metric	Value
1 Node Energy Usage per hour	2.62 kWh
1 US Household Energy Usage per hour	1.25 kWh
1 Node Hour Equivalency	2.1 US Houses

TKE Budgets – Pipe Flow Verification

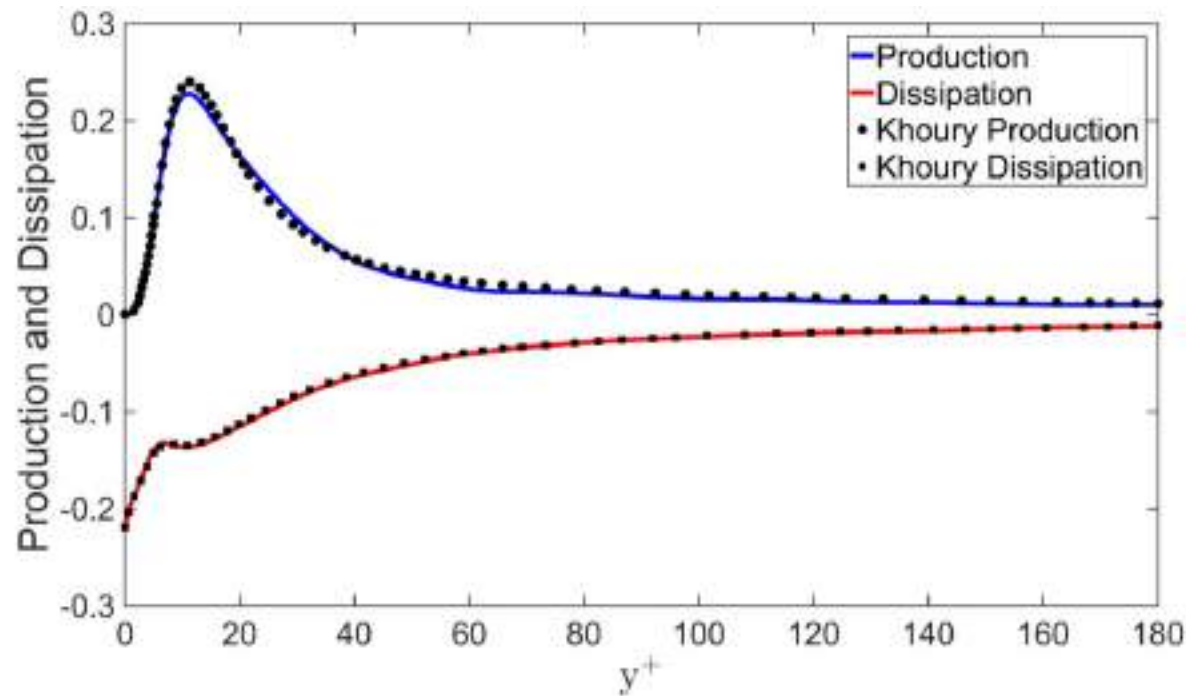
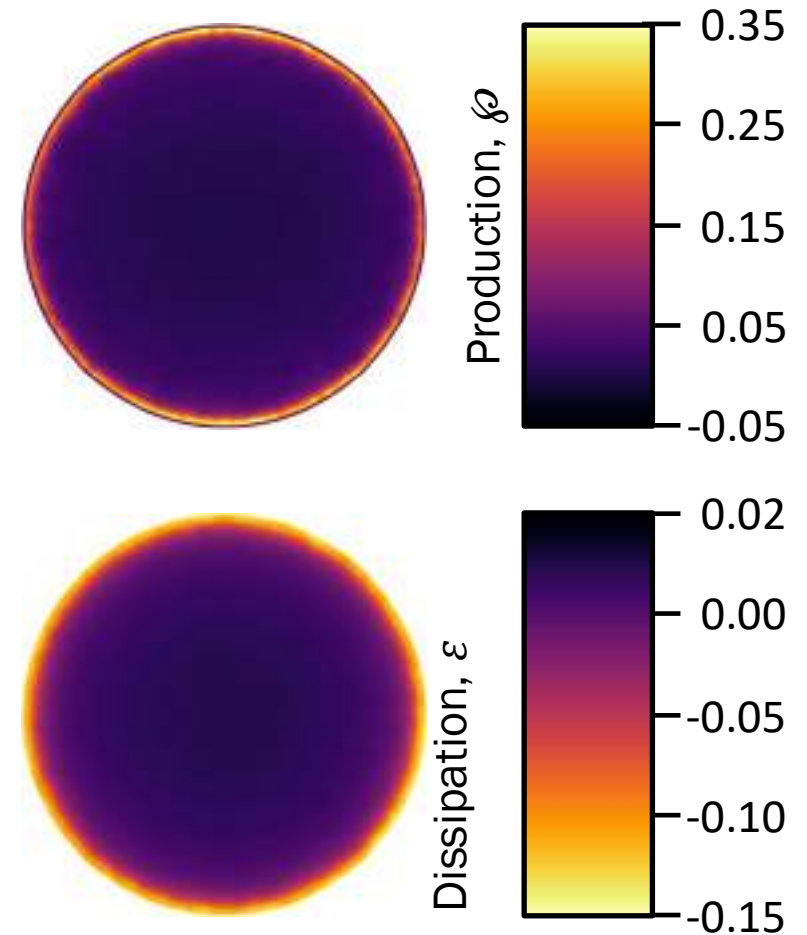
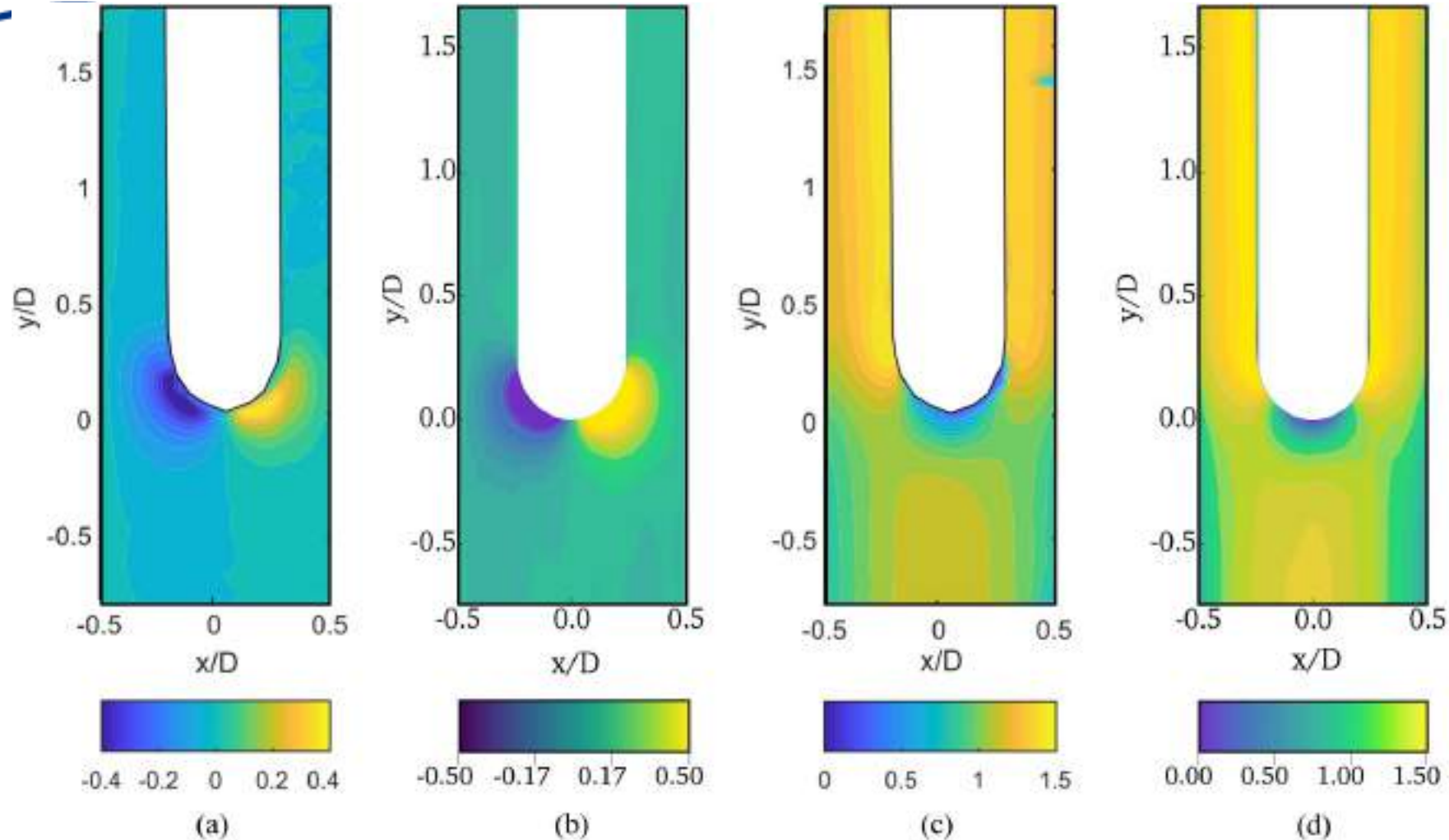


Figure 8. Production & dissipation of TKE normalized by u_τ^4 / ν
Simulation $Re_\tau = 484$ | Khoury $Re_\tau = 550$.

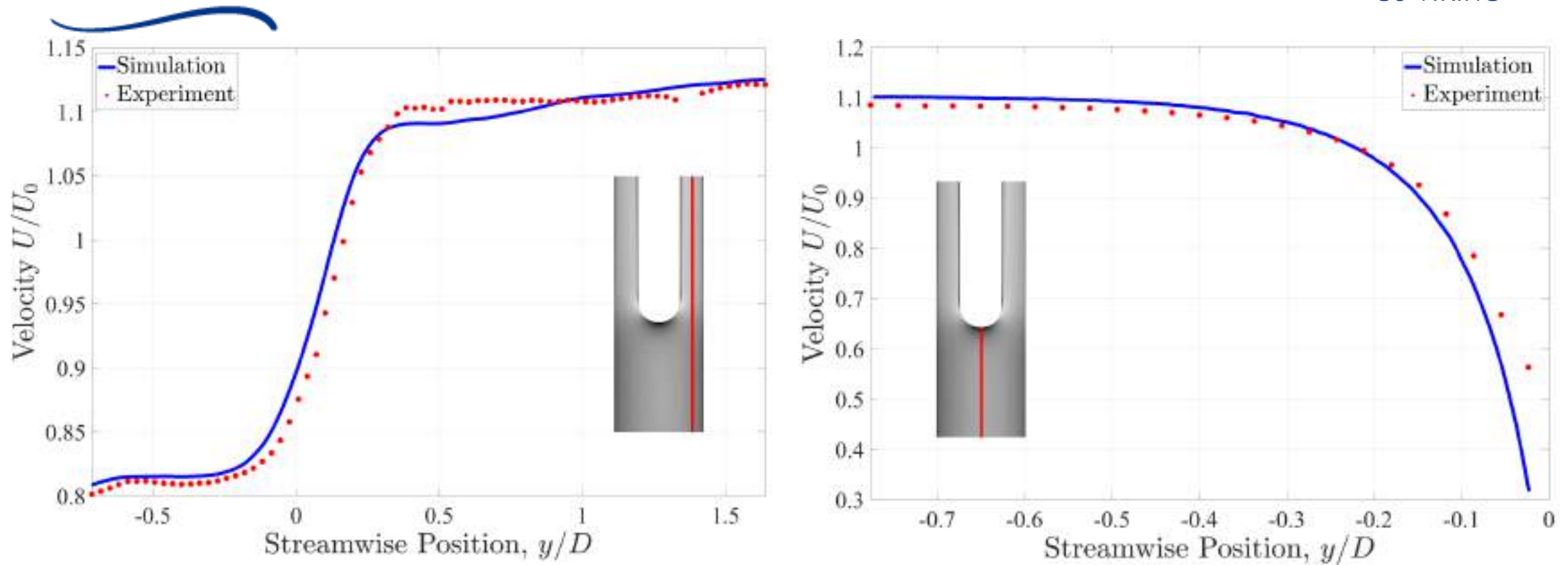


Velocity Flow Fields



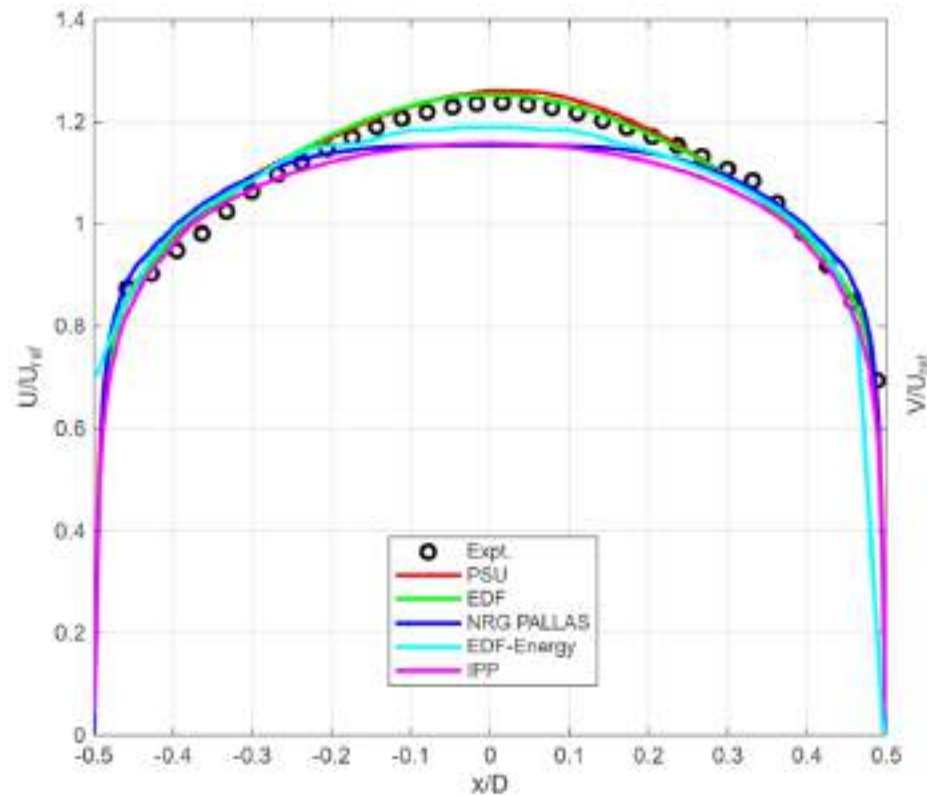
Velocity Flow Fields – (a) cross Exp. (b) cross Sim. (c) stream Exp. and (d) stream Sim.

PIV Comparison

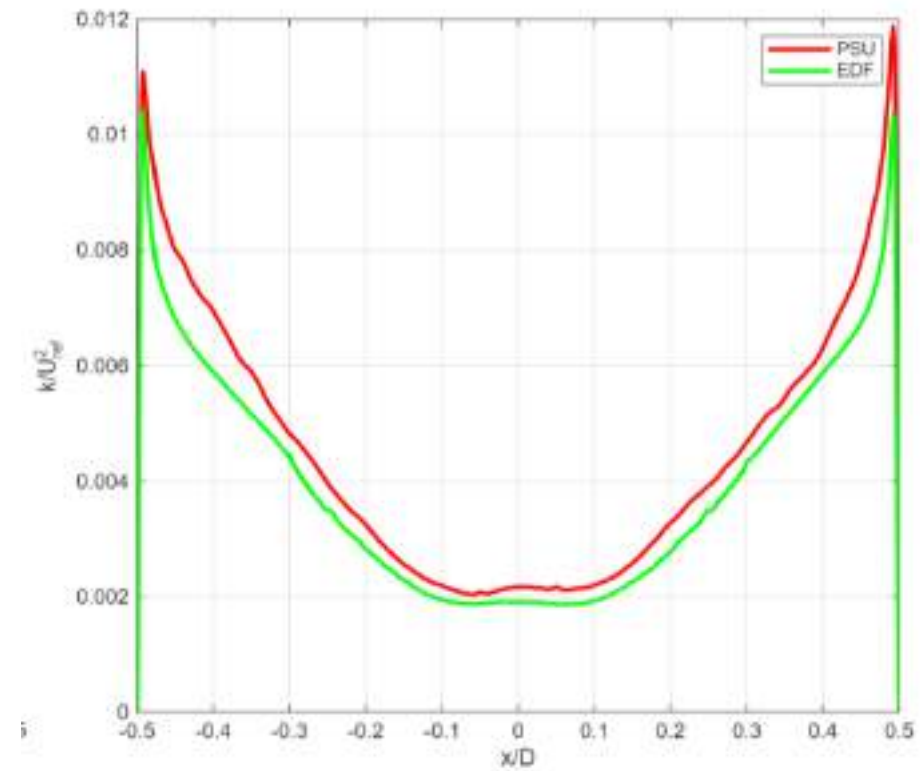


Direct comparison between the average non-dimensional axial velocity of the experiment and simulation

Other comparisons



Velocity profile



TKE

Time Series Data – Locations

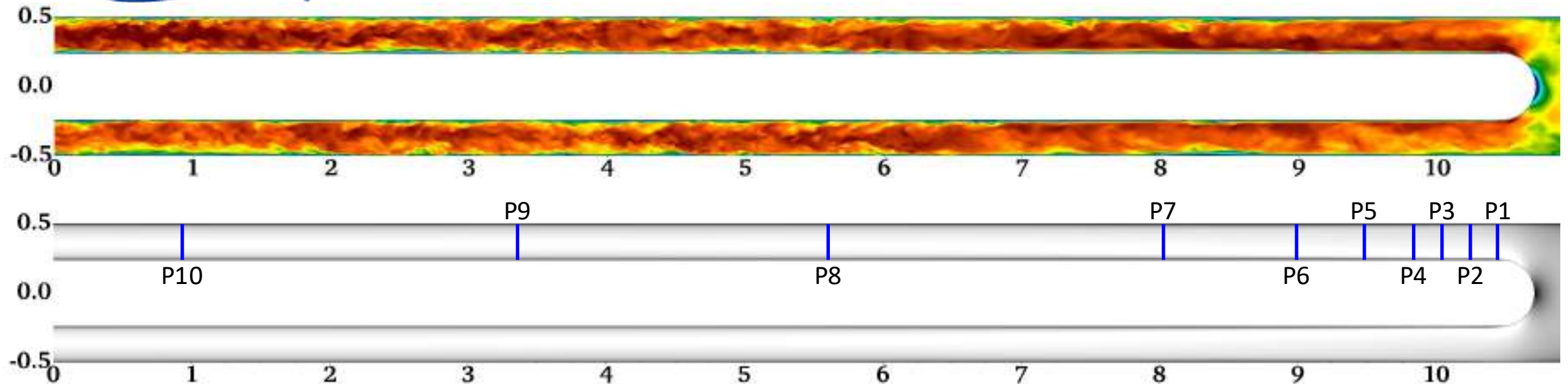
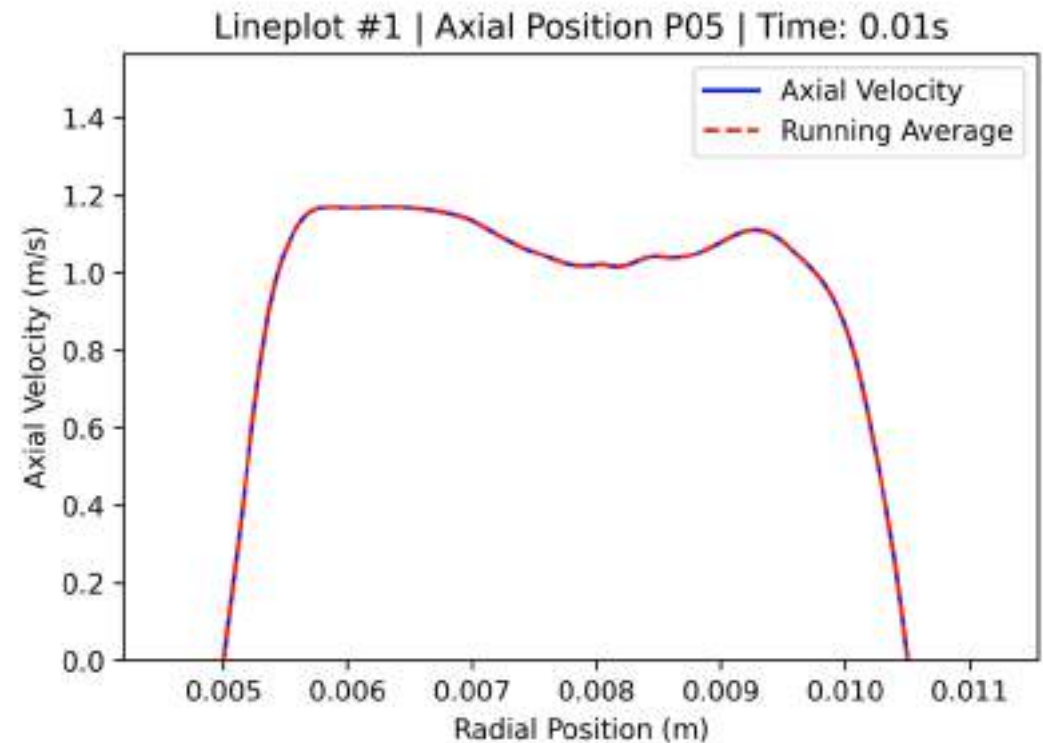


Table. Axial locations of radial lines – dimensional origin is beam tip without semi-spherical end.

Radial Lines	P1	P2	P3	P4	P5	P6	P7	P8	P9	P10
PSU (x/D_{pipe})	10.45	10.29	10.10	9.81	9.52	9.05	8.10	5.71	3.33	0.95
Dimensional (m)	0.0005	0.004	0.008	0.014	0.02	0.03	0.05	0.1	0.15	0.2

Time Series Data – Requested

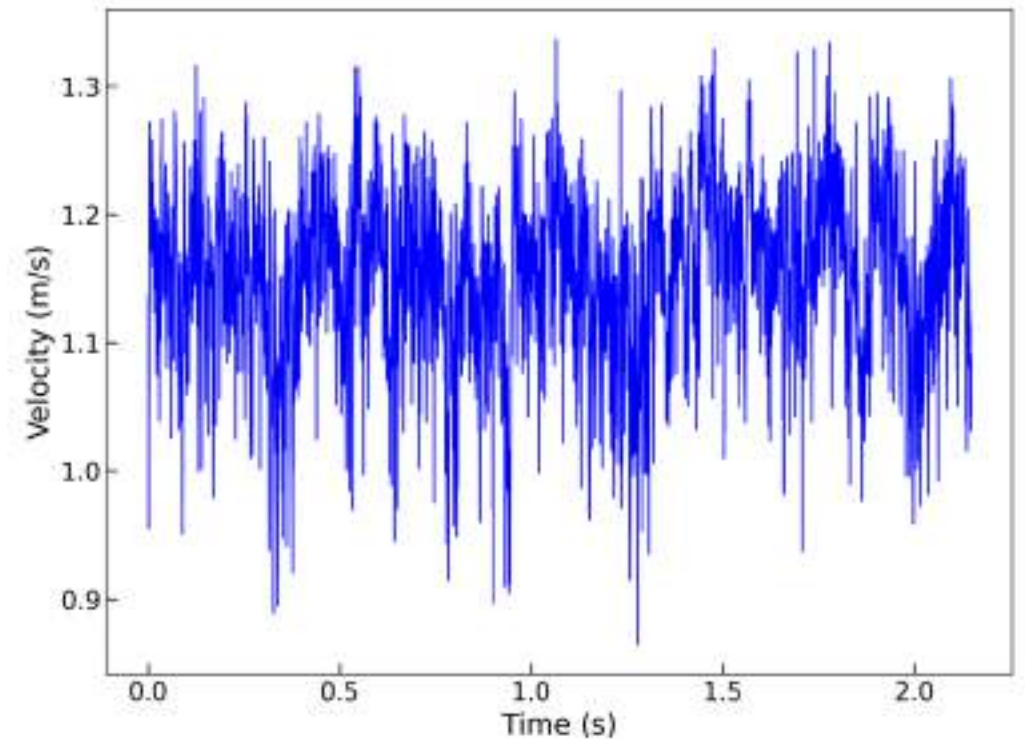
- **Frequency** – 100 Hz
- **Time** – 10 s
- **Mesh**
 - Polynomial Order – 7
 - Grid Points – 1.35 billion



Line plot accumulation of
dimensional time series data.

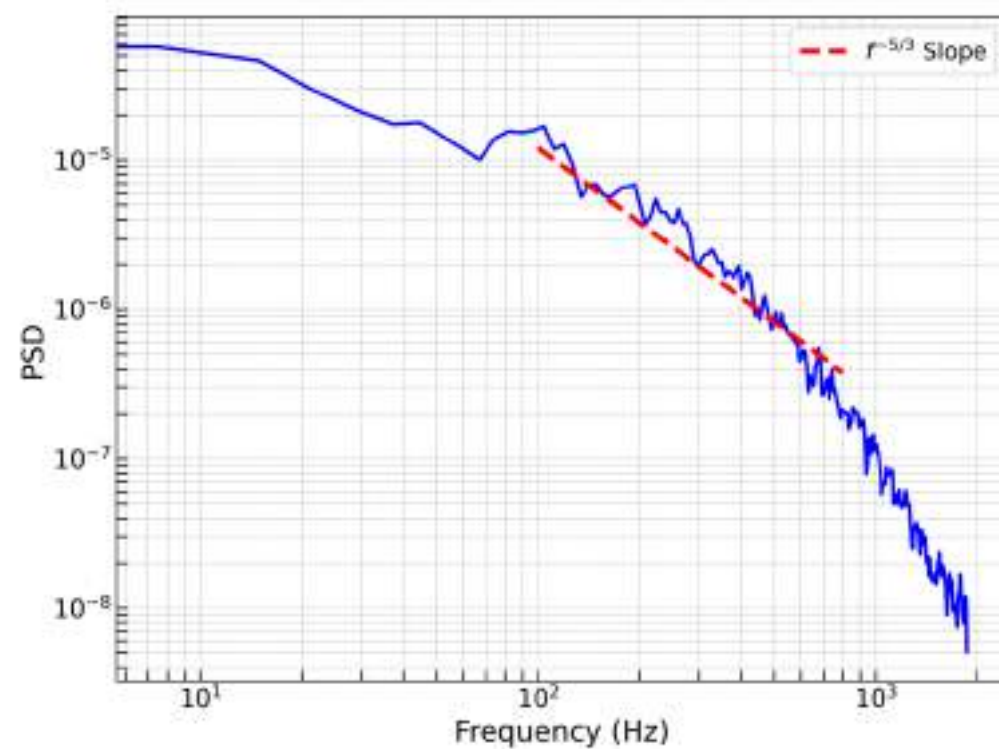
Time Series Data – Additional

- **Frequency** – 3725 Hz
- **Time** – 2.15 s
- **Mesh**
 - Polynomial Order – 7
 - Grid Points – 1.35 billion



Dimensional velocity time series –
axial position P09 & radial position 0.008 m.

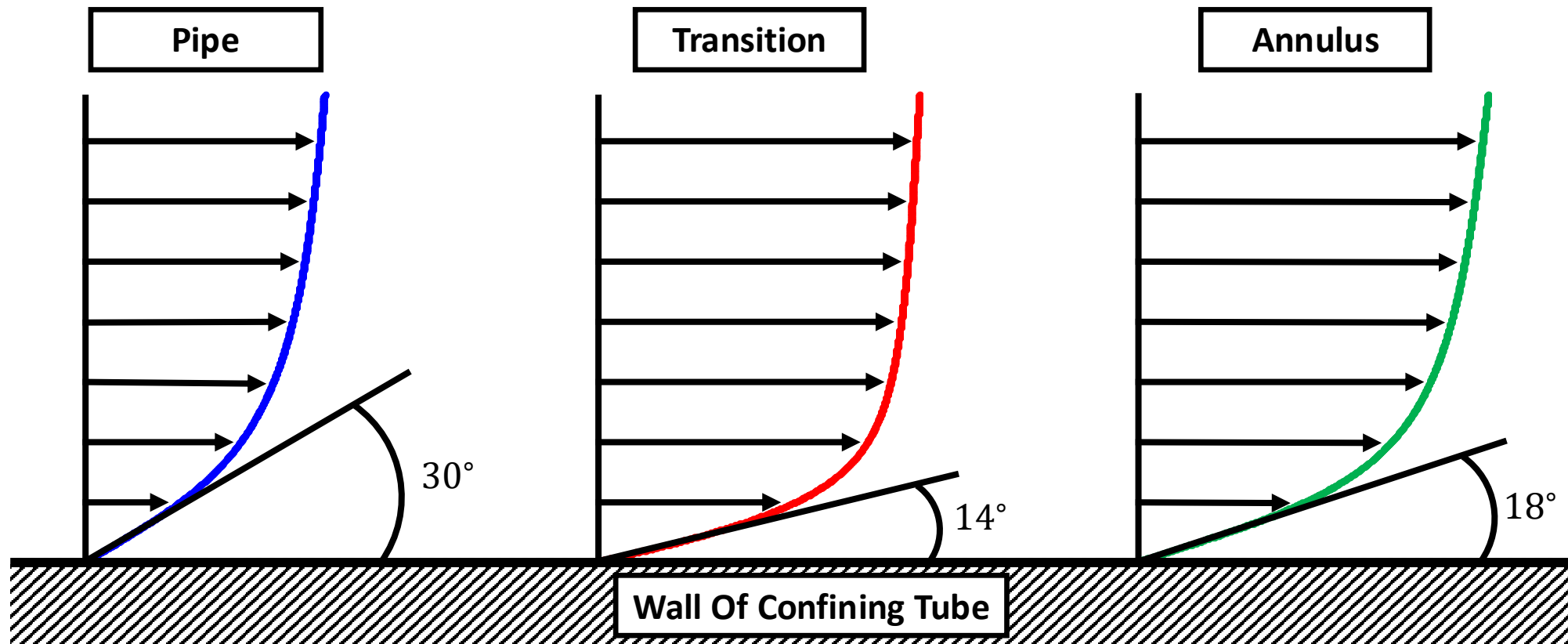
Time Series Data – PSD



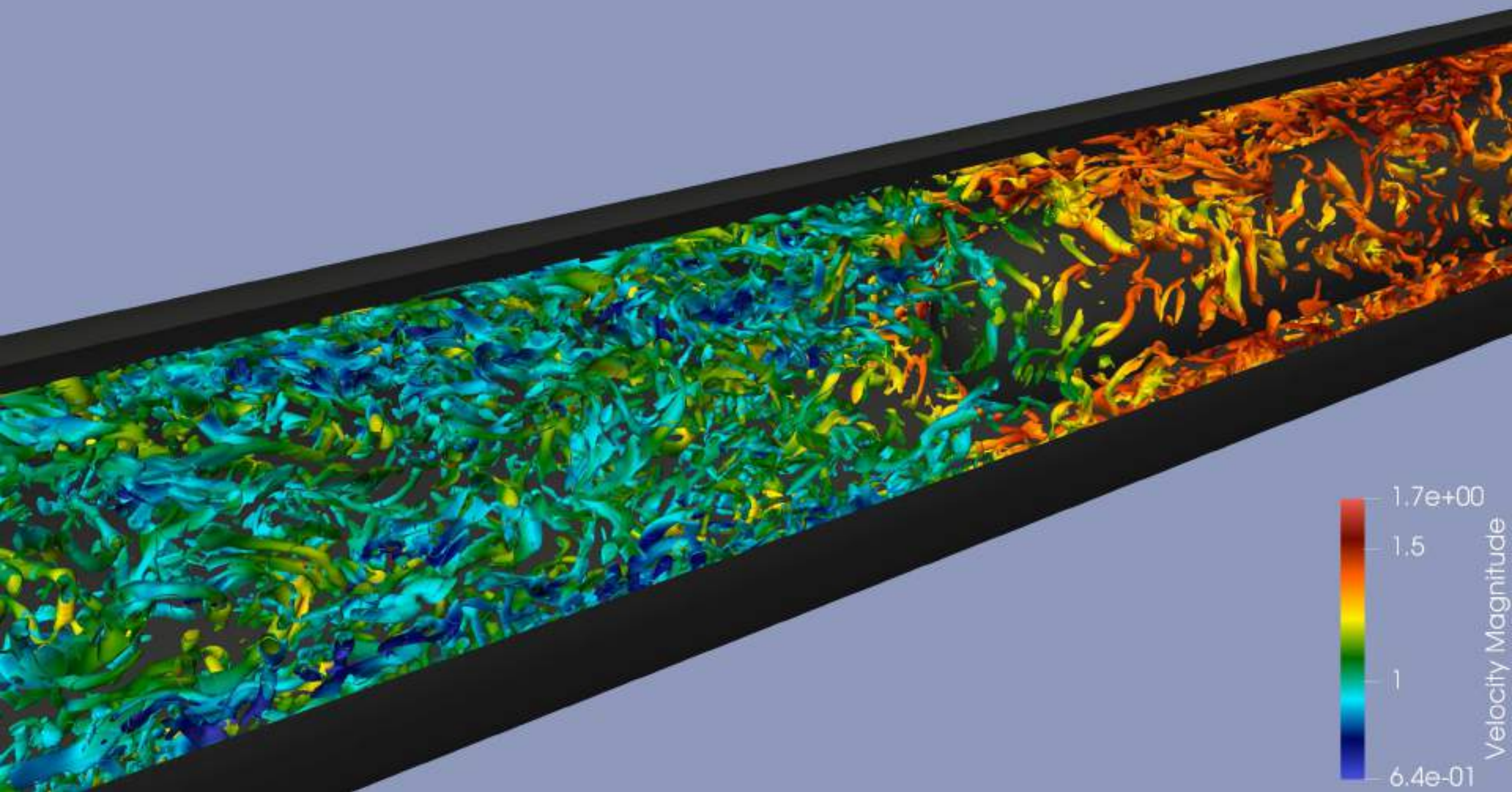
Streamwise Velocity

Power spectral density (PSD) – axial position P09 & radial position 0.008 m.

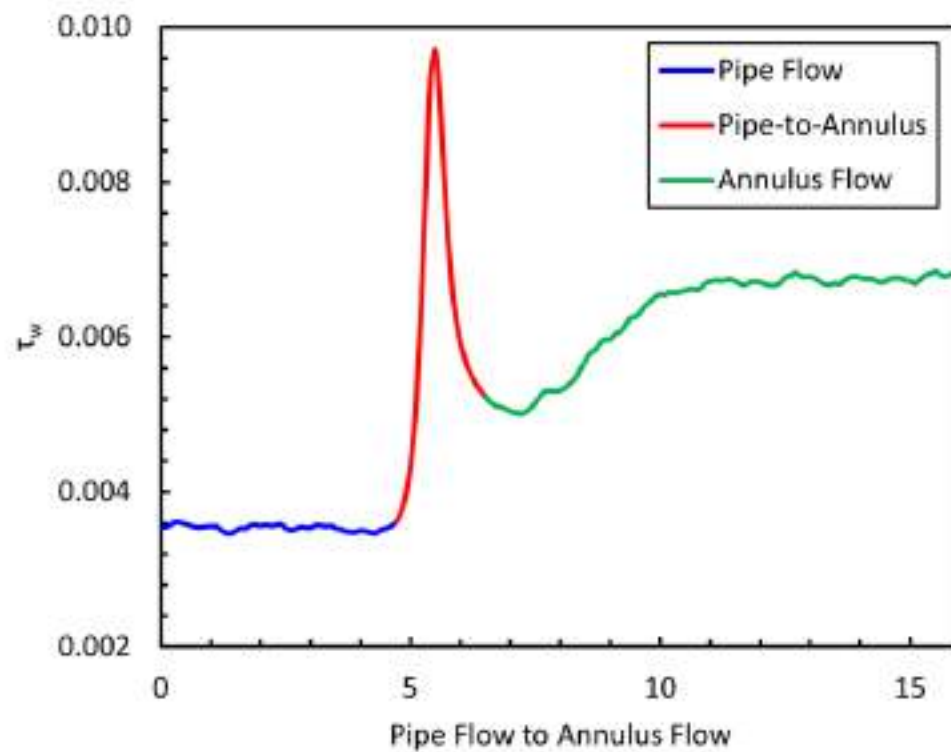
Flow Acceleration



Time averaged velocity profile comparing wall velocity gradients and the expected effects on turbulence.



Friction Reynolds Number



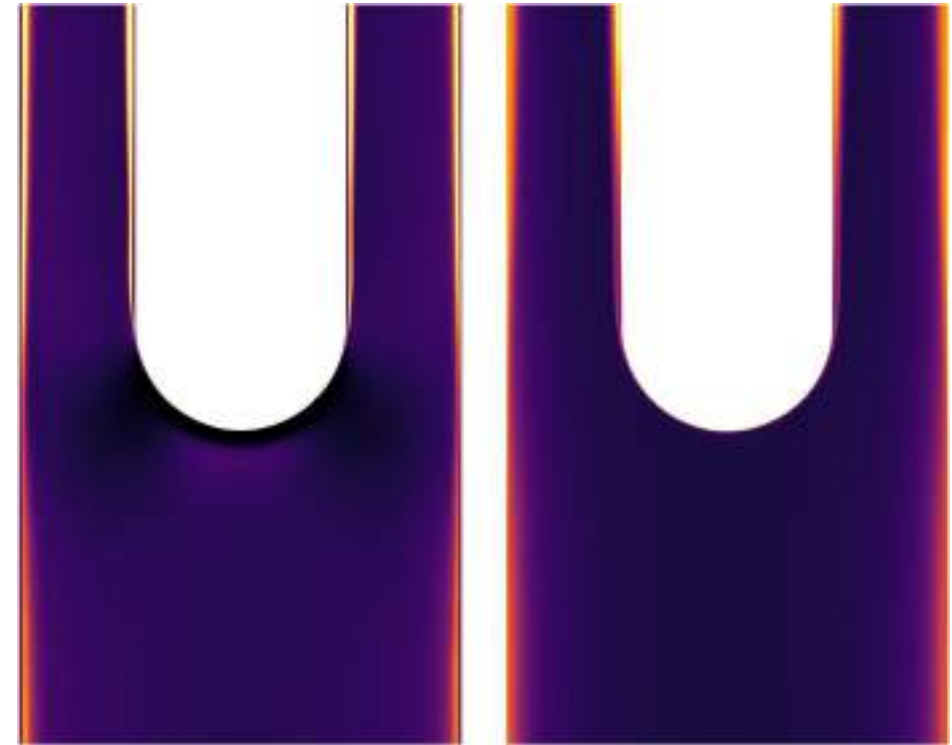
Dimensionless wall shear stress as a function of axial dimensionless position.

Pipe flow analysis – Re_τ

Parameter	Value	Difference
Re_{pipe}	16393	–
Analytical $Re_{\tau,pipe}$	484.57	–
Simulation $Re_{\tau,pipe}$	484.53	0.008%

Budgets of Turbulent Kinetic Energy

- **Flow Throughs** – 22
- **Time Step** – $1.34 \times 10^{-06} \text{ s}$
- **Averaging Time** – 8.5 s
- **Mesh**
 - Polynomial Order – 7
 - Grid Points – 1.35 billion
- **Computational Cost**
 - 20,400 node hours
 - 42,840 US homes

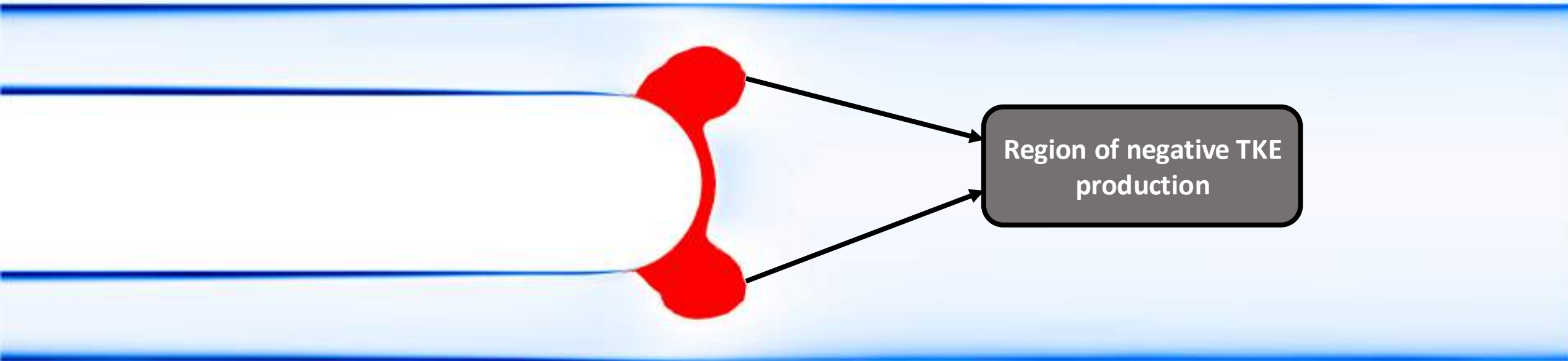


(a) Production

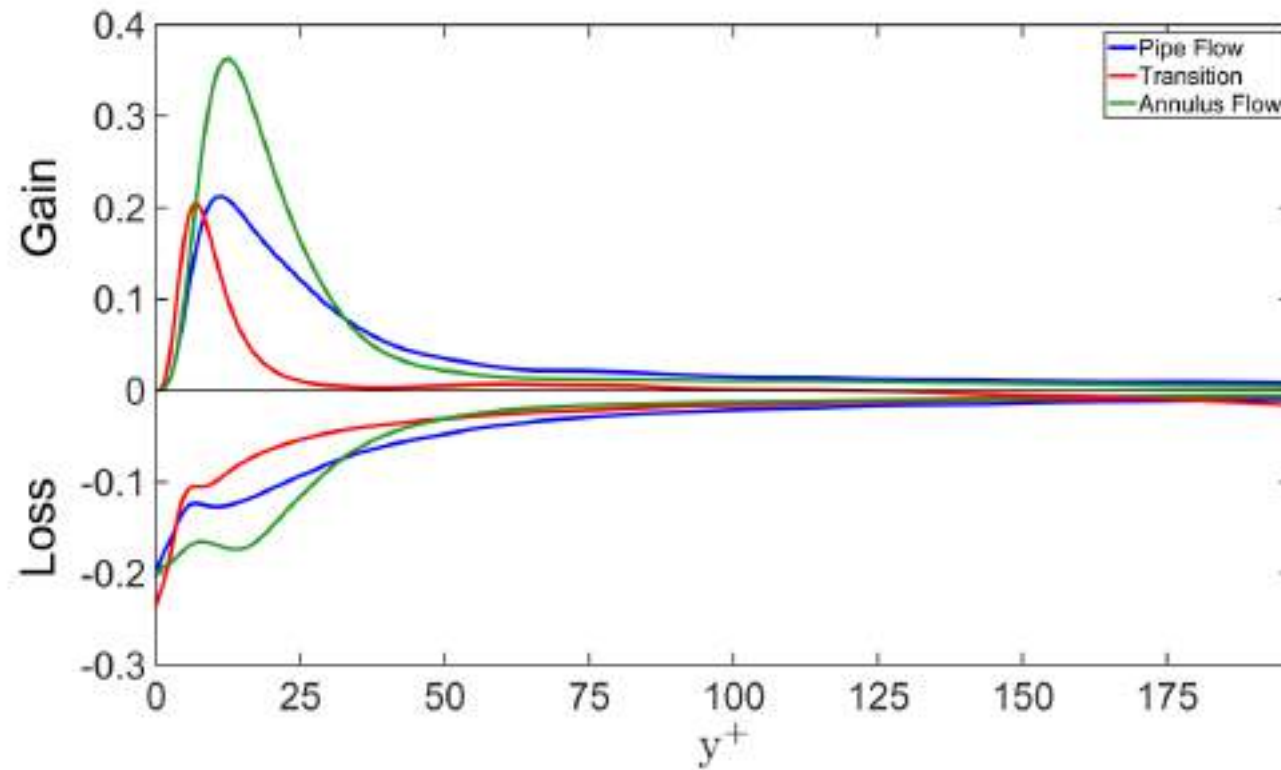
(a) Dissipation

Effect of flow acceleration on TKE.

TKE Budgets – Negative Production

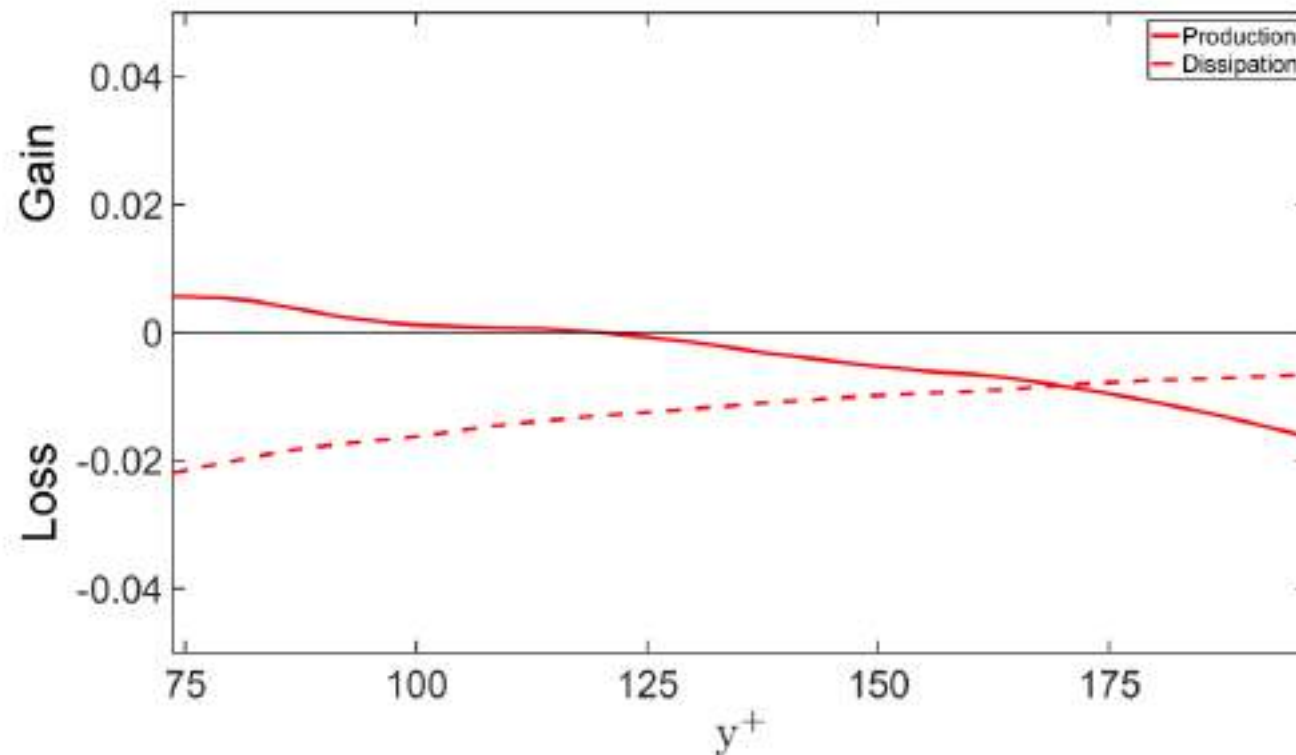


TKE Budgets – Flow Acceleration



Production & dissipation of TKE normalized by u_τ^4/ν
as a function of axial position.

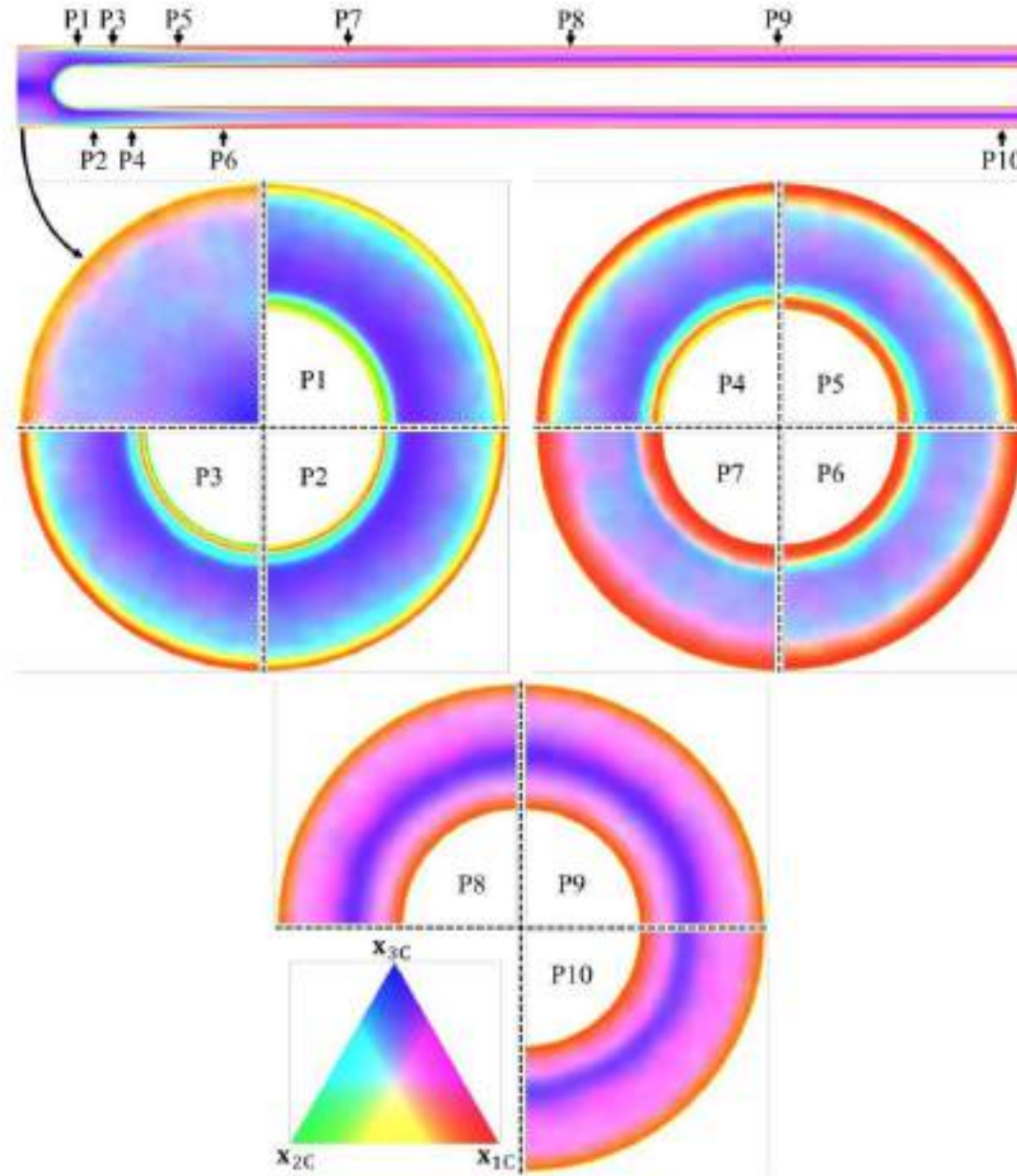
TKE Budgets – Flow Acceleration



Negative production of TKE in the transition region,
flow acceleration leads to loss of TKE.

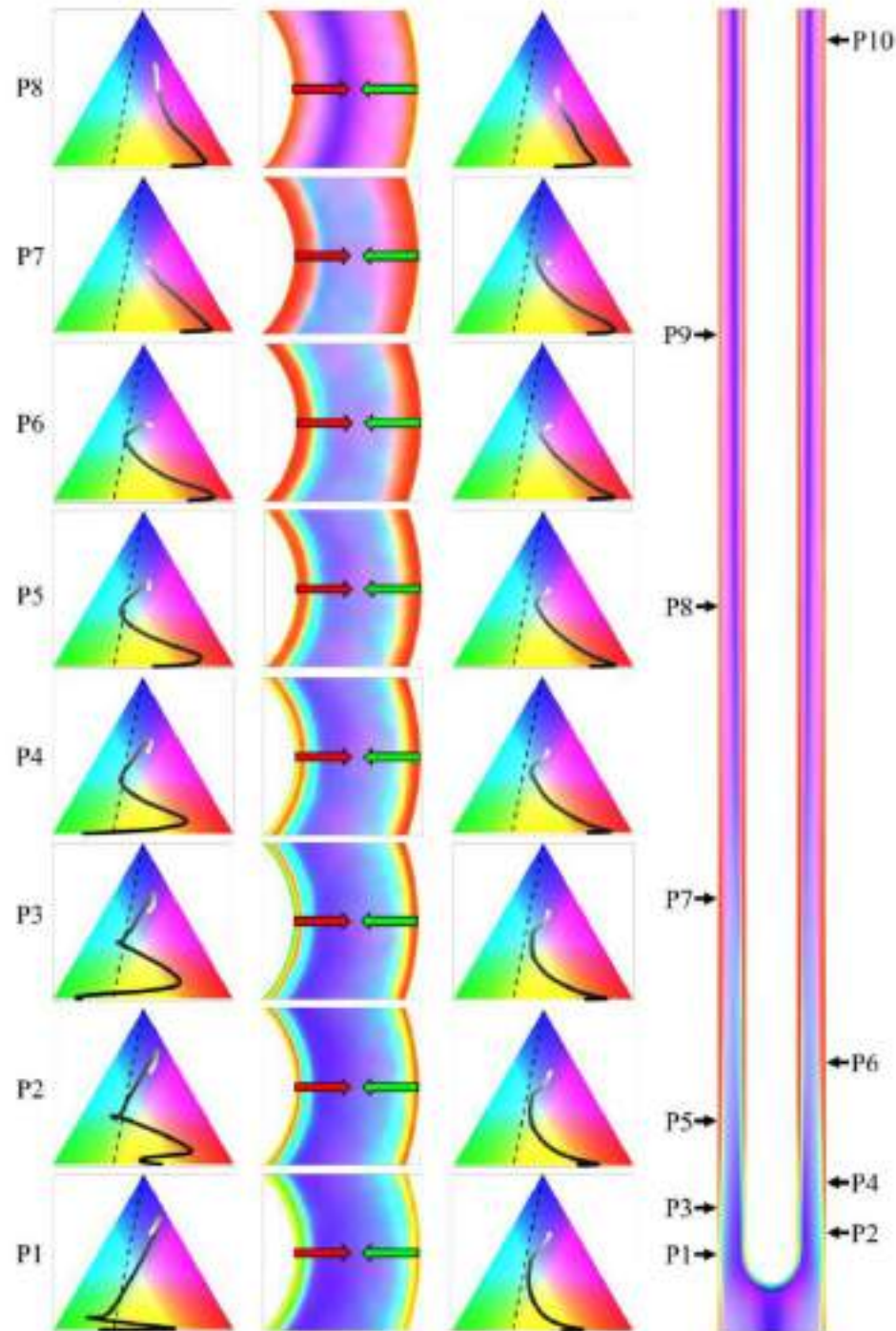
Anisotropy (1/2)

In the transition region we see a more prominent two-component behavior

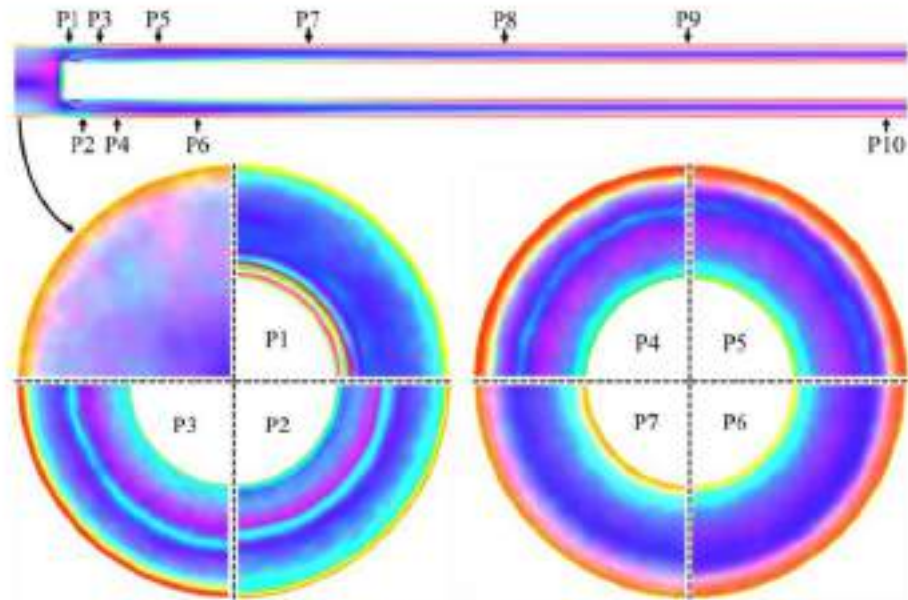
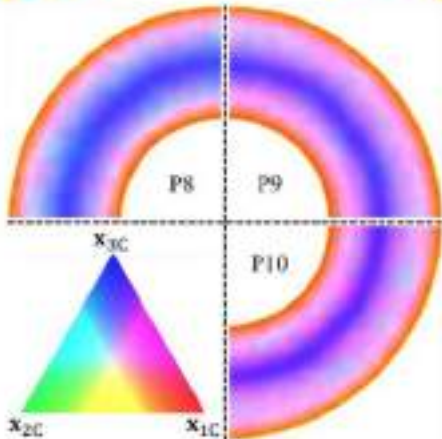
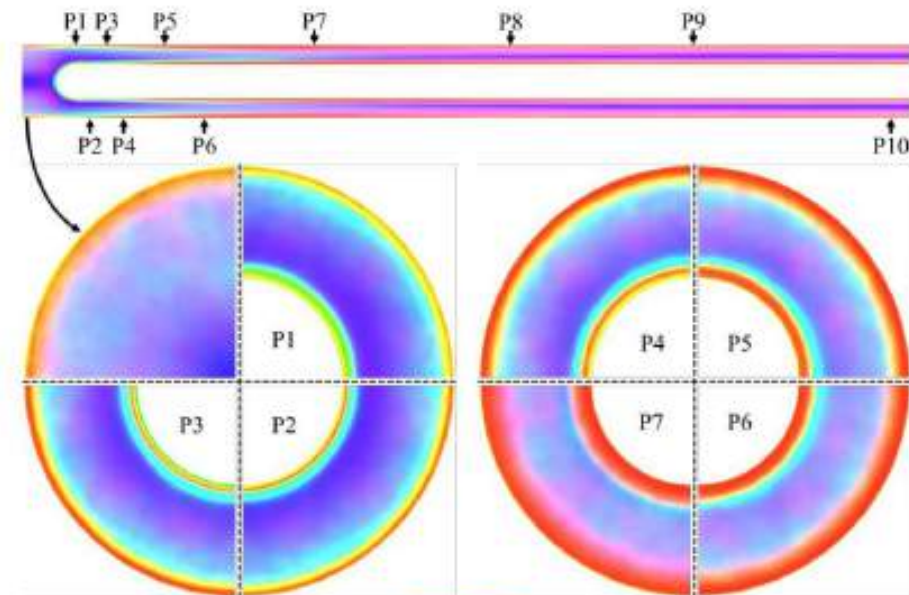
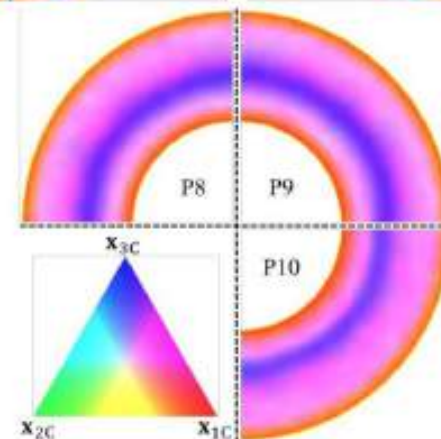


Anisotropy (2/2)

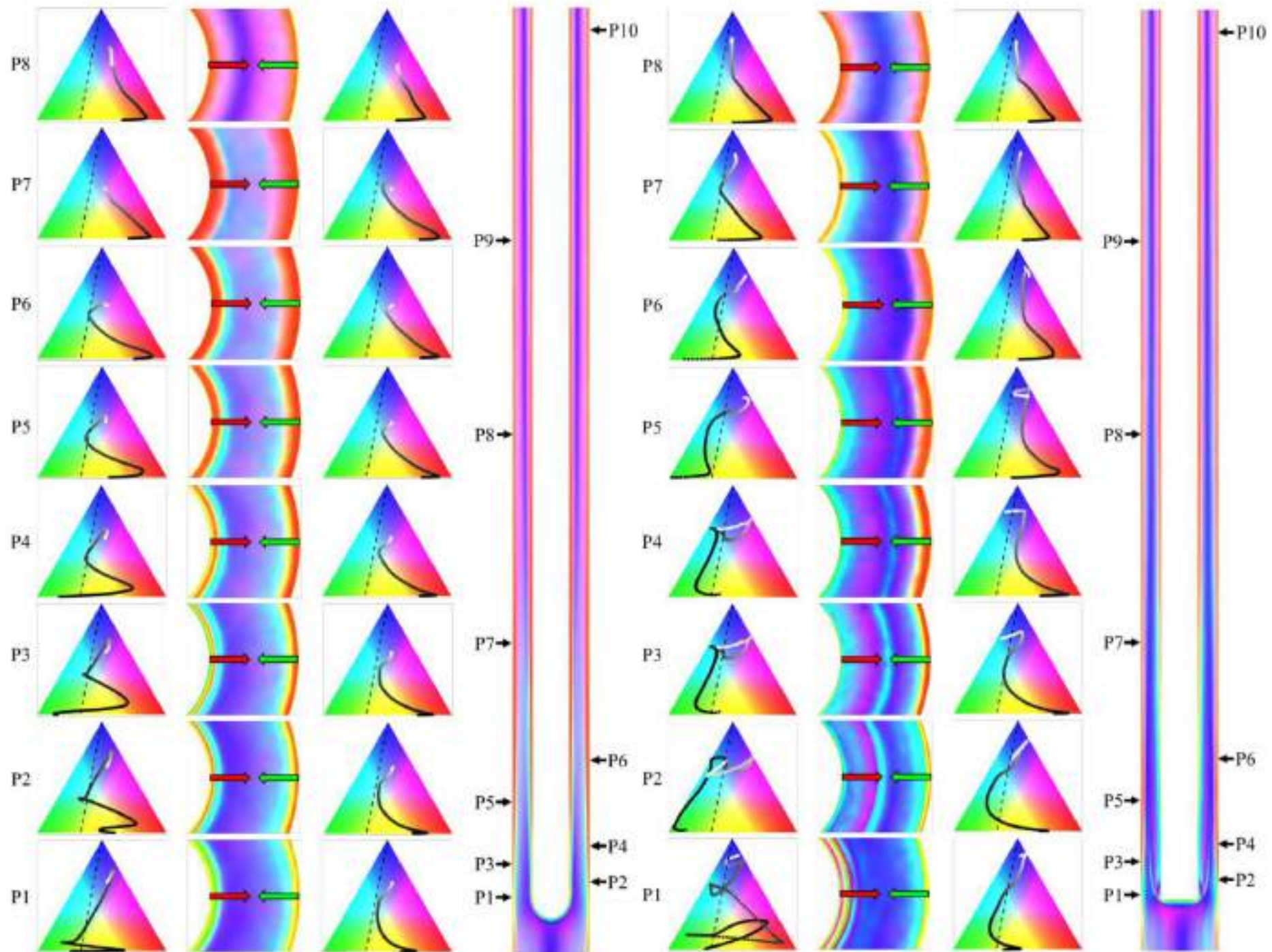
In the transition region we see a more prominent two-component behavior



Anisotropy – Comparison with Blunt end (1/2)



Anisotropy – Comparison with Blunt end (2/2)



Effect of FSI on budgets



- A test was performed by applying a fixed sinusoidal oscillation to the beam in two directions, comparable to what was observed experimentally.
- First order and rms statistics showed less than 1% change! **It was evaluated that an FSI campaign would be very expensive for this case and unnecessary.**
 - o We did not evaluate budgets in this exercise.
 - o We did not test the blunt end.
 - o Consistent with the comparison with LES.



Conclusions and next steps

- Data for DNS of fluid only simulations is available upon request.
 - Smooth-end results were published in IJHFF: Bistany, Sean, Victor Coppo Leite, Carolina Bourdot Dutra, Sinan Okyay, Anshuman Chaube, Tri Nguyen, and Elia Merzari. "Direct numerical simulation of a cantilevered rod experiment." *International Journal of Heat and Fluid Flow* 117 (2026): 110112.
 - Additional publications on blunt and conic ends are expected.
- Results demonstrate interesting and rich turbulence behavior near the ends.
- *Results were occasionally taken from $Re \sim 20k$ and $Re \sim 11k$ interchangeably, as they are very similar (not clearly distinguishable).*



GO-VIKING

Thank you!



Elia Merzari



Penn State



ebm5351@psu.edu



Website:
www.go-viking.eu



Mail:
contact@go-viking.eu



Twitter:
[@GOVIKING_EU](https://twitter.com/GOVIKING_EU)



LinkedIn:
[GO-VIKING](https://www.linkedin.com/company/go-viking)



**Funded by the
European Union**

Funded by the European Union. Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union or European Atomic Energy Community ('EC-Euratom'). Neither the European Union nor the granting authority can be held responsible for them.



WP3 – Flow-Induced Vibrations in Steam Generators

The AMOVI experiment on FIV in cross-flow

Domenico Panunzio, Lucas Prévost (CEA)



Funded by
the European Union

GO-VIKING Final workshop (GRS, Garching, Germany), 20-22/05/2026

Summary



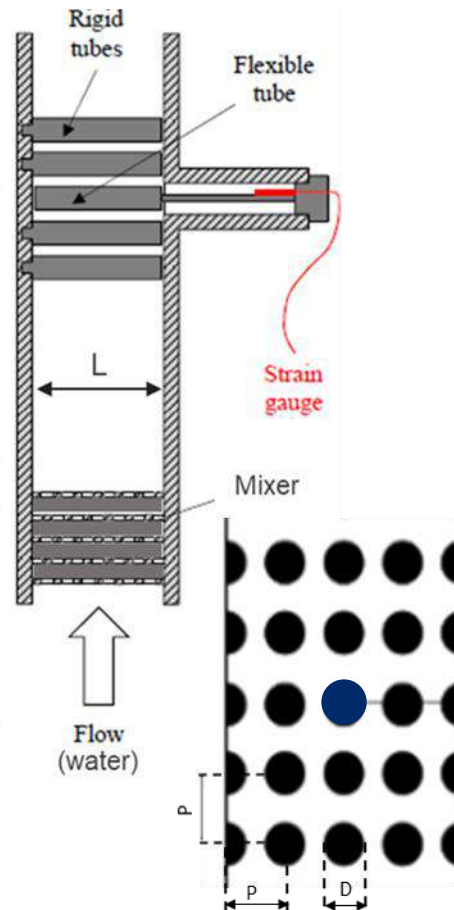
- **Introduction**
- **Experimental setup**
- **Results**
 1. Force and displacement measurements
 2. LDV measurements
 3. PIV measurements
- **Conclusions and perspectives**

Introduction



- In the nuclear field, a frequently experienced phenomena is the flow-induced vibrations (FIV) in tube bundles.
- The WP3 of GO-VIKING aims to improve understanding of the FIV phenomena in tube bundles exposed to cross-flows, mainly relevant to SG applications.
- CEA produces experimental results to support the numerical activities within GO-VIKING.
- A square tube bundle, named AMOVI, was tested in single-phase (water) cross-flow.

Experimental setup



- Water loop supplied by a centrifugal pump, ($Re_{max} \sim 23000$ inter-tubes)
- Test section of 70x100x600 (width x depth x height) mm^3
- Mixer at the inlet section.
- 5x3 square tube bundle (plus two columns of semi-tubes):
 $D = 12.15 \text{ mm}$, $L = 100 \text{ mm}$, $P = 17.5 \text{ mm}$ ($P/D = 1.44$)
- Central tube (steel 304L) mounted in a flexible (blade + strain gauge) or rigid (tri-axial force sensor) arrangement
- Other tubes (plexiglas) rigid and interchangeable.

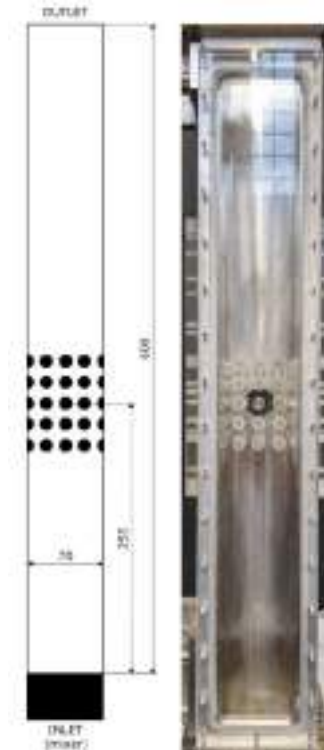
Experimental setup



Single tube in a reduced channel



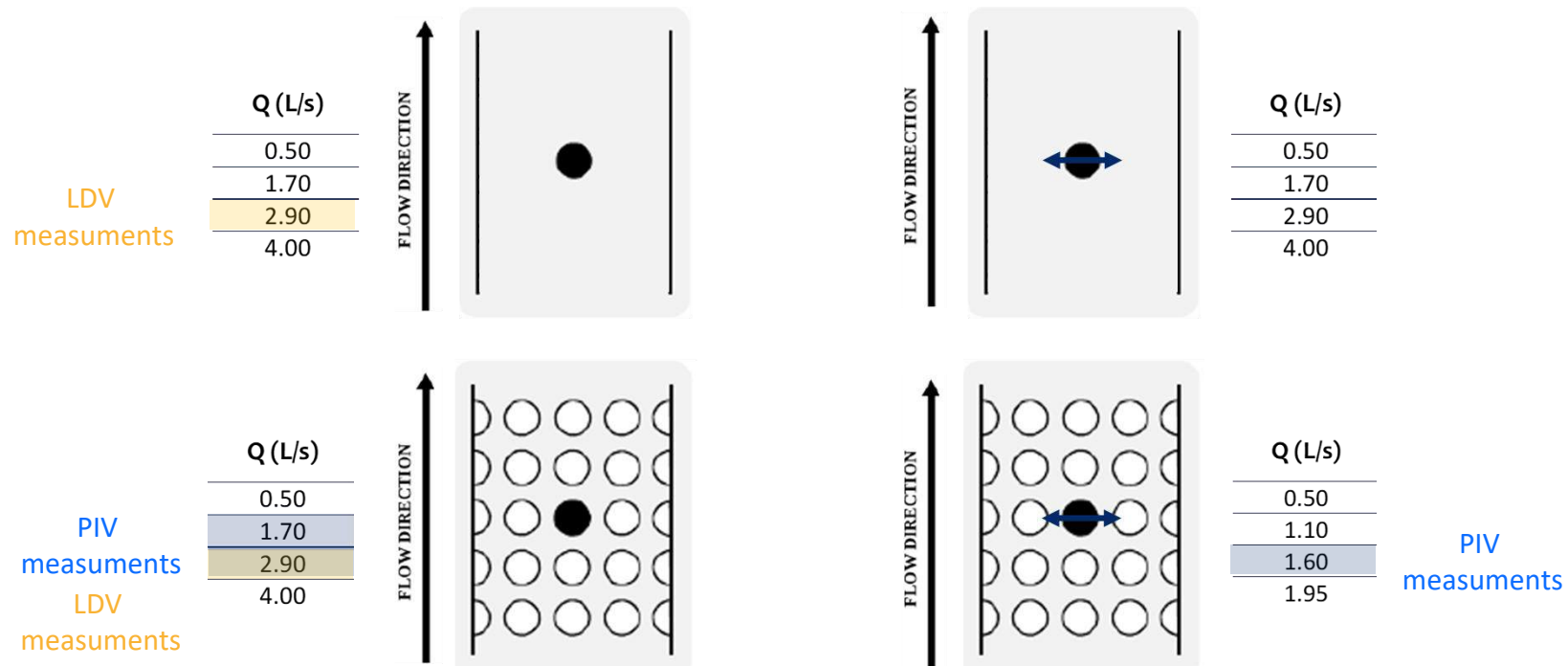
Full tube bundle



Experimental setup



2 basic configurations x 2 possible assemblies = 4 target configurations in total.

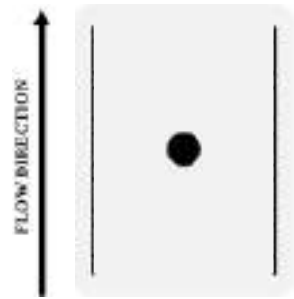
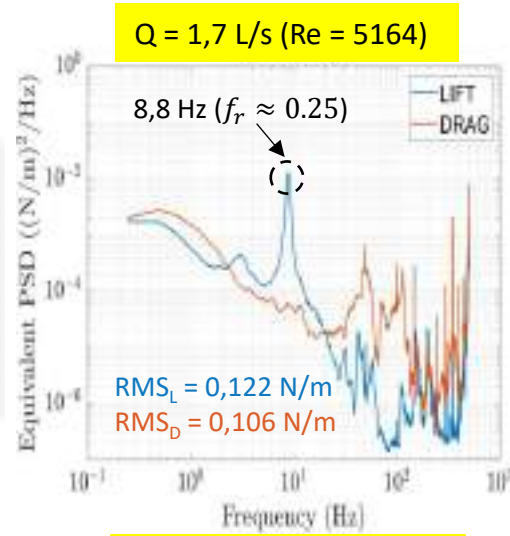
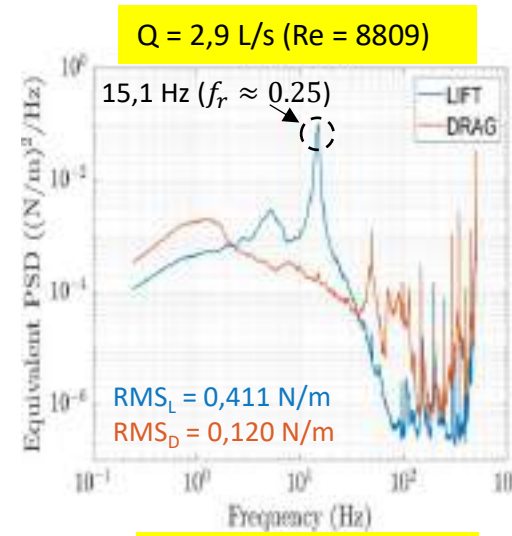
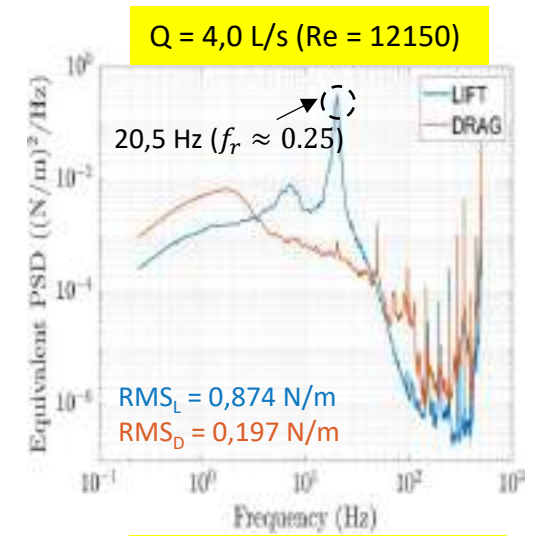


Results

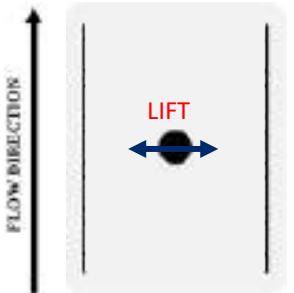
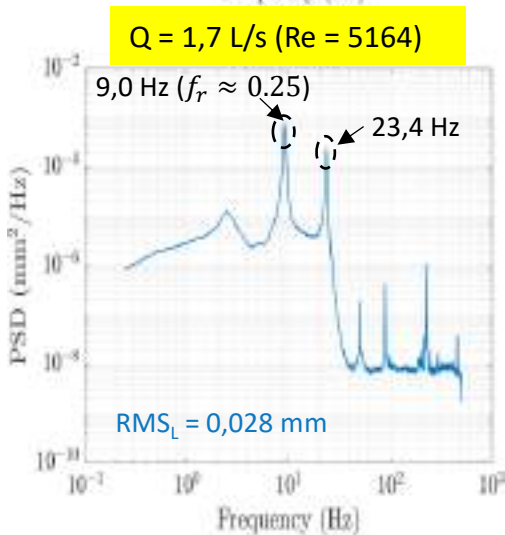
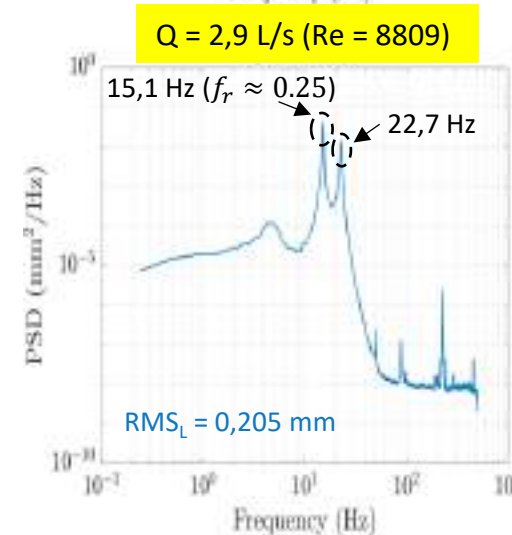
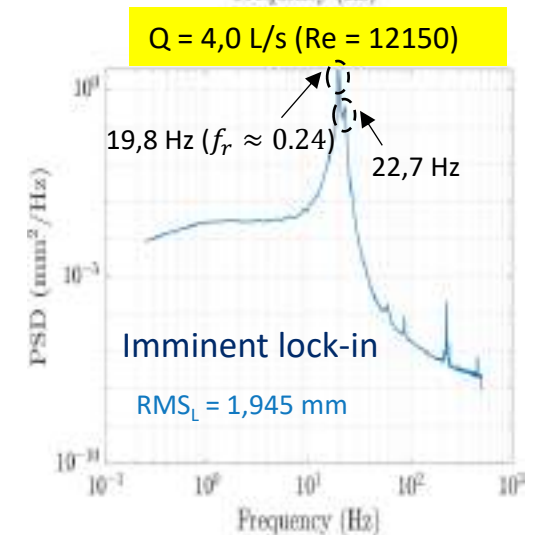
Force and displacement measurements



Force measurements



Displacement measurements

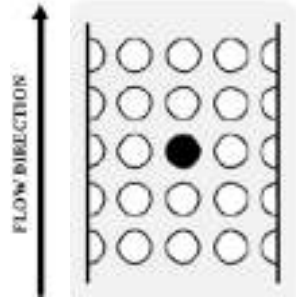
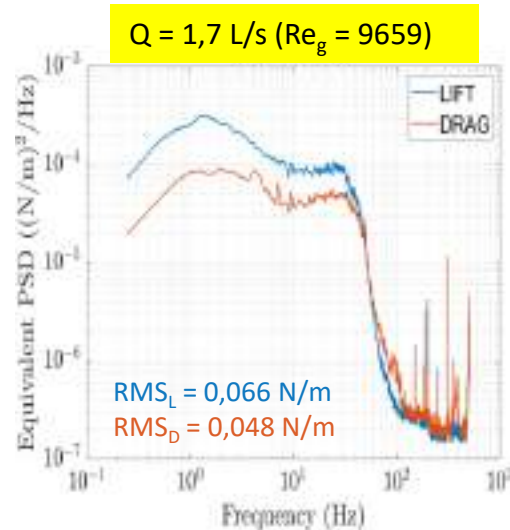
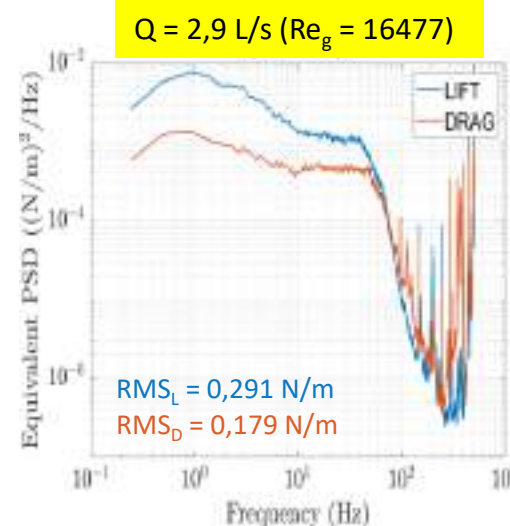
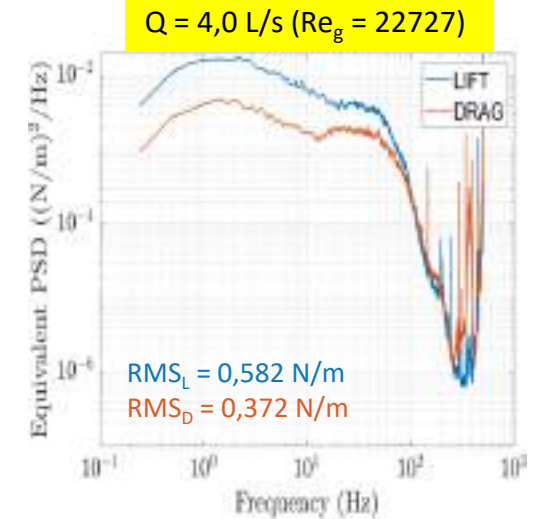


Results

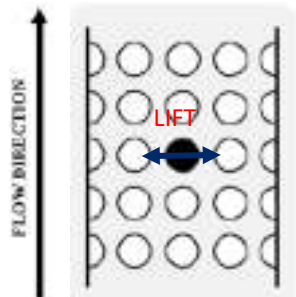
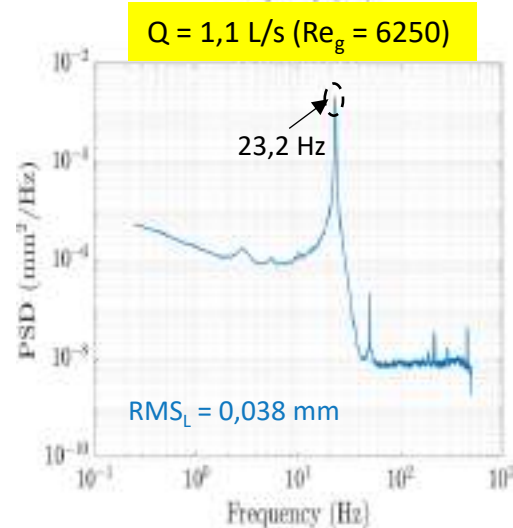
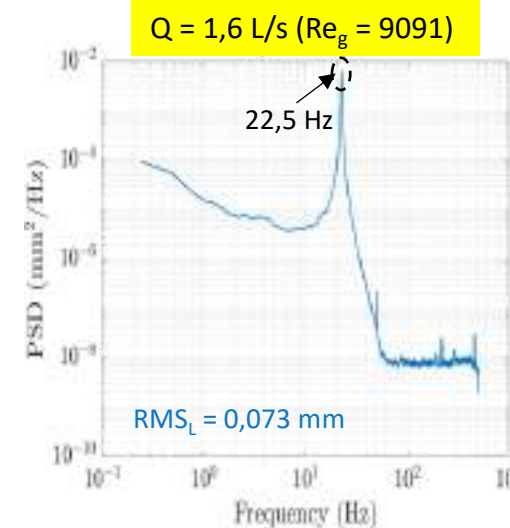
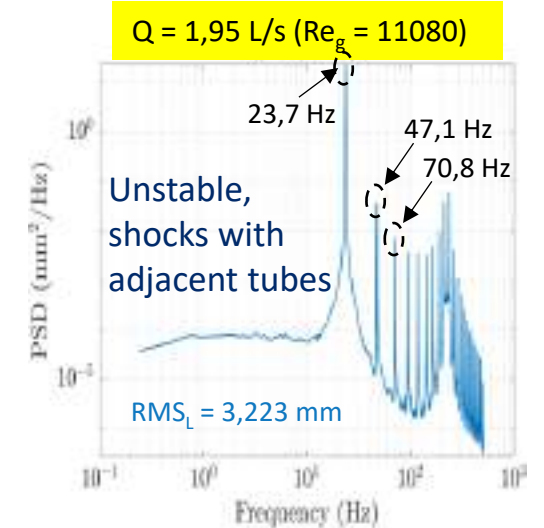
Force and displacement measurements



Force measurements



Displacement measurements

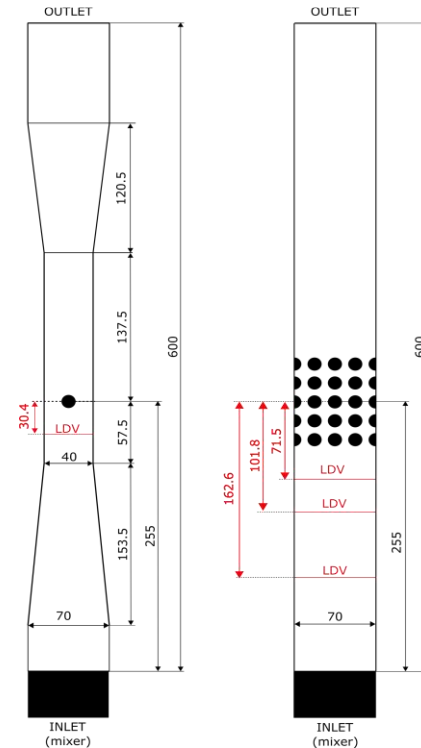


Results

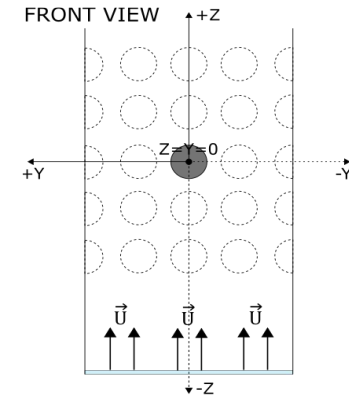
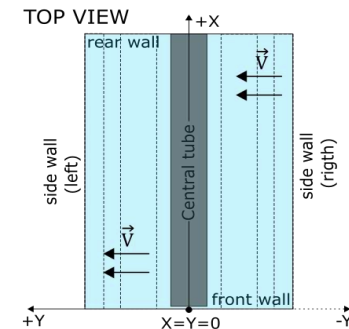
LDV measurements



Location of LDV measurements in the two tested configurations



(a)



(b)

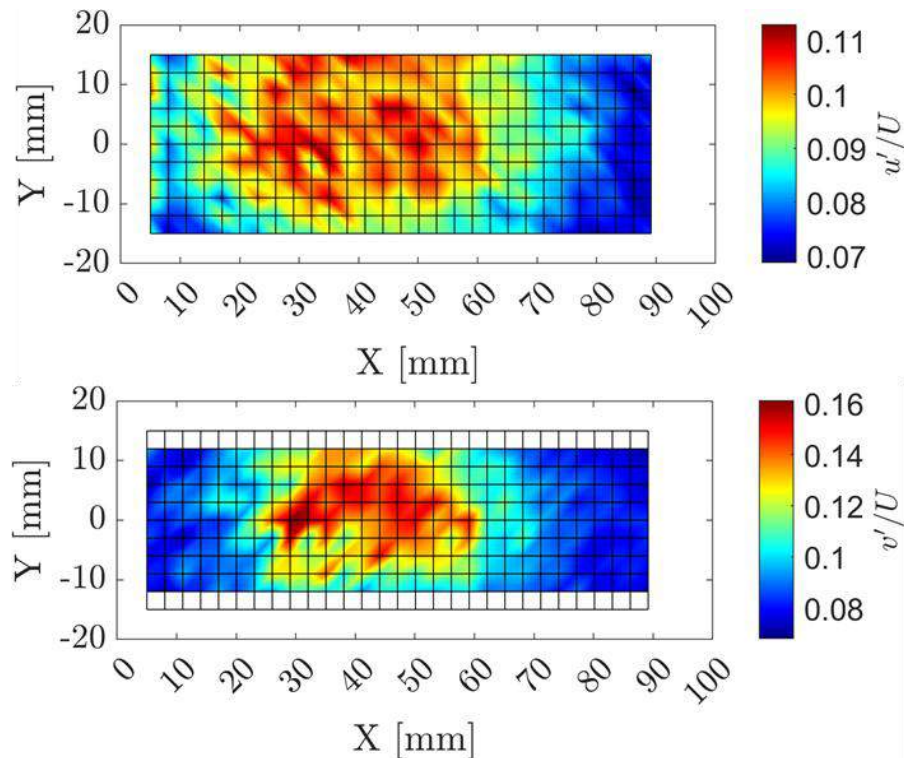
Results

LDV measurements

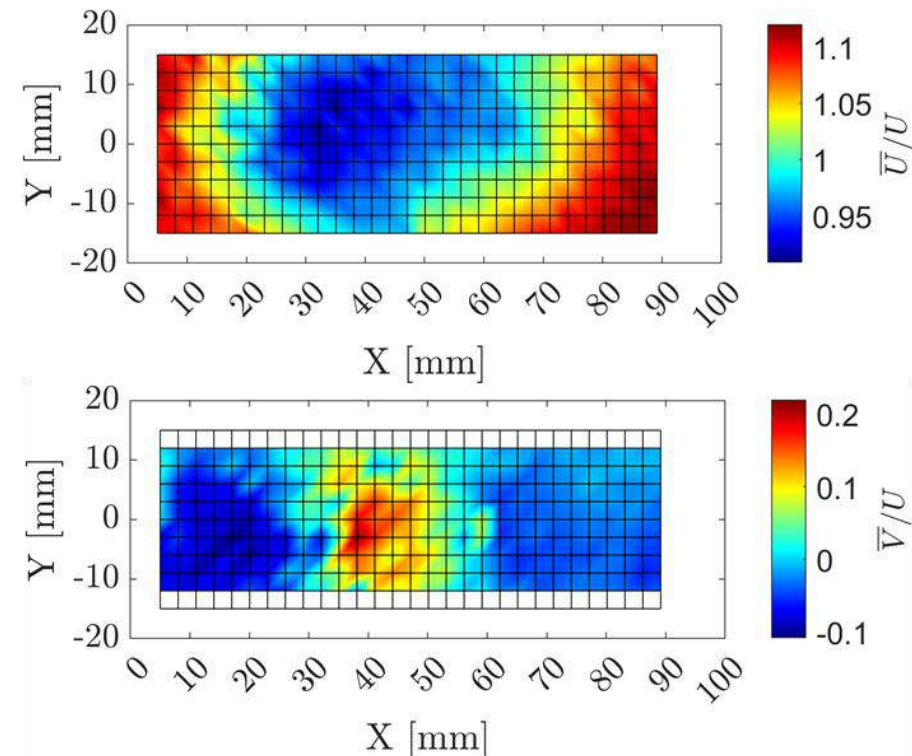


Single tube configuration at $Z = -30.4$ mm, $U = 0.725$ m/s ($Q = 2.9$ L/s)

Turbulence fluctuations (u', v')



Time-averaged velocities ($\bar{U}, \bar{V}$)



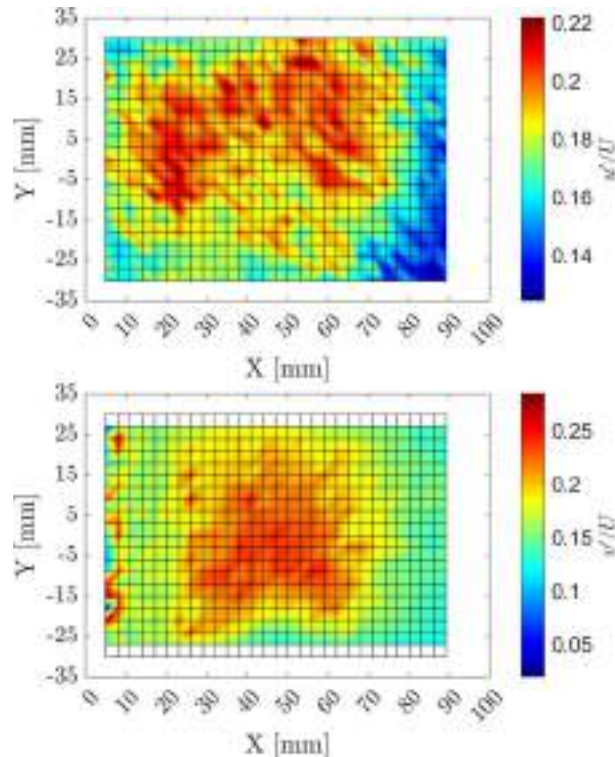
Results

LDV measurements

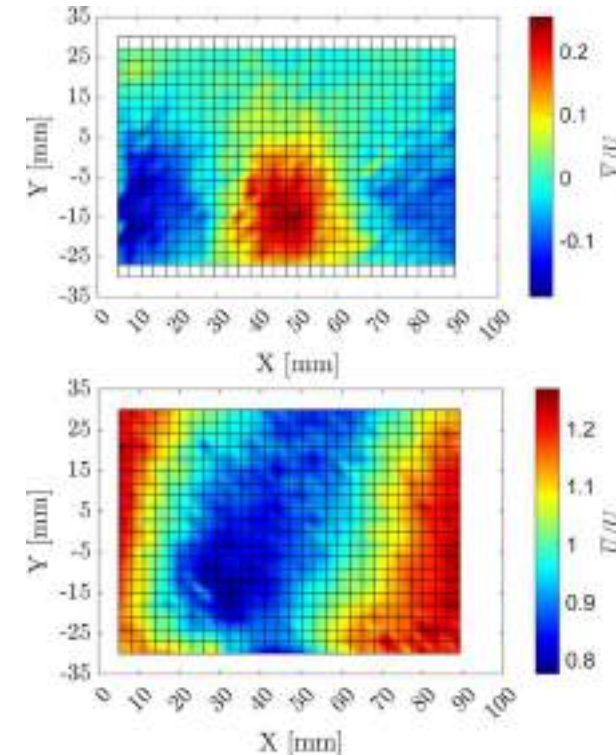


Tube bundle configuration at $Z = -71.5$ mm, $U = 0.414$ m/s ($Q = 2.9$ L/s)

Turbulence fluctuations (u', v')

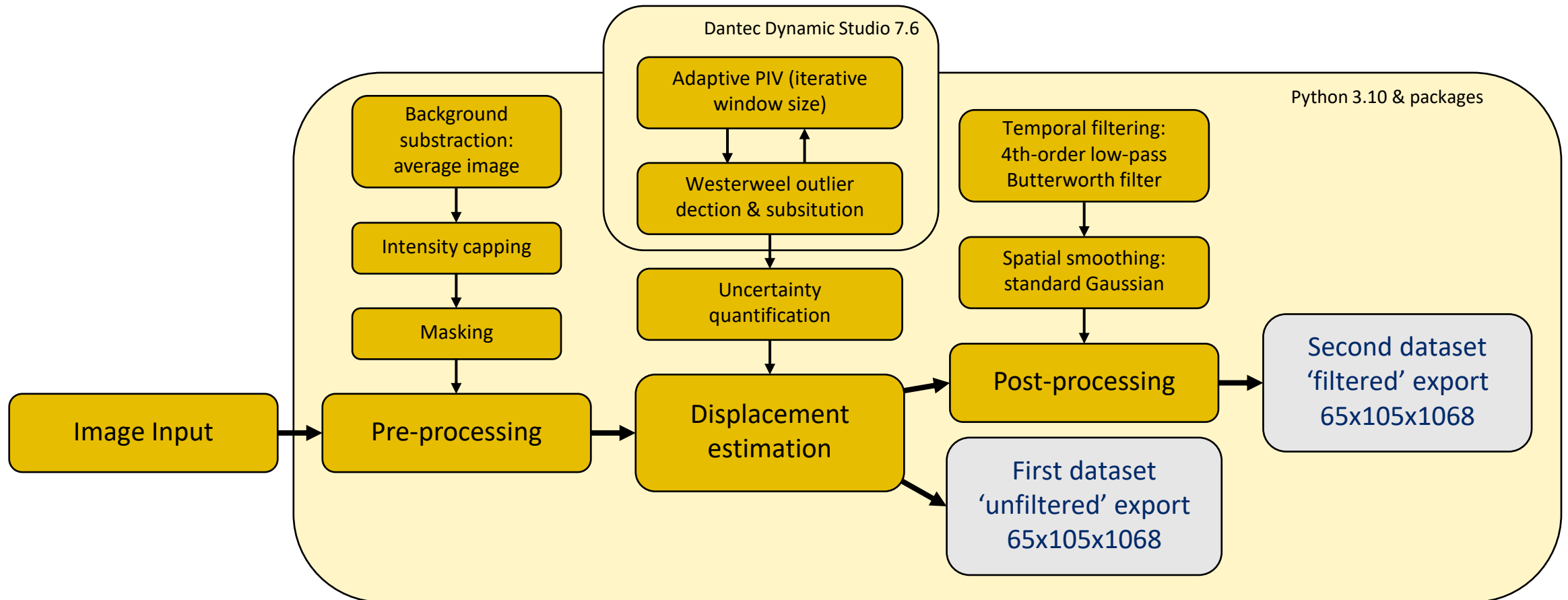


Time-averaged velocities ($\overline{U}, \overline{V}$)



Results

PIV measurements

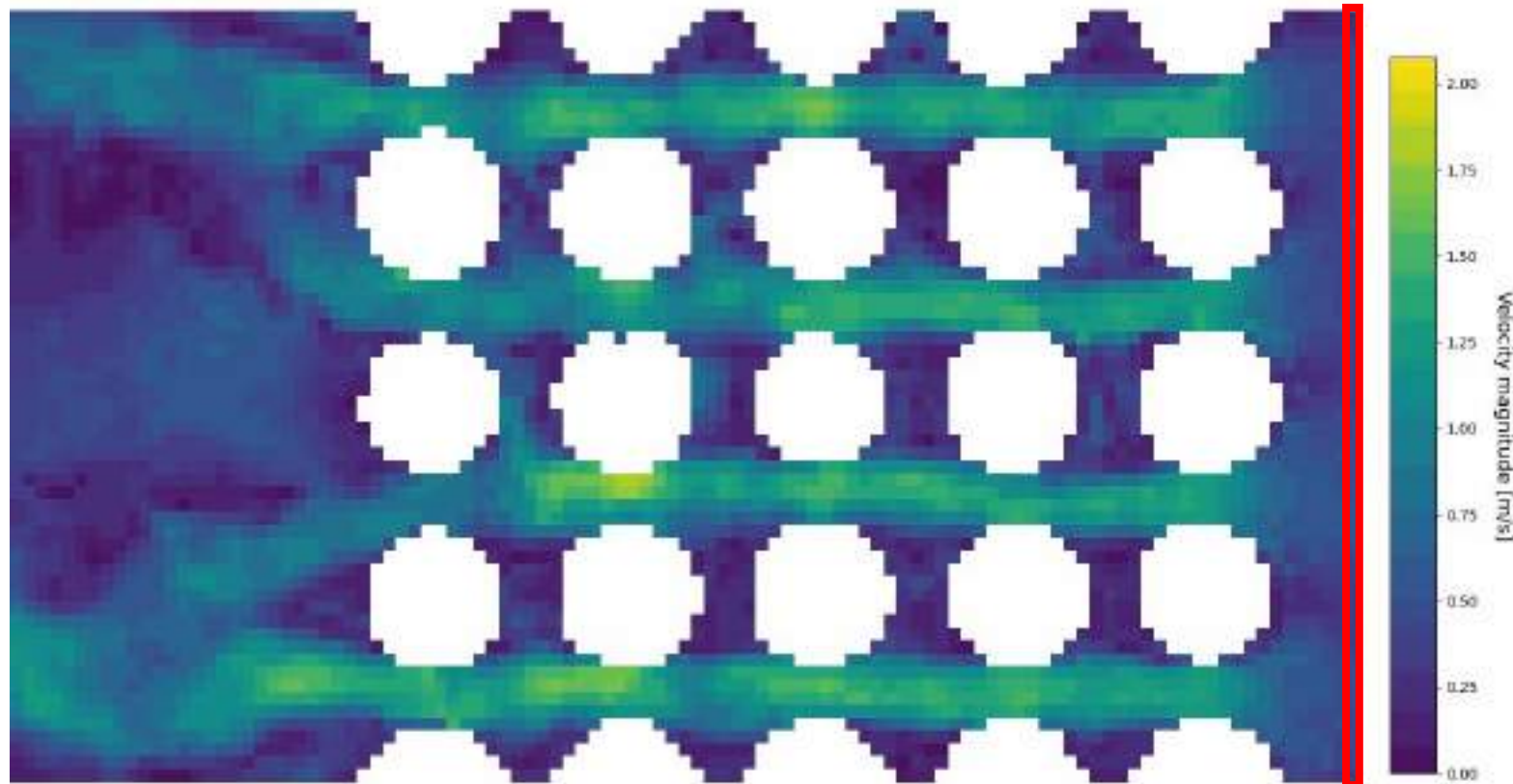


Results

PIV measurements



Velocity magnitude for $Q = 2.9 \text{ L/s} \rightarrow U_g = 1.35 \text{ m/s}$, rigid bundle



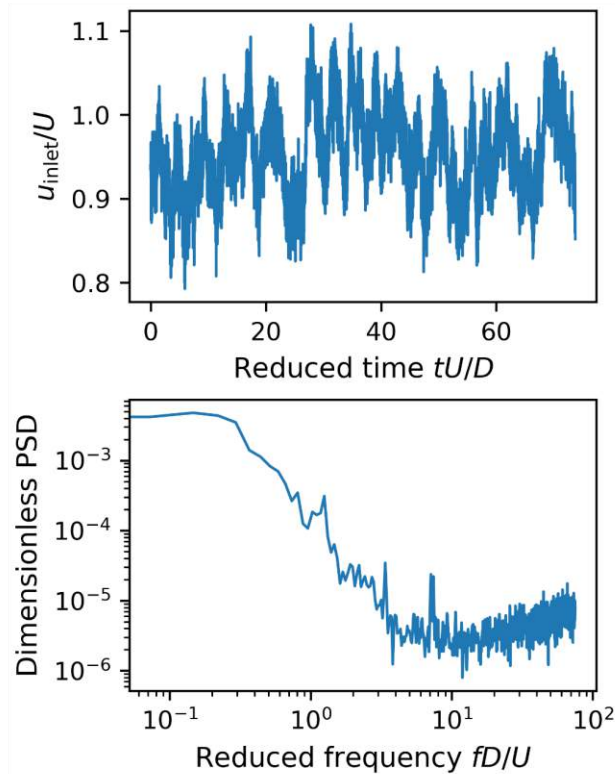
Incoming
velocity =
average over the
right-most
column

Results

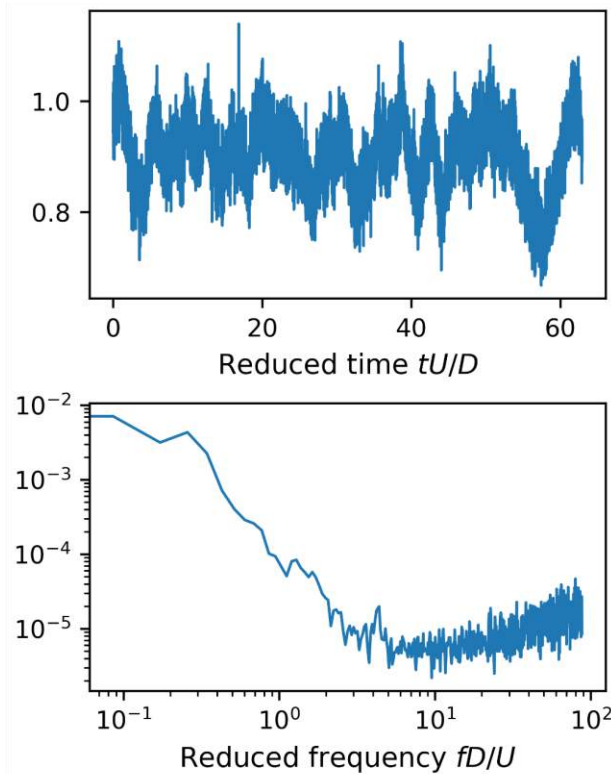
PIV measurements



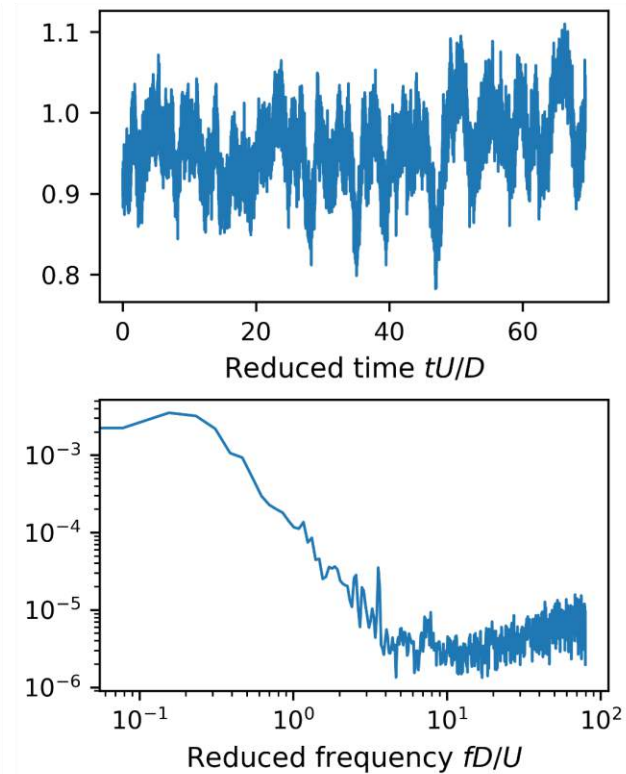
$Q = 1.7 \text{ L/s}$ ($Re_g = 9659$), rigid bundle



$Q = 2.9 \text{ L/s}$ ($Re_g = 16477$), rigid bundle



$Q = 1.6 \text{ L/s}$ ($Re_g = 9091$), one flexible tube



10 – 15 % fluctuations in inlet velocity → sign of 3D turbulent structures + perturbations from pump

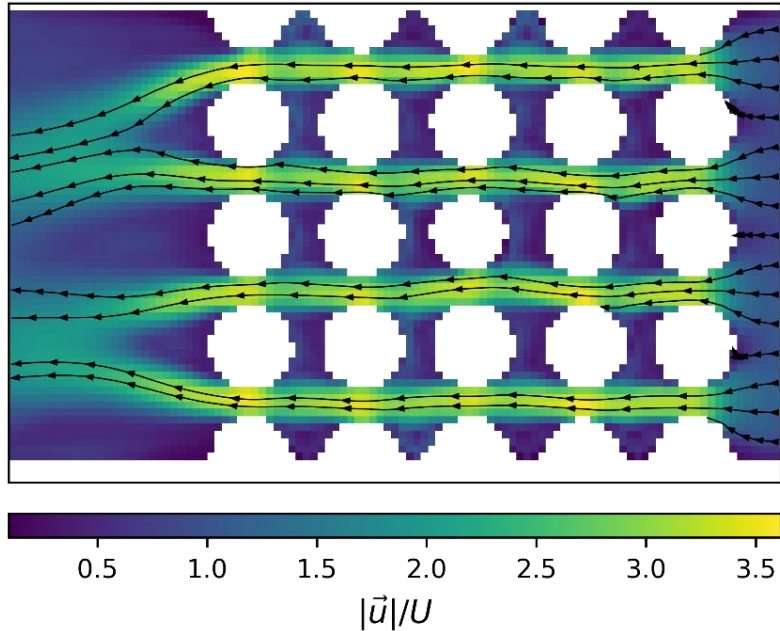
Results

PIV measurements

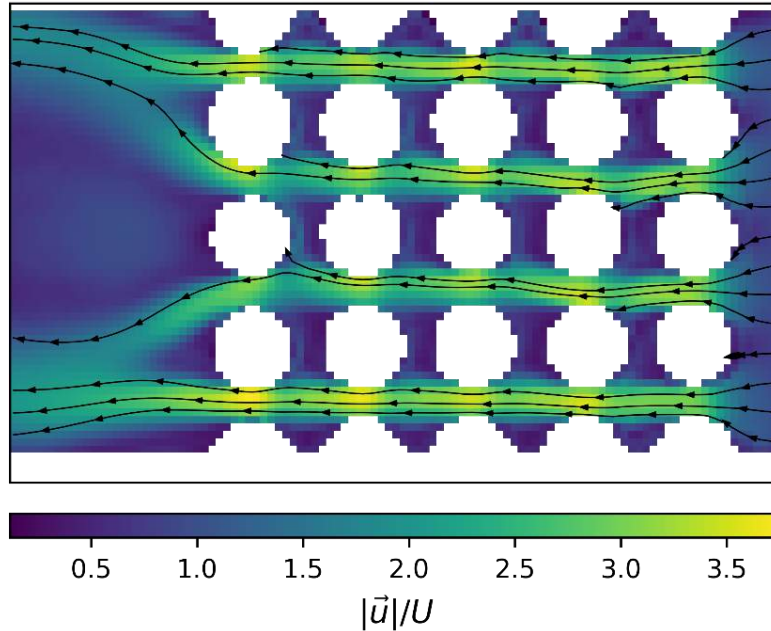


Mean velocity field

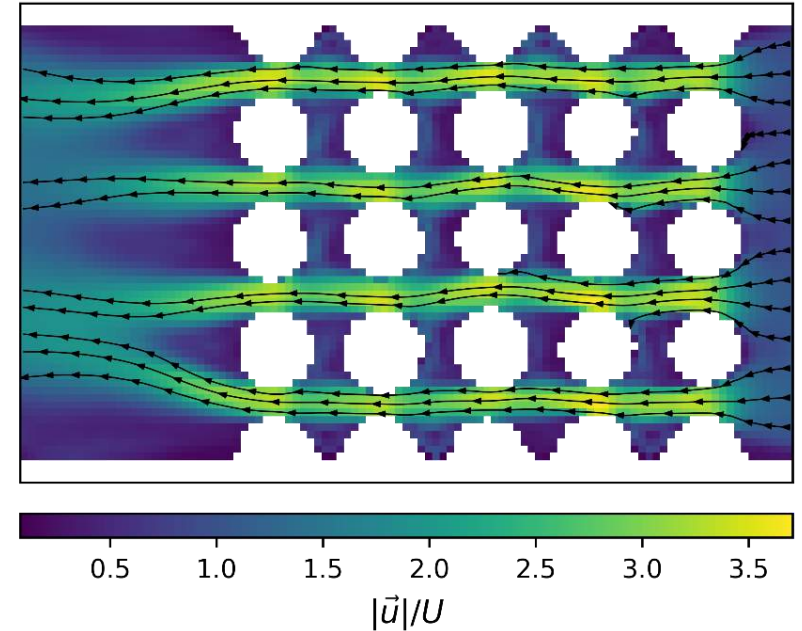
$Q = 1.7 \text{ L/s}$ ($Re_g = 9659$), rigid bundle



$Q = 2.9 \text{ L/s}$ ($Re_g = 16477$), rigid bundle



$Q = 1.6 \text{ L/s}$ ($Re_g = 9091$), one flexible tube

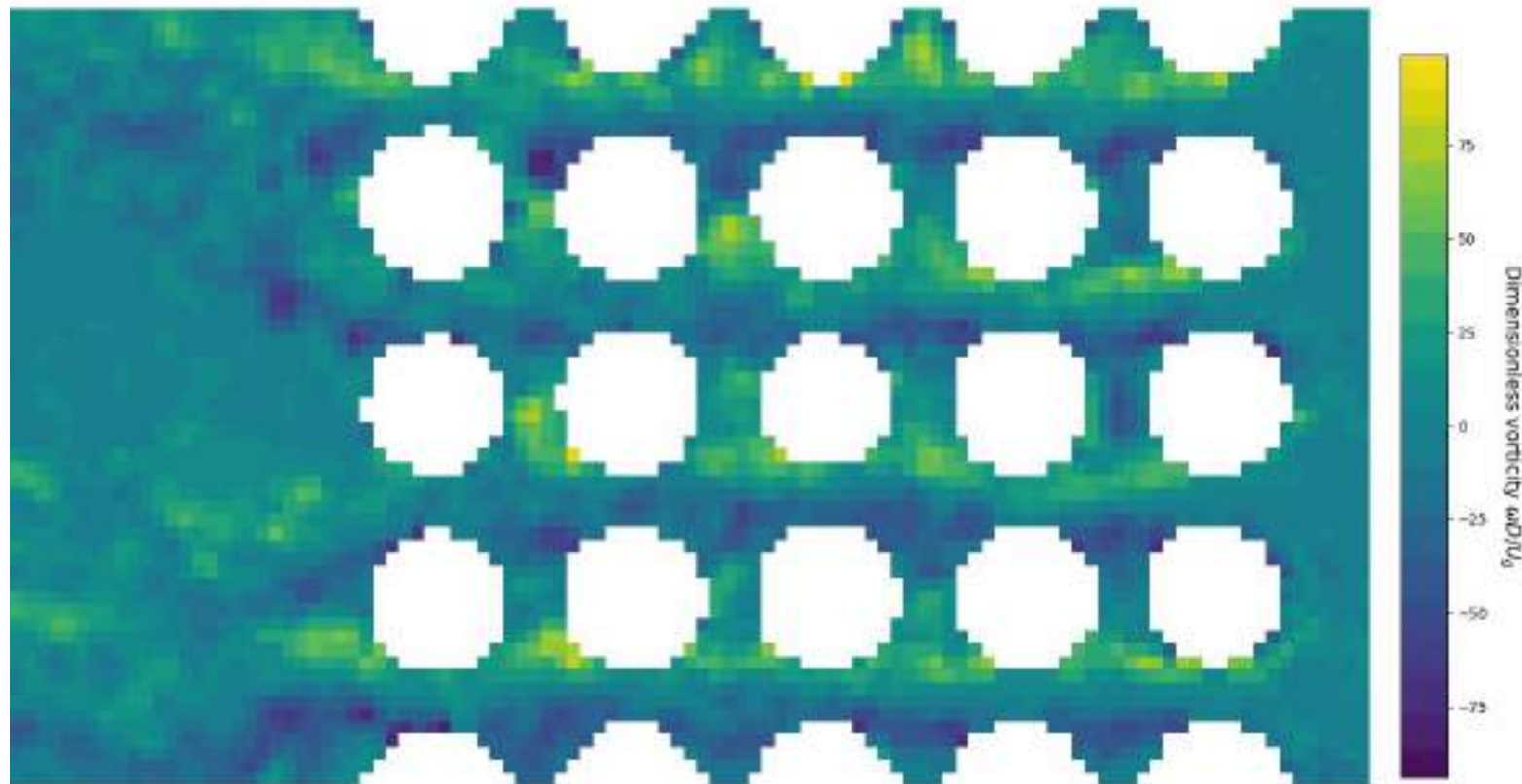


Results

PIV measurements



Flow vorticity for $Q = 2.9 \text{ L/s} \rightarrow U_g = 1.35 \text{ m/s}$, rigid bundle



Conclusions



- **Experimental setup AMOVI : square tube bundle $P/D = 1,44$ in water cross-flow**
 - Single tube in a reduced channel
 - Full tube bundle (5x3 tubes + 10 half-tubes)
- **Force measurements:**
 - Single tube: single dominant peak for lift component ($f_r \approx 0,25$)
 - Tube bundle: « double-slope » excitation spectra
- **Displacement measurements:**
 - Single tube: two dominant peaks coalescing at ≈ 4 L/s (imminent lock-in)
 - Tube bundle: single dominant peak unstable before 1,95 L/s upcoming LDV measurements
- **LDV measurements :**
 - Non-uniform velocity profiles
 - Significant turbulence levels
- **PIV measurements:**
 - Fluctuations in inlet velocity
 - Complex convection & diffusion of vorticity



GO-VIKING

Thank you!



Domenico Panunzio
Lucas Prévost



CEA



domenico.panunzio@cea.fr
lucas.prevost@cea.fr

www.go-viking.eu



Funded by the
European Union

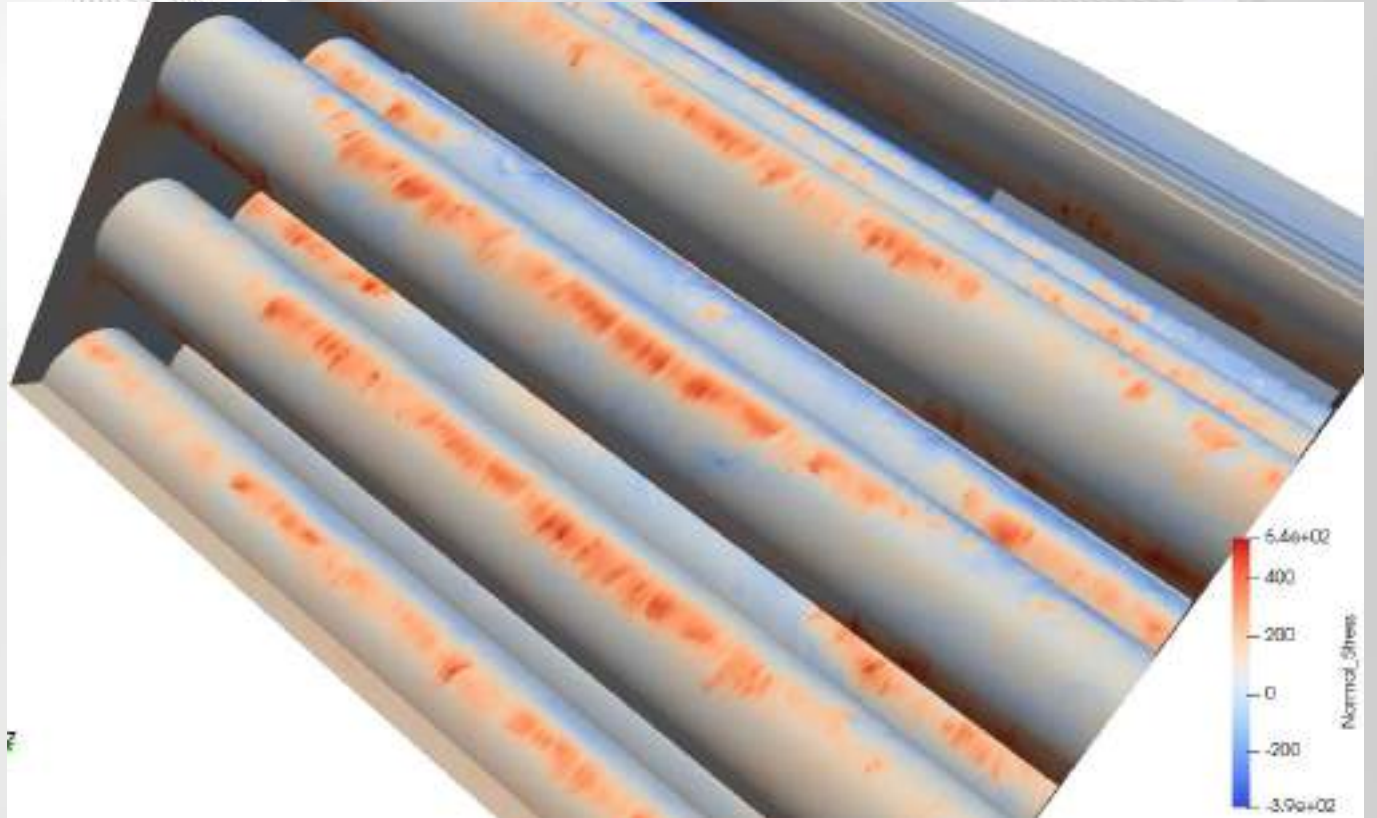


High-fidelity LES analysis of AMOVİ

Sofiane Benhamadouche

EDF R&D, Fluid Mechanics, Energy and Environment
Dept, EDF Lab Chatou, France
sofiane.benhamadouche@edf.fr

GO-VIKING FINAL WORKSHOP, 20-22 May 2026
GRS gGmbH, Garching n. Munich, Germany





High-fidelity LES analysis of AMOVİ

Sofiane Benhamadouche

EDF R&D, Fluid Mechanics, Energy and Environment
Dept, EDF Lab Chatou, France
sofiane.benhamadouche@edf.fr

GO-VIKING FINAL WORKSHOP, 20-22 May 2026
GRS gGmbH, Garching n. Munich, Germany



Overview

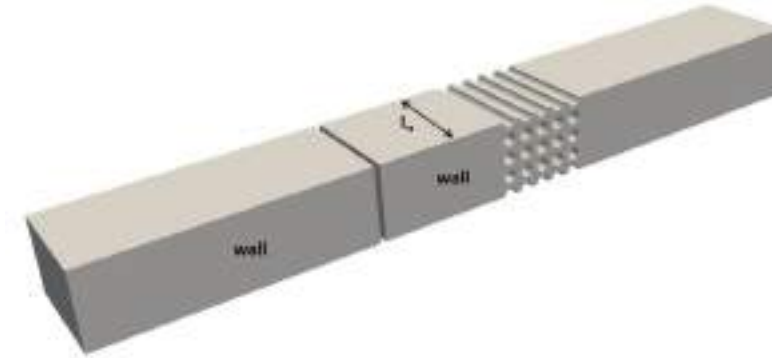


- I. **5x5 rigid tube bundle**
 - a. **Computational domain / Numerics / BC / Meshes**
 - b. Asymmetry of the flow / Flow reversal in the span-wise direction
 - c. Central cylinder Lift and drag forces fluctuations results
- II. 5x5 tube bundle with a flexible central cylinder
- III. Conclusions and perspectives

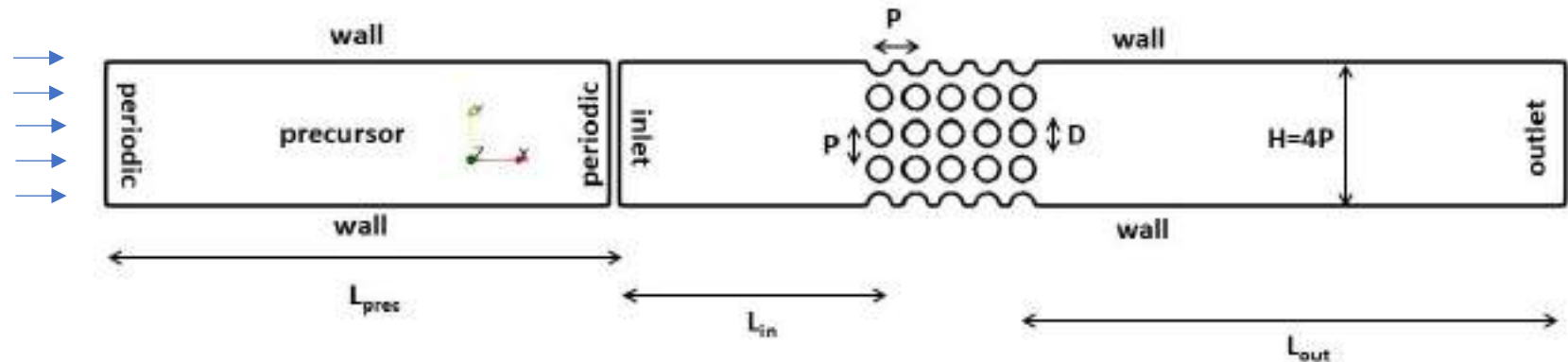
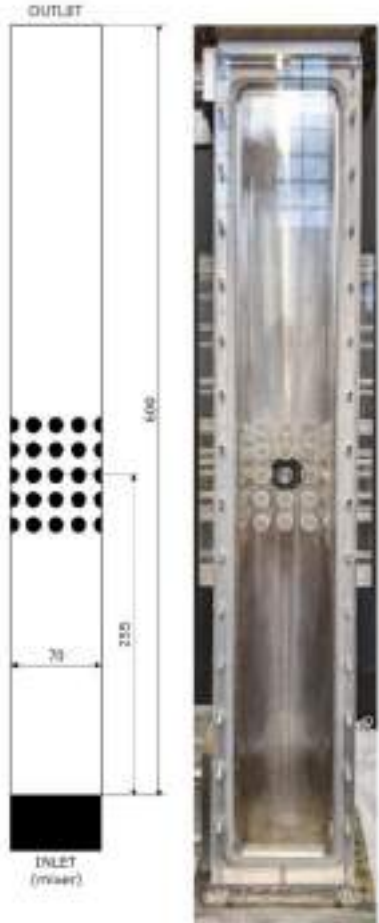
5x5 rigid tube bundle - computational domain

Amovi Experiment (CEA):

- Tube bundle $D = 0.01215$ m, $P = 0.0175$ m ($P/D = 1.44$)
- Water properties, the density and the kinematic viscosity are $\rho = 1000$ kg/m³ and $\nu = 10^{-6}$ m²/s, respectively.
- In the computations: L_{in} , L_{out} and L_z are equal to $10D$, $22D$ and 0.1 m, respectively.

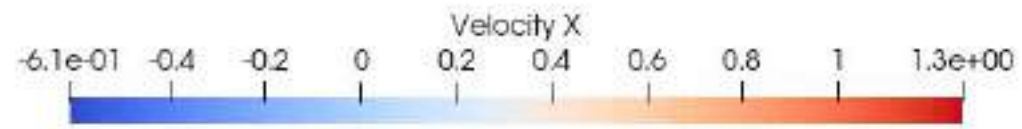
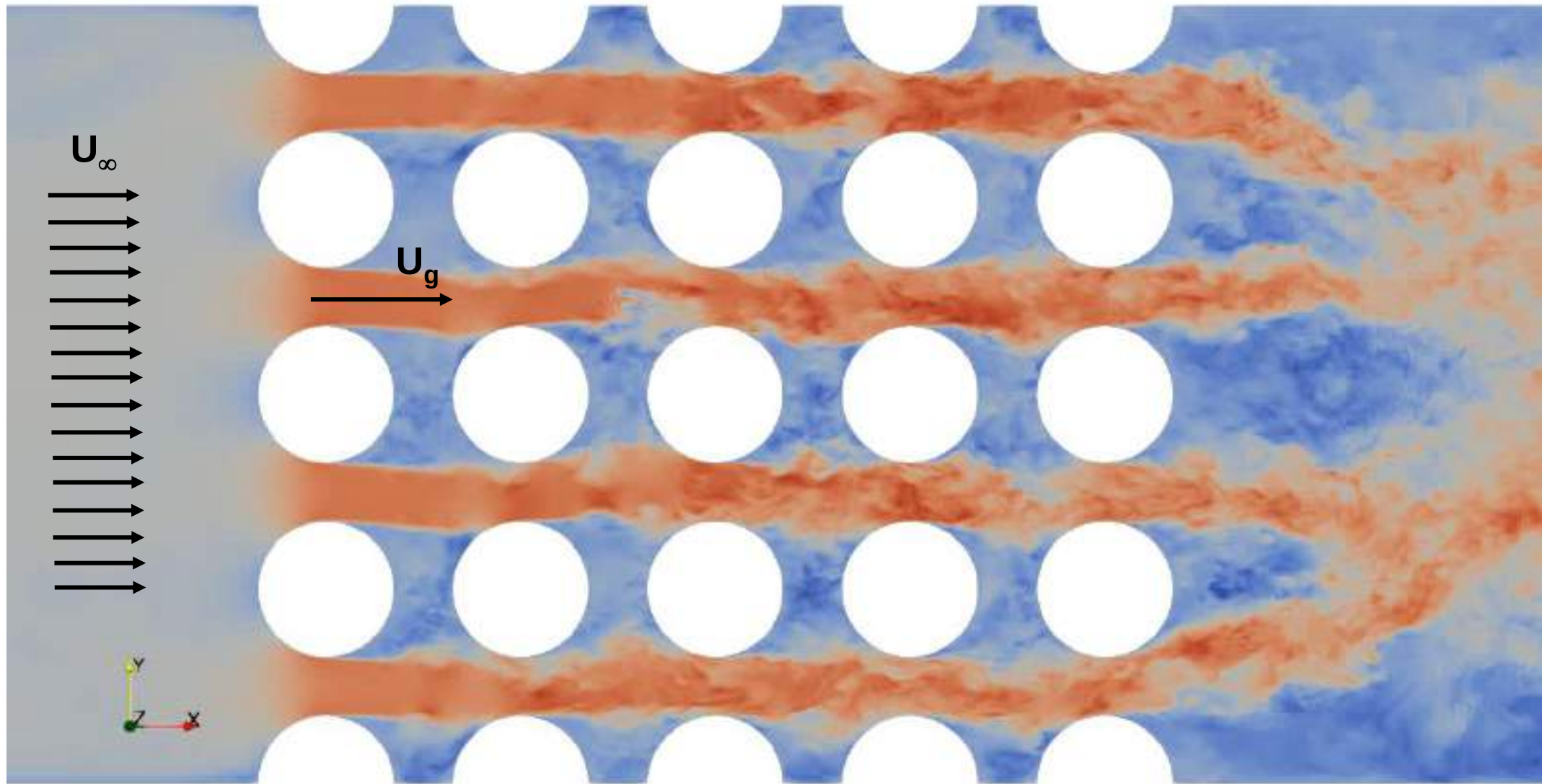


Q (l/s)	U_g (m/s)	U_∞ (m/s)	Re_g	Re_∞	Re_{inlet}	Re_τ
0,5	0,234	0,071	2 840	868	5 882	94
1,7	0,795	0,243	9 657	2 951	20 000	282
2,9	1,356	0,414	16 474	5 034	34 118	457



Amovi (CEA)

Time: 158.750000 s



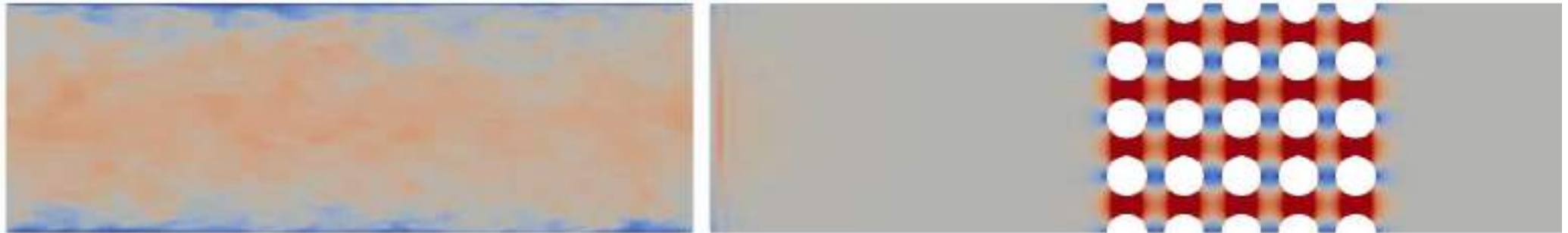
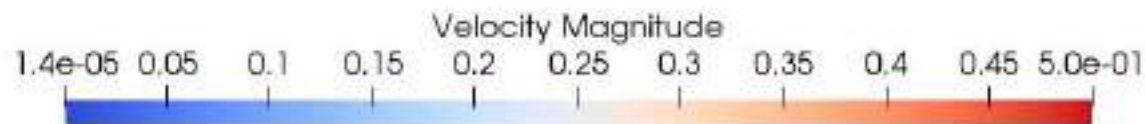
$Q = 1.7 \text{ l/s}$

5x5 rigid tube bundle – Wall-resolved LES, numerical options

- Collocated Finite Volume using EDF open source CFD solver code_saturne V8.0.
- Second order in time
 - Crank Nicolson for the convected velocity and the diffusion term.
- Second order in space
 - Two points Laplacian and pure centered convection.
- Gradient correction: Gauss with implicit handling of non-orthogonalities.
- Pressure/Velocity coupling : SIMPLEC (no outer iterations on pressure/velocity coupling)
- The dynamic model based on explicit filtering using the neighborhood of a computational cell sharing a node.
- Targeted maximum CFL around 1.
- Maximum normal non-dimensional distance to the wall ~ 2 .
 - Aspect ratio in the stream-wise direction at the inlet/outlet ~ 30 .
 - Aspect ratio in the stream-wise direction at the inlet/outlet < 10 (due to the refinements along the cylinders).

5x5 rigid tube bundle – Wall-resolved LES, Boundary conditions

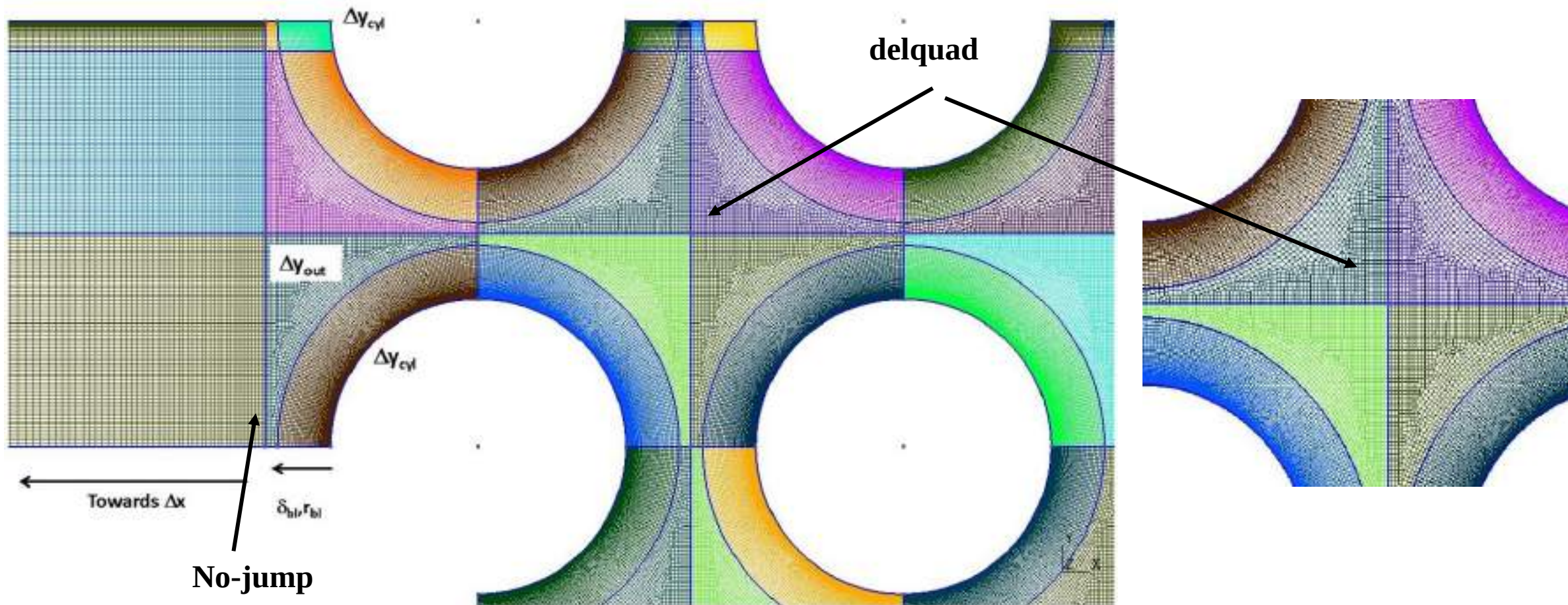
- Inlet/outlet/walls: no periodic boundary conditions except for the precursor computation in the stream-wise direction.
- All the walls are resolved
- Fully developed flow at the inlet: embedded precursor calculation
 - Imposed pressure gradient obtained from a separate computation (precursor alone)
 - The precursor stream-wise length = $6 D_H$, where D_H is the hydraulic diameter of the inlet.



Embedded precursor computation (no data saving)

5x5 rigid tube bundle – Wall-resolved LES, meshes

- Massive use of **delquad** algorithm available in Gmsh 4.7.1. Mesh totally parametrized.



5x5 rigid tube bundle – Wall-resolved LES, meshes

Q (l/s)	mesh	δ_{bl} (m)	Δy_{cyl} (m)	Δy_{out} (m)	$\Delta x_{cyl} / \Delta y_{cyl}$
0.5	ref.	1.958E-02	6.200E-05	2.570E-04	4
1.7	ref.	1.814E-03	2.000E-05	2.000E-04	9
2.9	ref.	1.247E-03	1.300E-05	1.370E-04	10

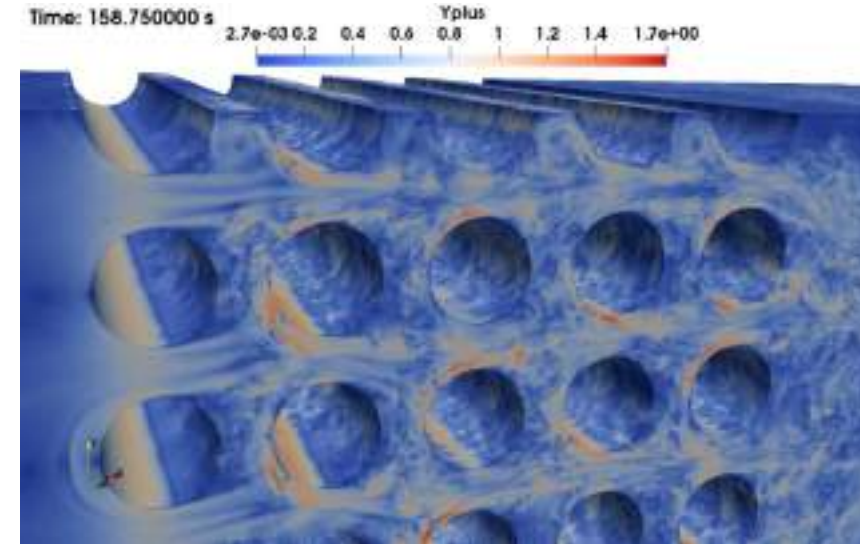
$$r_{bl}=1.1$$

$$\Delta z_{cyl} / \Delta y_{cyl}=15$$

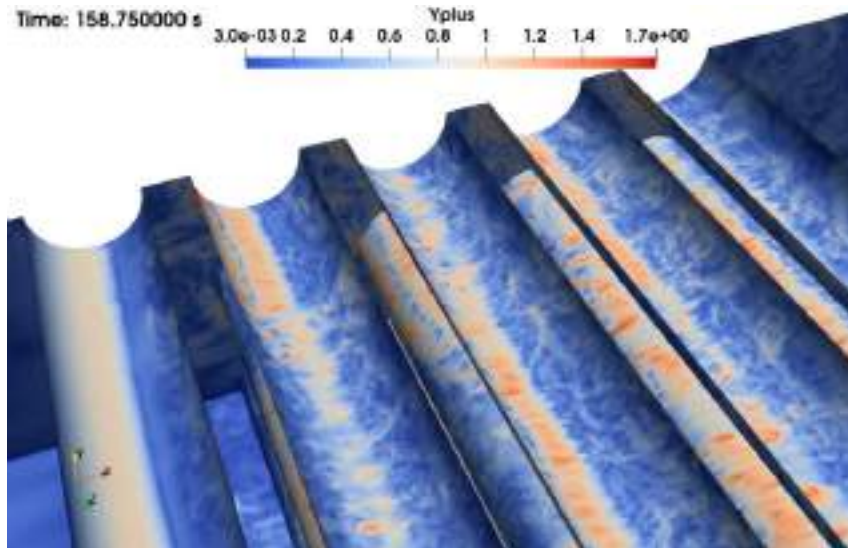
Q (l/s)	mesh	n_z	n_θ	n_{prec} (Million)	n_{cells} (Million)
0.5	ref.	104	204	2.1	27.7
1.7	ref.	400	268	17.3	144.7
2.9	ref.	602	356	59.8	430.6

5x5 rigid tube bundle – Wall-resolved LES, meshes

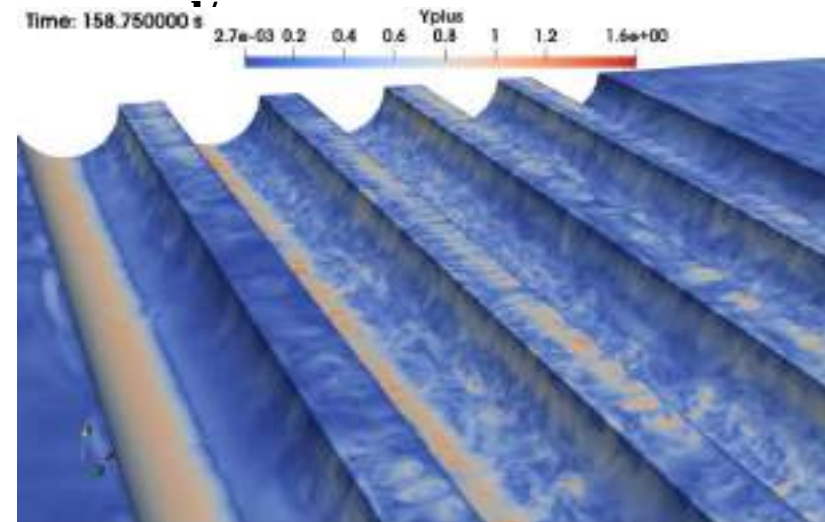
Q (l/s)	Δt (s)	CFL _{max}	CFL _{max} Std.	CFL _{sp,av}	y ⁺ _{max}	y ⁺ _{max} Std.	y ⁺ _{sp,av}
0.5	0.00025	0.89	0.05	0.10	2.05	0.11	0.36
1.7	0.000035	1.03	0.12	0.06	2.03	0.11	0.36
2.9	0.00002	1.53	0.18	0.08	2.05	0.11	0.35



Lateral side-wall Q = 1.7



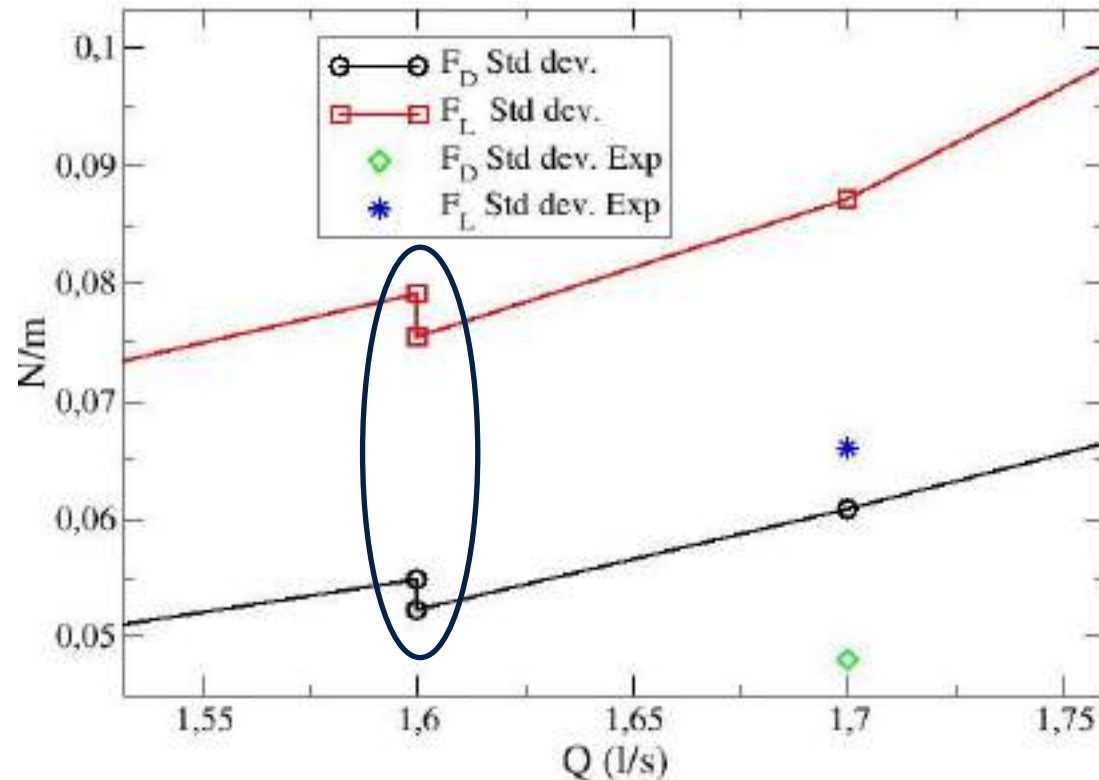
Inner cylinders Q = 1.7 l/s



Upper side-wall Q = 1.7 l/s

5x5 rigid tube bundle – Wall-resolved LES, Mesh sensitivity

Q (l/s)	r_{bl}	δ_{bl}/D	$\Delta y_{cyl}/D$	$\Delta y_{out}/D$	n_z	n_θ	n_{cells} (M)	$\Delta t U_g / D$
1,60	1,10	0,16	0,0018	0,02	298	252	126	0,0025
1,60	1,05	0,18	0,0018	0,01	384	268	257	0,0025



Overview



I. 5x5 rigid tube bundle

- a. Computational domain / Numerics / BC / Meshes
- b. **Asymmetry of the flow / Flow reversal in the span-wise direction**
- c. Central cylinder Lift and drag forces fluctuations results

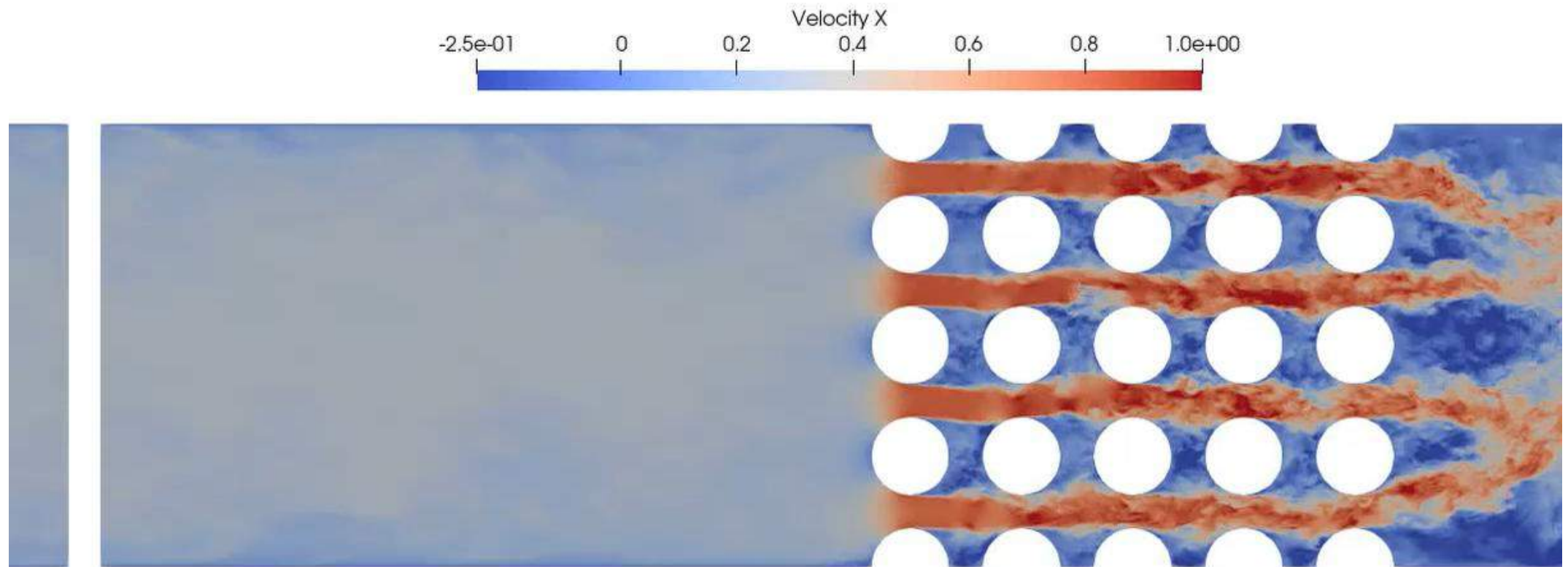
II. 5x5 tube bundle with a flexible central cylinder

III. Conclusions and perspectives

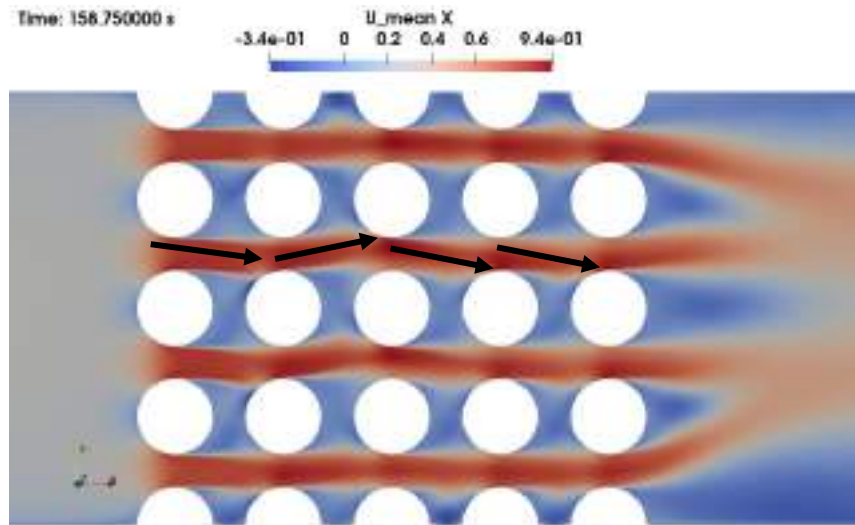
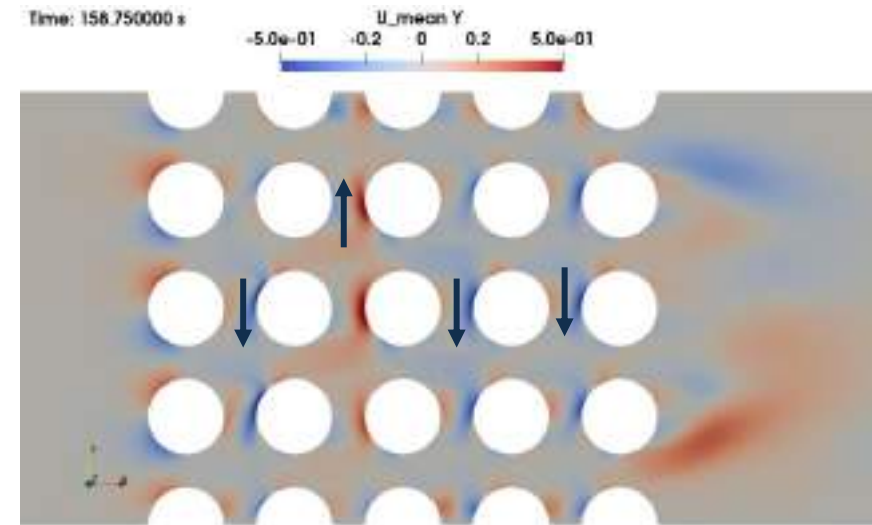
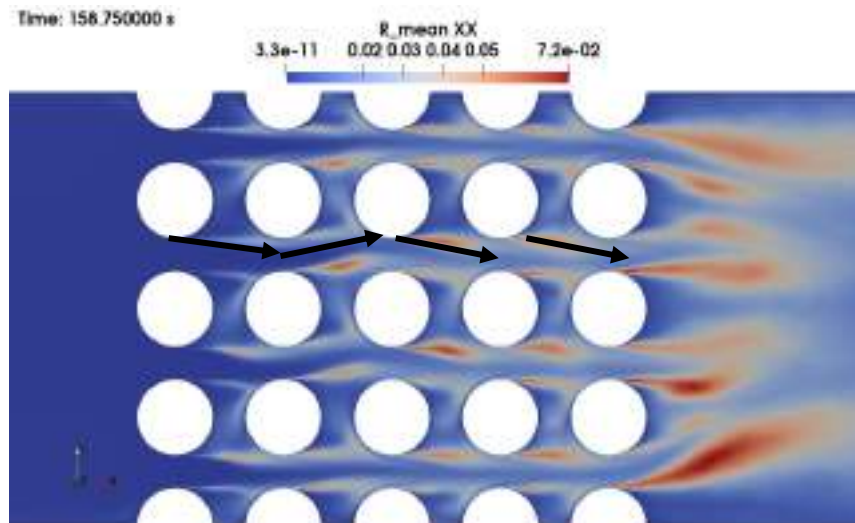
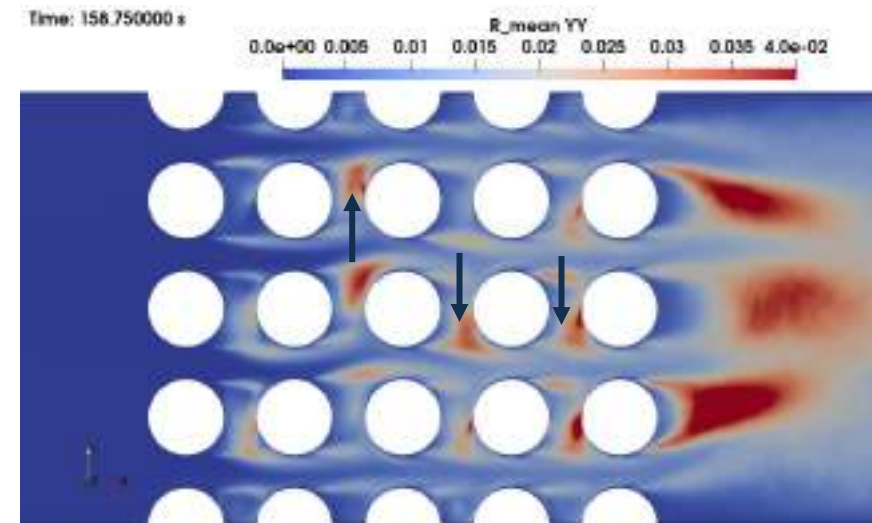
5x5 rigid tube bundle – Asymmetry of the flow

$$Q = 1.7 \text{ l/s}$$

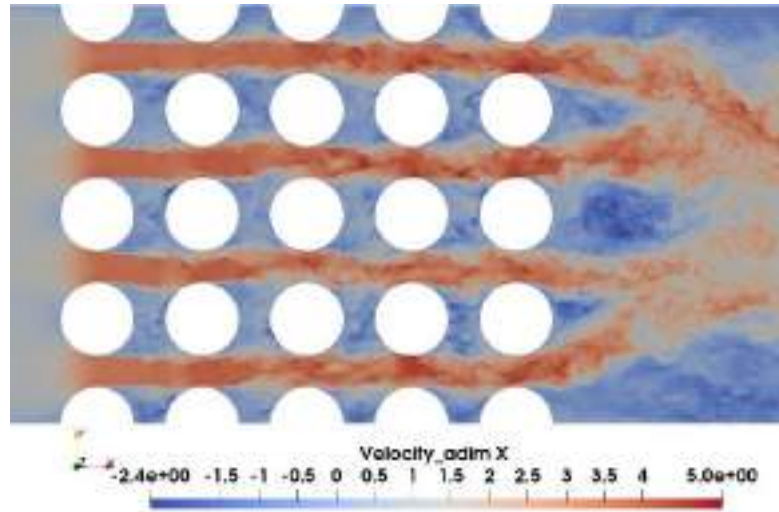
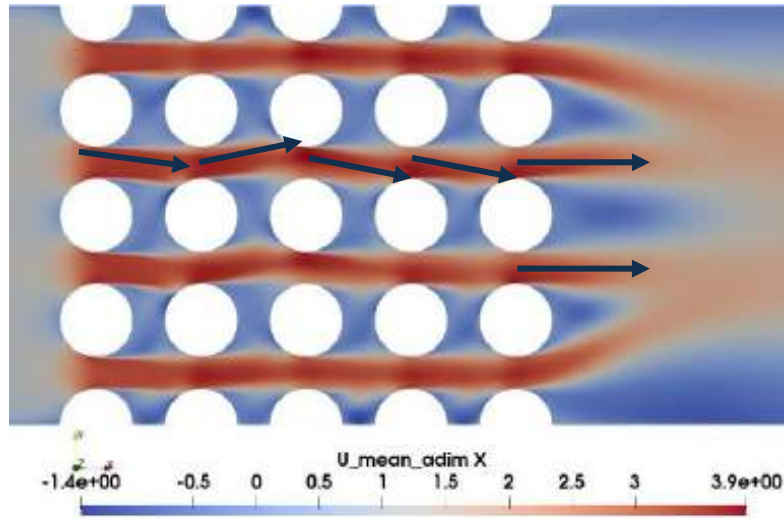
Time: 158.750000 s



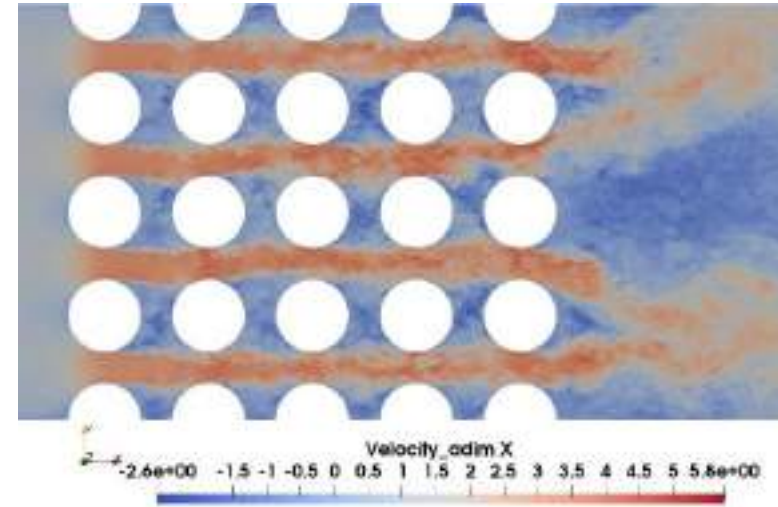
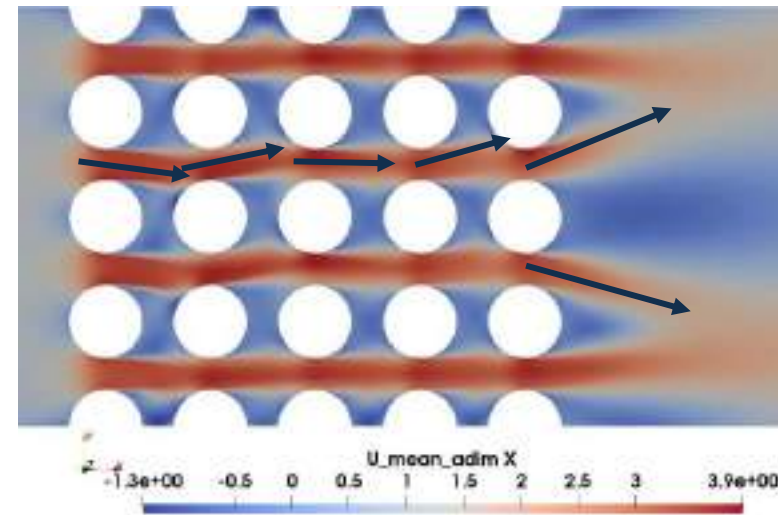
- Different patterns (alternating jets) affecting the mean-flow and the turbulent statistics ($Q = 1.7$ l/s)



- Different patterns (alternating jets) affecting in the bundle and down-stream the bundle

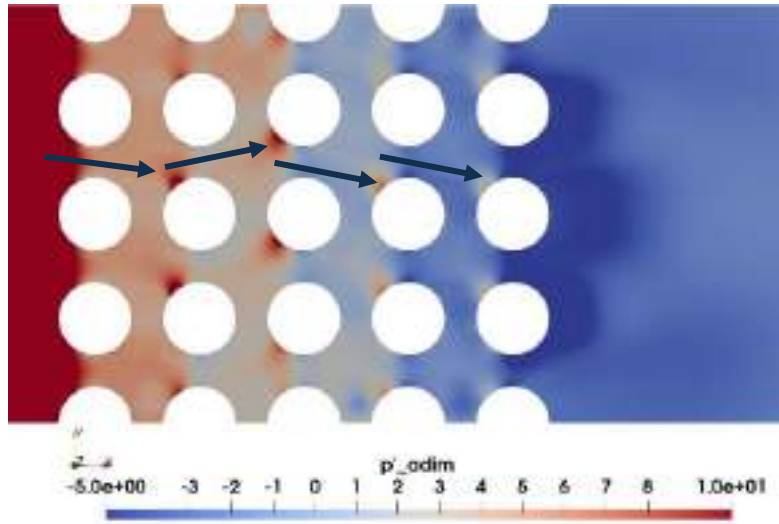
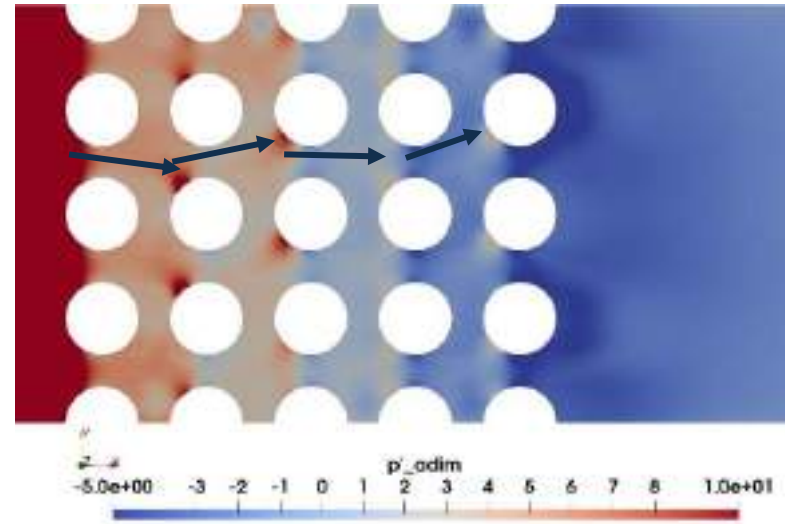
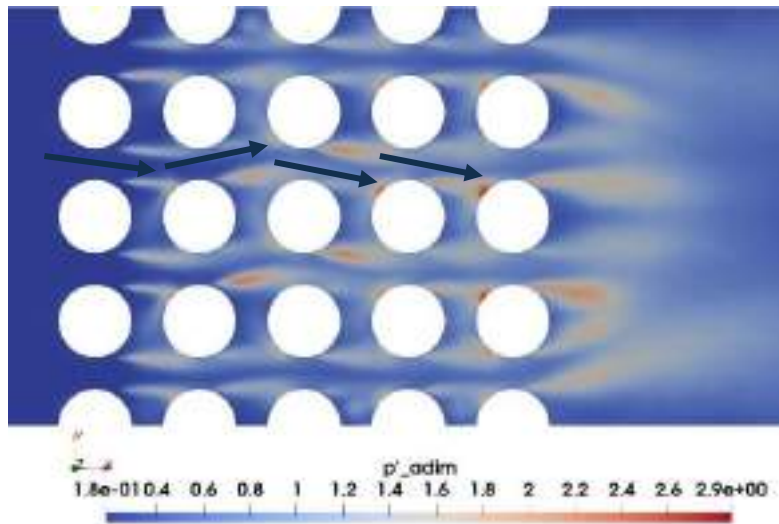


$Q = 1.7 \text{ l/s}$

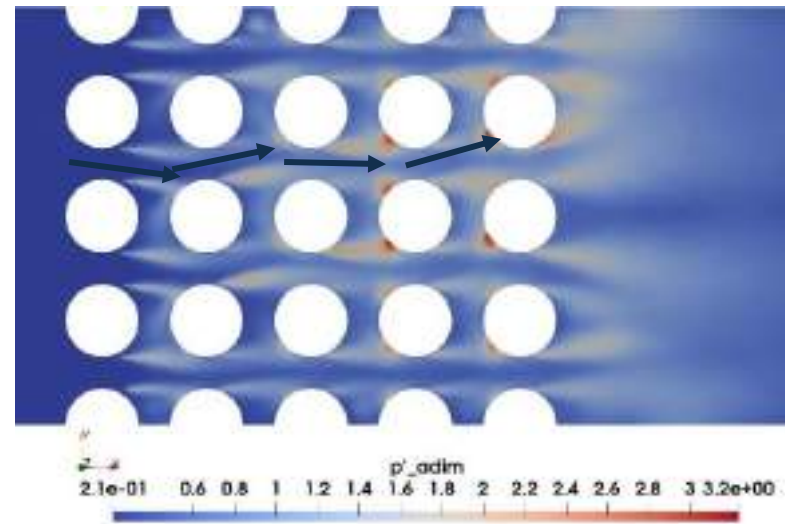


$Q = 2.9 \text{ l/s}$

- This naturally affects the pressure distribution and its standard deviation



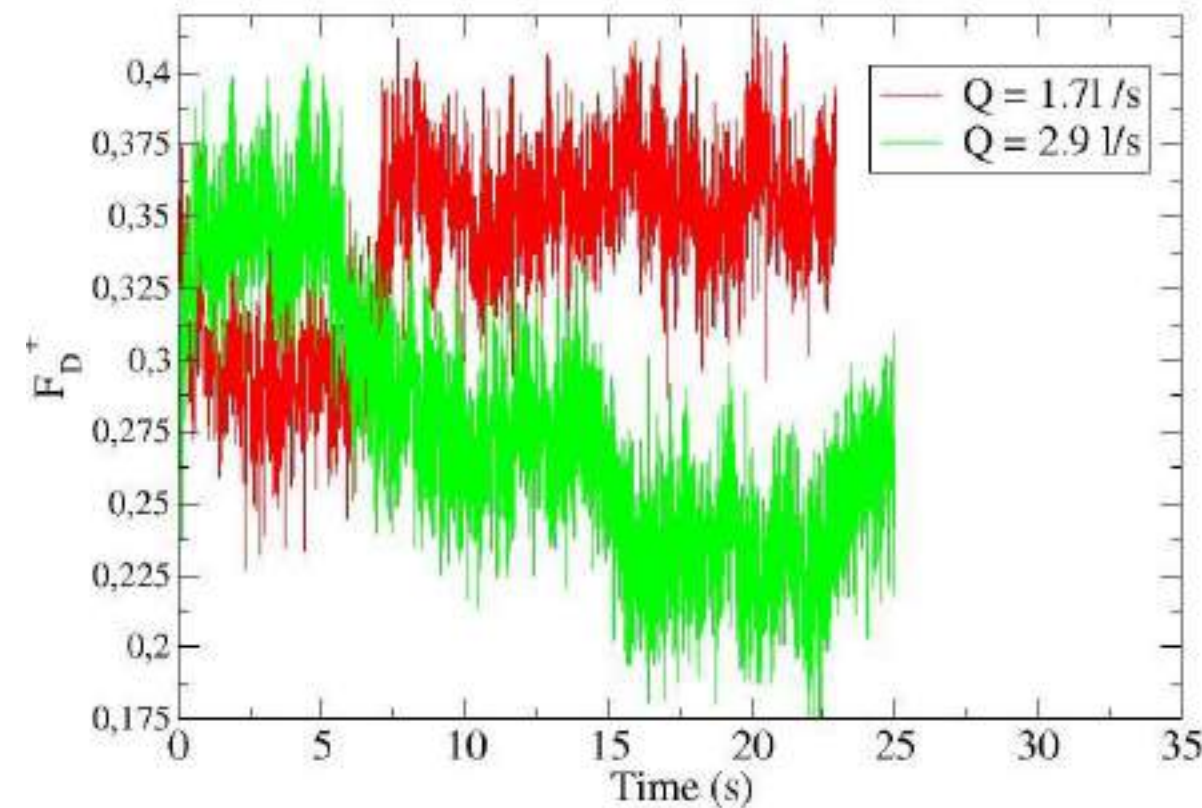
$Q = 1.7 \text{ l/s}$



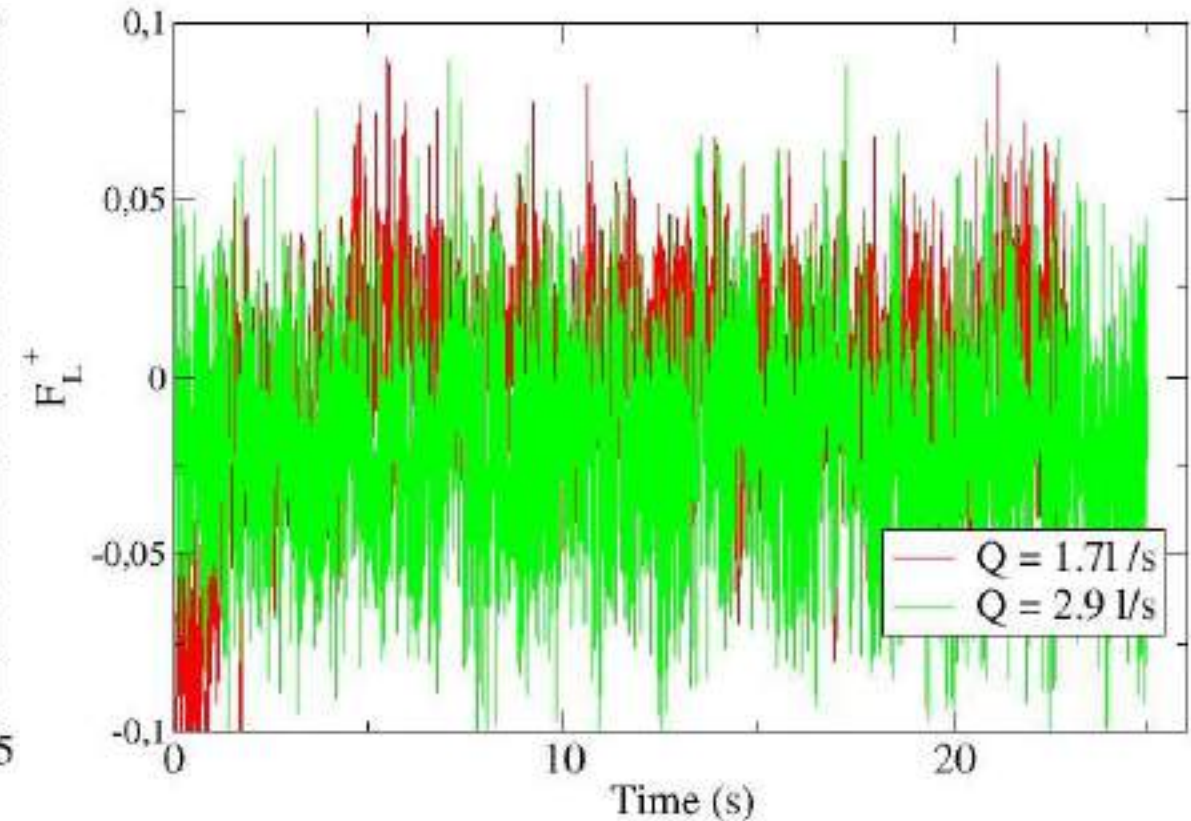
$Q = 2.9 \text{ l/s}$

5x5 rigid tube bundle – Asymmetry of the flow – possible switches

- Special care while computing the statistics (long enough: few tens of seconds, in a given stable position)



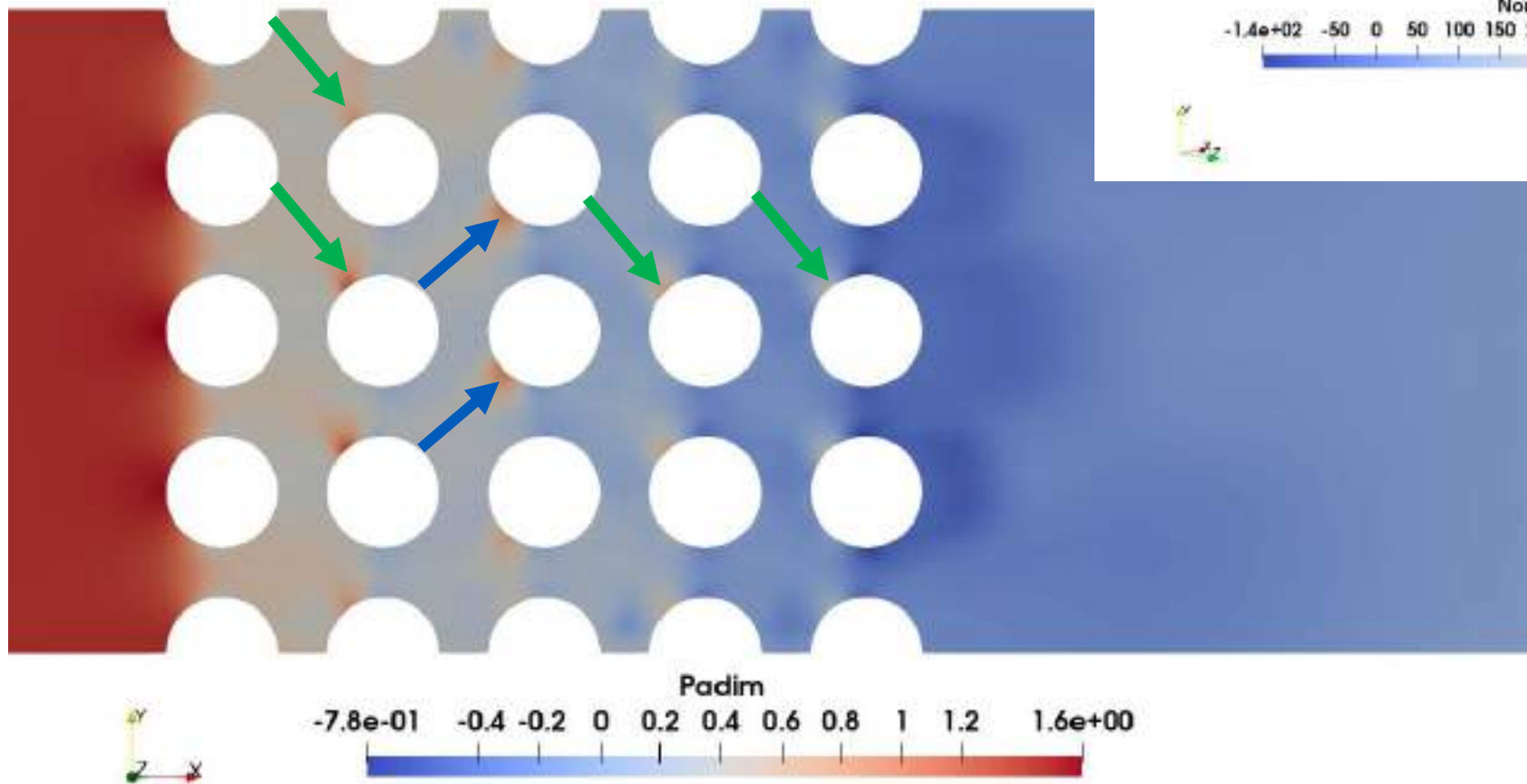
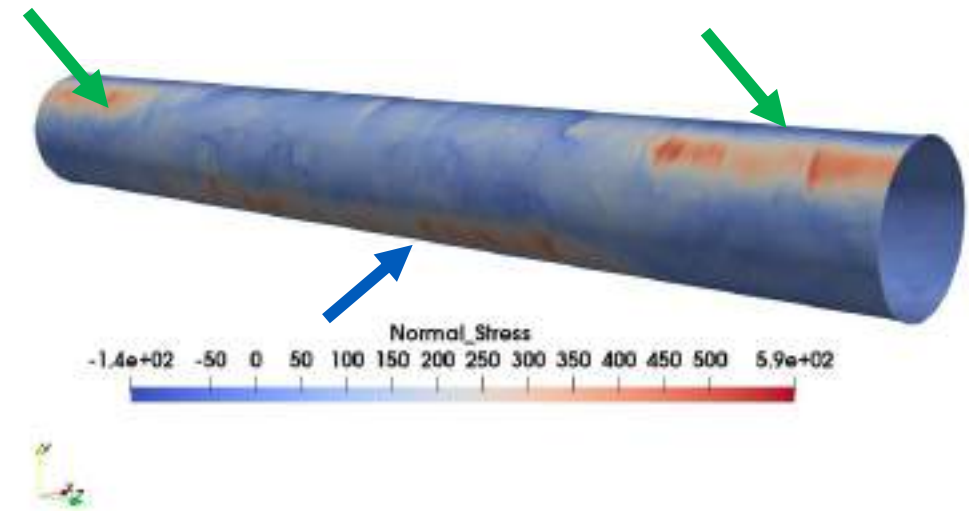
Tube (5,2) – last row

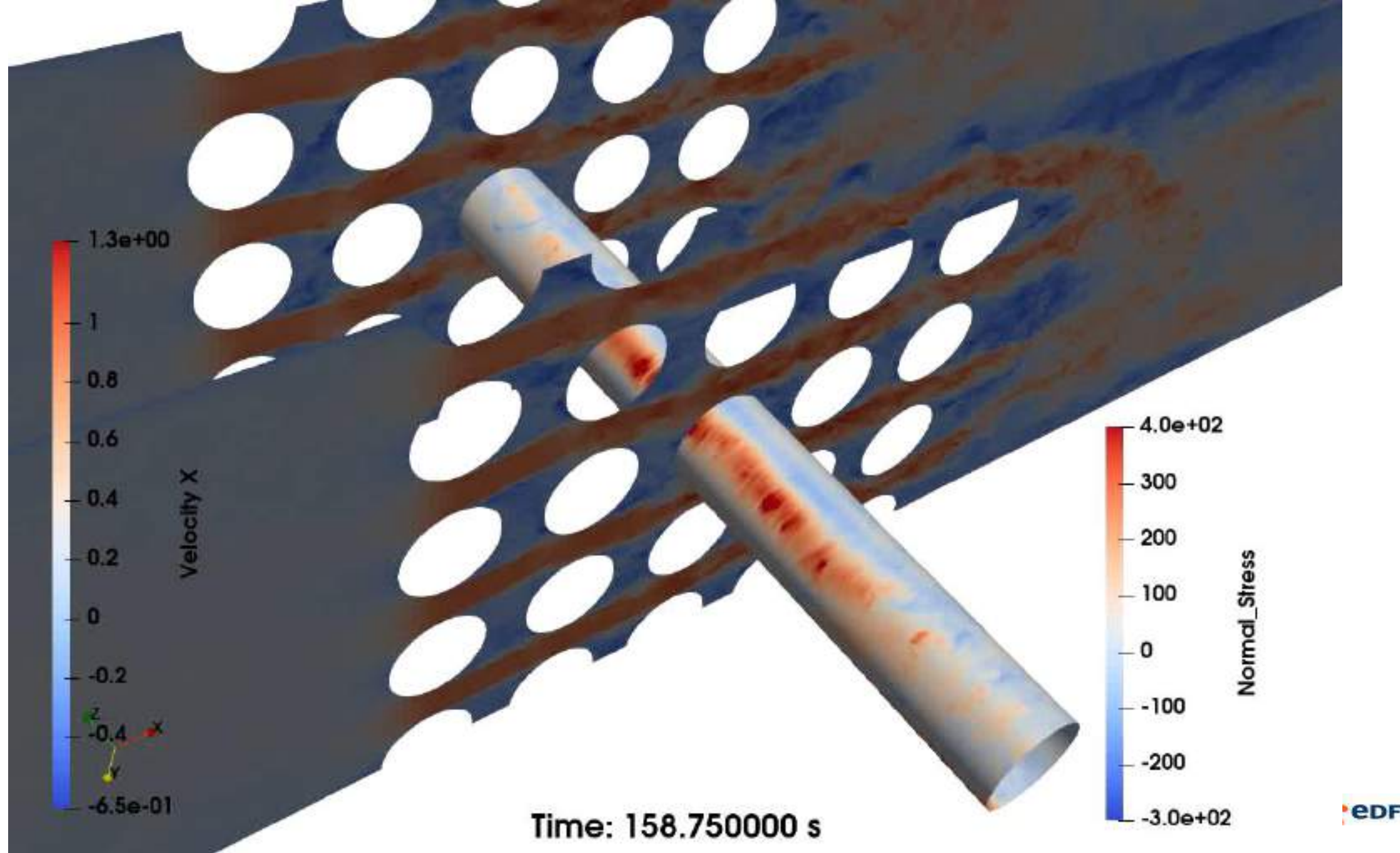


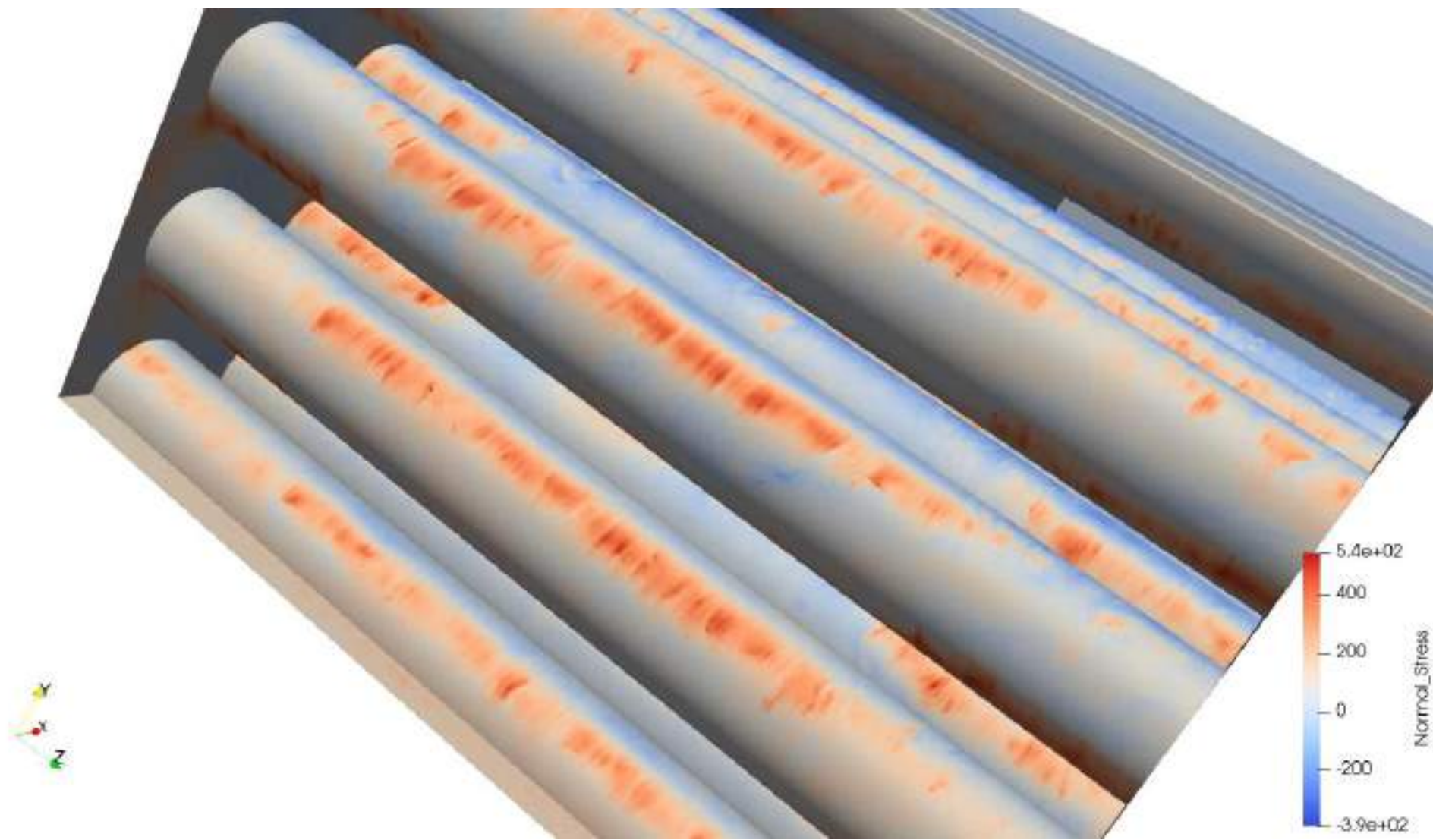
Tube(3,2) (central)

5x5 rigid tube bundle – Flow reversal in the span-wise direction

- Flow reversal close to the side-walls!

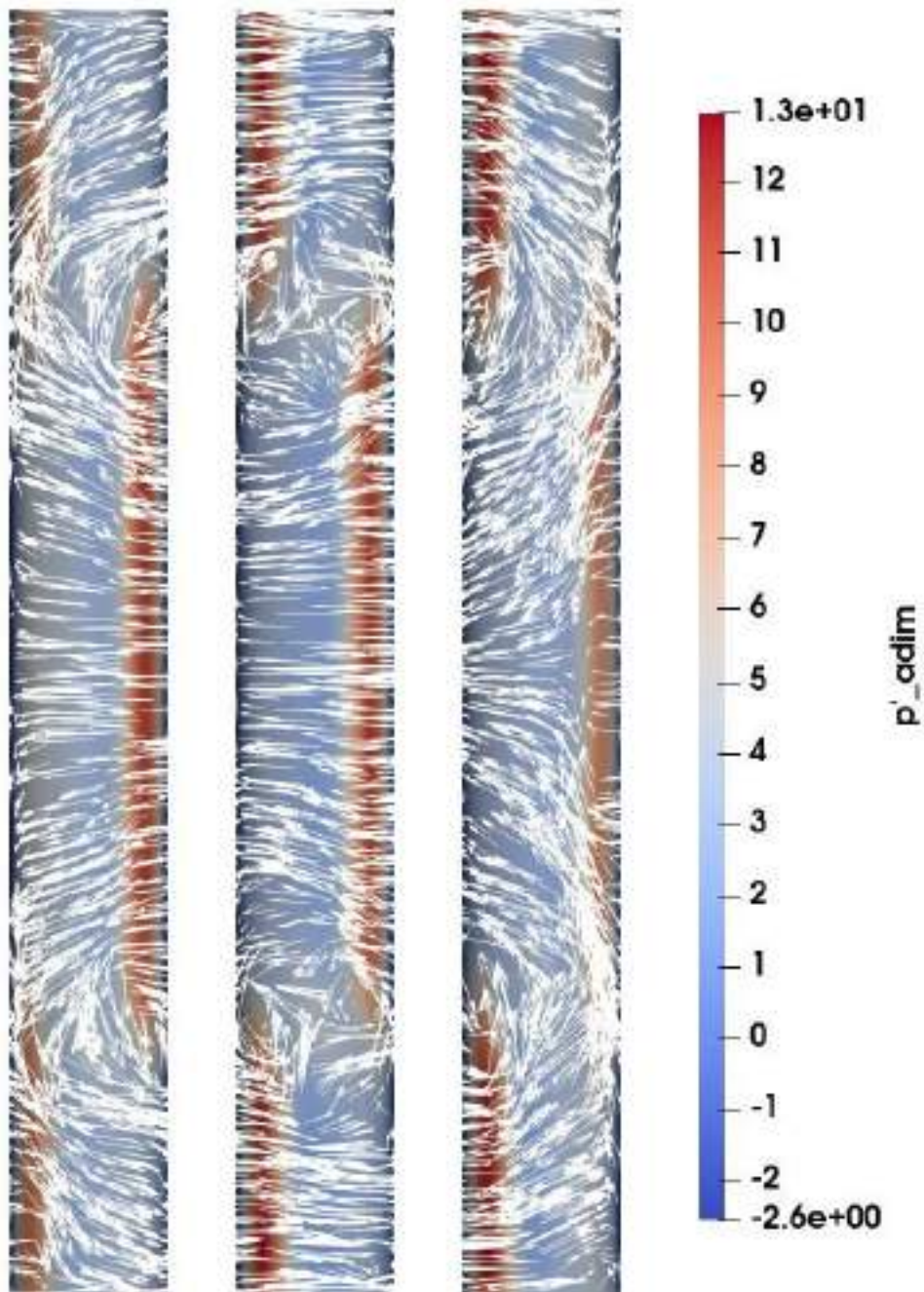
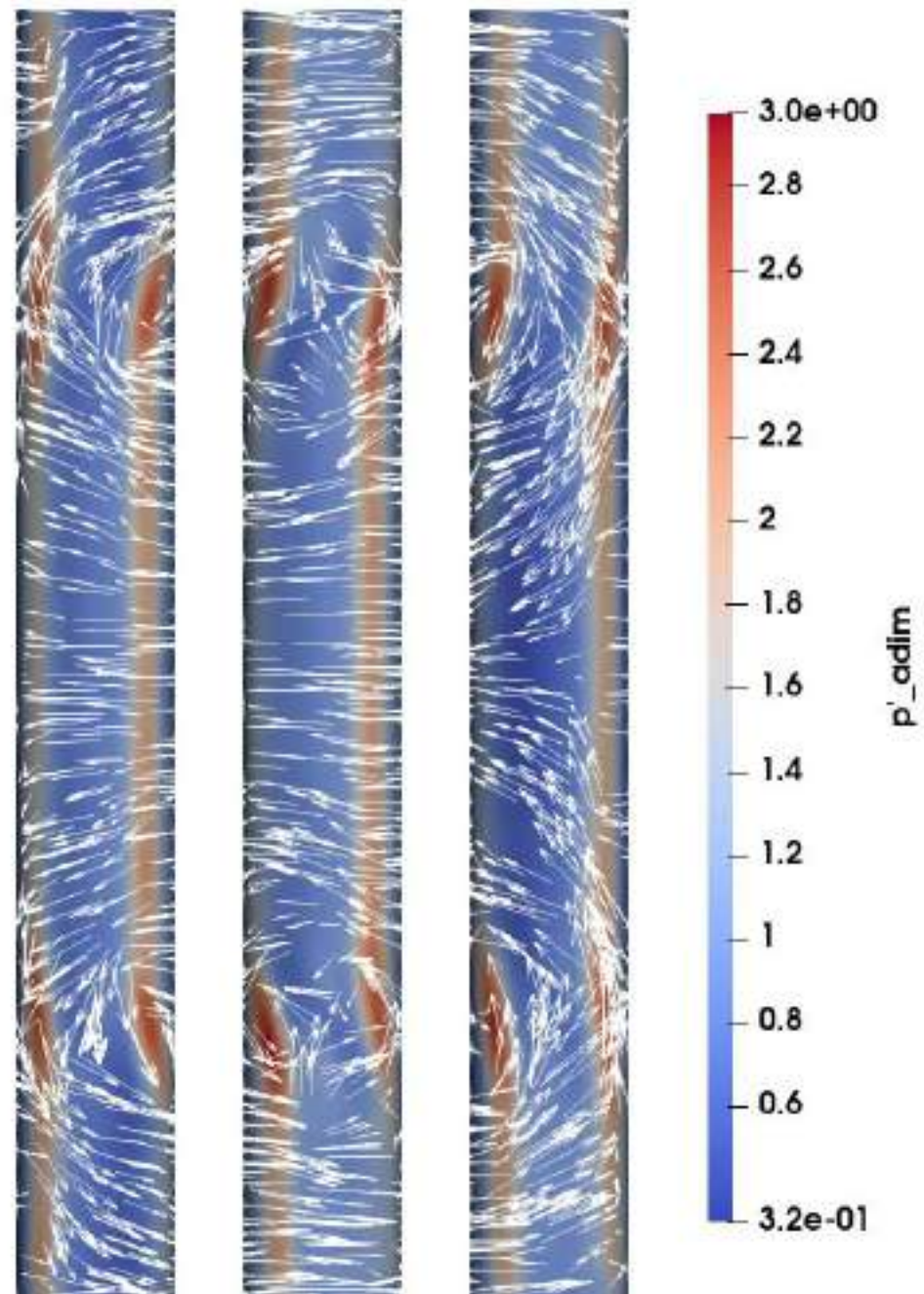






- Strong secondary structures up-stream the tubes





Overview



I. 5x5 rigid tube bundle

- a. Computational domain / Numerics / BC / Meshes
- b. Asymmetry of the flow / Flow reversal in the span-wise direction
- c. **Central cylinder Lift and drag forces fluctuations results**

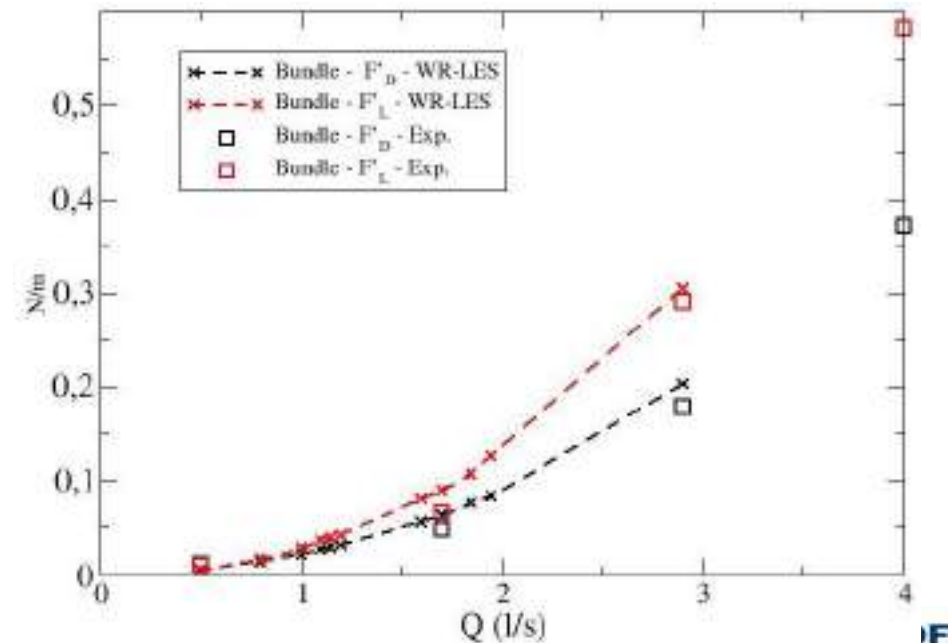
II. 5x5 tube bundle with a flexible central cylinder

III. Conclusions and perspectives

5x5 rigid tube bundle – central cylinder lift and drag forces fluctuations results

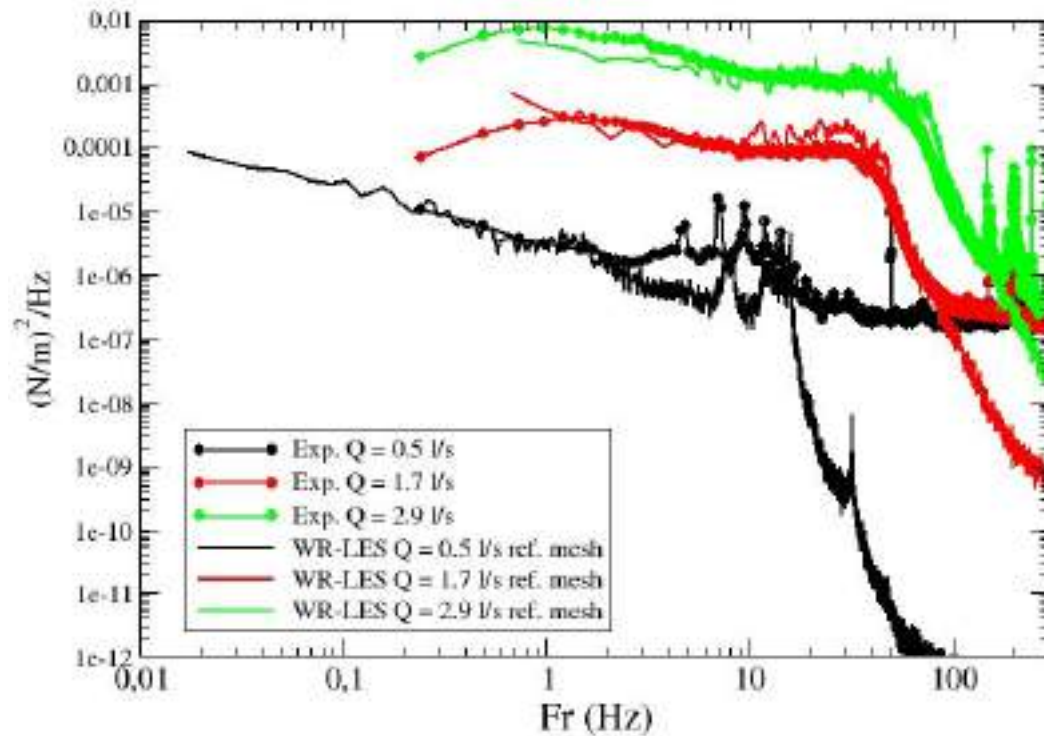
- The errors of the predicted drag and lift forces per unit of length compared to the experiment are equal to 29% and 35%, respectively for $Q = 1.7$ l/s.
- The errors drop to 14% and 5% for $Q = 2.9$ l/s. This is a **very satisfactory results**, which shows that the more turbulent the case is, the closest the results are to the experimental data.

Q (l/s)	CFD/Exp.	time (s)	$\overline{F_D}$ (N/m)	$\overline{F_L}$ (N/m)	$\overline{F_D'^2}$ (N/m)	$\overline{F_L'^2}$ (N/m)
0.5	Exp.	-	-	-	0.011	0.01
0.5	CFD	570.9	0.072	-0.001	0.004	0.005
1.7	Exp.				0.048	0.066
1.7	CFD	28.9	1.394	0.0174	0.062	0.089
2.9	Exp.				0.179	0.291
2.9	CFD	27.1	3.802	-0.2153	0.204	0.305

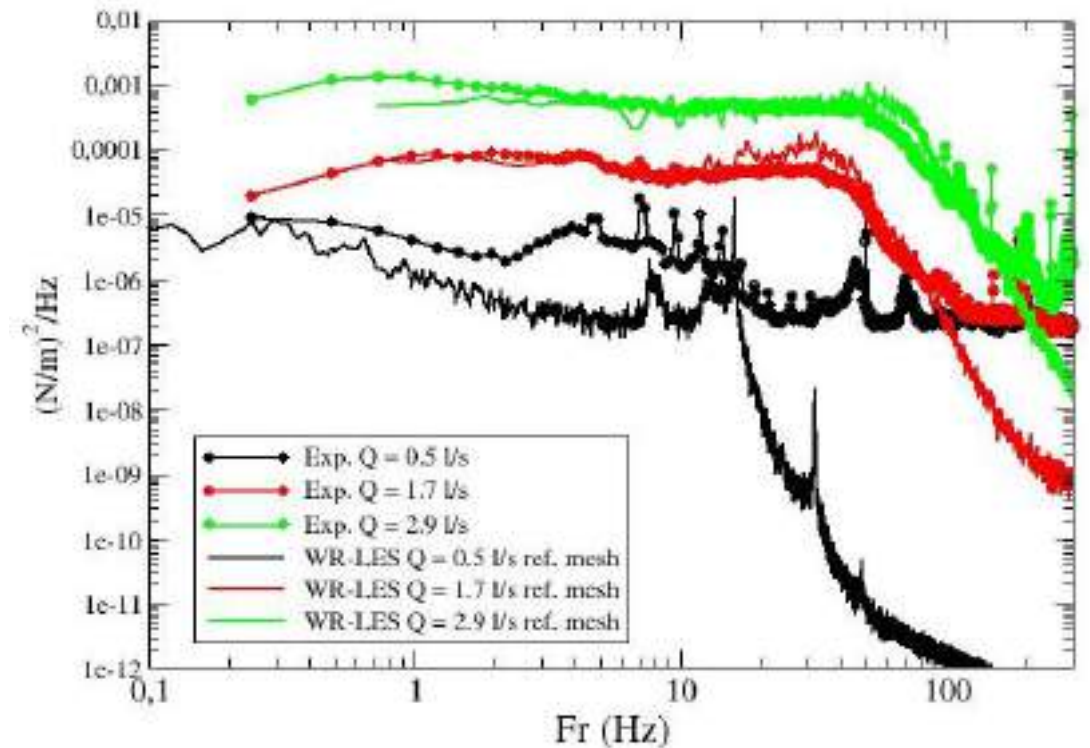


5x5 rigid tube bundle – central cylinder lift and drag forces fluctuations results

- Very good agreement for the DSP of the lift fluctuation at $Q = 1.7$ l/s and 2.9 l/s



PSD of the lift force per unit of length



PSD of the drag force per unit of length

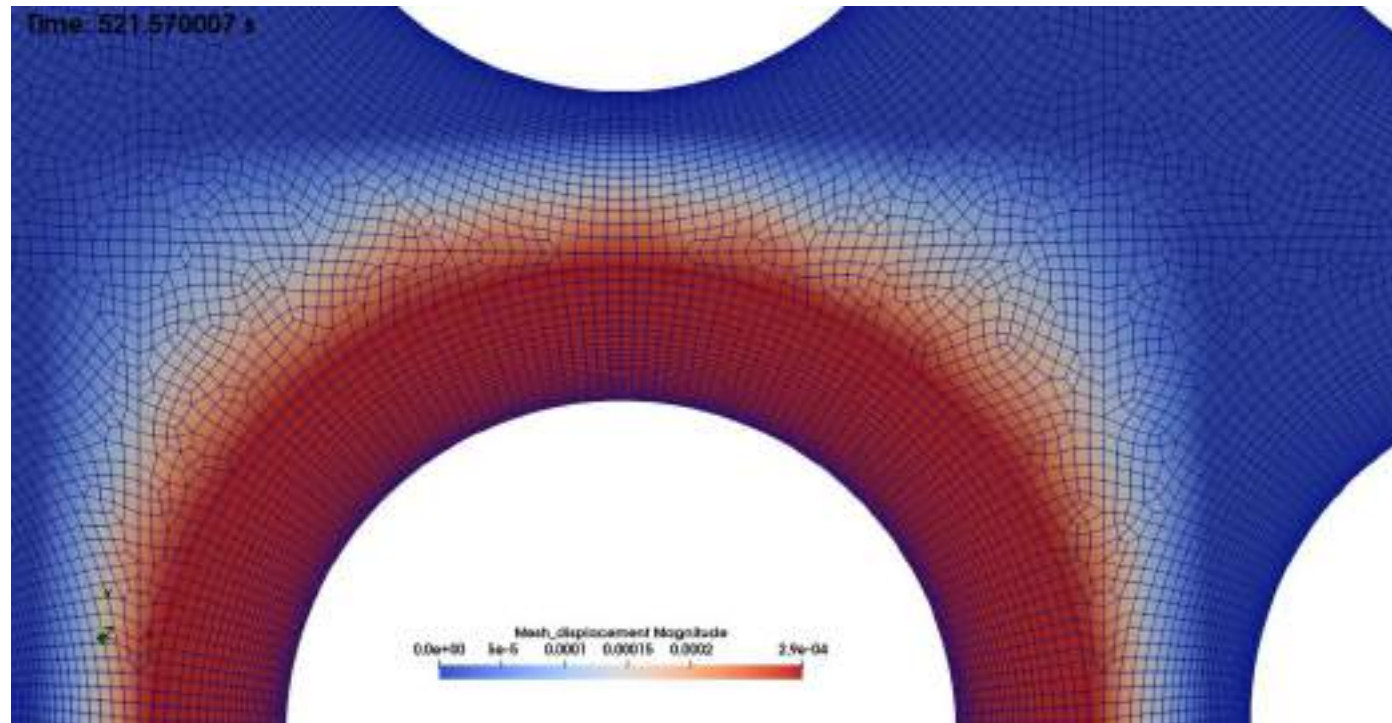
Overview



- I. 5x5 rigid tube bundle
 - a. Computational domain / Numerics / BC / Meshes
 - b. Asymmetry of the flow / Flow reversal in the span-wise direction
 - c. Central cylinder Lift and drag forces fluctuations results
- II. **5x5 tube bundle with a flexible central cylinder**
- III. Conclusions and perspectives

5x5 tube bundle with a flexible central cylinder – displacements in the lift direction results

- For all the computations with moving cylinder $M = 45,2622 \text{ g}$, $k = 1542,92 \text{ N/m}$: $f = \frac{1}{2\pi} \sqrt{\frac{k}{M}} \approx 29.38 \text{ Hz}$
- Rigid body mass-spring (no-damping) with Newmark time scheme.
- Arbitrary Lagrangian Eulerian approach (ALE) for moving the mesh nodes.
- Two-way coupling: maximum of 10 outer iterations with a precision of 10^{-5} .
- The forces on the cylinder are computed at each time step.



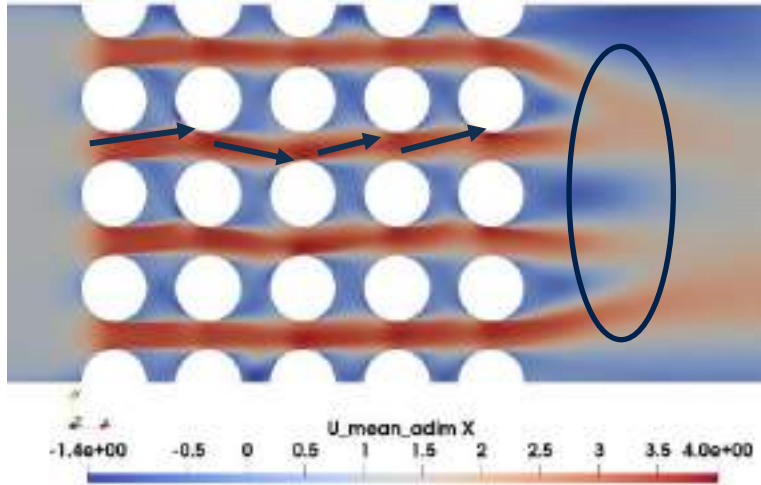
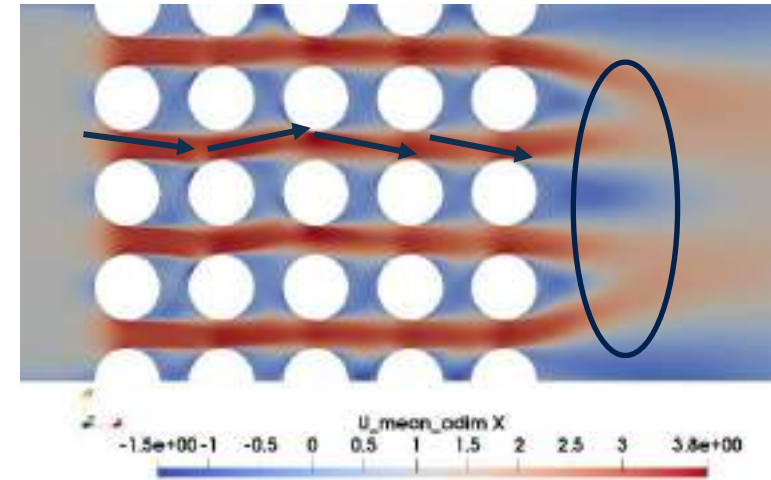
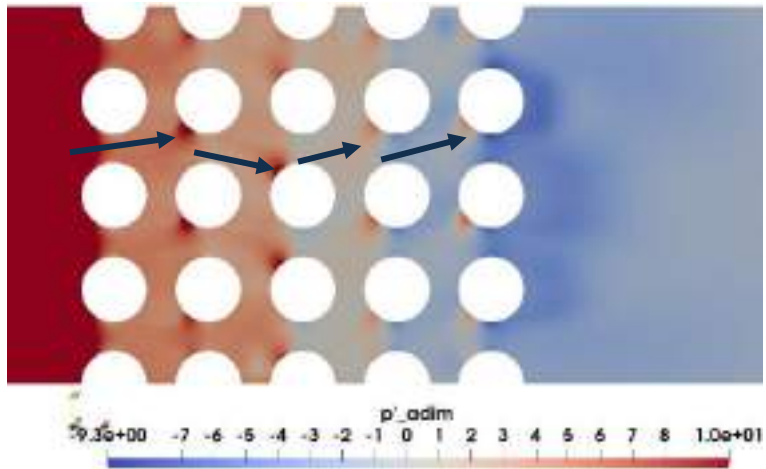
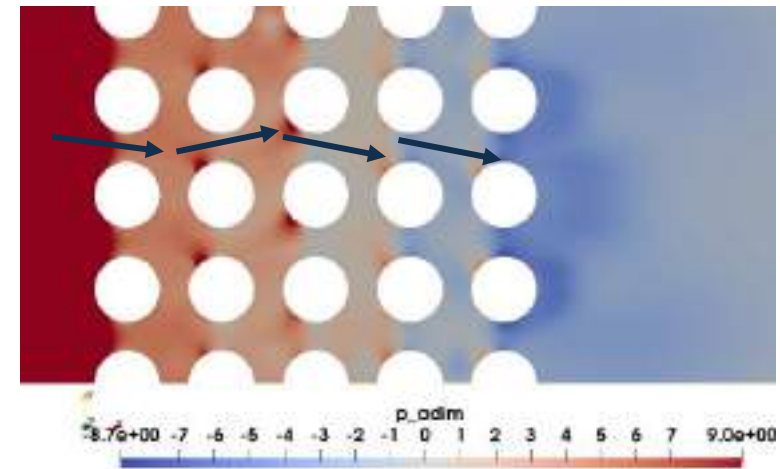
5x5 tube bundle with a flexible central cylinder – displacements in the lift direction results

- 4 flowrates 0.5 l/s, 1.1 l/s, 1.6 l/s and 1.95 l/s (a lock-in occurs at 1.1 l/s; not observed experimentally)
- Need to coarsen the mesh at 1.95 l/s (the one of 1.6 l/s the reference one)
- The results are lot affected but the topology of the flow is not exactly the same

Q (l/s)	mesh	δ_{bl} (m)	Δy_{cyl} (m)	Δy_{out} (m)	$\Delta x_{cyl} / \Delta y_{cyl}$
1.95	ref.	1.782E-03	1.800E-05	1.960E-04	10
1.95	coarse	2.431E-03	1.800E-05	2.610E-04	14

Q (l/s)	mesh	n_z	n_θ	n_{prec} (Million)	n_{cells} (Million)
1.95	ref.	444	268	22.3	174.4
1.95	coarse	444	212	16.6	124.5

- Coarse vs. ref. mesh: different jets in the bundle / similar wake of the bundle

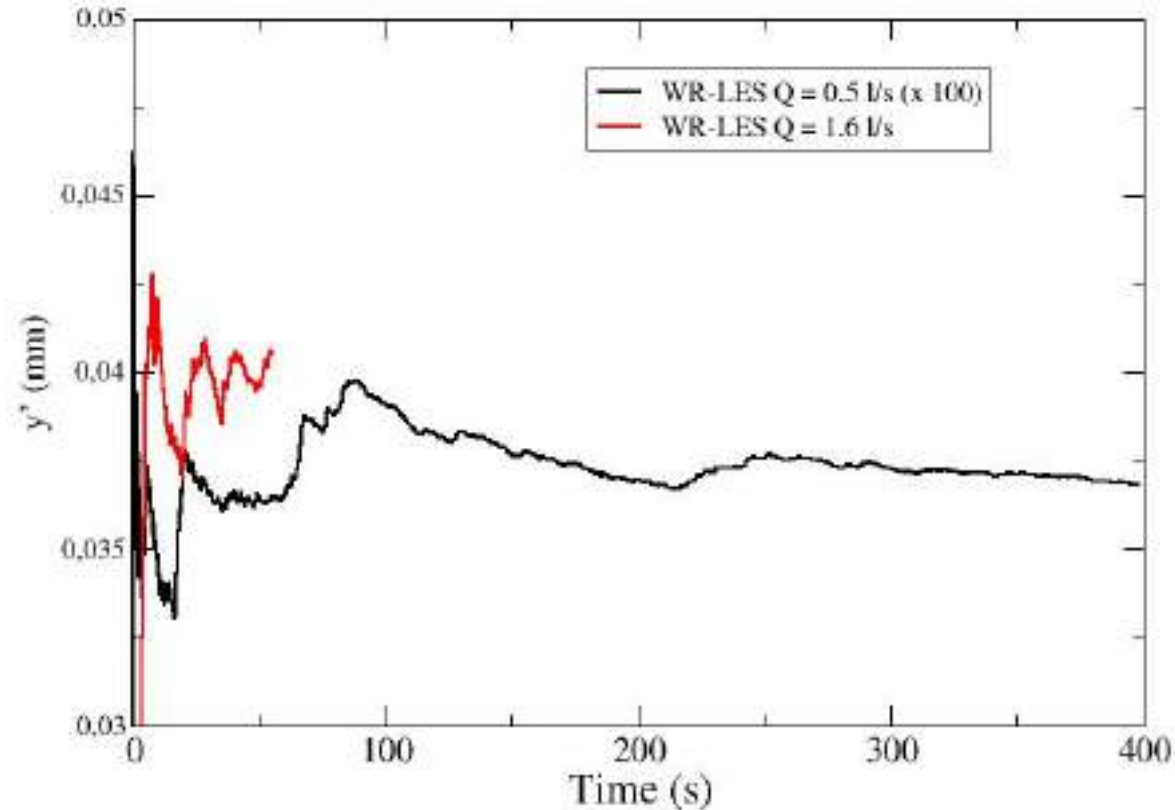


$Q = 1.95$ l/s – coarse mesh

$Q = 1.95$ l/s – ref. mesh

5x5 tube bundle with a flexible central cylinder – displacements in the lift direction results

- Special care while computing the statistics (long enough: few tens of seconds, in a given stable position)

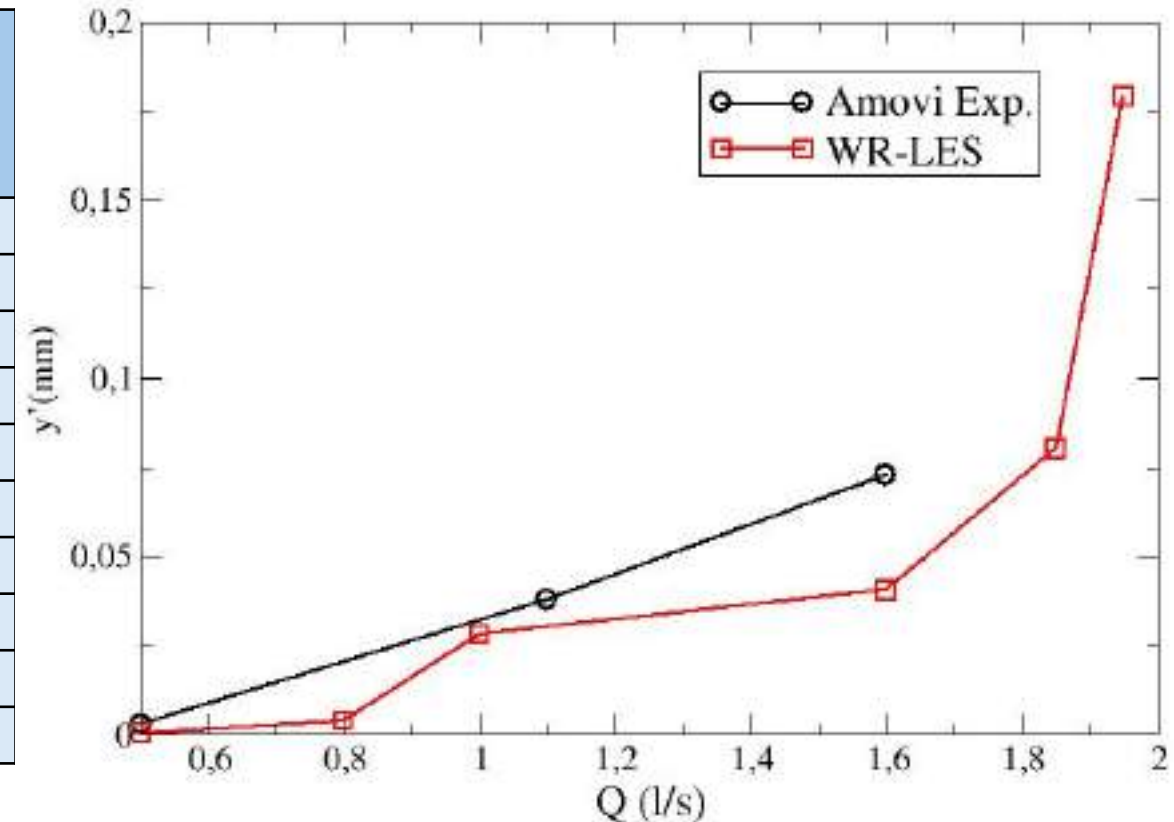


The standard deviation of the displacement as a function of time for $Q = 0.5\ l/s$ and $Q = 1.6\ l/s$ – reference meshes -

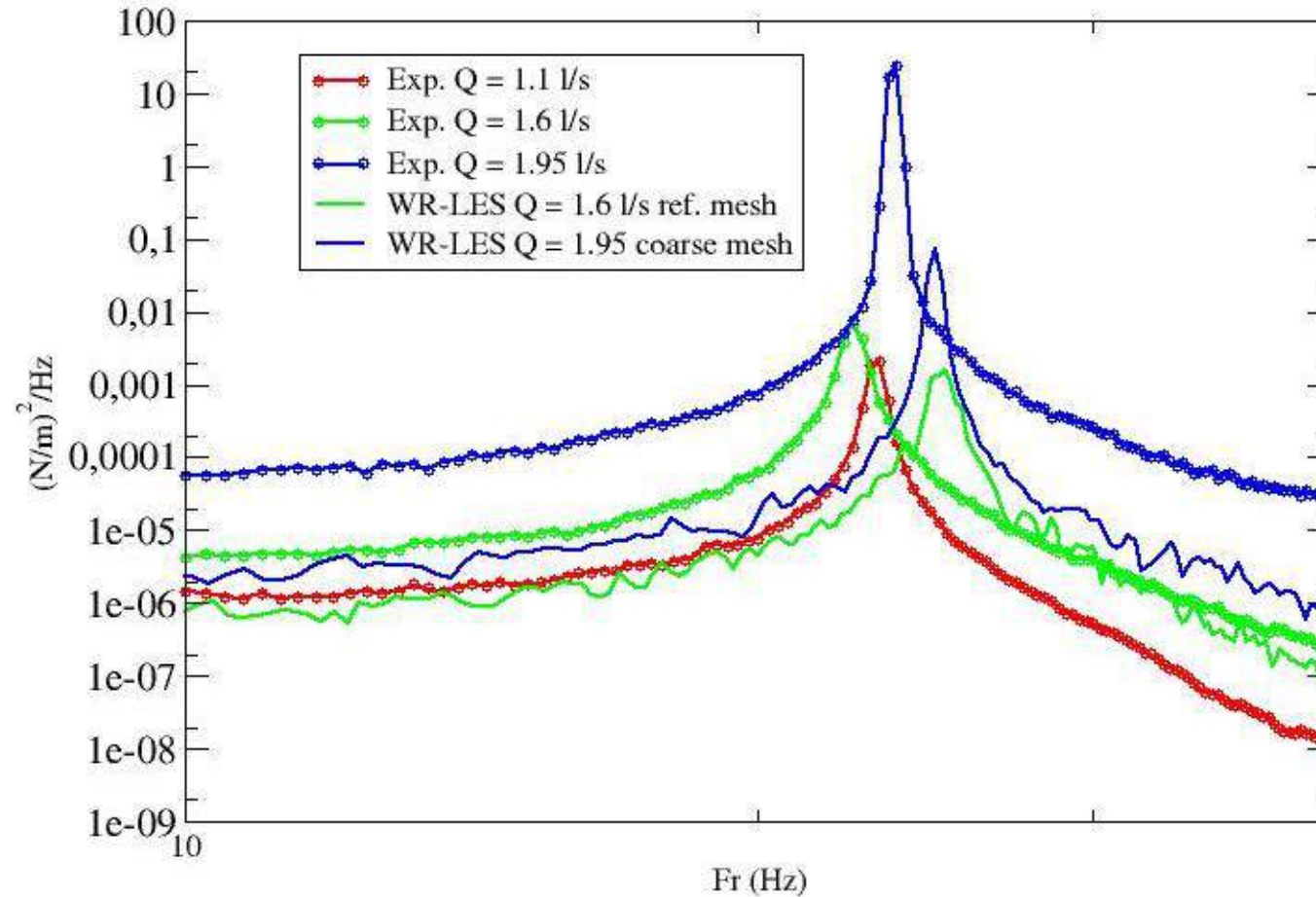
5x5 tube bundle with a flexible central cylinder – displacements in the lift direction results

- Fluid elastic instability is predicted

Q (l/s)	CFD/Exp.	$\bar{y}$ (mm)	$\overline{y'^2}$ (mm)	Fr_y (Hz)
0.5	Exp.	-	0.003	-
0.5	CFD	-0.0001	0.0003	-
0.8	CFD	0.0005	0.00385	24.86
1	CFD	0.0012	0.0282	24.85
1.1	Exp.	-	0.038	23.05
1.6	Exp.	-	0.073	22.53
1.6	CFD	0.0036	0.0405	25
1.85	CFD	-0.003	0.0802	24.85
1.95	Exp.	-	3.223	23.56
1.95	CFD	-0.0039	0.1791	24.81

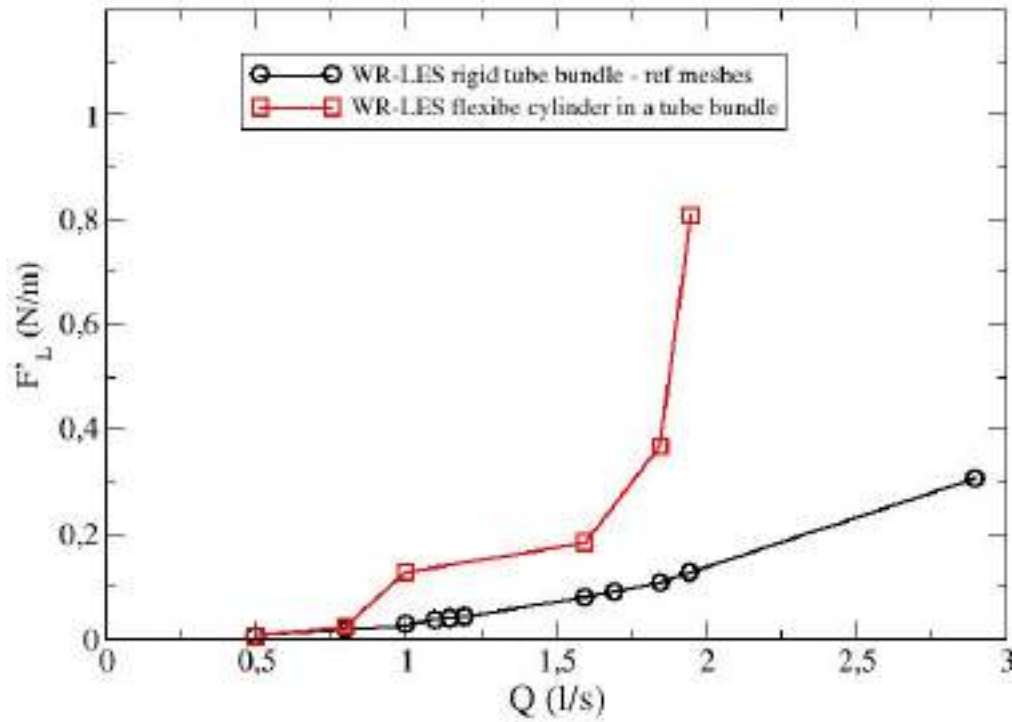


5x5 tube bundle with a flexible central cylinder – displacements in the lift direction results

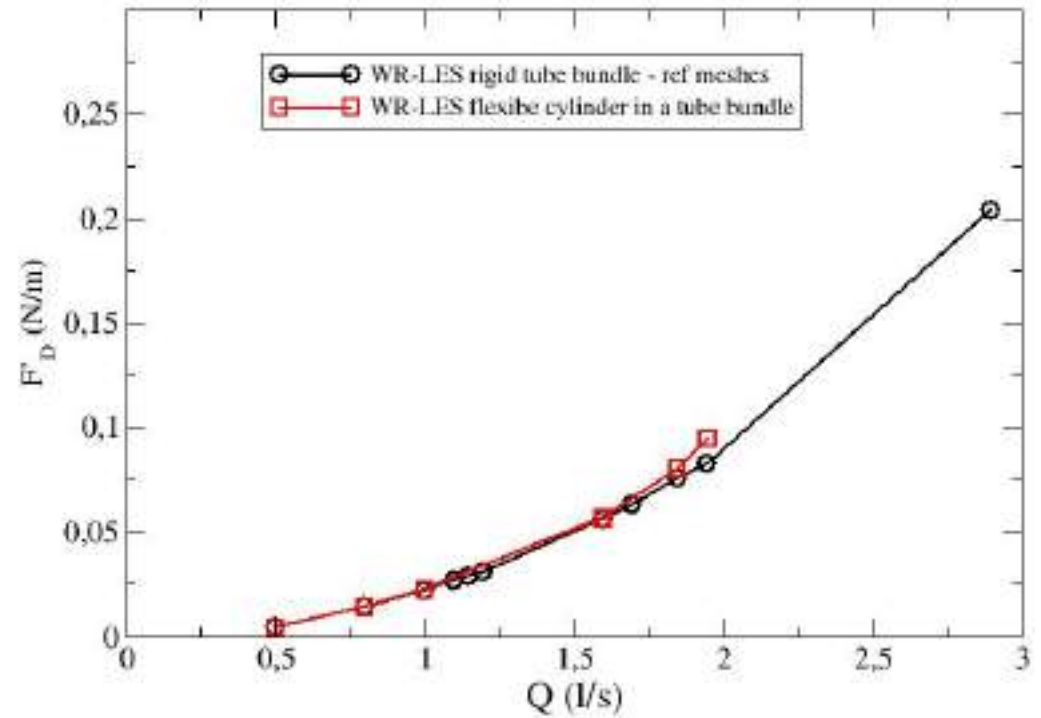


Power Density Spectra (PDS) of the tube displacement at $Q = 1.6$ l/s and $Q = 1.95$ l/s – flexible cylinder in a rigid tube bundle configuration – comparison with the experimental data

5x5 tube bundle with a flexible central cylinder – displacements in the lift direction results

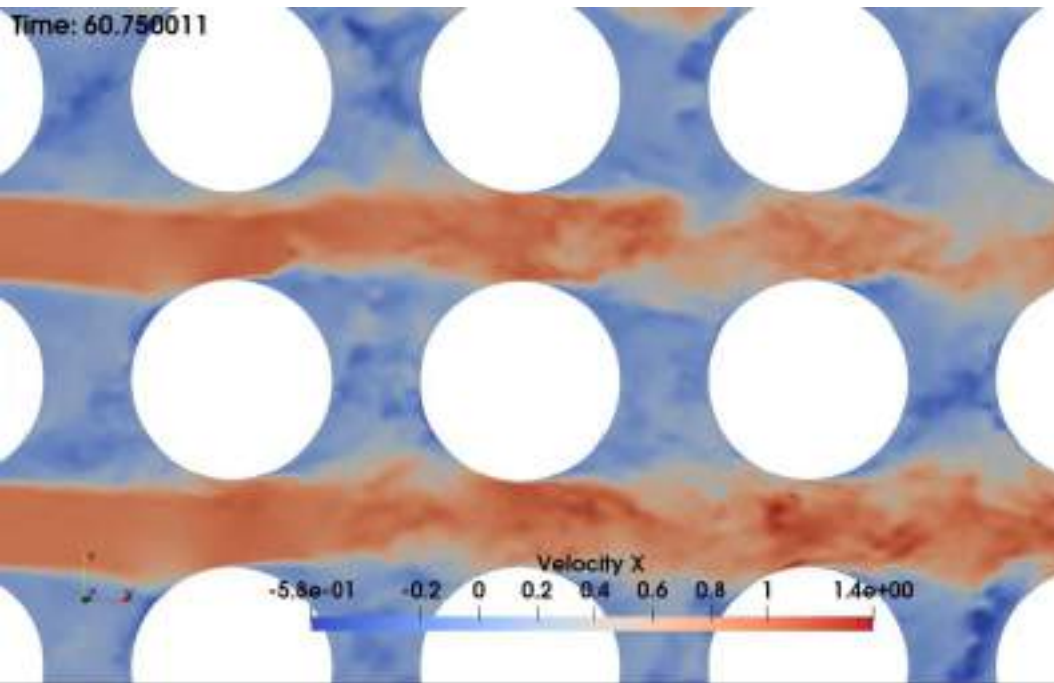
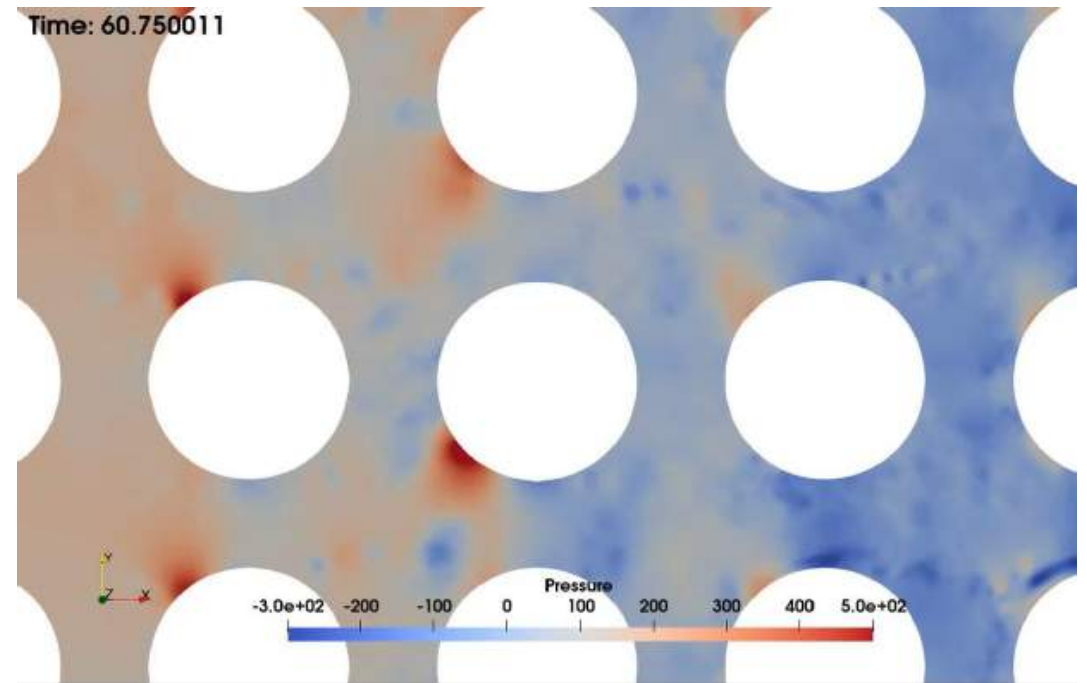


Lift force



Drag force

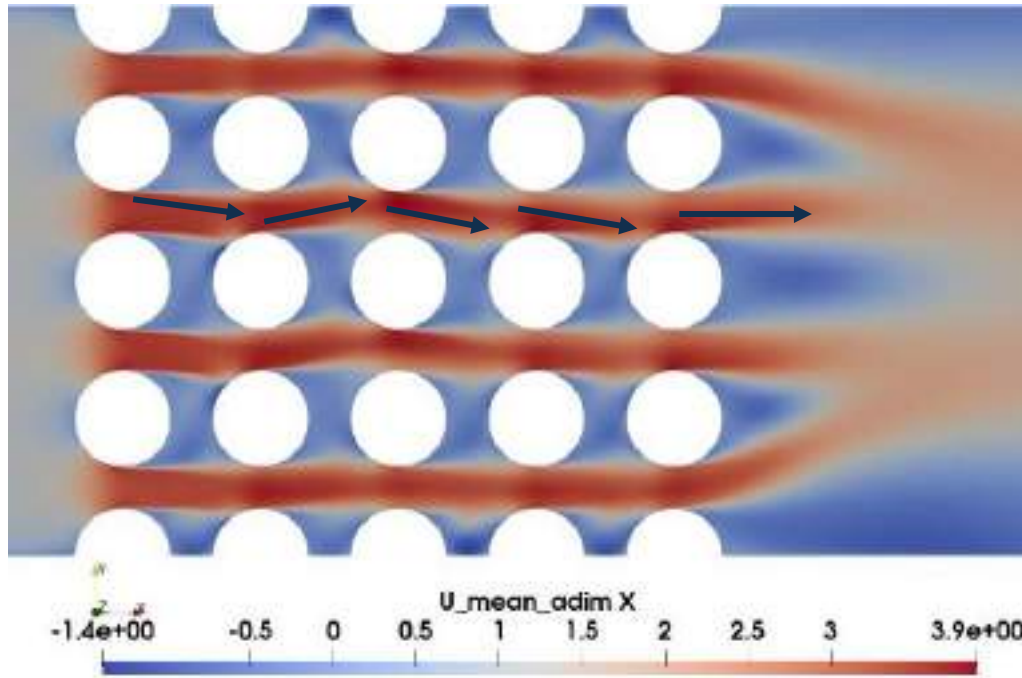
5x5 tube bundle with a flexible central cylinder – displacements in the lift direction results



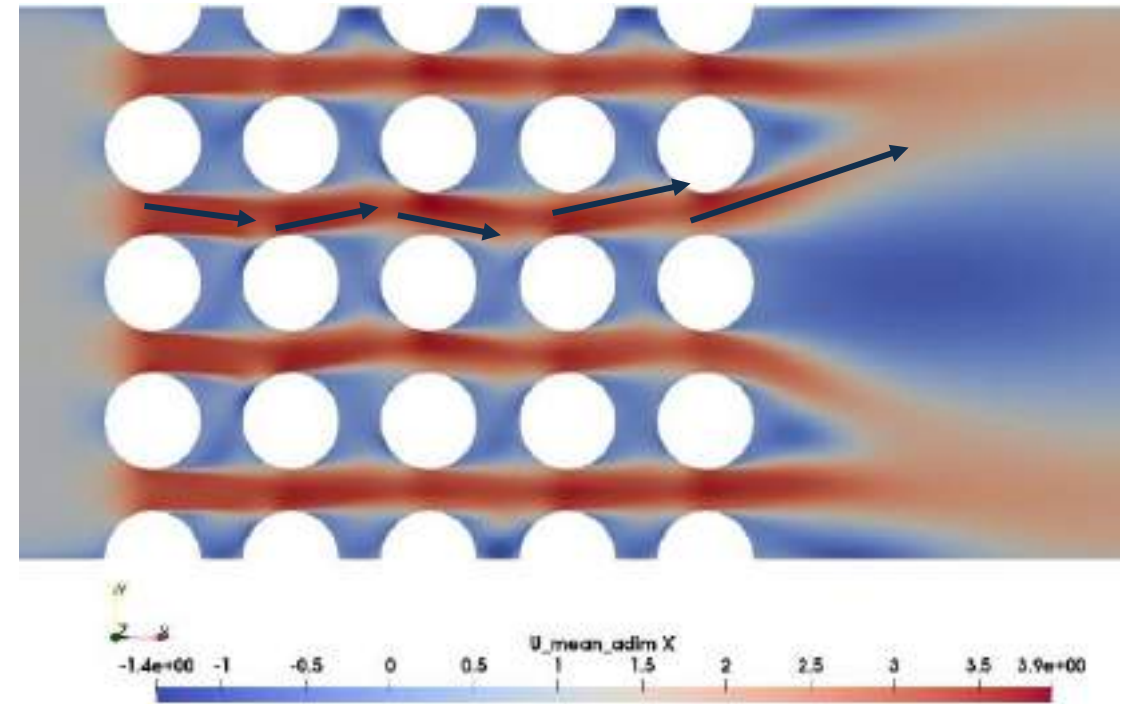
$Q = 1.95 \text{ l/s}$ – coarse mesh

5x5 tube bundle with a flexible central cylinder – displacements in the lift direction results

- $Q = 1.7$ l/s rigid vs. $Q = 1.6$ l/s flexible with ref. mesh: almost jets in the bundle / different wake of the bundle



$Q = 1.6$ l/s – flexible configuration



$Q = 1.7$ l/s – rigid configuration

Overview



- I. 5x5 rigid tube bundle
 - a. Computational domain / Numerics / BC / Meshes
 - b. Asymmetry of the flow / Flow reversal in the span-wise direction
 - c. Central cylinder Lift and drag forces fluctuations results
- II. 5x5 tube bundle with a flexible central cylinder
- III. **Conclusions and perspectives**

Conclusions and perspectives

- For the Rigid tube bundle configuration
 - The results of the Std. dev. of the lift and drag forces are better as the Reynolds number increases (of course there is a sensitivity to the inlet boundary conditions and this should be kept in mind).
 - They are however satisfactory and exhibit the right trends.
 - The observed asymmetry consists in
 - impinging jets on the upper and lower parts upstream a cylinder.
 - Multiple solutions are possible in the wake of the bundle
 - These jets show a reversal while approaching the side-walls. This has to be deeply investigated and is related apparently to a secondary flow.

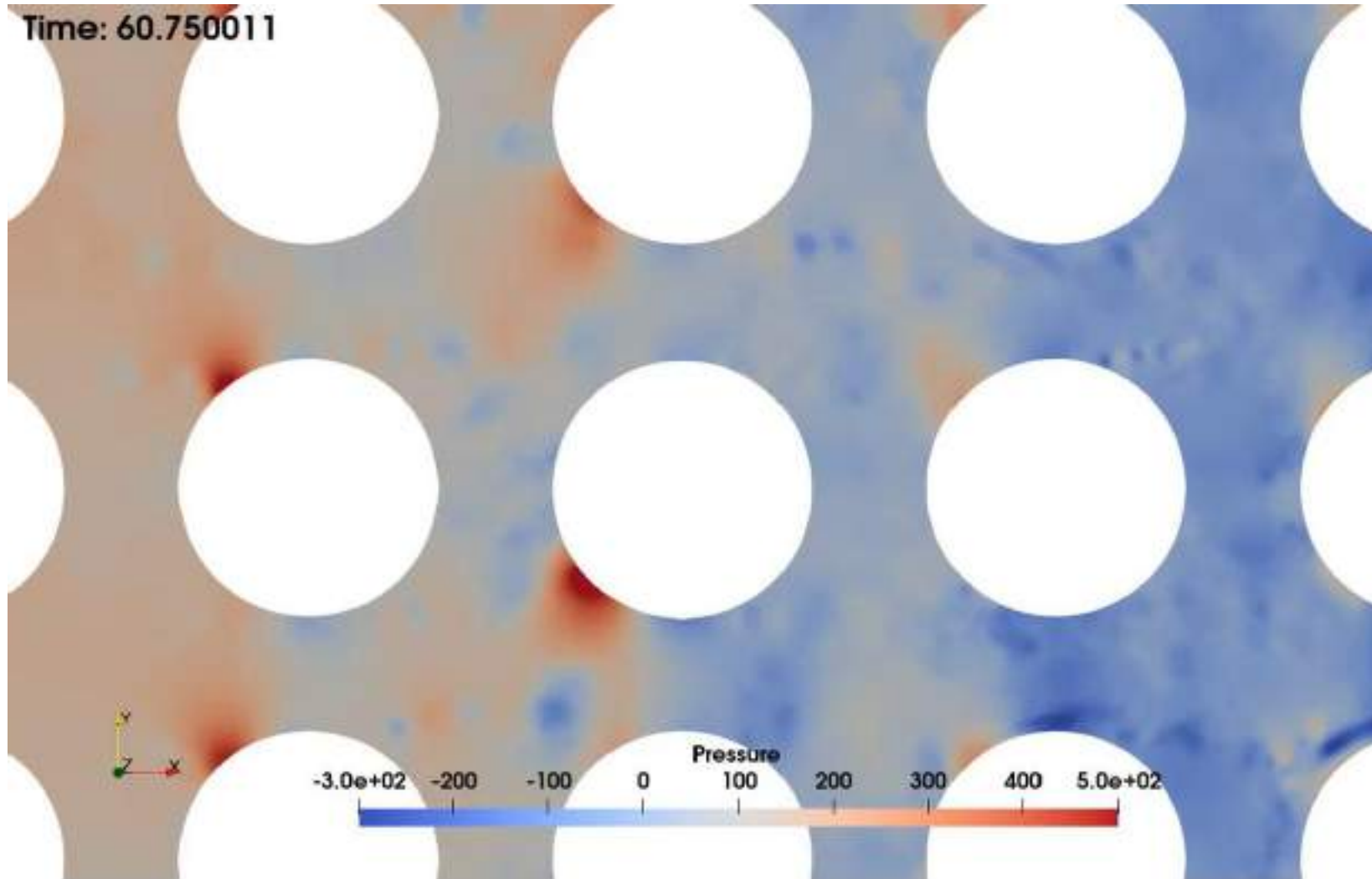


Conclusions and perspectives

- For the tube bundle configuration with a central flexible tube
 - The results obtained with wall-resolved LES are overall satisfactory both for the displacements amplitudes and the frequencies.
 - The configuration of the moving cylinder in a bundle at $Q = 1.95$ l/s has been successfully run with a coarsened mesh to allow larger displacements: the fluid elastic instability is predicted.
 - A lock-in at $Q = 1.1$ l/s is observed (and the computation is also unstable at $Q=1.05$ and 1.15 l/s). This has to be more deeply investigated.



Thank you!





Validation of ANSYS CFX using the Rigid Body Motion approach on the AMOVI experiment

Hemish Mistry, Angel Papukchiev, GRS

GO-VIKING Final Workshop, 20-22.05.2026, GRS gGmbH, Germany



Funded by
the European Union

Outline



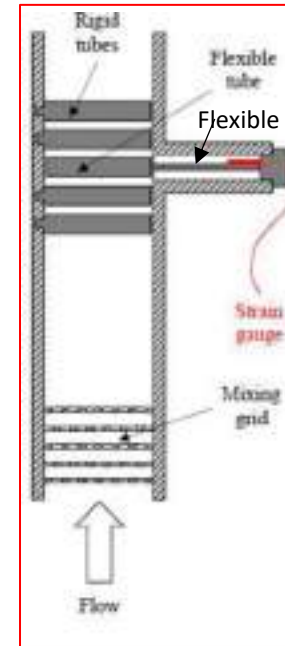
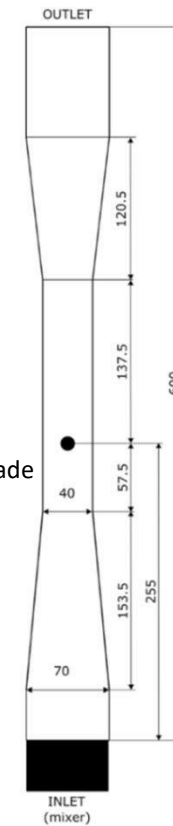
- AMOVI experiment
- CAD modelling
- Mesh generation
- Single tube and tube bundle cases
 - Force comparisons
 - Displacement comparisons
 - Flow pattern comparison
- Conclusion

AMOVl experiment

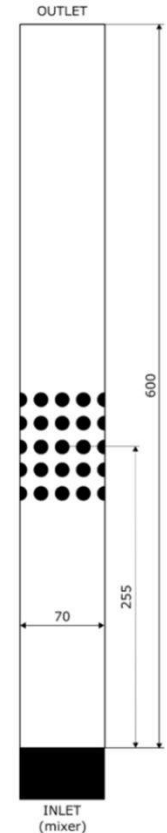
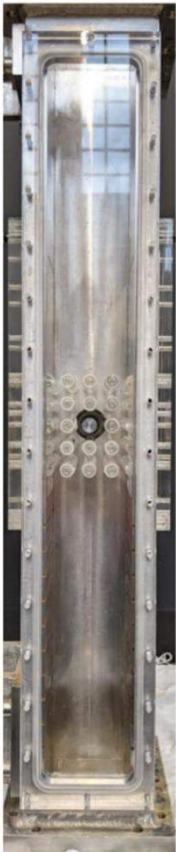


- AMOVl is built to measure tube vibration in single phase water flow. [1]
- Components:
 - Pump with max. flow rate of 4 L/s.
 - Storage tank of 200 L for pump.
 - Test section with 600 x 70 x 100 (length x width x depth) mm³.
 - Tube diameter: 12.15 mm
 - Tube bundle pitch: 17.496 mm

Experiment	Flow Rate, L/s	Pressure, bar	Temperature, °C
Single rigid tube	2.9	2.88	26
Single flexible tube (lift direction)	1.7	3.33	21
	2.9	3.20	25
Rigid tube bundle	1.7	0.90	18
	2.9	3.19	41
Flexible tube (lift direction) in a rigid tube bundle	1.6	1.07	17



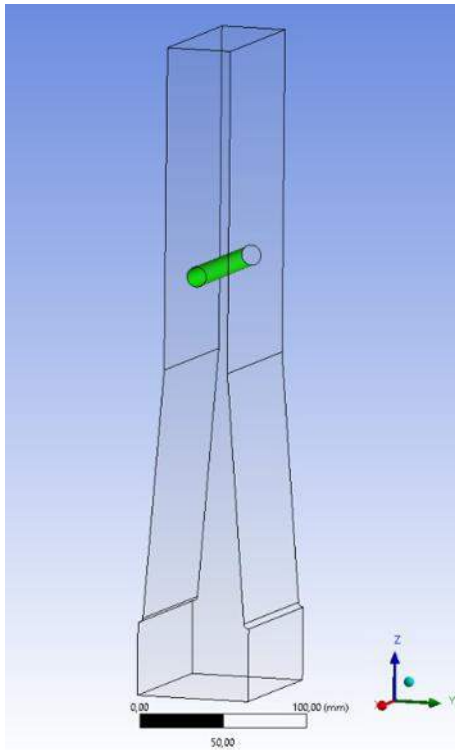
Flow direction



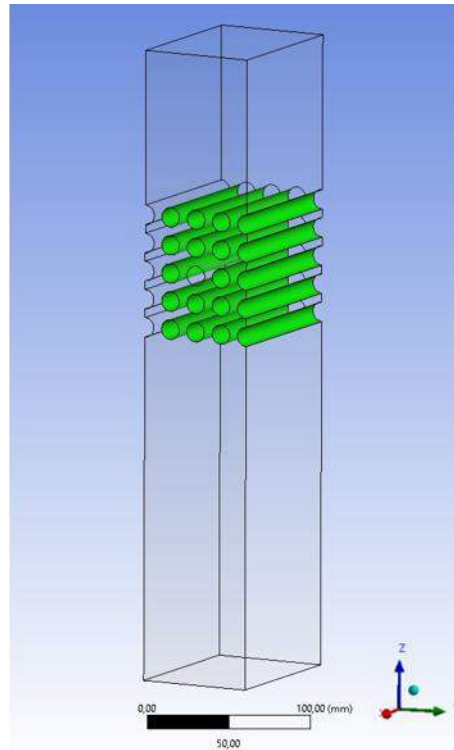
CAD modelling and meshing of AMOVI



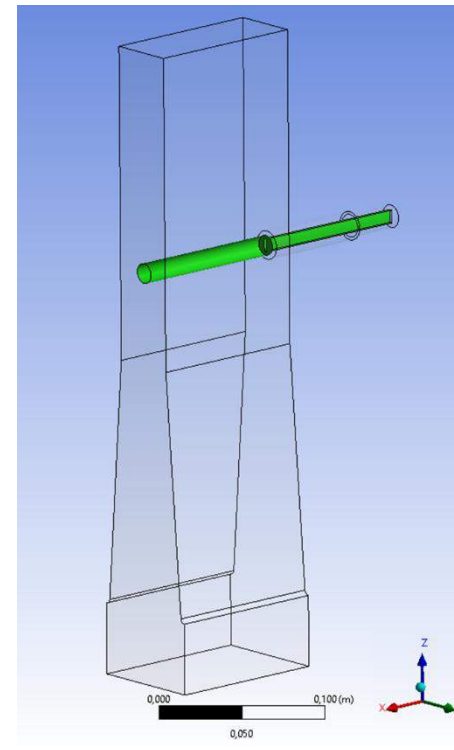
- CAD models (100 x 70 x 392.5 mm) generated based on CEA drawings



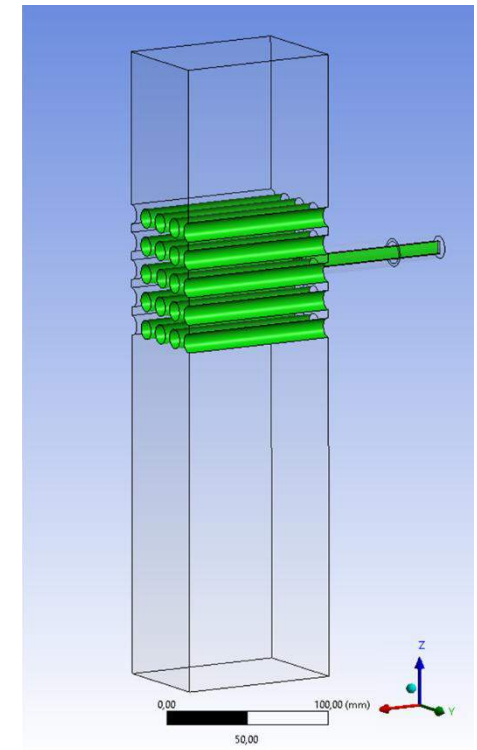
Force measurement



Flow direction



Displacement measurement

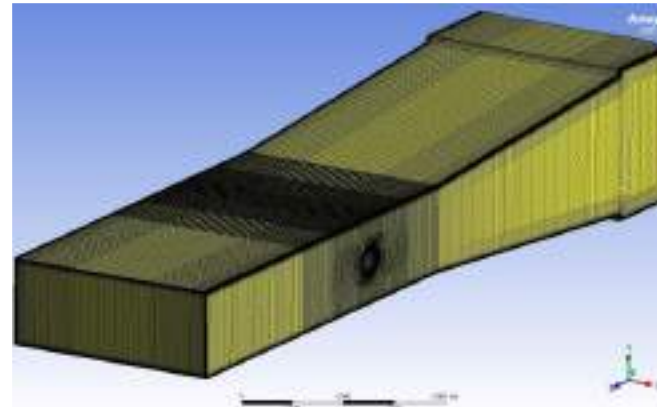
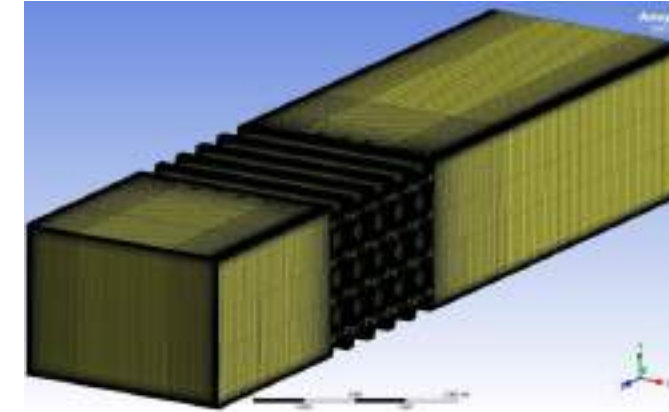


CAD modelling and meshing of AMOVI

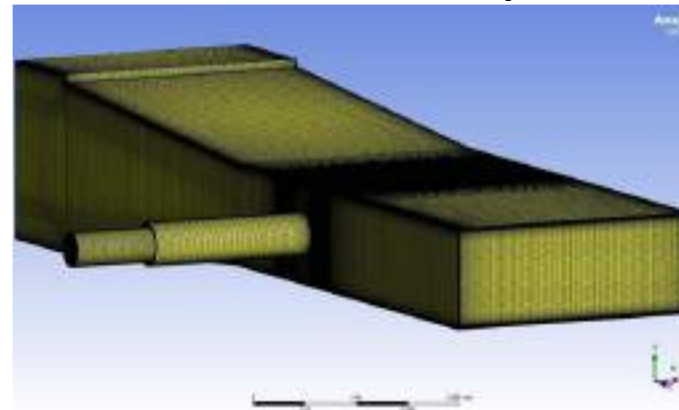
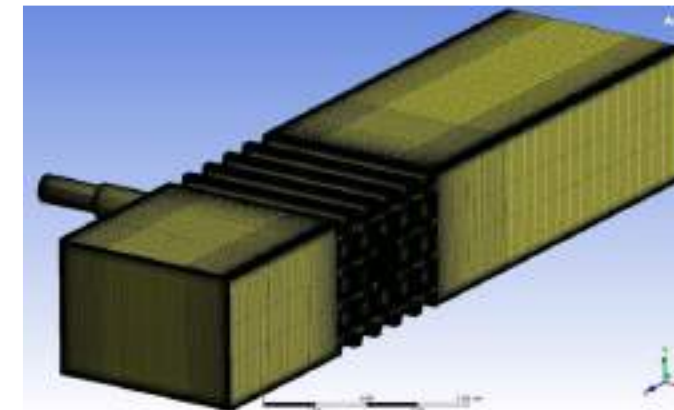


- Structured hex mesh- ANSYS Meshing
- Force measurement
 - Single tube: 5.15 M.
 - Tube bundle: 19 M.
- Displacement measurement
 - Single tube: 3.95 M.
 - Tube bundle: 19.6 M.
- Quality of meshes
 - Aspect ratio < 344
 - Min. orthogonality angle: 13°
 - Expansion factor: 88

Force measurement



Displacement measurement



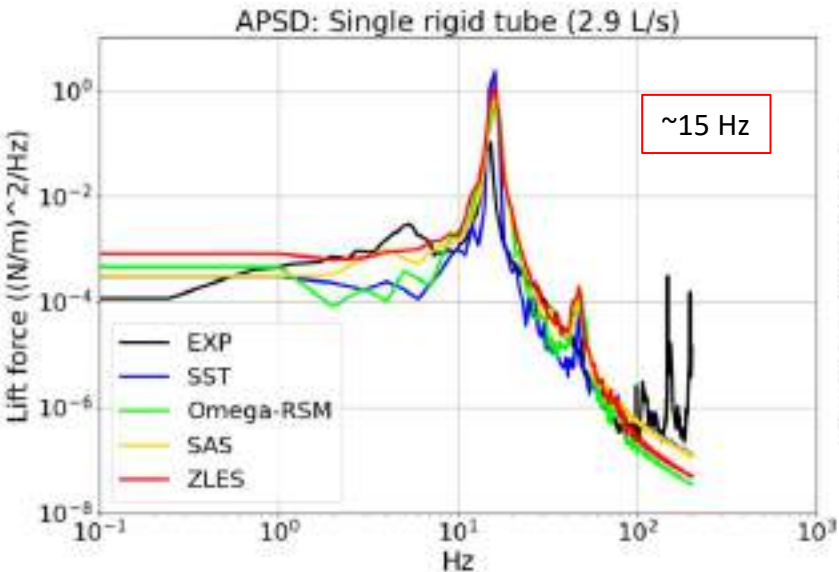
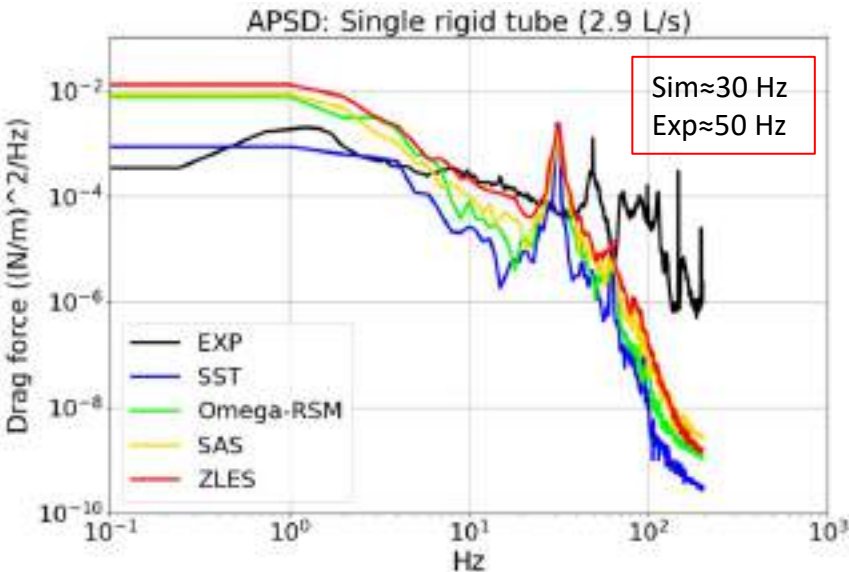
AMOVl: Force comparisons

- Results of Single Rigid Tube ($Q= 2.9 \text{ L/s}$)



- URANS and hybrid-URANS/LES simulation
 - Transport of quantities: High-Resolution advection scheme
 - Time discretization: Second Order Backward Euler scheme
- Different turbulence models: SST, RSM, SAS, ZLES
- Δt : 0.2 ms, t : 10 s
- Convergence, $<1E-5$
- $CFL < 0.5$

	Lift force (N/m)	Drag force (N/m)
EXP	0.411	0.12
SST	1.87	0.09
Omega-RSM	1.11	0.153
SAS	1.31	0.156
ZLES	1.5	0.197



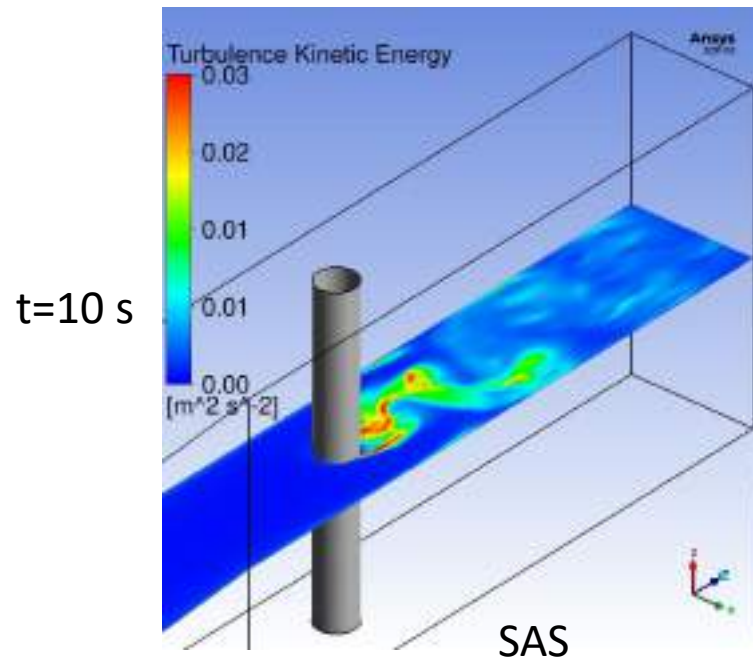
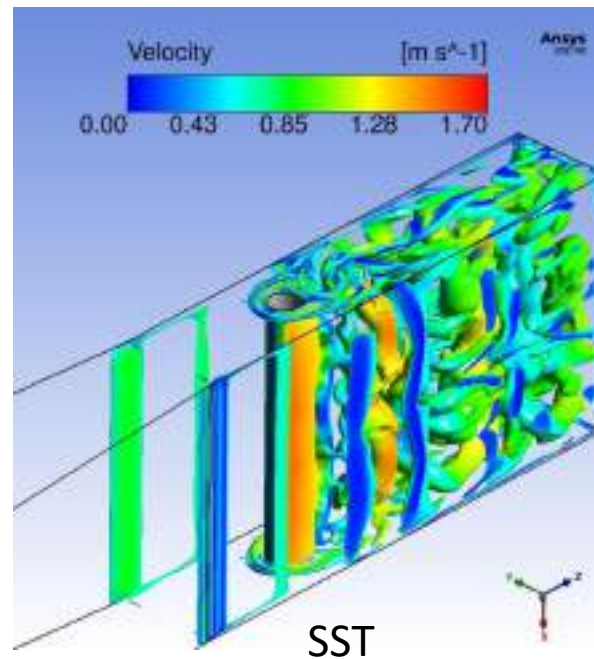
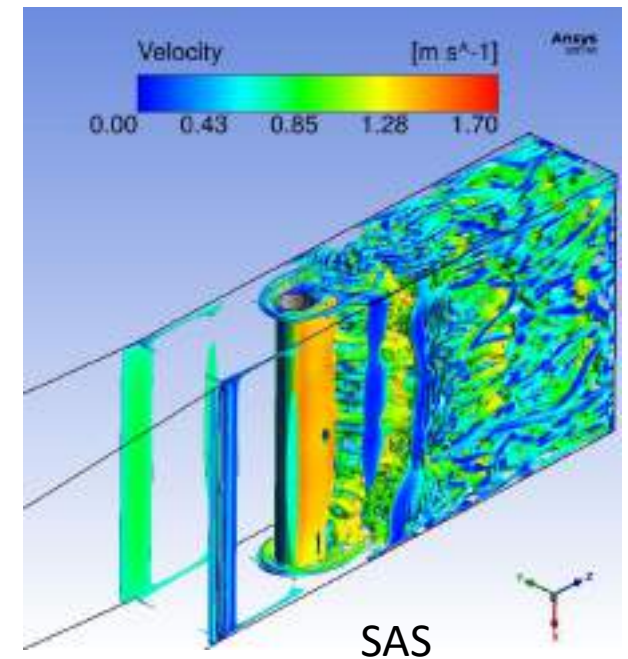
AMOVl: Flow pattern

- Results of Single Rigid Tube ($Q= 2.9 \text{ L/s}$)



- Von Kármán vortex street develops behind the tube with periodically detaching eddies.
- URANS simulation produces only the large-scale unsteadiness, whereas the SAS model adjusts to the already resolved scales in a dynamic way and allows the development of a turbulent spectrum in the detached regions.^[3]

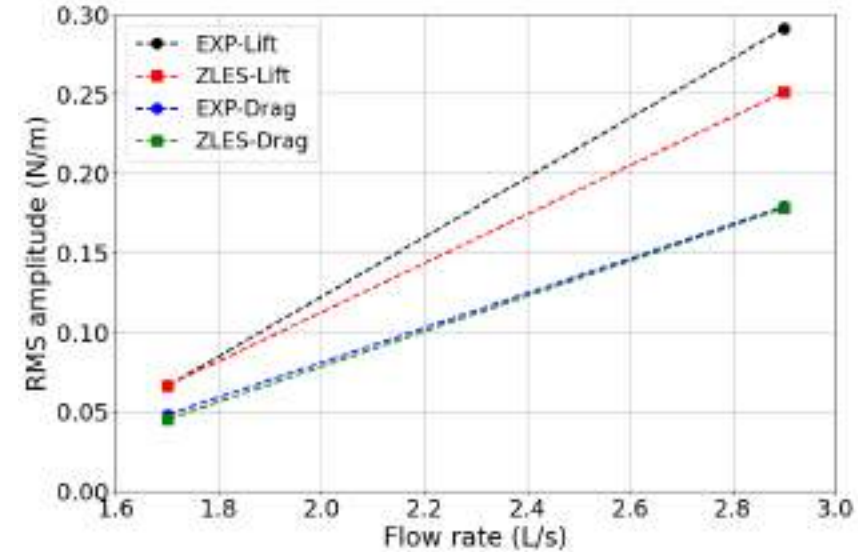
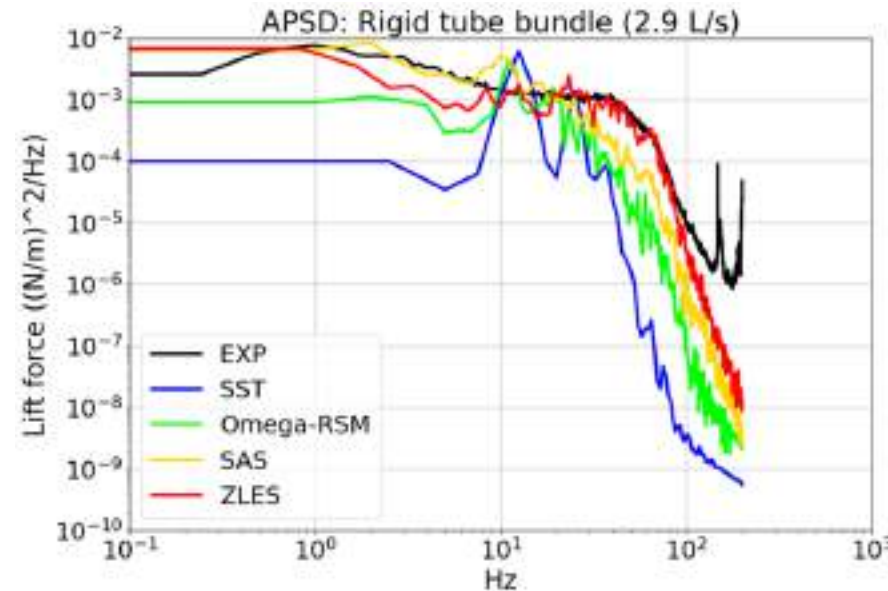
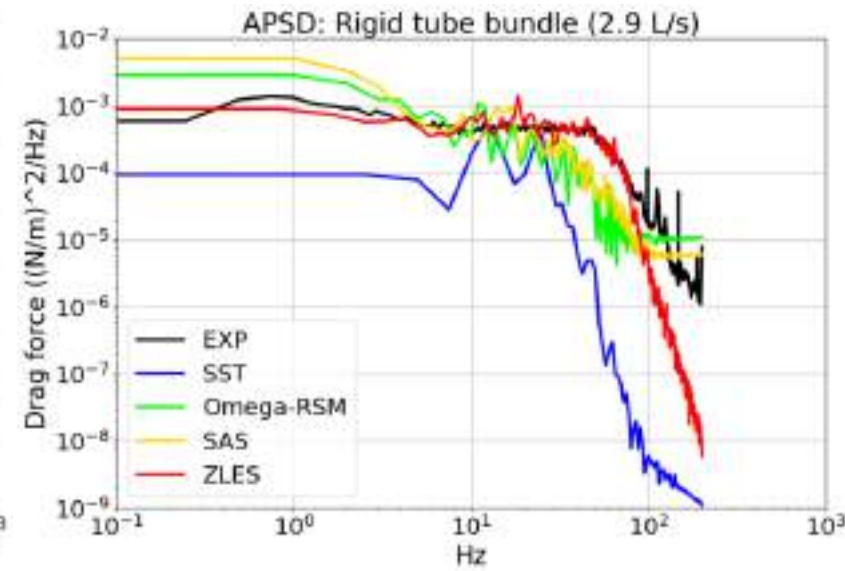
$$S^2 - \Omega^2 = 200 \text{ [s}^{-2}\text{]}$$



AMOVl: Force comparisons

- Results of Rigid Tube Bundle ($Q= 2.9$ & 1.7 L/s)

- Relatively broad distribution of energy without a sharp dominant peak
- Lift and drag force fluctuations mainly associated with turbulent wake interactions and vortex-shedding suppression within the tube array
- Very good agreement for both flow rates, shape and magnitude of spectrum of lift and drag forces reproduced

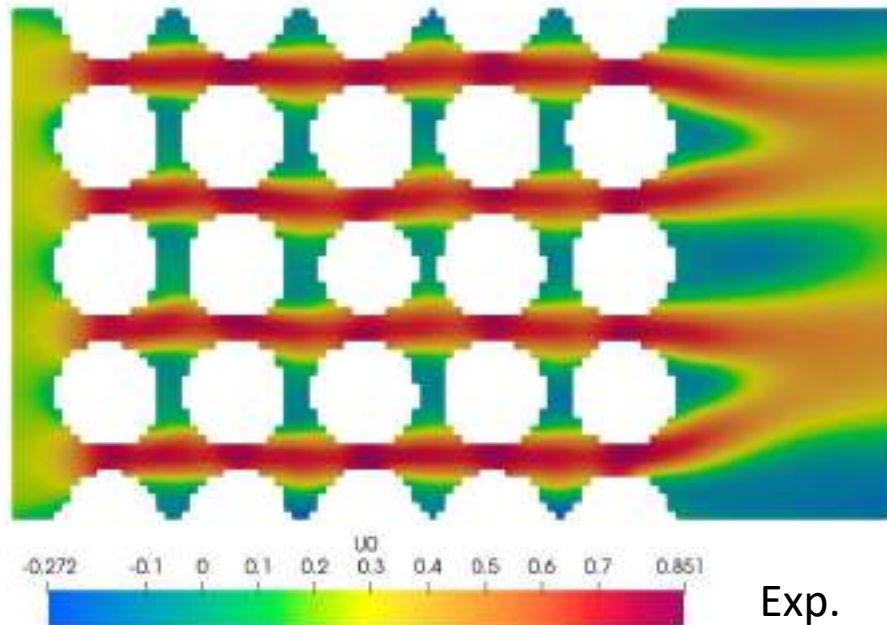
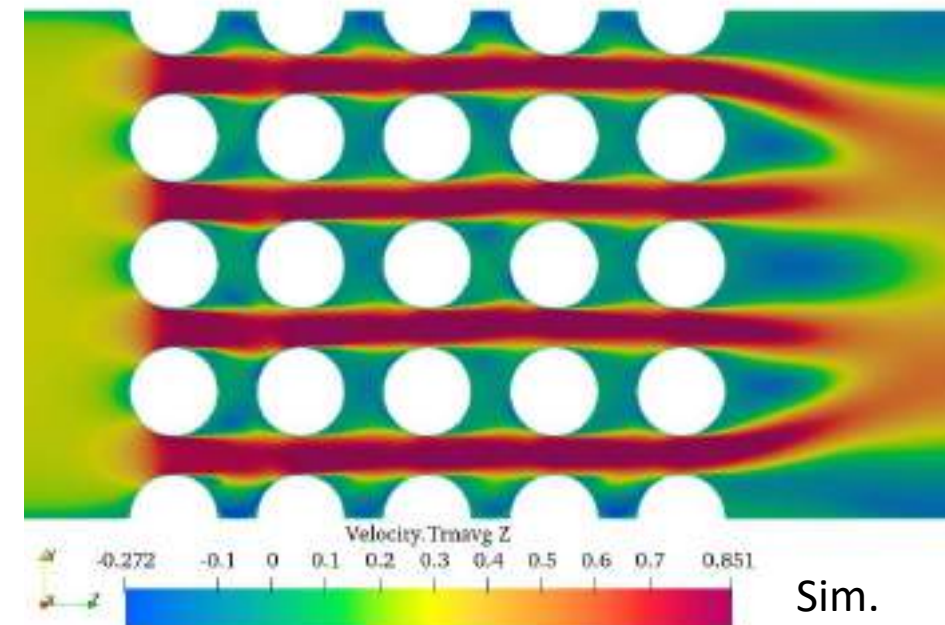


AMOVl: Flow pattern

- Comparison of PIV data with simulation for $Q = 1.7$ L/s



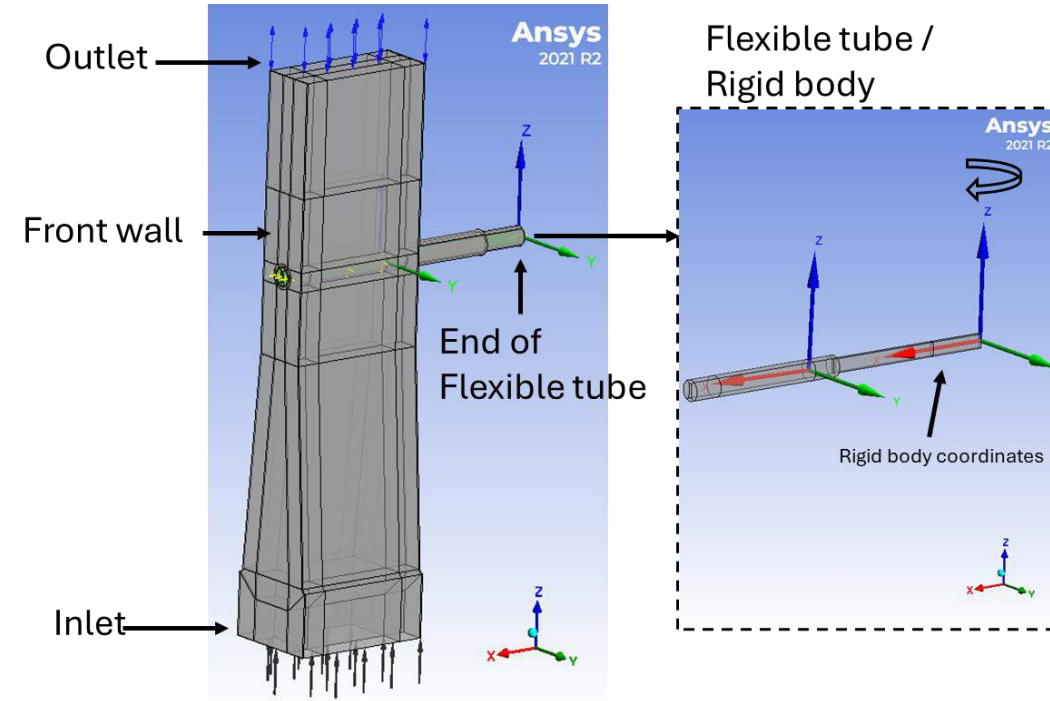
- Qualitative good agreement found between simulation and PIV data
- The upper two and the lower two jets merge shortly downstream the tube bundle. ANSYS CFX slightly overpredicts the maximal jet velocity and suggests larger recirculation zones behind the last tube row
- Averaging performed for 8 s, while experiment over 410 s.



FSI: Rigid Body Motion Modelling



- Flexible Tube:
 - Natural frequency of vibration $f_n=29.37$ Hz validated against CEA data using ANSYS Mechanical.
- Rigid Body Motion Modelling:
 - A rigid body is a collection of 2D faces which moves through fluid forces and torques acting upon it [3].
 - It solves the equation of motion at the center of mass using linear or angular equation of motion.
- The Rigid Body Motion solver requires
 - Mass moment of inertia
 - Mass
 - Stiffness /- torque (as external forces), gravity
 - One degree of freedom



CFX: Angular Momentum Solver

$$M = \dot{\theta} * I \dot{\theta} + I \ddot{\theta}$$
$$M = M_{Aero} - k_{Spring(Rotate)} (\theta - \theta_0) - M_{External}$$

AMOVl: Displacement comparison

- Results of Single Flexible Tube ($Q = 2.9$ & 1.7 L/s)

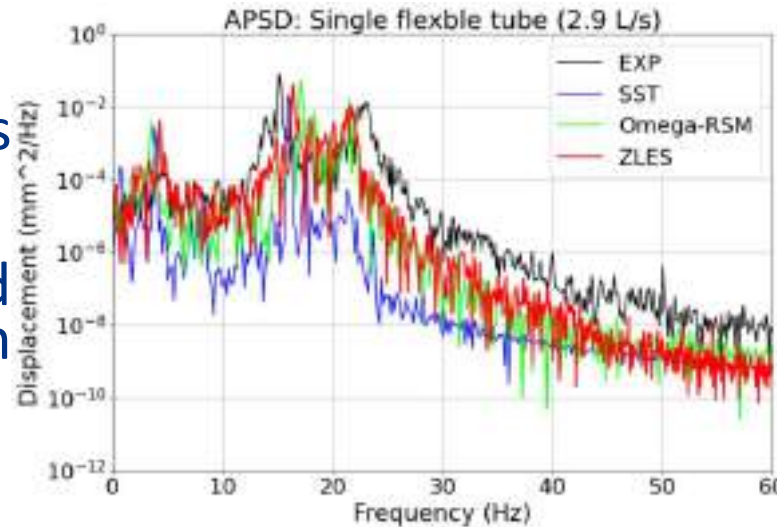
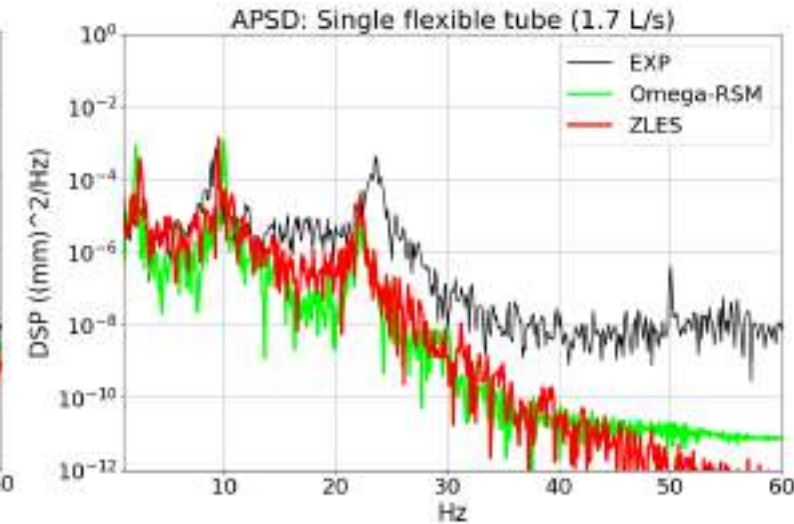
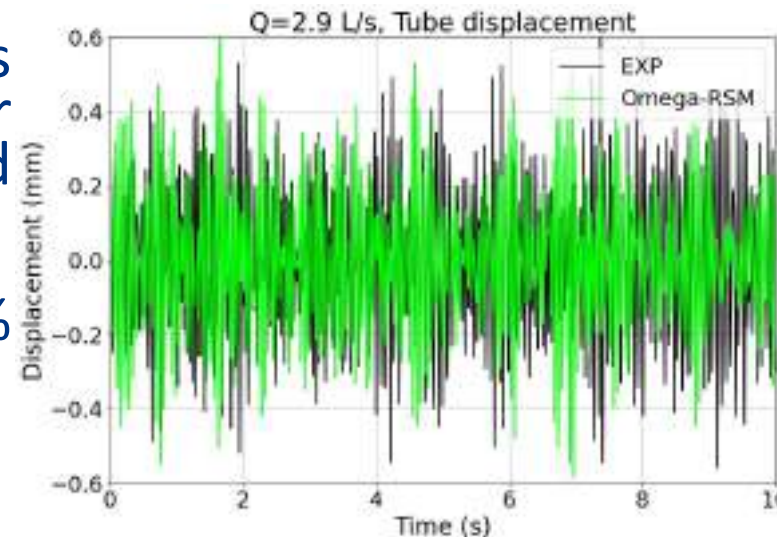
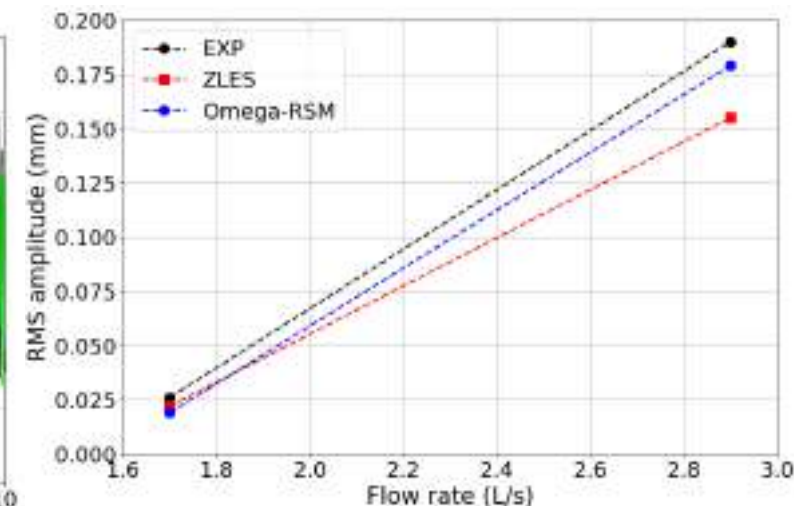


$Q = 2.9$ L/s:

- SST underpredicted the quantities in general.
- Omega-RSM accurately reproduced the max. displacement (0.61 mm vs. 0.6 mm).

$Q = 1.7$ L/s:

- Omega-RSM and ZLES models calculate 26% and 14% smaller displacements than the measured ones (0.026 mm).
- Max. displacement is about 34% lower than experiment (0.096 mm)

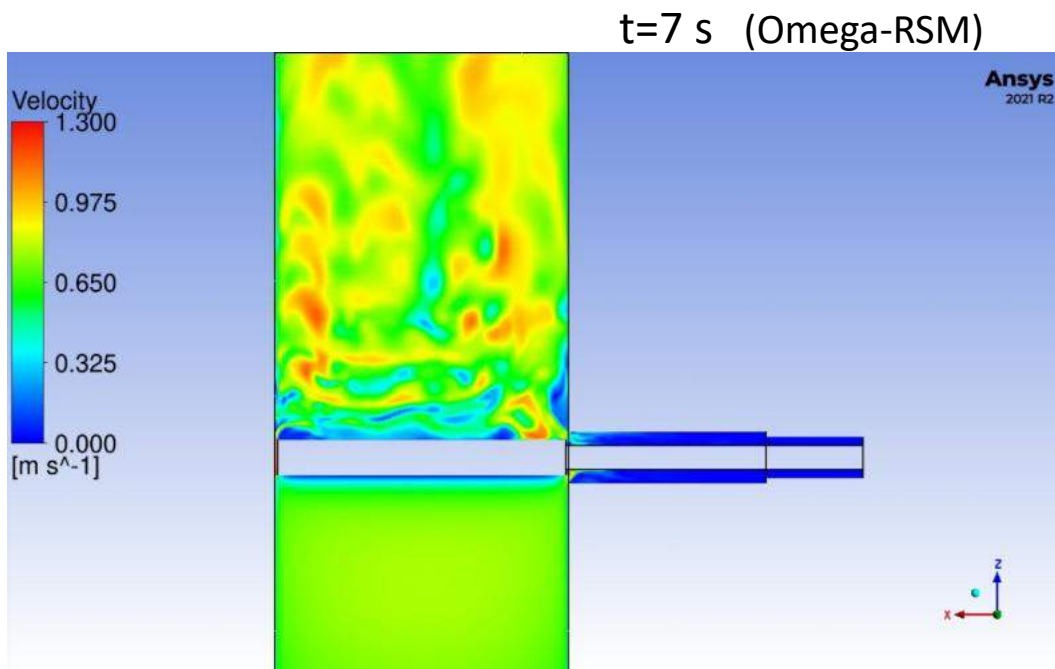
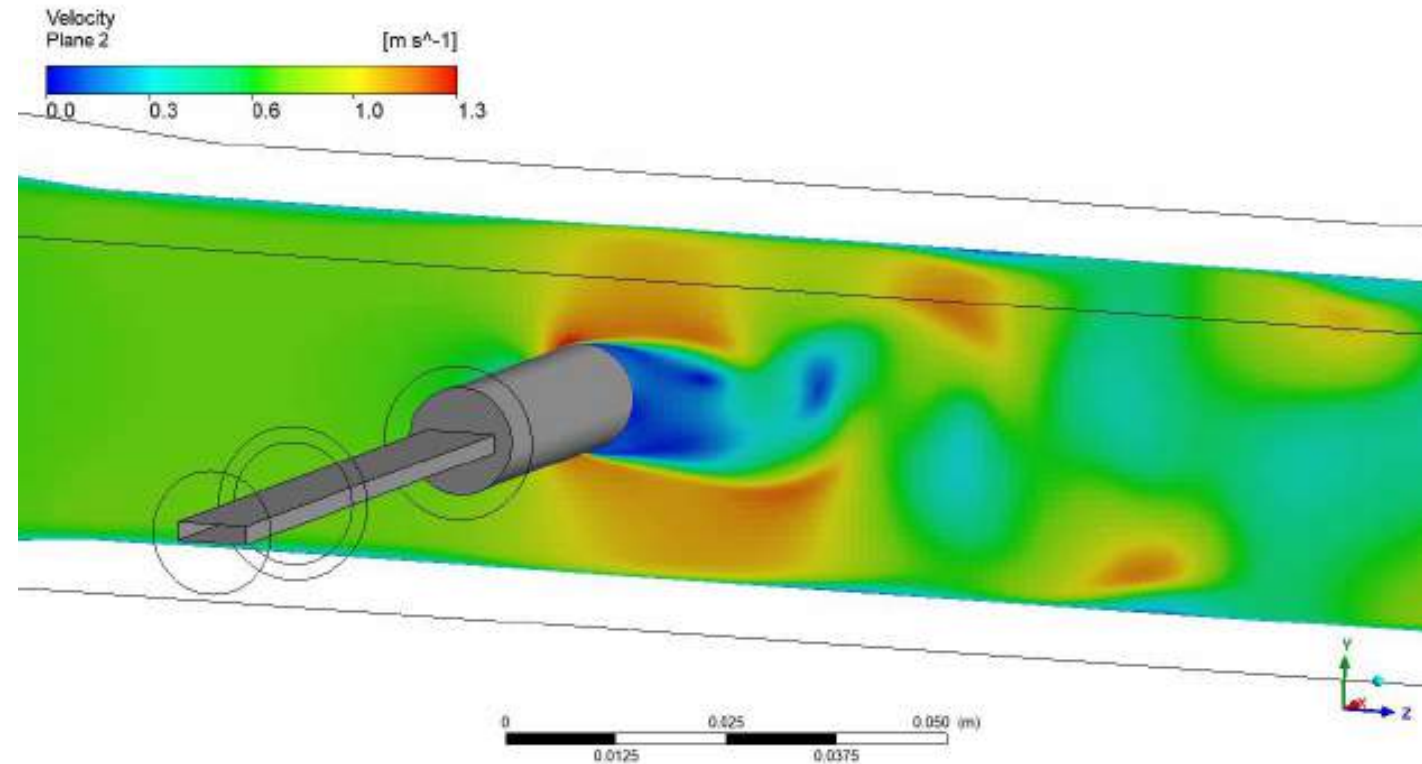


AMOVl: Flow pattern

- Results of Single Flexible Tube ($Q = 2.9 \text{ L/s}$)



- The maximum velocity of 1.3 m/s is observed in the 1 mm gap between the left tube end and the channel wall. The flow in the right part of the blade channel, close to the blade fixation point, is stagnant.

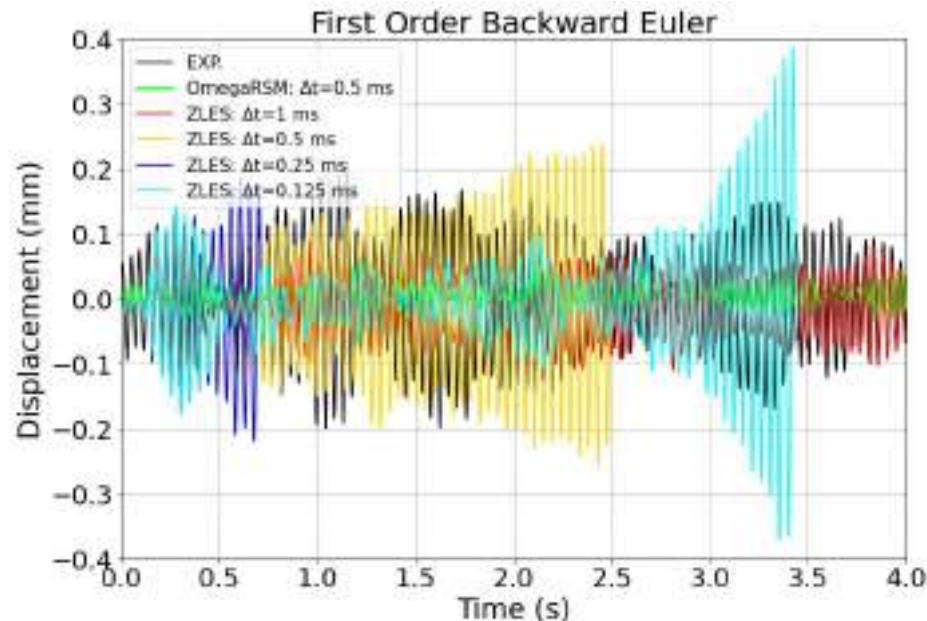
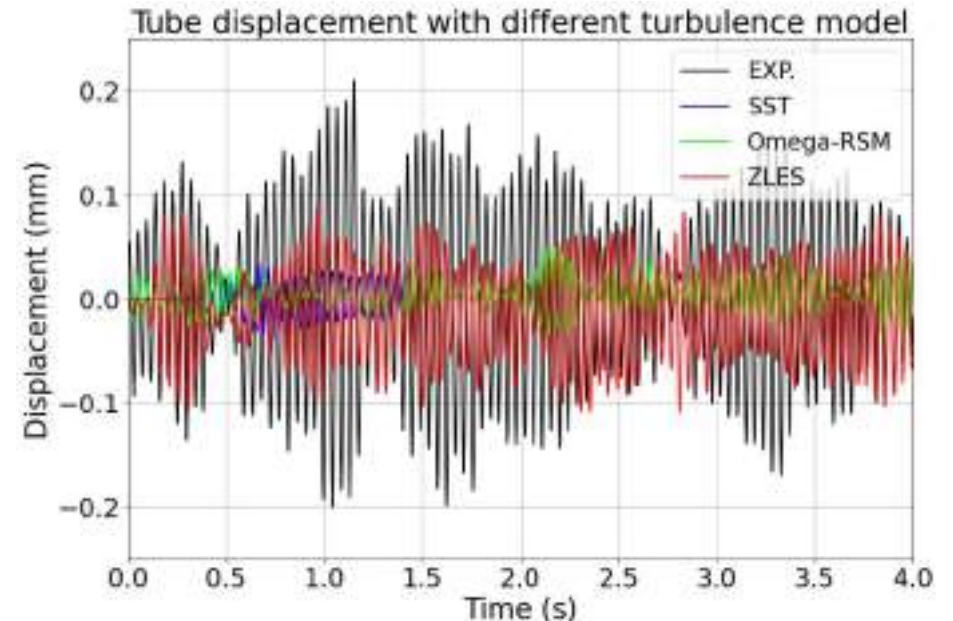


AMOVl: Displacement comparison

- Results of Flexible Tube in Rigid Tube Bundle ($Q = 1.6 \text{ L/s}$)



- After the introduction of the flexible tube in the bundle, time-step size independency of the results could not be achieved.
- With $\Delta t = 0.5$ and 1 ms , Omega-RSM and ZLES were able to run. The ZLES led to $\sim 38 \%$ underprediction of the tube displacement (0.045 vs 0.073).
- Validation with other FSI methods necessary.



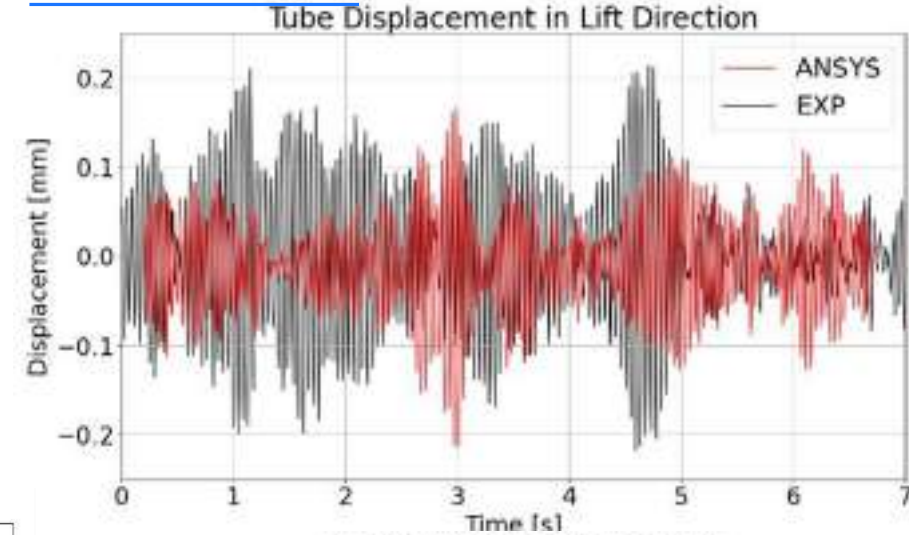
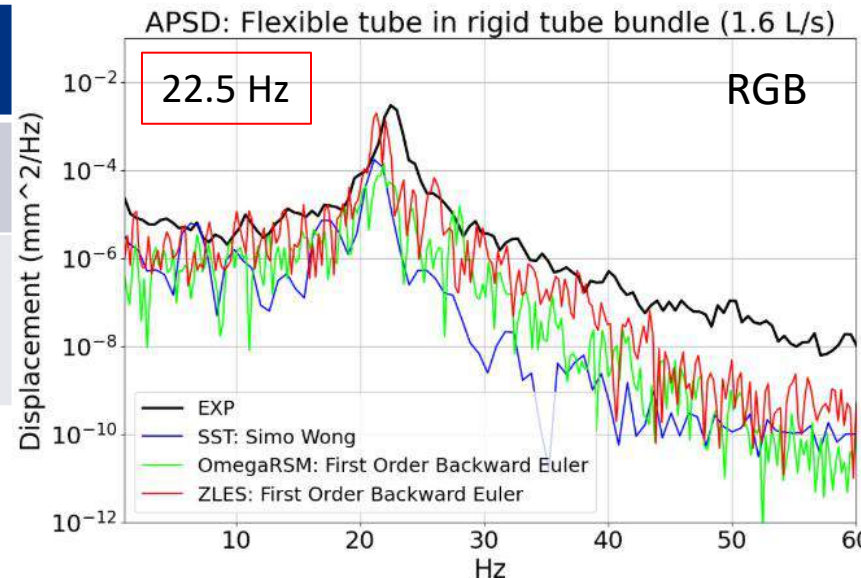
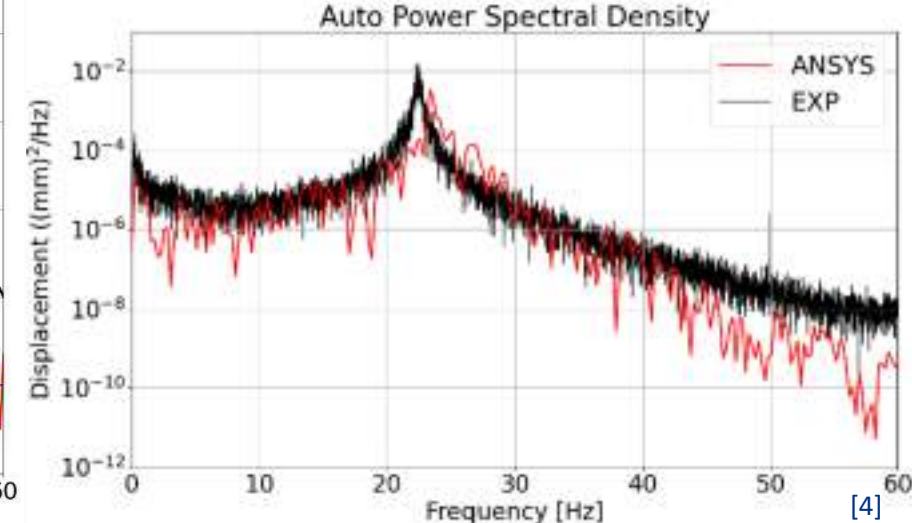
AMOVl: Displacement comparison

- Results of Flexible Tube in Rigid Tube Bundle ($Q= 1.6$ L/s) with CFX-MOR



ANSYS CFX-MOR

- Using ANSYS CFX-MOR code, the simulation was performed and compared with RGB method. [4]
- MOR deviates by only $\sim 27\%$ compared to experiment and predicts well the max. displacement of 0.22 mm (exp. 0.3 mm).



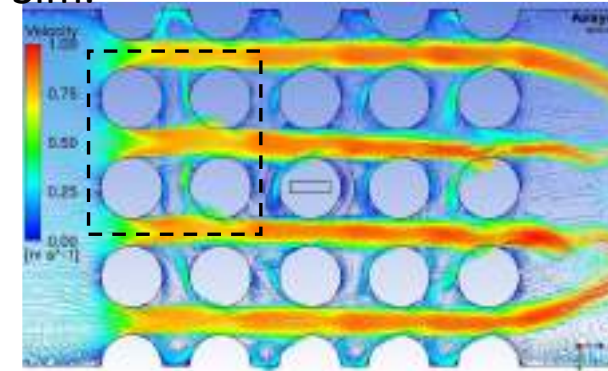
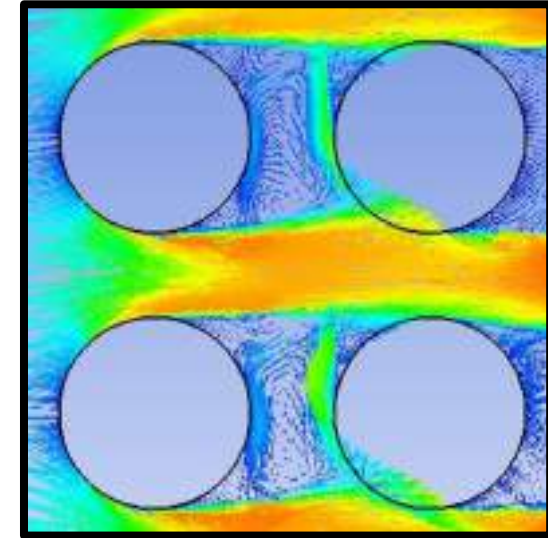
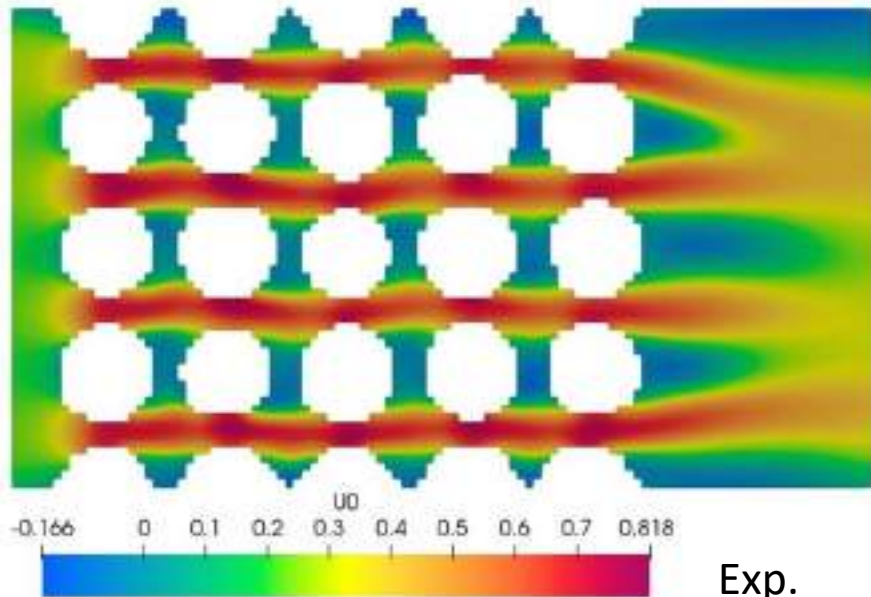
AMOVl: Flow pattern comparison

- Comparison of PIV data with simulation ($Q = 1.6 \text{ L/s}$)

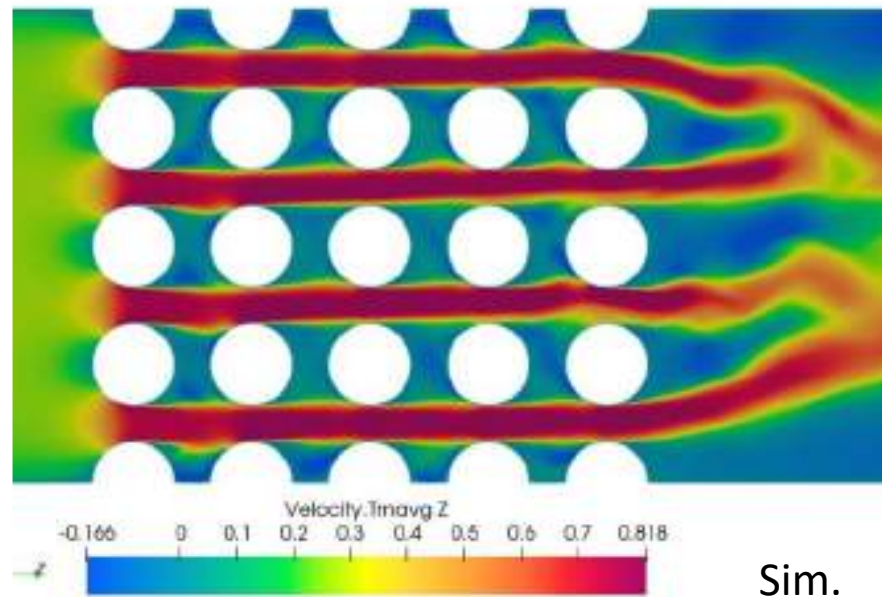


- Positions of high-velocity jets and recirculation zones very similar
- Qualitative agreement found between simulation and experiment with PIV data, some deviations downstream the bundle
- Averaging was performed over 3 s (RGB) and 410 s (exp).

Sim.



Exp.



Sim.

Conclusion



- FSI validation activities performed on different AMOVI experiments
- SST showed underestimation in all cases, while Omega-RSM and ZLES turbulence models were found to be the most accurate for the prediction of forces and displacements.
- RGB method is applicable to simpler FIV phenomena (displacement in lift direction). For more complex cases with higher velocities, further validation is necessary.
- Comparison with PIV data requires longer simulation times for improved statistics.

References



- [1]. D. Panunzio, P . Piteau , R. Lagrange, C . Gauthier. AMOVI tube bundle Experiment, CEA. (15th September 2022).

- [2]. D. Panunzio, P . Piteau , R. Lagrange, C . Gauthier. Description of AMOVI tube bundle Experiment, Version 3, CEA.

- [3] ANSYS 2021 R2 User guide.

- [4] Papukchiev, A. & Mistry, H. (Eds.) (2025): Application of an FSI approach based on structural reduced-order model for the analysis of flow-induced vibrations in nuclear power plants. NURETH-21 Konferenz, August, 31 - September, 5: Busan, Südkorea

Partners



Associated Partners



*EDF R&D UK Center



GO-VIKING

Thank you!



Hemish Mistry



GRS

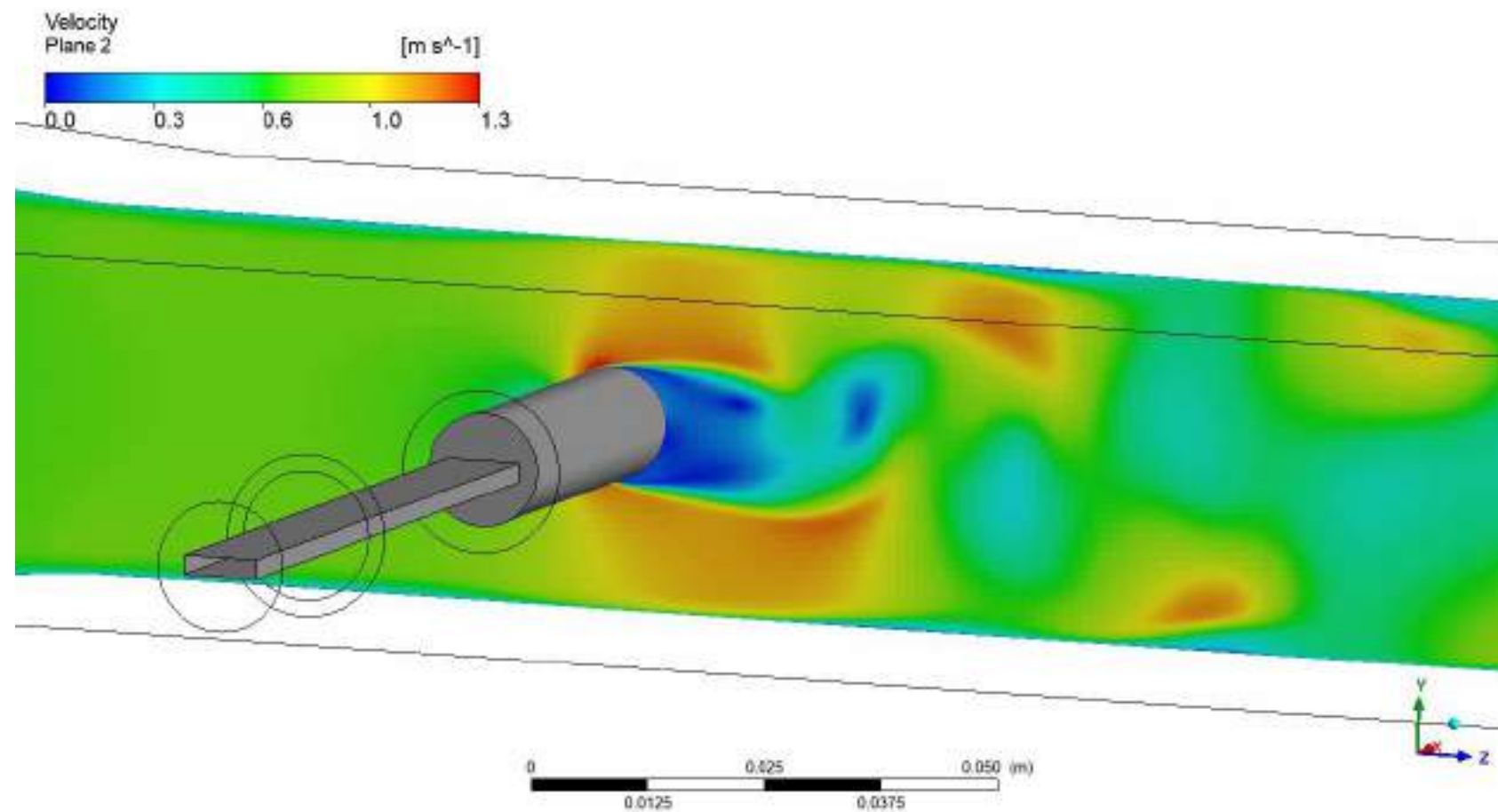


hemish.mistry@grs.de

www.go-viking.eu



Funded by the
European Union





WP3: Comparison of different numerical approaches within the AMOVI benchmark

Vladislav Filonov, Yuliia Filonova, Yaroslav Dubyk, Hemish Mistry, Angel Papukchiev, Sofiane Benhamadouche



Funded by
the European Union

GO-VIKING FINAL WORKSHOP 20-22 May 2026, Garching n. Munich, Germany

WP3. Flow-induced vibrations in steam generators

Task 3.1. Analysis of the AMOVI cross flow tube bundle (M1-M46)

[CEA, EDF, GRS, IPP]



- **D3.1 Single-phase AMOVI experiments and benchmark setup (M18) [CEA].**

Description of the AMOVI experimental facility and experimental data.

- **D3.2 AMOVI model preparation and simulation results (M42) [IPP, GRS, EDF]**

Description of the high- and medium-resolution methods and tools, model development, and simulation results.

- Perform high-resolution simulations of AMOVI in Code_Saturne using a wall-resolved LES model using the ALE approach. The generated results will be compared with experimental data and utilized for the development and validation of medium-resolution models. (EDF)
- Perform medium-resolution simulations of AMOVI using hybrid turbulence methods (ZLES, SBES), and URANS based tools to improve the current state-of-the-art in the modeling of cross-flow induced vibrations in tube bundles, relevant for SGs. (GRS, IPP)

Content



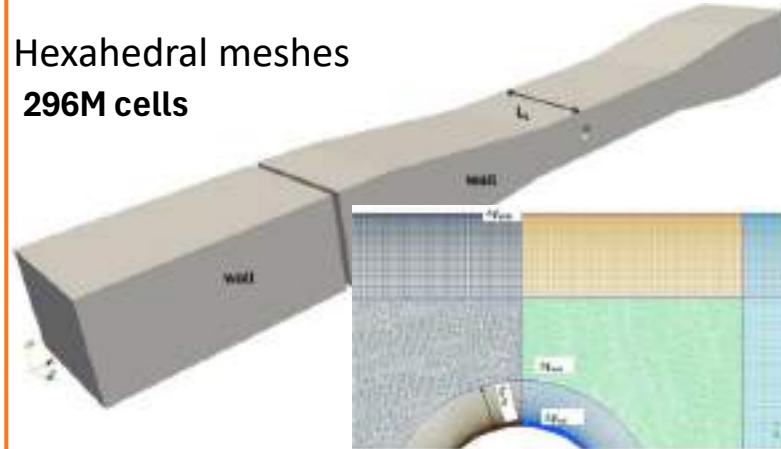
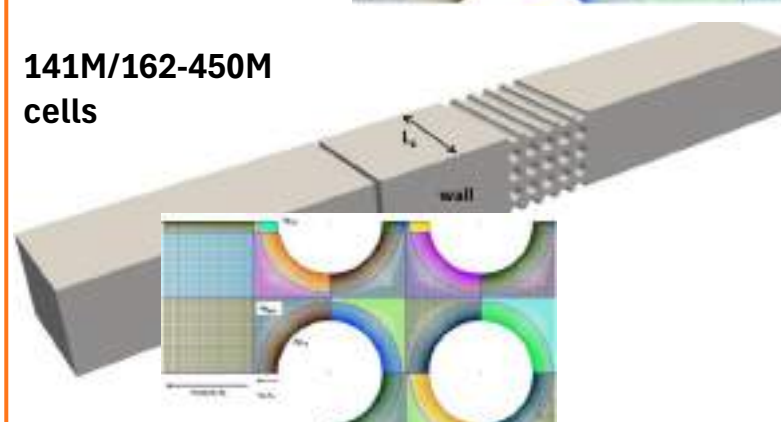
- Numerical approaches used by partners within the AMOVI simulations
- Experimental cases agreed for the AMOVI Benchmark
- Simulations comparison (GRS, IPP and EDF results):
 - Single Rigid Tube
 - Single Flexible Tube
 - Rigid Tube Bundle
 - Flexible Tube in a Rigid Tube Bundle
- Overall comparison of results and computational costs
- Conclusions

Numerical approaches used by partners within the AMOVI simulations

High-fidelity simulations

EDF

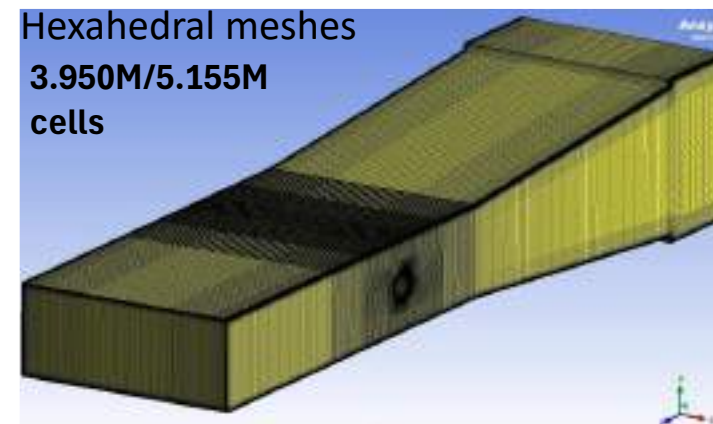
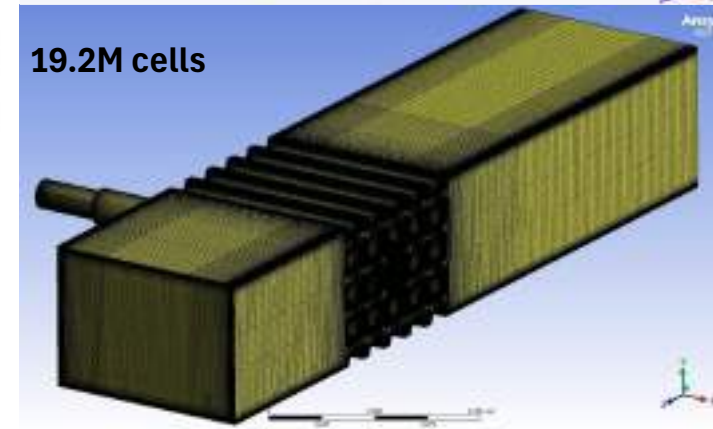
- Code Saturne Ver 8.0
- 3D formulation
- FSI - **Rigid Body** (subroutine in code_saturne)
- **Wall-resolved LES**
- Fully developed inlet flow (embedded precursor LES)



Medium resolution simulations

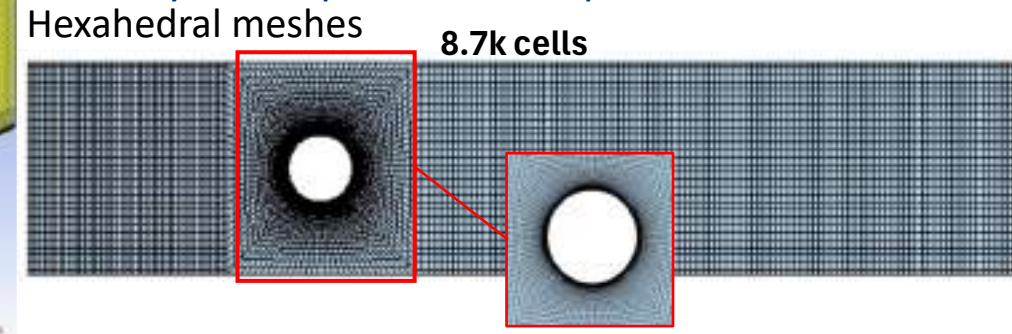
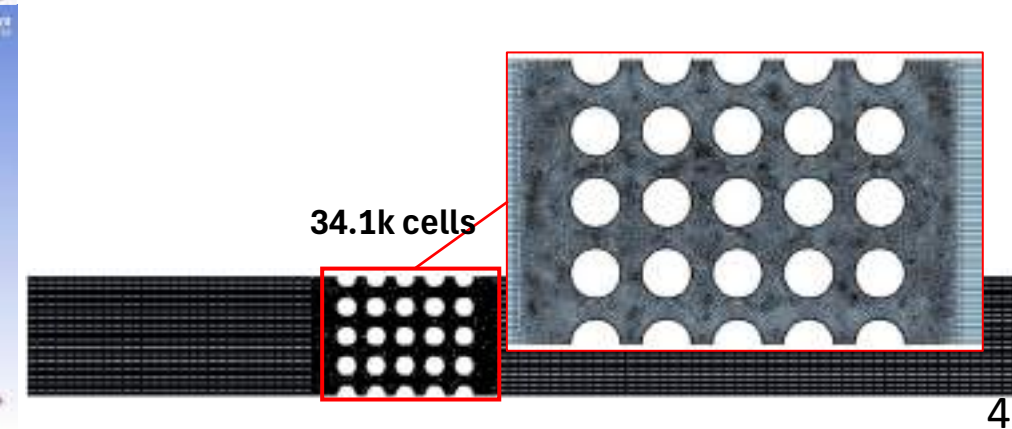
GRS

- ANSYS CFX 2021 R2
- 3D formulation
- FSI - **Rigid Body** (inbuilt)
- Hybrid LES-URANS/ZLES/OmegaRSM
- Fully developed inlet flow profile (x,y,z)



IPP

- ANSYS CFX 2018
- 2D formulation
- **Rigid Body FSI (inbuilt) & User Beam Libs** (implemented using FORTRAN)
- URANS SST/SAS SST/OmegaRSM
- Fully developed inlet flow profile



Experimental cases agreed for the AMOVI Benchmark



Sixteen different test cases are made available by CEA to the GO-VIKING partners (four water flow rates for each configuration).

Participants agreed on a set of 7 cases of interest and provided their results for comparison.

	Flow Rate (l/s)	Pressure (bar)	Temperature (°C)	PIV/LDV data available	Simulations performed		
					GRS	IPP	EDF
Single <u>rigid</u> tube	0.5	1.33	18	-	-	-	+
	1.7	3.80	25	-	-	-	+
	2.9	2.88	26	LDV	+	+	+
	4.0	3.5	26	-	-	-	-
<u>Rigid</u> tube bundle	0.5	0.88	20	-	-	-	+
	1.7	0.90	18	PIV	+	+	+
	2.9	3.19	41	PIV,LDV	+	+	+
	4.0	3.6	40	-	-	-	-
Single <u>flexible</u> tube (lift direction)	0.5	2.19	18	-	-	+	+
	1.7	3.33	21	-	+	+	+
	2.9	3.20	25	-	+	+	+
	4.0	3.53	26	-	-	+	-
<u>Flexible</u> tube (lift direction) in a rigid tube bundle	0.5	1.48	17	-	-	+	+
	1.1	0.9	17	-	-	+	+
	1.6	1.07	17	PIV	+	+	+
	1.95	3.17	23	-	-	+	+

SRT-290

RTB-170

RTB-290

SFT-170

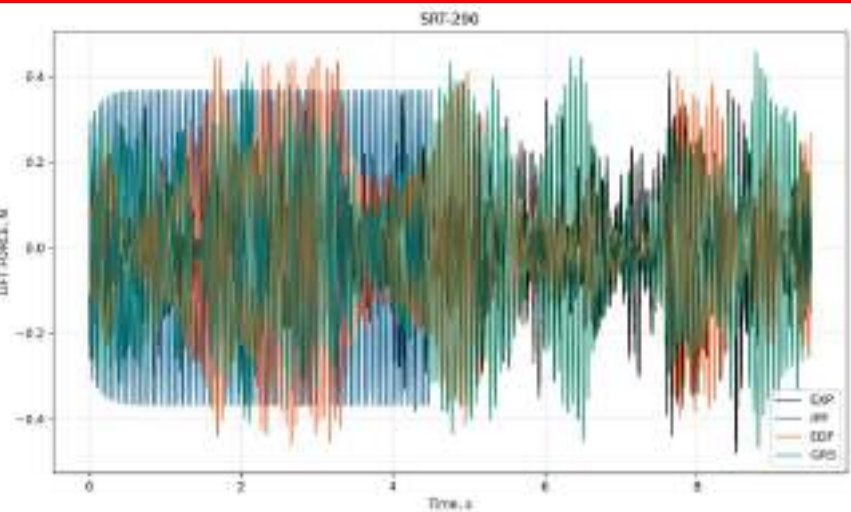
SFT-290

FTRTB-160

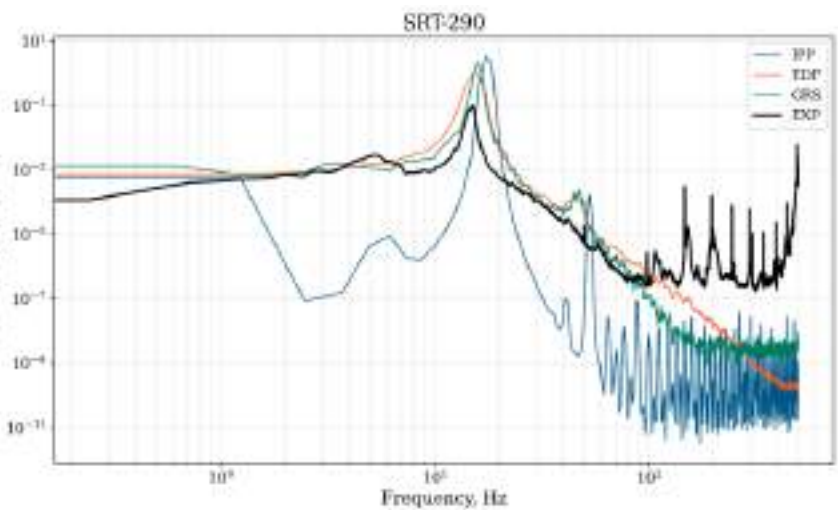
FTRTB-195

Simulations comparison (Single Rigid Tube) SRT-290

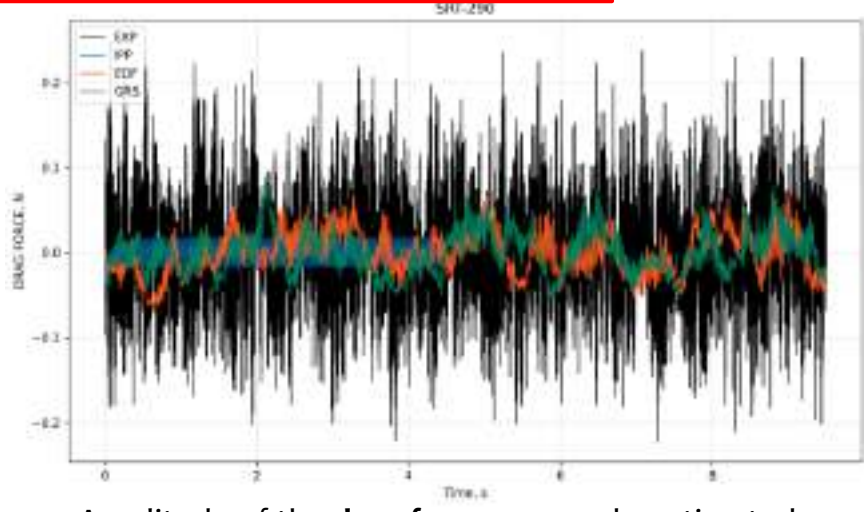
EDF: 66.5 s (dT=0.04ms) **GRS:** 10.0 s (dT=0.2 ms) **IPP:** 5.0 s (dT=0.1 ms)



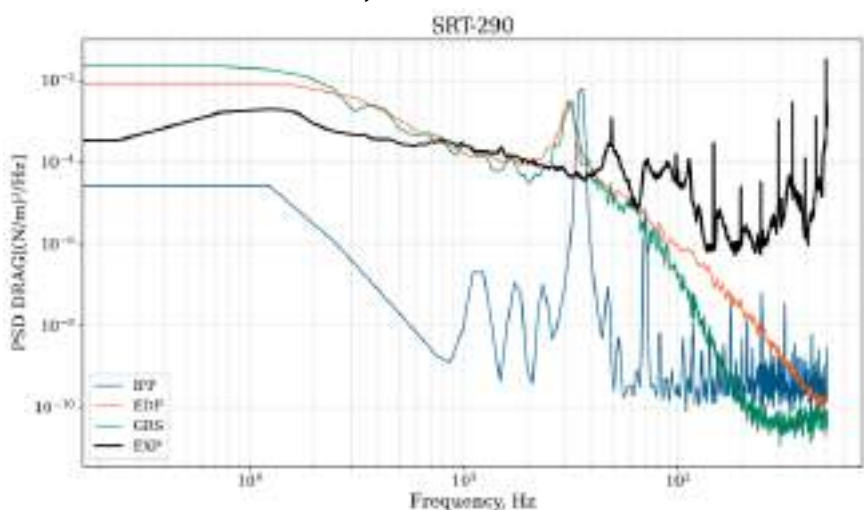
Amplitude of the **lift forces** has a quite good agreement with experiment (EDF and GRS captured the modulation in time).



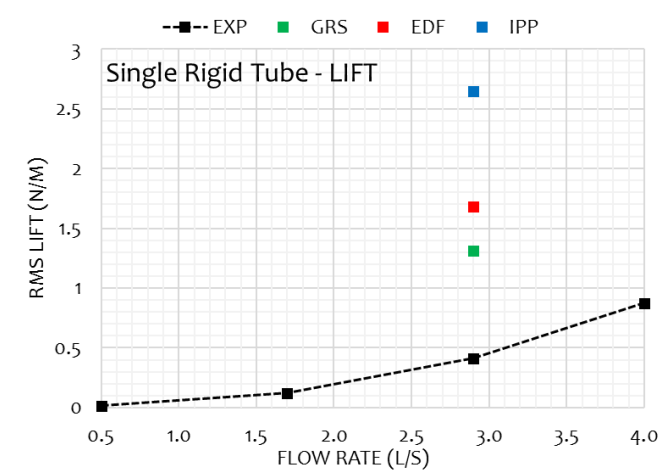
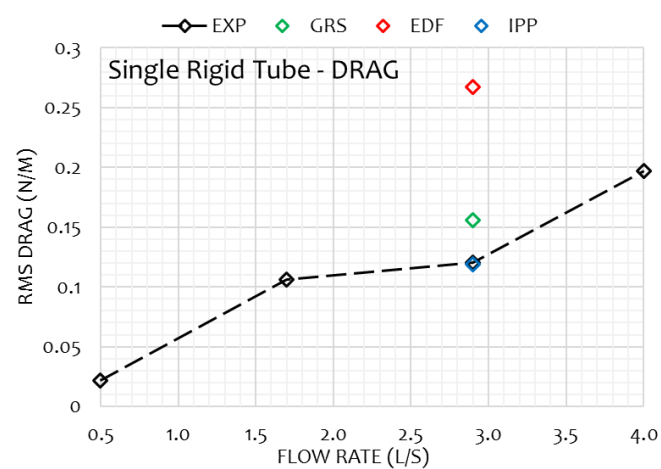
Calculated APSDs **capture the peak** present in the experimental spectrum with a sufficiently good accuracy



Amplitude of the **drag forces** are underestimated compared to experiment (EDF and GRS captured the modulation in time).



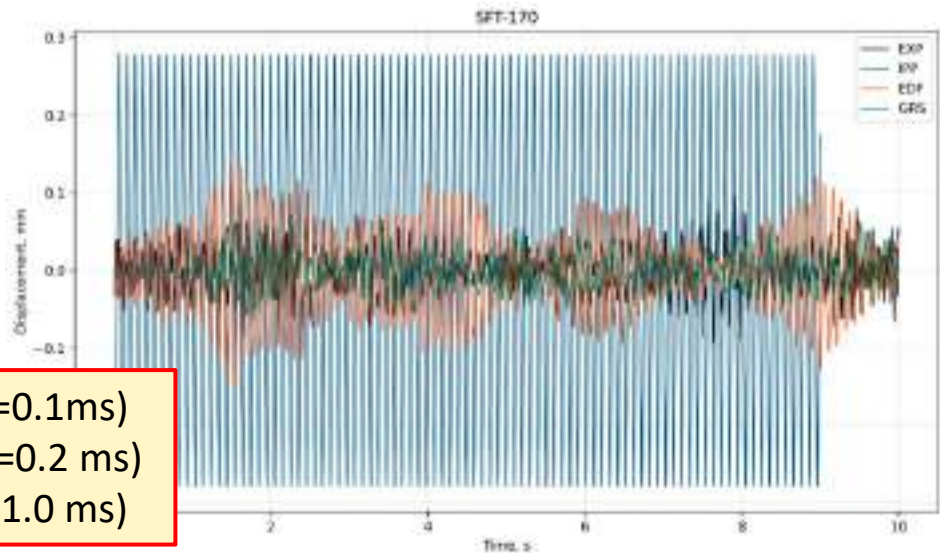
Calculated APSDs captured visible peak in the interval ~30-35Hz, while in experimental spectrum this peak is at ~50Hz



Institution	RMS LIFT	RMS DRAG
EXP	0.411	0.120
EDF	1.684	0.267
GRS	1.312	0.156
IPP	2.644	0.119

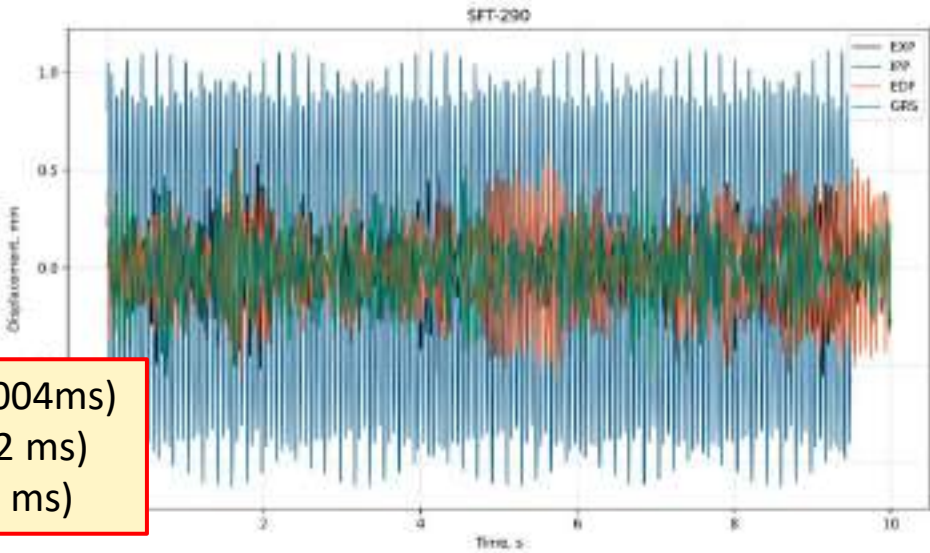
Simulations comparison (Single Flexible Tube)

SFT-170



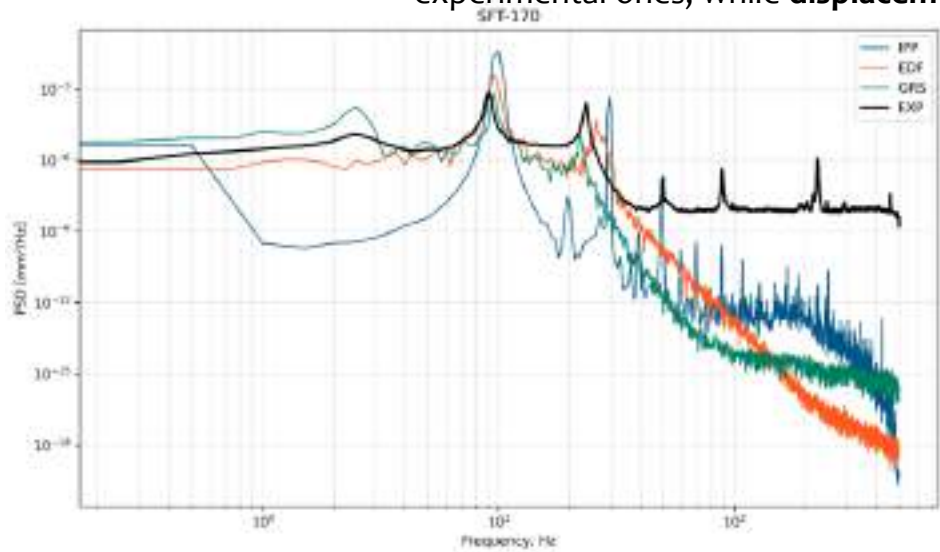
EDF: 43.5 s (dT=0.1ms)
GRS: 10.0 s (dT=0.2 ms)
IPP: 10.0 s (dT=1.0 ms)

SFT-290

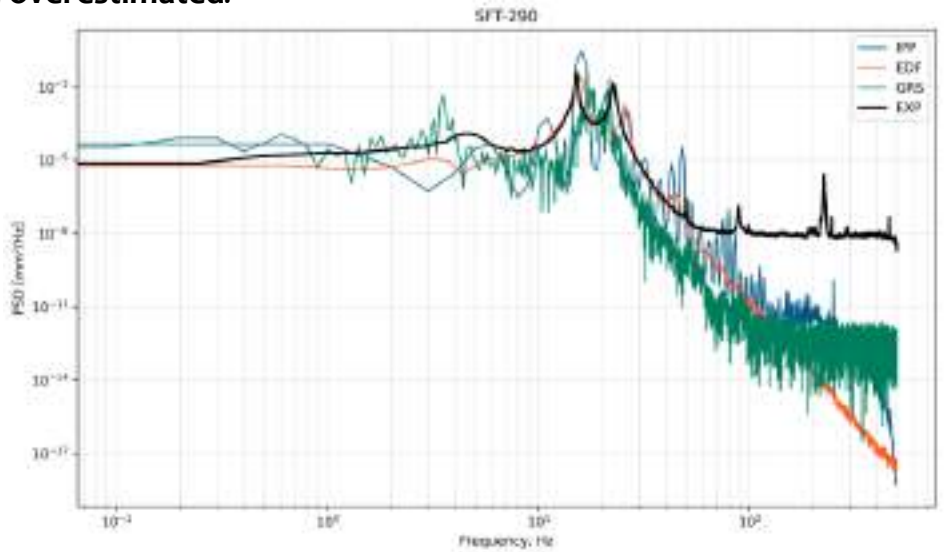


EDF: 14.8 s (dT=0.004ms)
GRS: 12.0 s (dT=0.2 ms)
IPP: 10.0 s (dT=1.0 ms)

The amplitudes obtained by GRS and EDF have **good agreement** with the experimental ones, while **displacements obtained by IPP are overestimated**.

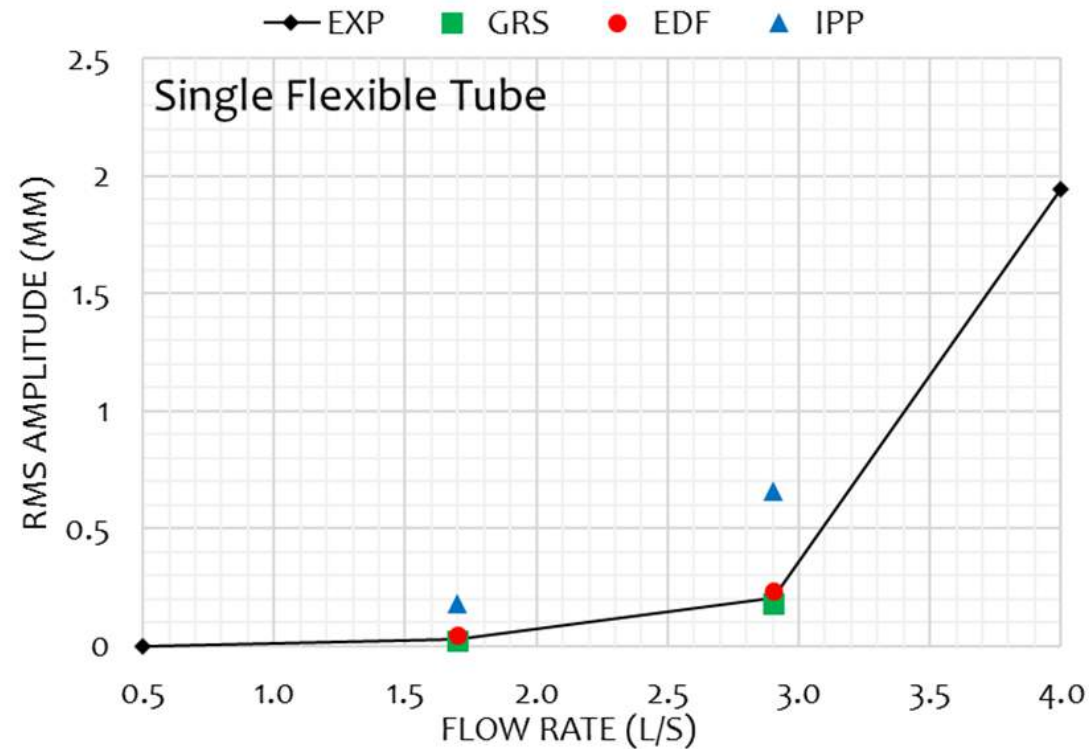


Captured both peaks present in the experimental spectrum – the first peak is reproduced with a sufficiently good accuracy in all three simulations, while the second one has small discrepancy both towards slightly lower and slightly higher frequencies



Calculated APSD captured both peaks present in the experimental spectrum with a sufficiently good accuracy in all three simulations

Simulations comparison (Single Flexible Tube)



SFT-170

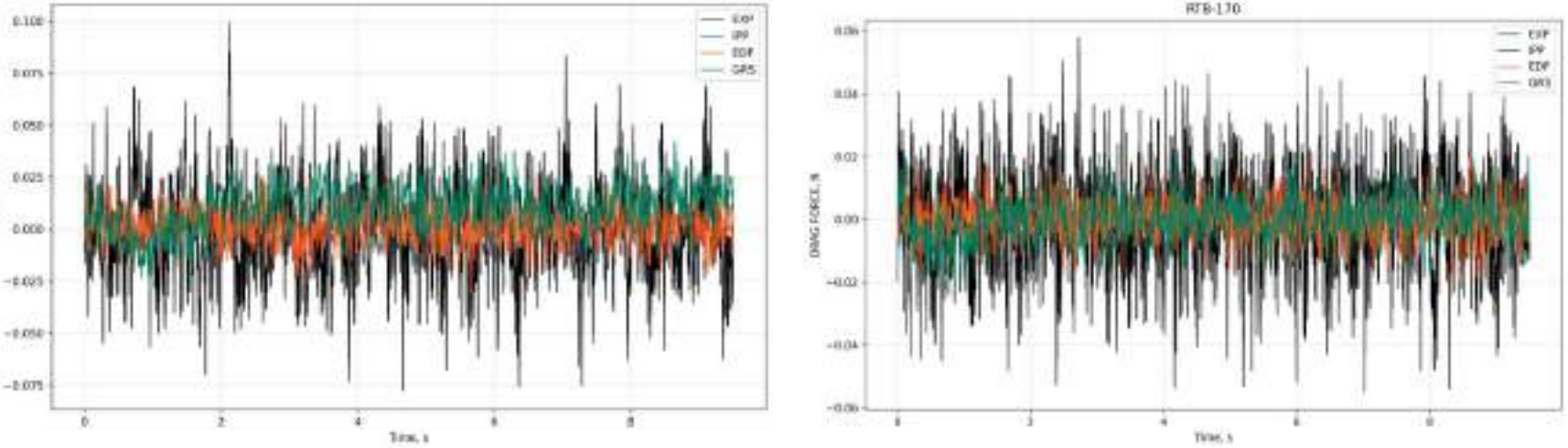
Institution	RMS	1 st Peak Frequency	2 nd Peak Frequency
EXP	0.027	9.03	23.43
EDF	0.045	9.5	26.0
GRS	0.022	9.49	22.24
IPP	0.18	10.0	29.5

SFT-290

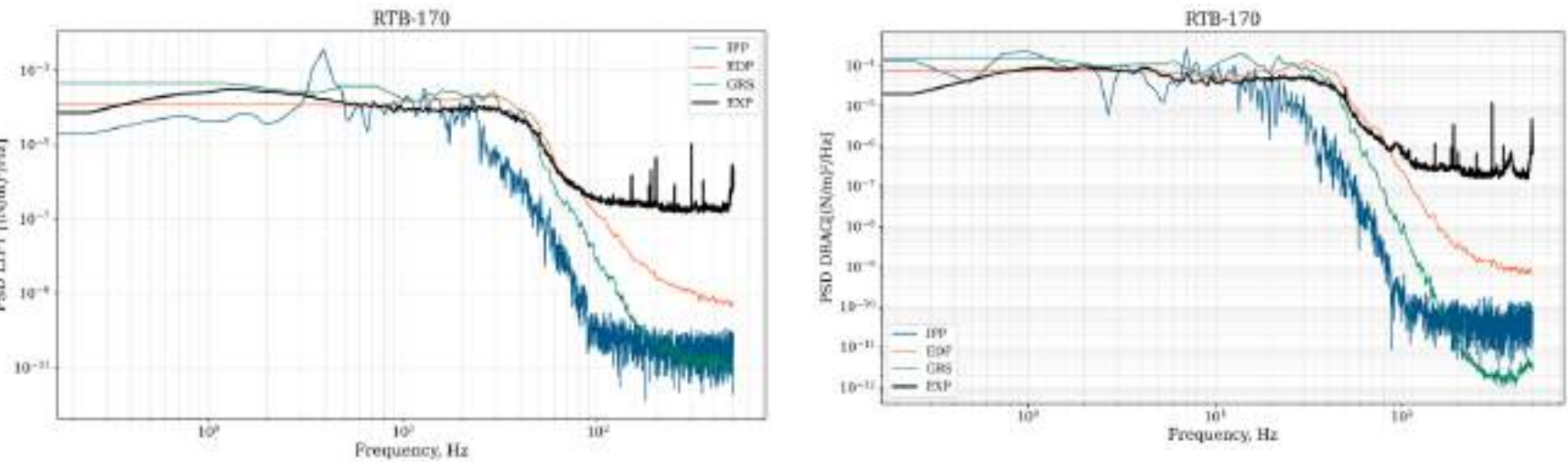
Institution	RMS	1 st Peak Frequency	2 nd Peak Frequency
EXP	0.205	15.14	22.7
EDF	0.235	15.62	25.62
GRS	0.179	17.00	22.3
IPP	0.658	16.78	21.4

Simulations comparison (Rigid Tube Bundle) RTB-170

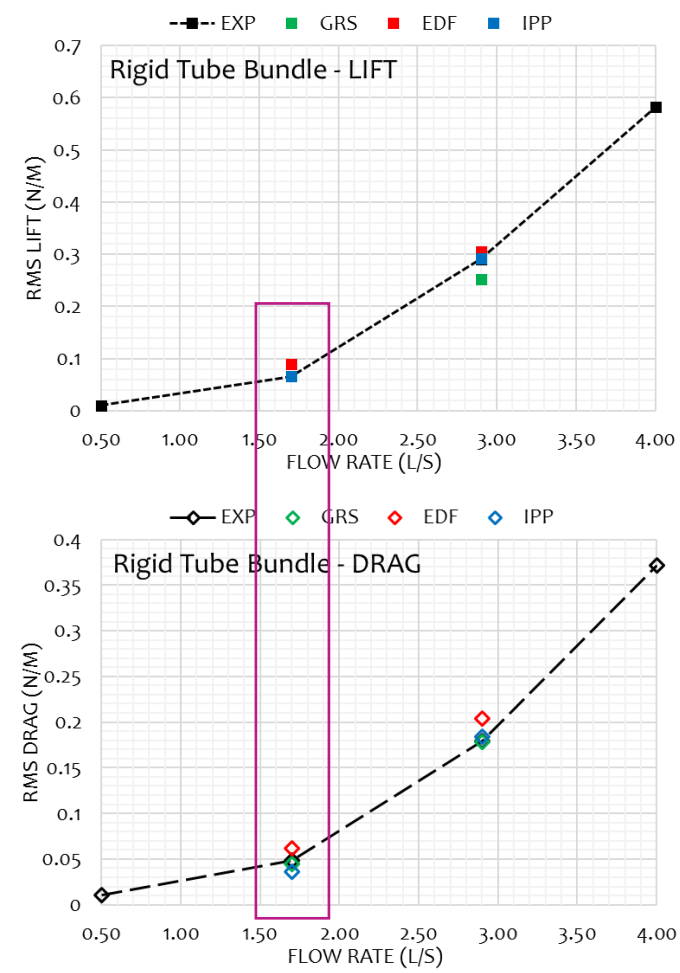
EDF: 39.7 s (dT=0.035ms) **GRS:** 10.0 s (dT=0.2 ms) **IPP:** 10.0 s (dT=1.0 ms)



Obtained amplitudes of the **lift and drag forces** are **slightly underestimated** compared to the experiment in all three simulations.



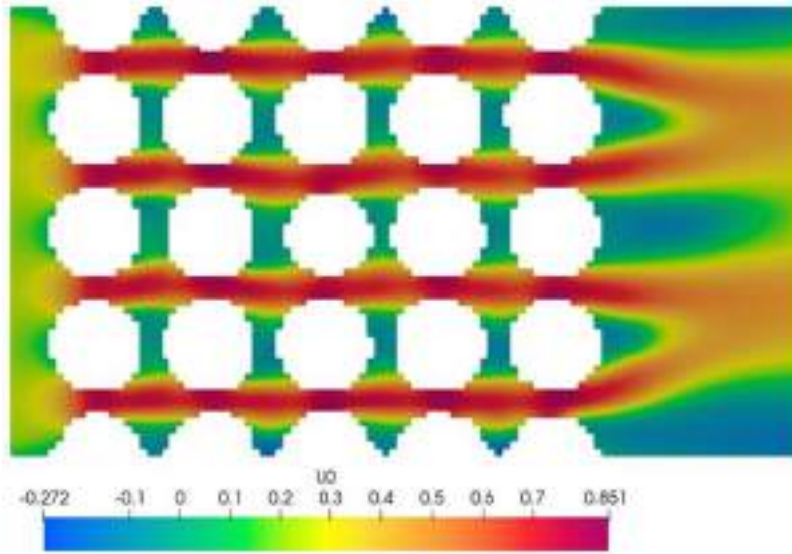
Calculated **APSD** have **sufficiently good agreement with experimental spectrum**, especially in simulations performed by GRS and EDF.



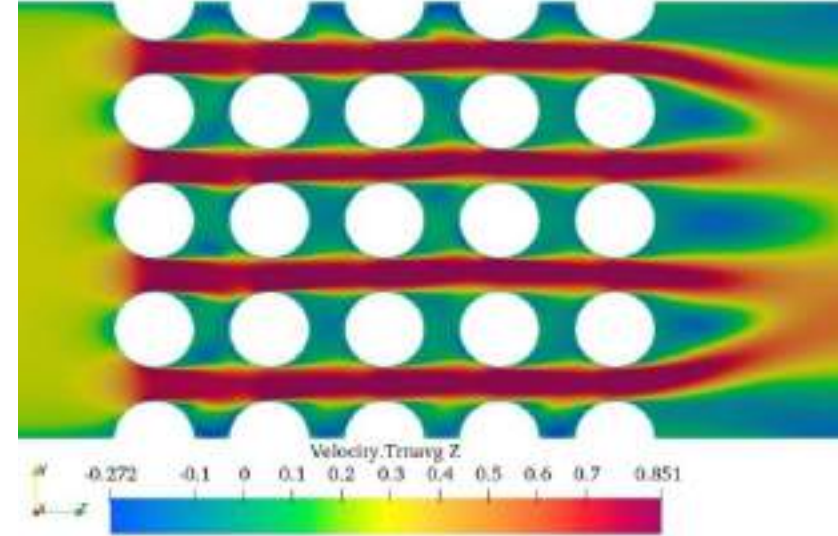
Institution	RMS LIFT	RMS DRAG
EXP	0.066	0.048
EDF	0.089	0.062
GRS	0.066	0.045
IPP	0.067	0.036

Simulations comparison (Rigid Tube Bundle) RTB-170

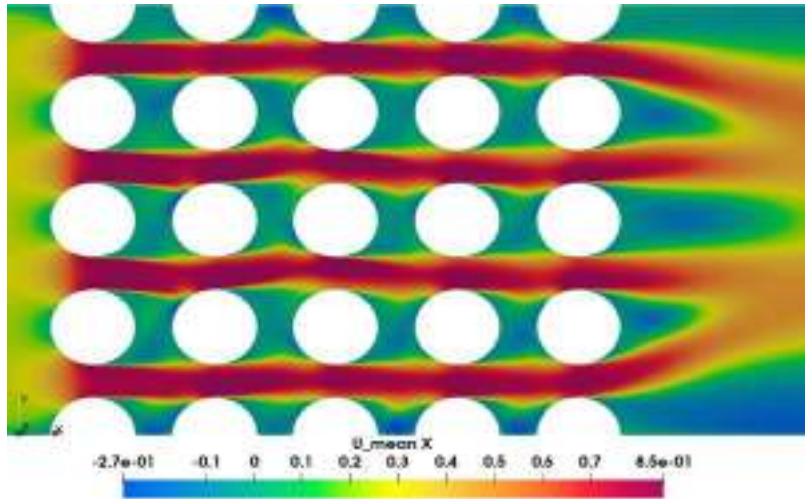
EXP



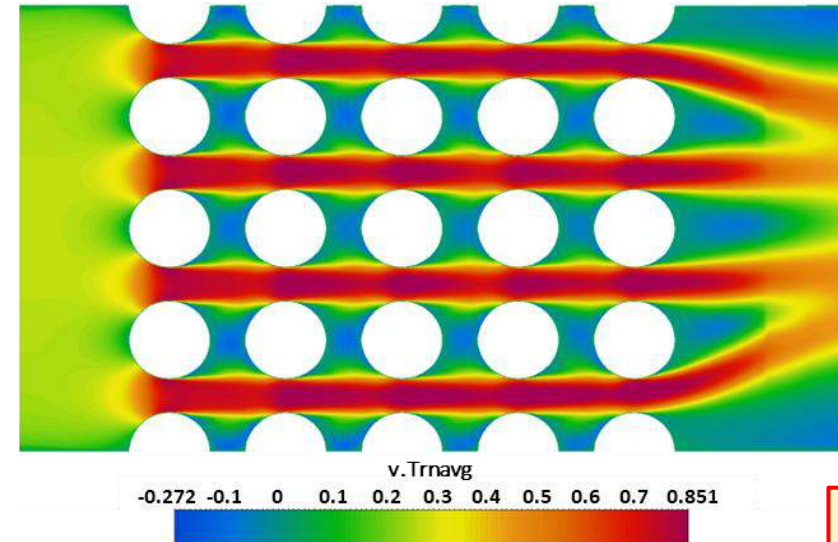
GRS



EDF



IPP



All three simulations performed by EDF, GRS and IPP provided **definitely good agreement** of calculated time-averaged velocity field with the experimental picture.

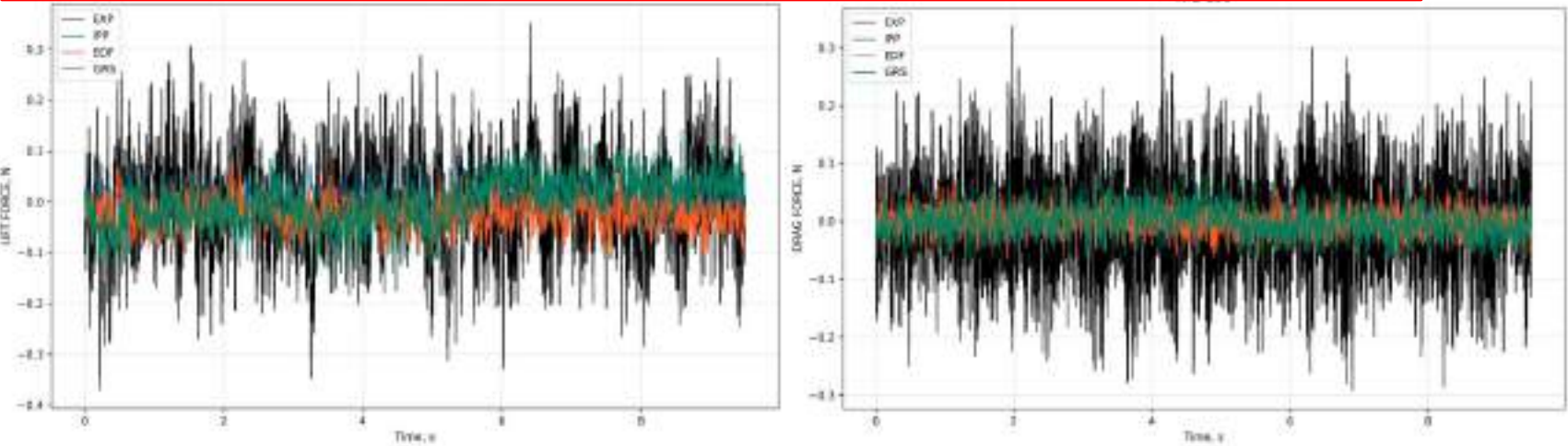
EDF: 39.7 s (dT=0.035ms)

GRS: 10.0 s (dT=0.2 ms)

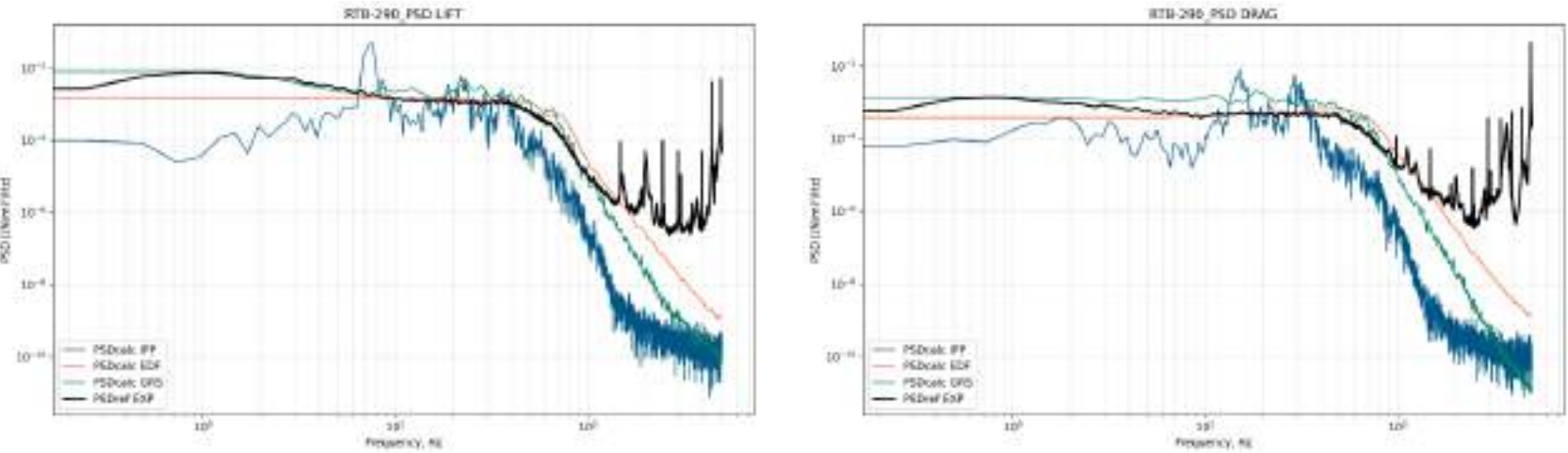
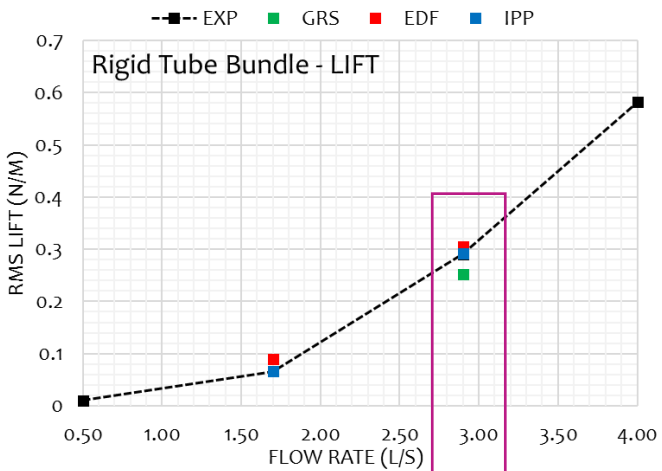
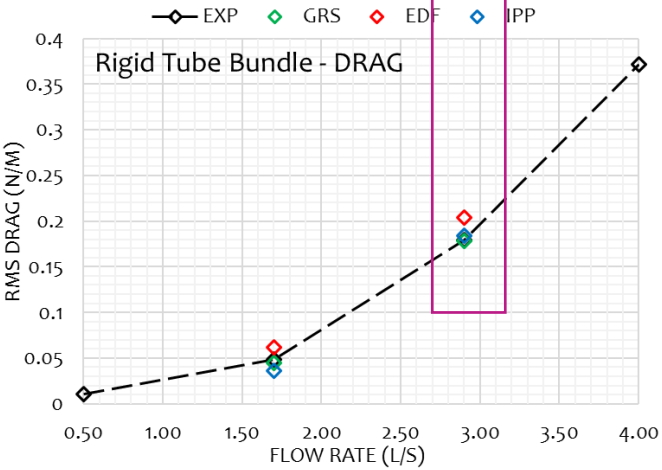
IPP: 10.0 s (dT=1.0 ms)

Simulations comparison (Rigid Tube Bundle) RTB-290

EDF: 41.7 s (dT=0.02ms) **GRS:** 16.6 s (dT=0.2 ms) **IPP:** 10.0 s (dT=1.0 ms)



Obtained amplitudes of the **lift and drag forces** are **slightly underestimated** compared to the experiment in all three simulations.

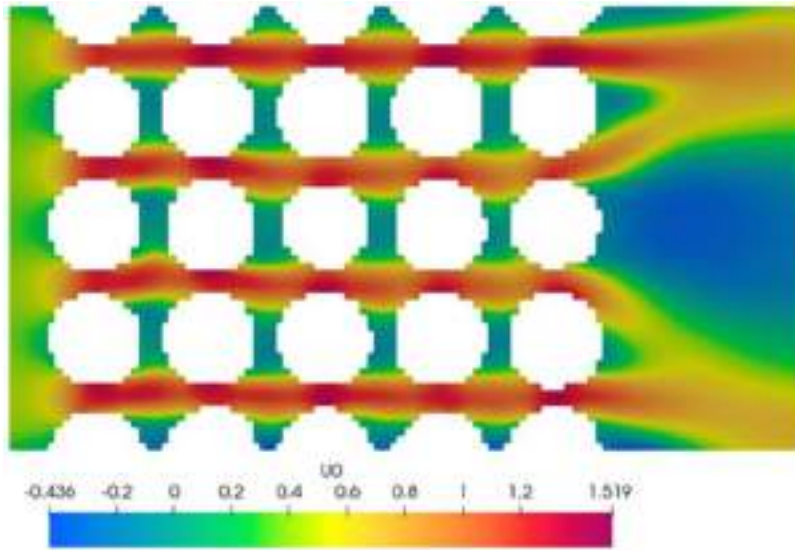


Institution	RMS LIFT	RMS DRAG
EXP	0.291	0.179
EDF	0.305	0.204
GRS	0.252	0.178
IPP	0.292	0.184

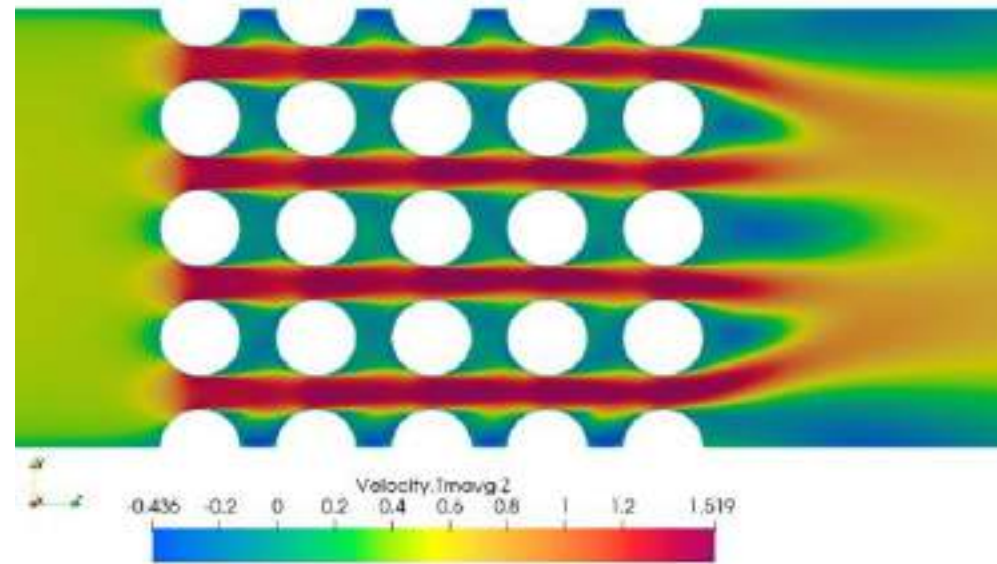
Calculated **APSD** have **sufficiently good agreement with experimental spectrum**, especially in simulations performed by GRS and EDF.

Simulations comparison (Rigid Tube Bundle) RTB-290

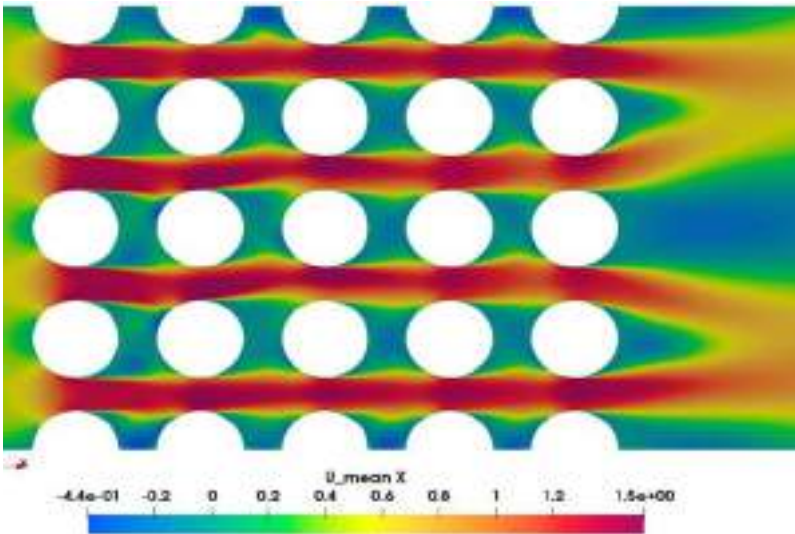
EXP



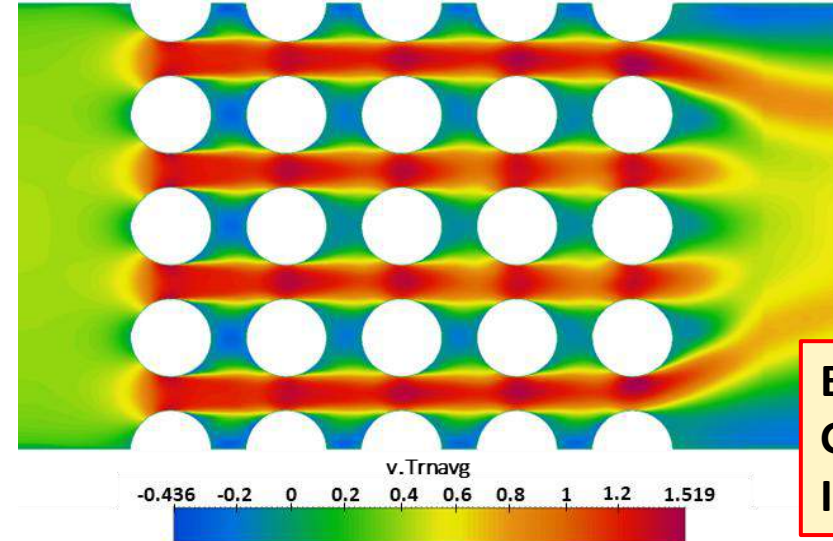
GRS



EDF



IPP

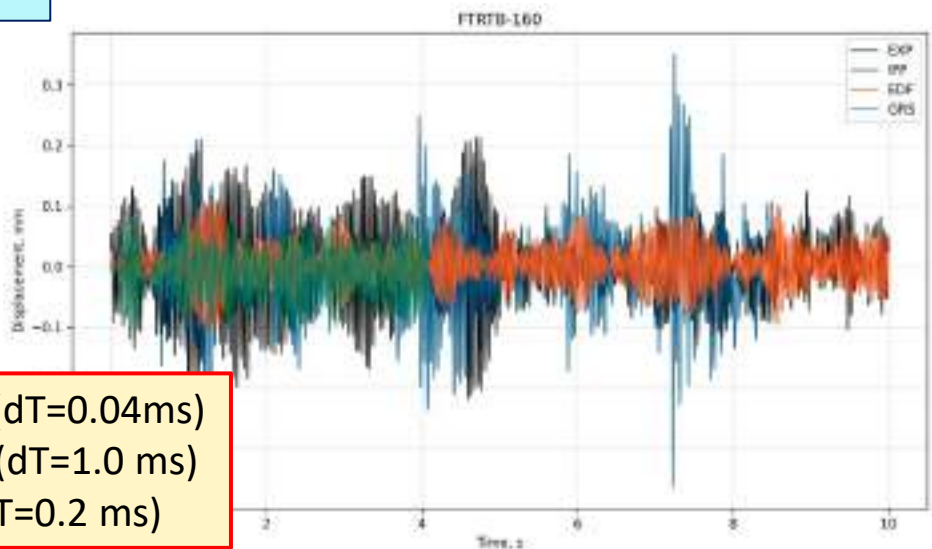


EDF: 41.7 s (dT=0.02ms)
GRS: 16.6 s (dT=0.2 ms)
IPP: 10.0 s (dT=1.0 ms)

The **best agreement of calculated time-averaged velocity field was obtained by EDF** where four high-velocity jets tend to attach to the side walls of the test section, while in **GRS** and **IPP** fields these jets coalesce behind the bundle.

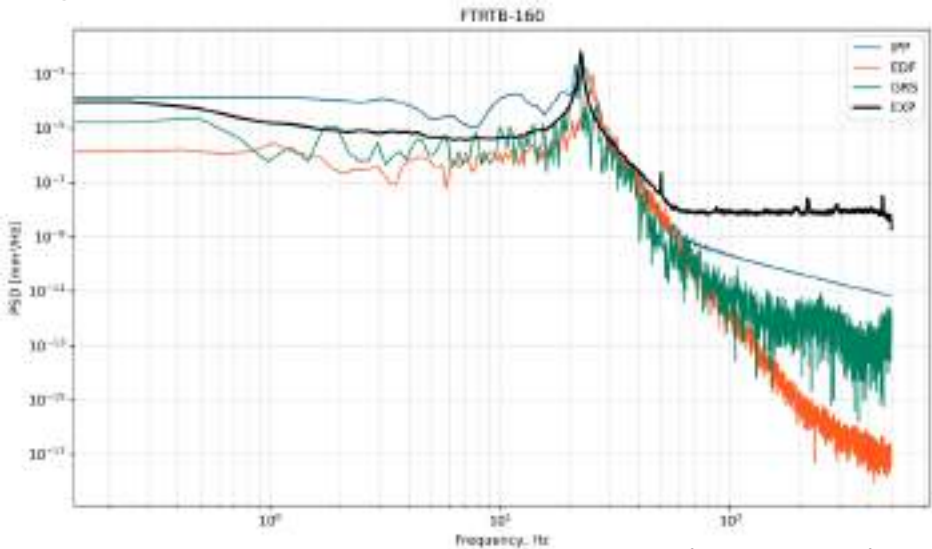
Simulations comparison (Flexible Tube in a Rigid Tube Bundle)

FTRTB-160



EDF: 55.0 s (dT=0.04ms)
GRS: 4.16 s (dT=1.0 ms)
IPP: 9.0 s (dT=0.2 ms)

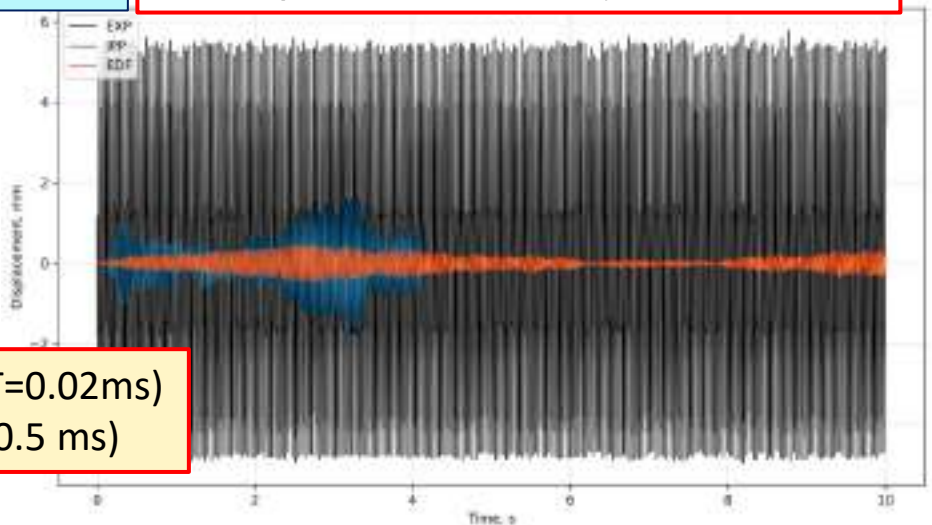
The **amplitudes** obtained in simulations have good **agreement** with the experimental ones.



Calculated **APSD** captured the **peak** present in the experimental **spectrum** with a good accuracy in all three simulations

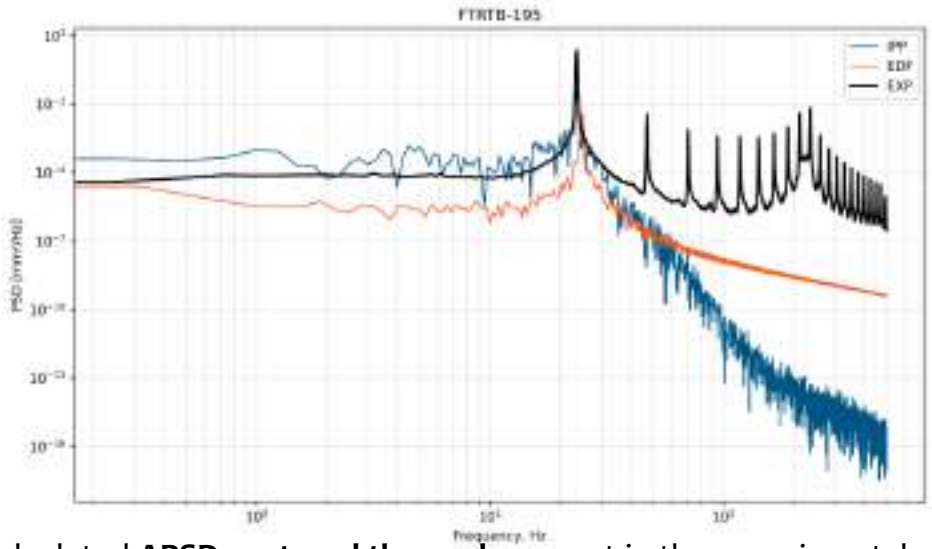
FTRTB-195

FEI occurs, and this is extremely challenging to be reproduced numerically



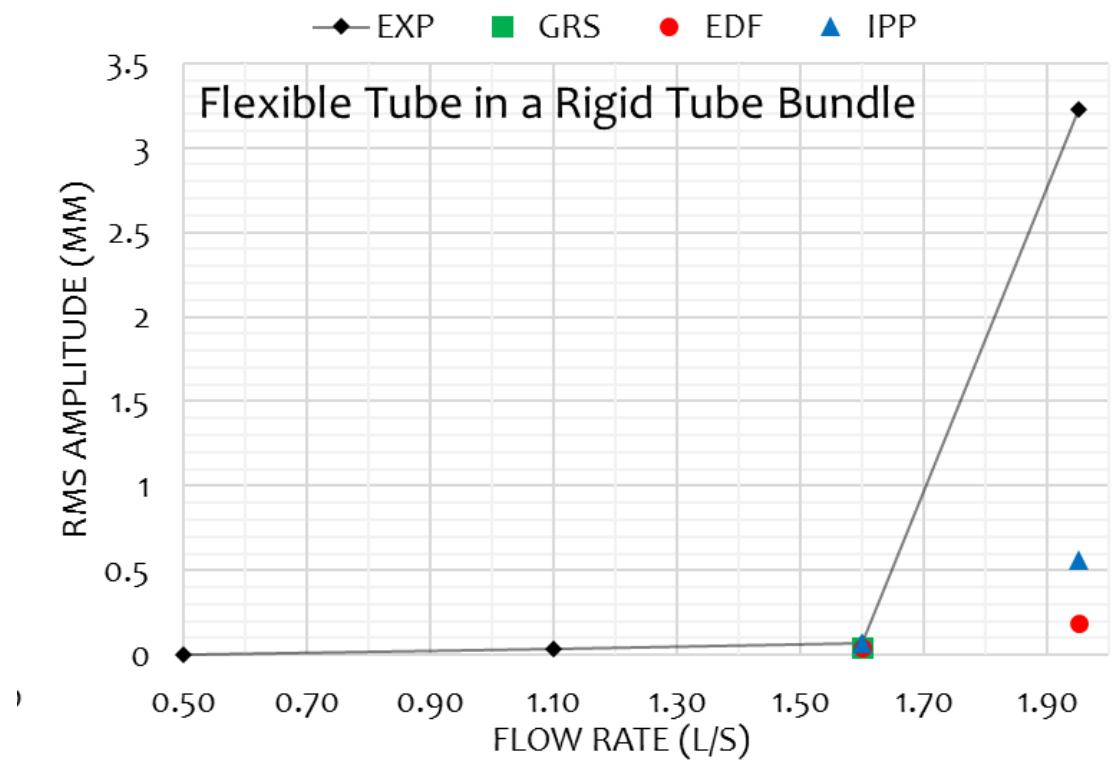
EDF: 42.7 s (dT=0.02ms)
IPP: 4.5 s (dT=0.5 ms)

The **amplitudes** obtained in simulations are **underestimated** compared to the experimental ones in both EDF and IPP analyses.



Calculated **APSD** captured the **peak** present in the experimental spectrum with a good accuracy in both simulations

Simulations comparison (Flexible Tube in a Rigid Tube Bundle)



FTRTB-160

Institution	RMS	1 st Peak Frequency
EXP	0.073	22.46
EDF	0.04	25.21
GRS	0.044	21.31
IPP	0.069	22.59

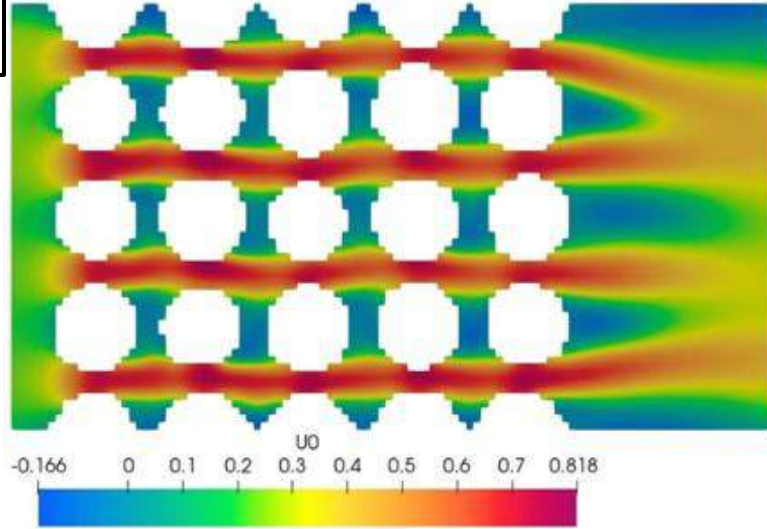
FTRTB-195

Institution	RMS	1 st Peak Frequency
EXP	3.223	23.68
EDF	0.191	24.69
IPP	0.561	23.75

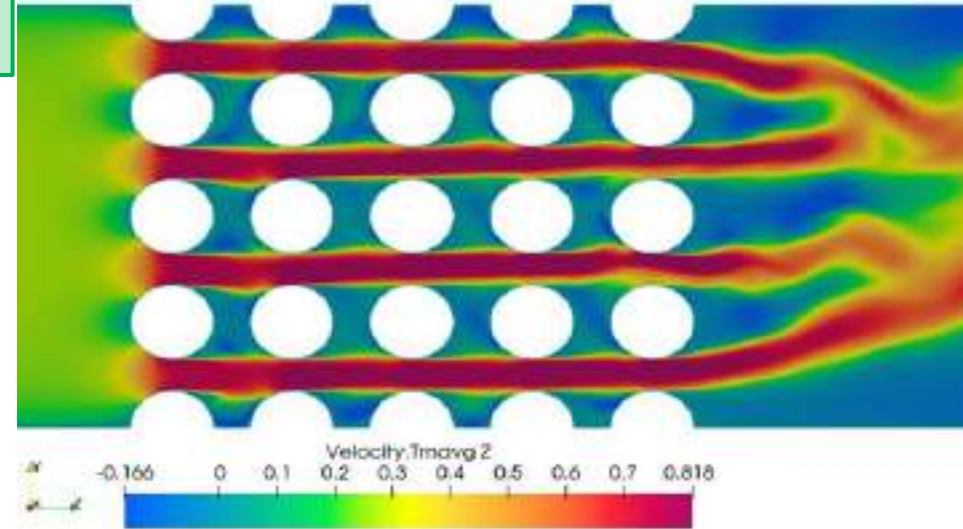
Simulations comparison (Flexible Tube in a Rigid Tube Bundle) FTRTB-160

FTRTB-160

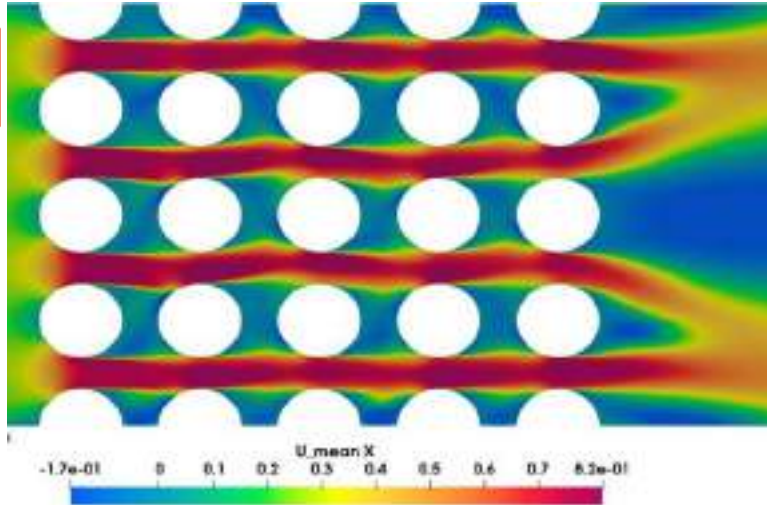
EXP



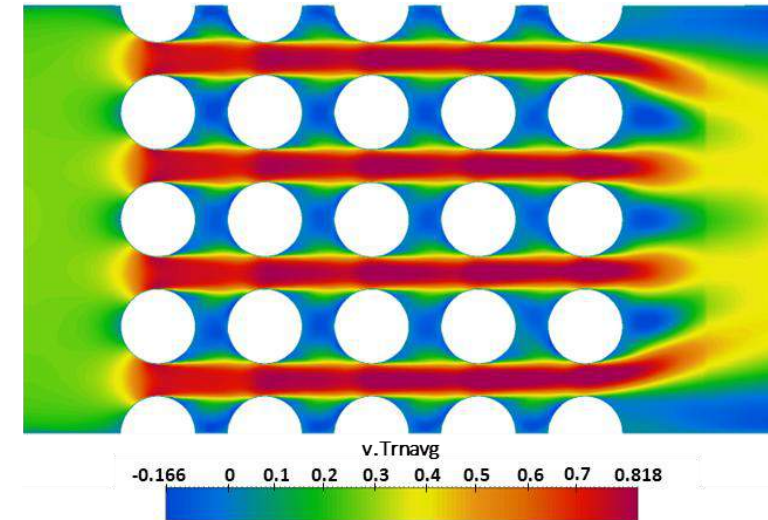
GRS



EDF



IPP



EDF: 55.0 s (dT=0.04ms)
GRS: 4.16 s (dT=1.0 ms)
IPP: 9.0 s (dT=0.2 ms)

The **best agreement of calculated time-averaged velocity field was obtained by EDF** where four high-velocity jets tend to form two jets after the bundle, while in **GRS** and **IPP** fields these jets tend to coalesce behind the bundle.

Overall comparison of results and computational costs



- **EDF performed the high-fidelity simulations** applying their CFD solver code_saturne v8.0 and **involving for this purpose supercomputers** (3D formulation)
- **GRS performed the medium-resolution simulations** applying the ANSYS CFX code with Rigid Body Solution for flexible rods configurations (3D formulation)
- **IPP** was aimed to test the possible application of the **least resource-intensive models** for simulations – for this purpose ANSYS CFX code with 2D formulation was applied for all simulated cases

The described differences in the approaches used are directly related to the computing resources required for modeling.

Case	Equivalent CPU/Node hours (RAM/ No. of (CPU/GPU) Cores)		
	EDF	GRS	IPP
SRT-290	4 M (57.7 Tb/ CPU=64)	233 (10.46 Gb/ CPU=30)	24.83 (1.23 Gb/ CPU=12)
SFT-170	3.7 M (45 Tb/ CPU=64)	374.75 (23.0 Gb/ CPU=50)	118.3 (0.49 Gb/ CPU=12)
SFT-290	1.8 M (36 Tb/ CPU=64)	336.16 (31.83 Gb/ CPU=96)	135.6 (0.49 Gb/ CPU=12)
RTB-170	2.7 M (155 Tb/ CPU=37)	562.08 (44.77 Gb/ CPU=70)	136.0 (2.59 Gb/ CPU=12)
RTB-290	13 M (155 Tb/ CPU=64)	975.96 (44.77 Gb/ CPU=70)	189.0 (2.59 Gb/ CPU=12)
FTRTB-160	4 M (42 Tb/ CPU=64)	267.166 (37.81 Gb/ CPU=50)	803.3 (1.82 Gb/ CPU=12)
FTRTB-195	6.5 M (30 Tb/ CPU=64)	-	280.6 (1.82 Gb/ CPU=12)

Case	Wall Clock Time needed for 1 second of simulation, s		
	EDF	GRS	IPP
SRT-290	35.6×10 ³	83.9×10 ³	1.49×10 ³
SFT-170	48.7×10 ³	135.0×10 ³	3.55×10 ³
SFT-290	94.7×10 ³	2.69×10 ³	4.07×10 ³
RTB-170	72.3×10 ³	202.0×10 ³	4.08×10 ³
RTB-290	88.0×10 ³	211.4×10 ³	5.67×10 ³
FTRTB-160	54.9×10 ³	231.0×10 ³	26.77×10 ³
FTRTB-195	142.4×10 ³		18.71×10 ³

Conclusions



Analyzing the obtained results, it could be concluded:

- some test cases with **Single Rigid Tube** turned out to be difficult to predict by all the applied models in terms of the obtained RMS for the Lift and Drag force components
- for the **Rigid Tube Bundle** cases the corresponding results demonstrate much better agreement with the experiment
- for flexible configurations, **excellent agreement with the experiment was obtained by EDF and GRS** for cases with **Single Flexible Tube**, while **IPP** have slightly overestimated predictions for the corresponding RMS values
- for **Flexible Tube in a Rigid Tube Bundle** all participants obtained excellent agreement in the displacements RMS for an average flow rate of 1.6 l/s
- when analysing **the FTRTB case with the highest flow rate of 1.95 l/s**, both **EDF** and **IPP**, although underestimating the RMS amplitude, still showed a nonlinear increase in displacement RMS when moving to a higher flow rate, indicating **capturing the Fluid Elastic Instability phenomenon**.

Despite the radically different modeling approaches, **each demonstrated reasonably good results and agreement with experiment**. Moreover, **the degree of agreement with experiment proved to be a function of the detail of the models used and simulation time length**.

The results obtained by EDF and GRS were closest to experiment in all the cases considered. While the IPP results in some cases (particularly with a single flexible rod) differed in their predictions of vibrations amplitudes, they still provided insight into the order of magnitude and spectrum of vibrations.

SUBMITTED



D3.2 AMOVI model preparation and simulation results

Version N°2
May / 2026

Madslav Filonov¹, Yulia Filonova¹, Yaroslav Dubyk², Hemish Mistry³, Angel Papadachiev¹, Sofiane Berhamadouchi¹

¹IPP Centre (IPP), Kiev, Ukraine

²Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Garching, Germany

³Electricité De France (EDF Lab Chatou), France



Funded by
the European Union



For the **most accurate simulations**, improved estimation methods using 3D models with sufficiently dense computational mesh discretization and resource-intensive turbulence models are clearly required.

For **engineering purposes**, the use of "lightweight" 2D models with specialized user functions is justified, which can be especially useful when analyzing more complex equipment configurations such as steam generators.





GO-VIKING

Thank you!



Yuliia Filonova



IPP-CENTRE



filonova-ys@ipp-centre.com.ua



Website:
www.go-viking.eu



Mail:
contact@go-viking.eu



Twitter:
[@GOVIKING_EU](https://twitter.com/GOVIKING_EU)



LinkedIn:
[GO-VIKING](https://www.linkedin.com/company/GO-VIKING)



Funded by the
European Union



GOKSTAD Facility

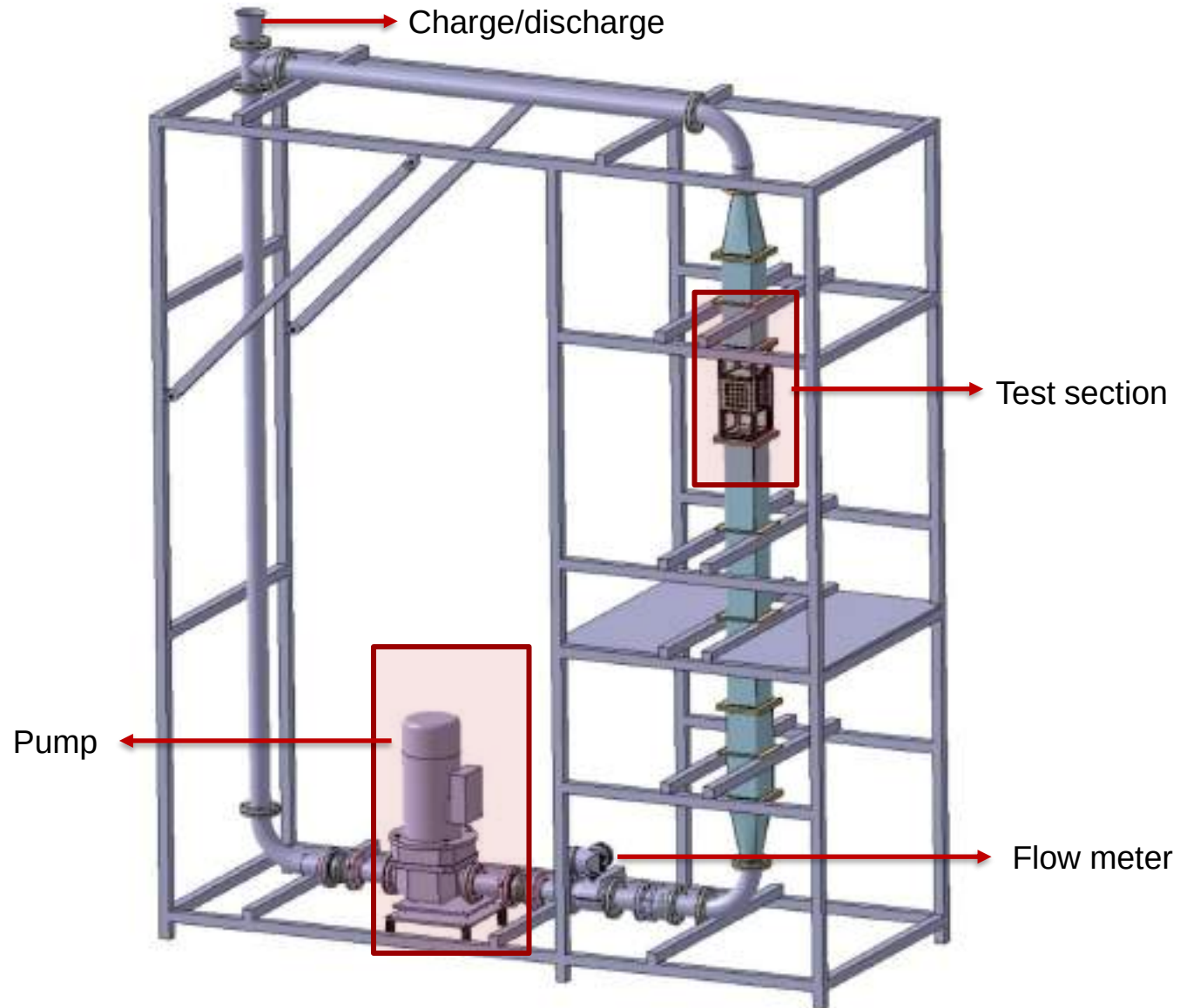
FIV in crossflow through bundles

Muhsin Can Akkurt, Philippe Planquart, Mathieu Delsipee

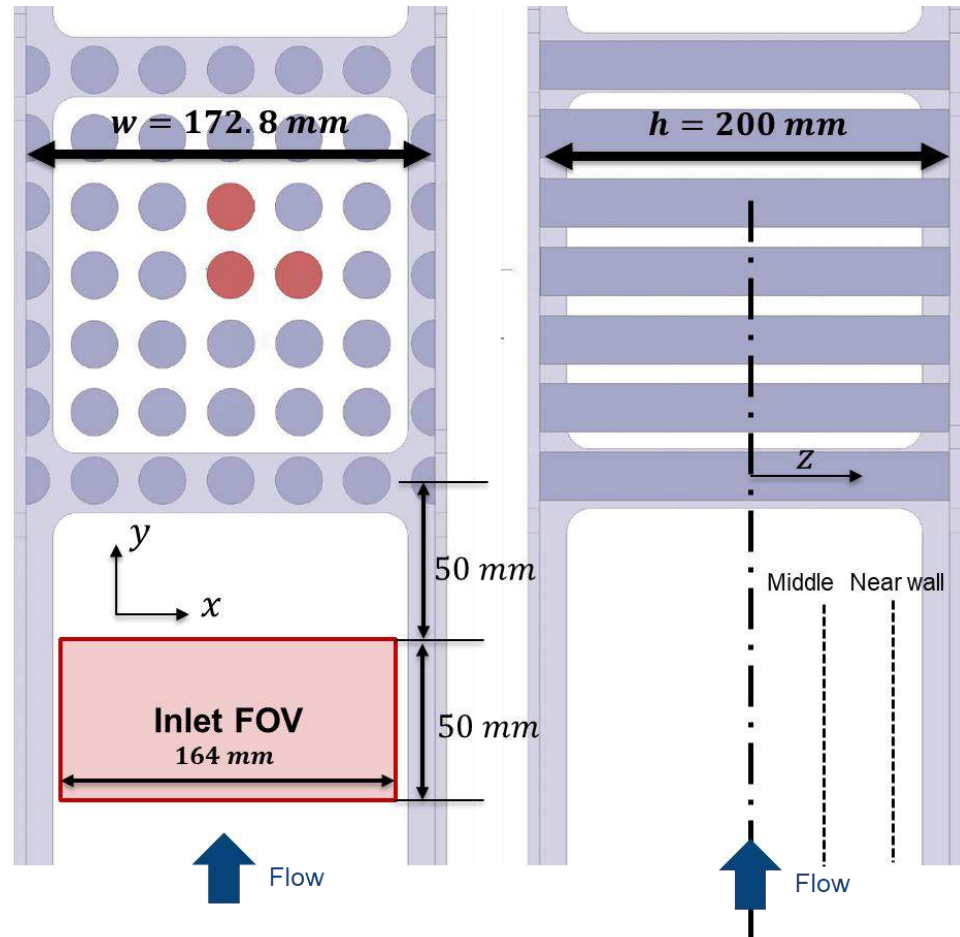


Funded by
the European Union

Experimental Facility



Test Conditions



Flow rates:

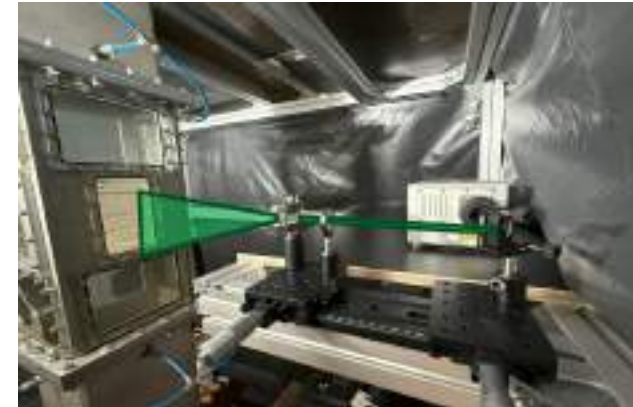
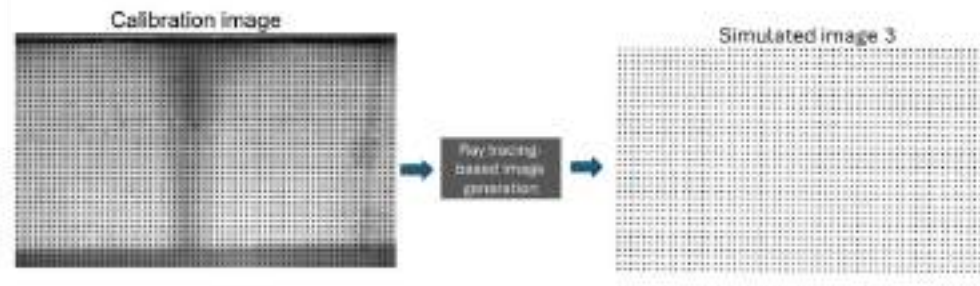
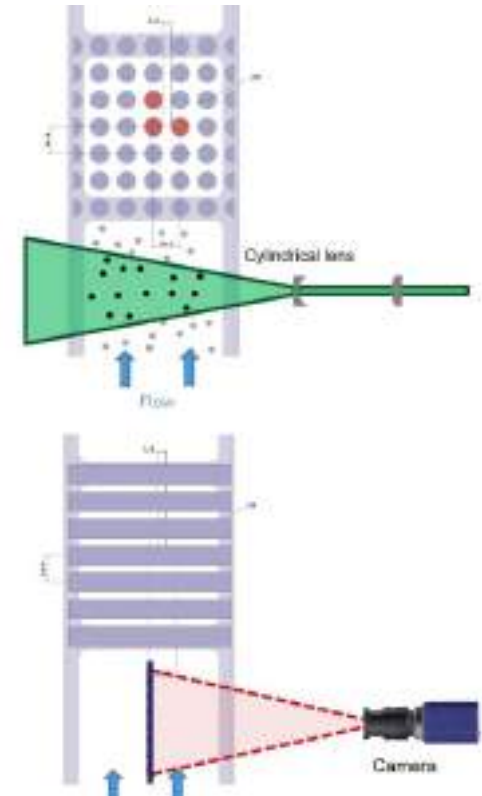
- 15 m³/h
- 30 m³/h
- 60 m³/h
- 90 m³/h
- 120 m³/h

Measurement planes:

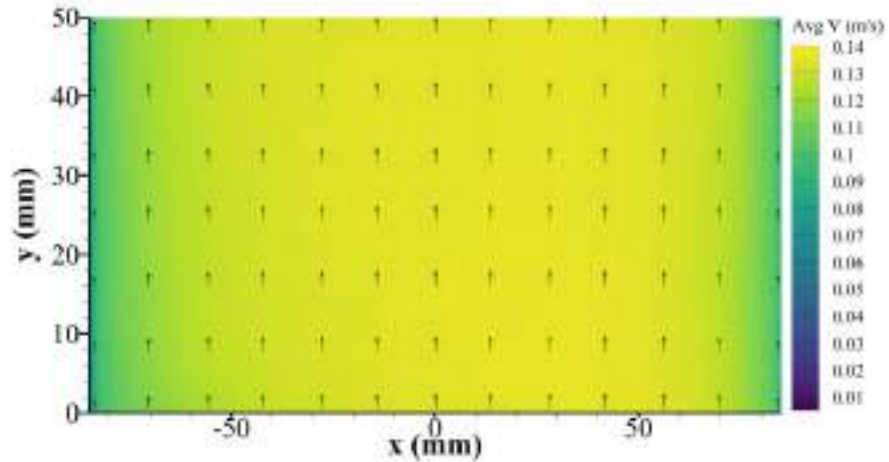
- Center ($z=10\text{cm}$)
- Middle ($z=7\text{cm}$)
- Near wall ($z=3.5\text{cm}$)

Measurement regions:

- Inlet
- Around the bundle
- Outlet



Results: Average velocity at center (symmetry) plane

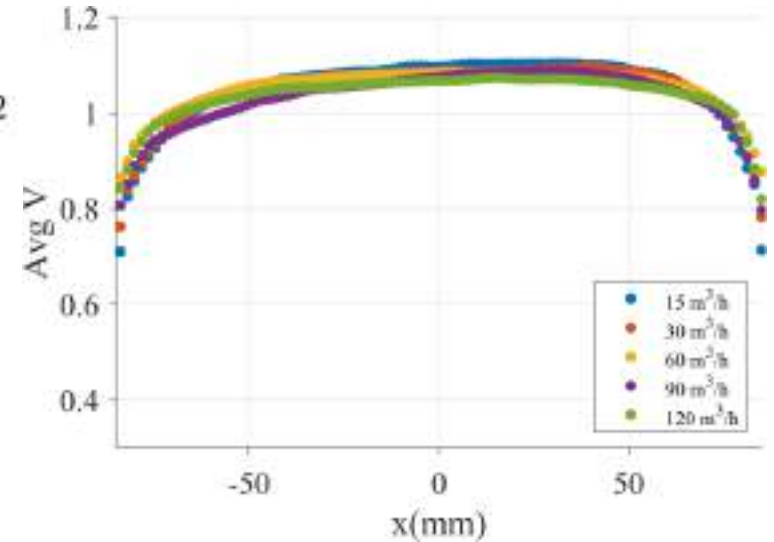
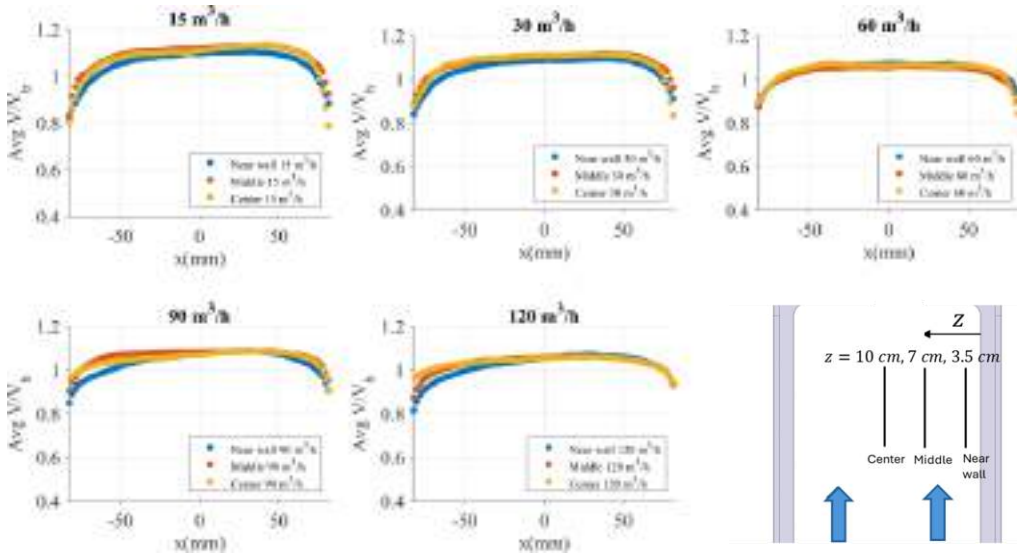
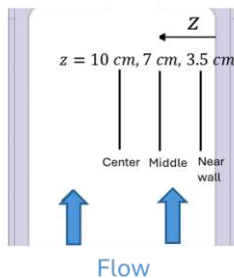


$$TKE = \frac{3}{4} (\overline{u'^2} + \overline{v'^2}) \quad \overline{u'^2} = \frac{1}{N} \sum_1^N u'^2$$

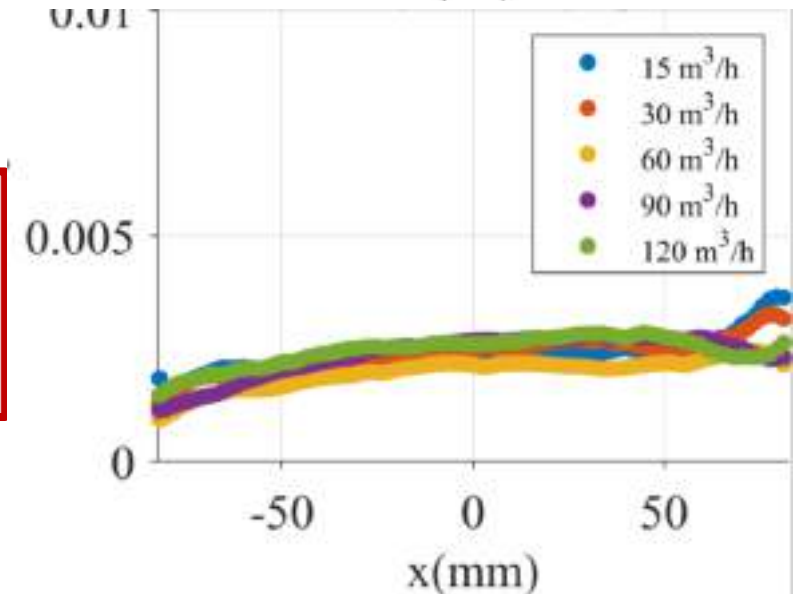
u : instantaneous velocity component in x direction

v : instantaneous velocity component in y direction

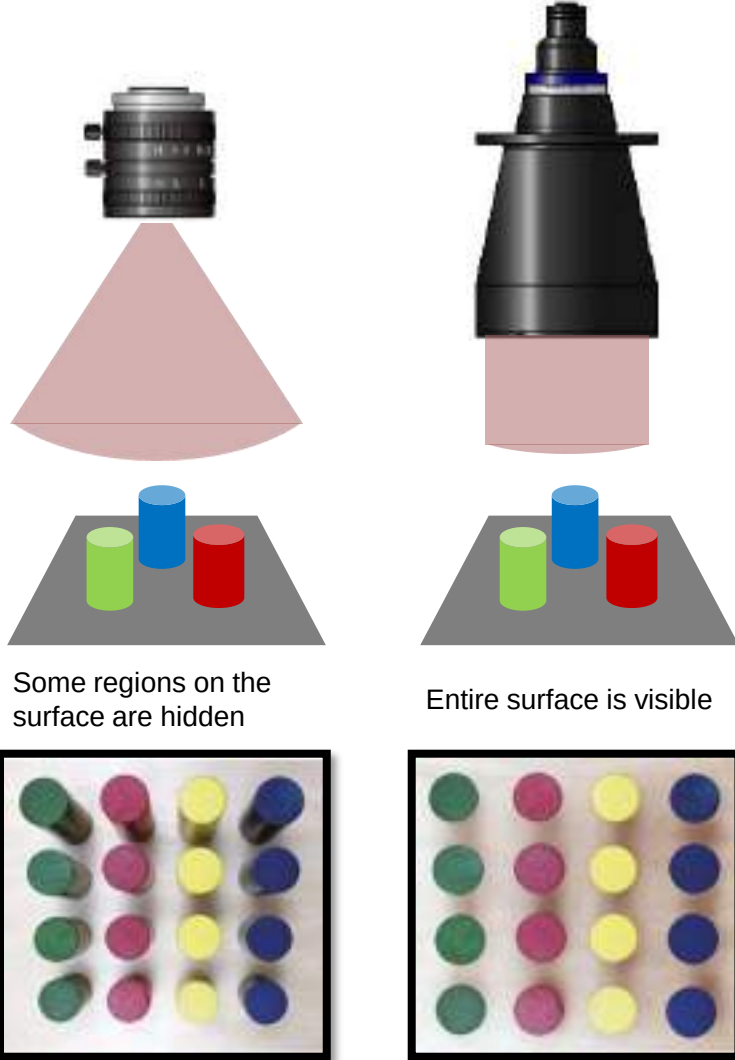
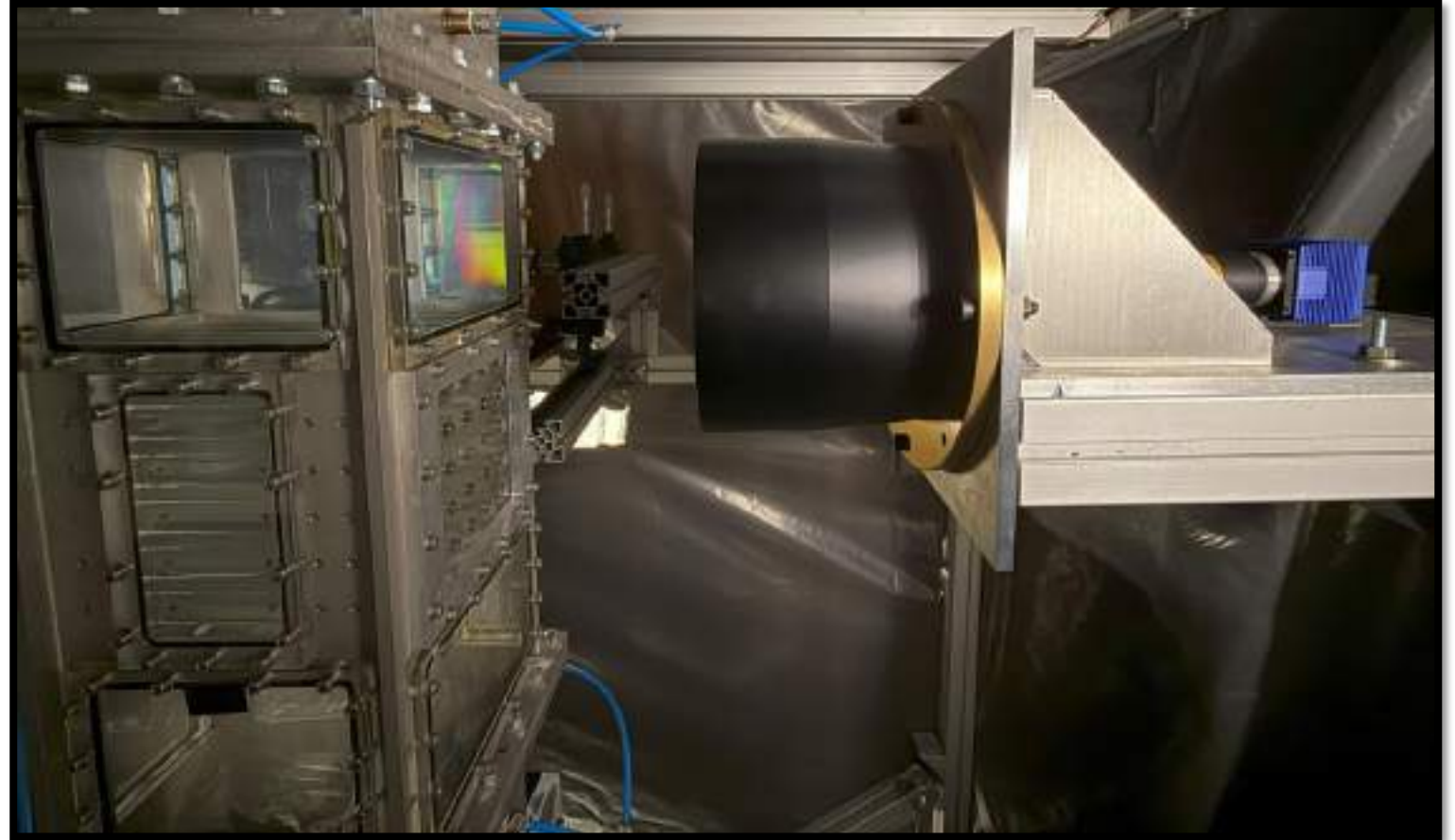
N : number of samples (1000 PIV image pairs)



Assumption for TKE: The third component (w) contains a similar turbulent contribution as the other two components (u and v).



PIV measurements in the bundle region



Results: Average velocity

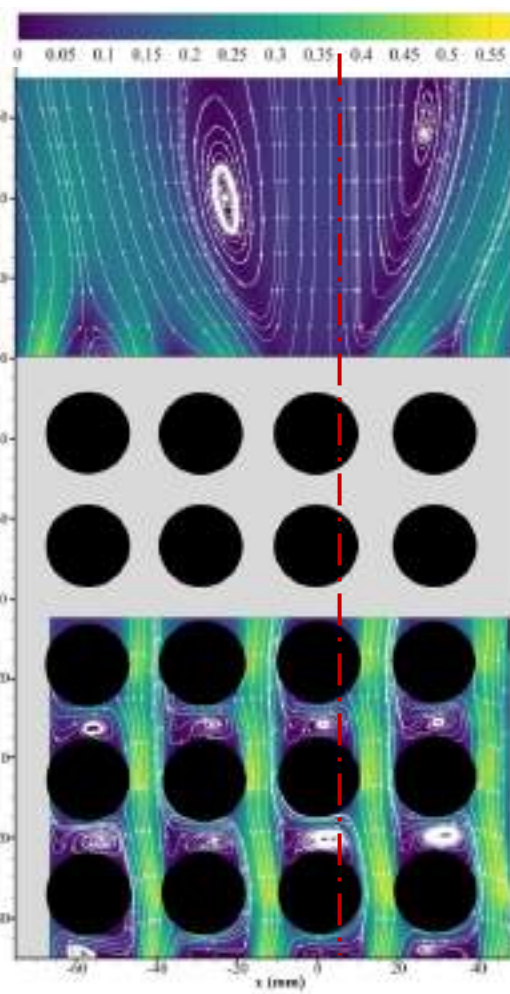
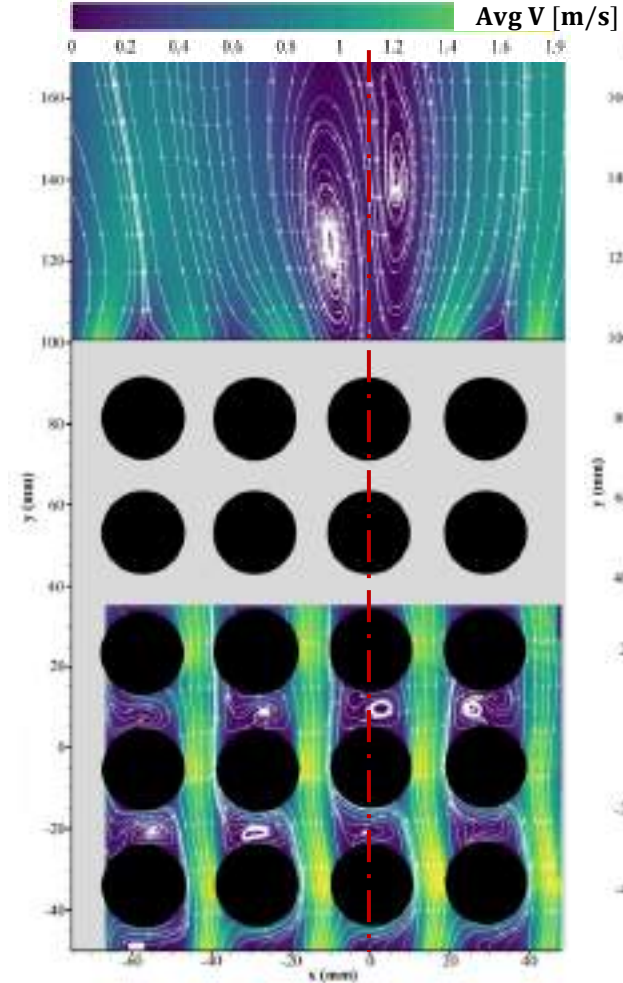
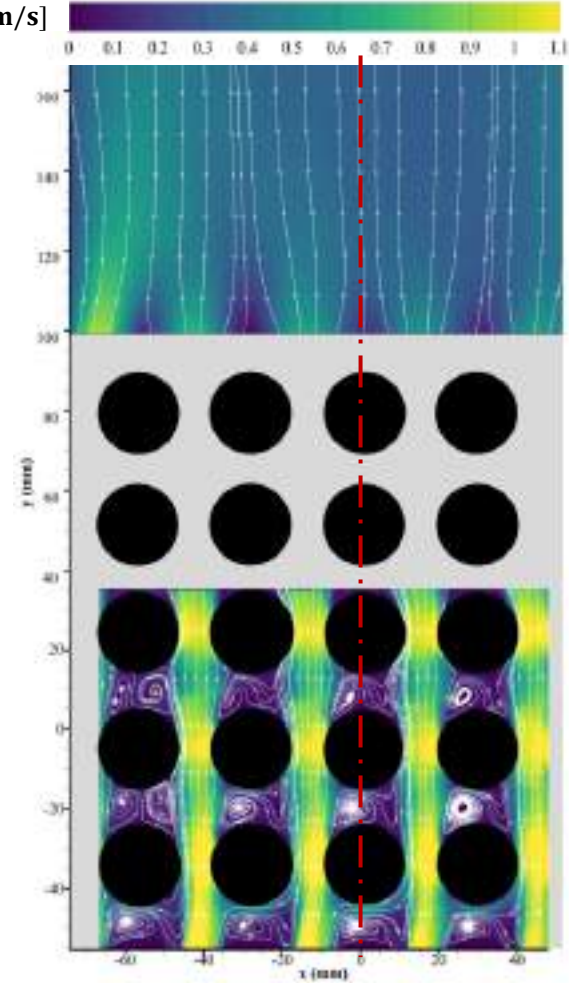
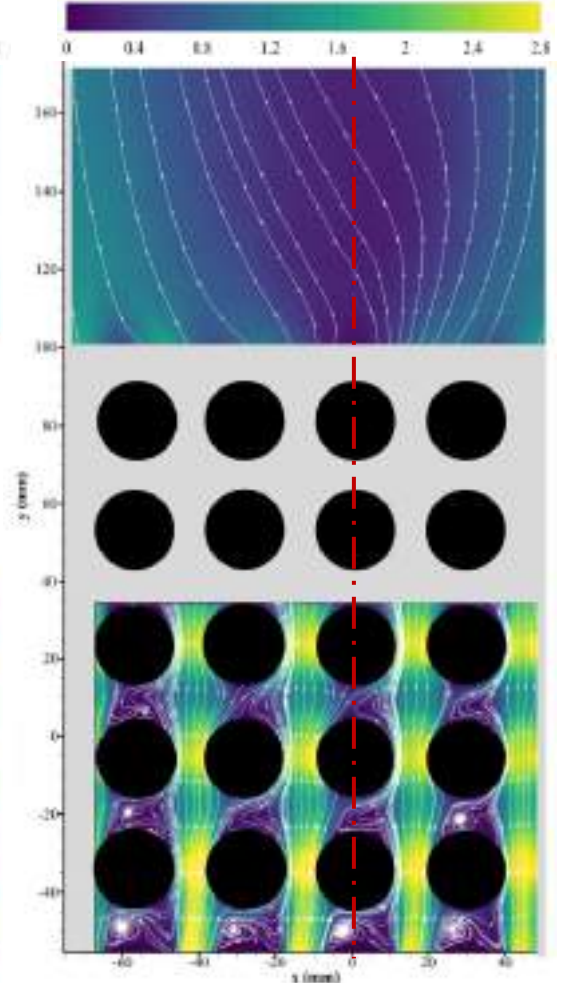
$$U_b = 0.125 \text{ m/s}$$

$$U_b = 0.25 \text{ m/s}$$

$$U_b = 0.49 \text{ m/s}$$

$$U_b = 0.73 \text{ m/s}$$

Avg V [m/s]



Results: Turbulent kinetic energy (TKE)

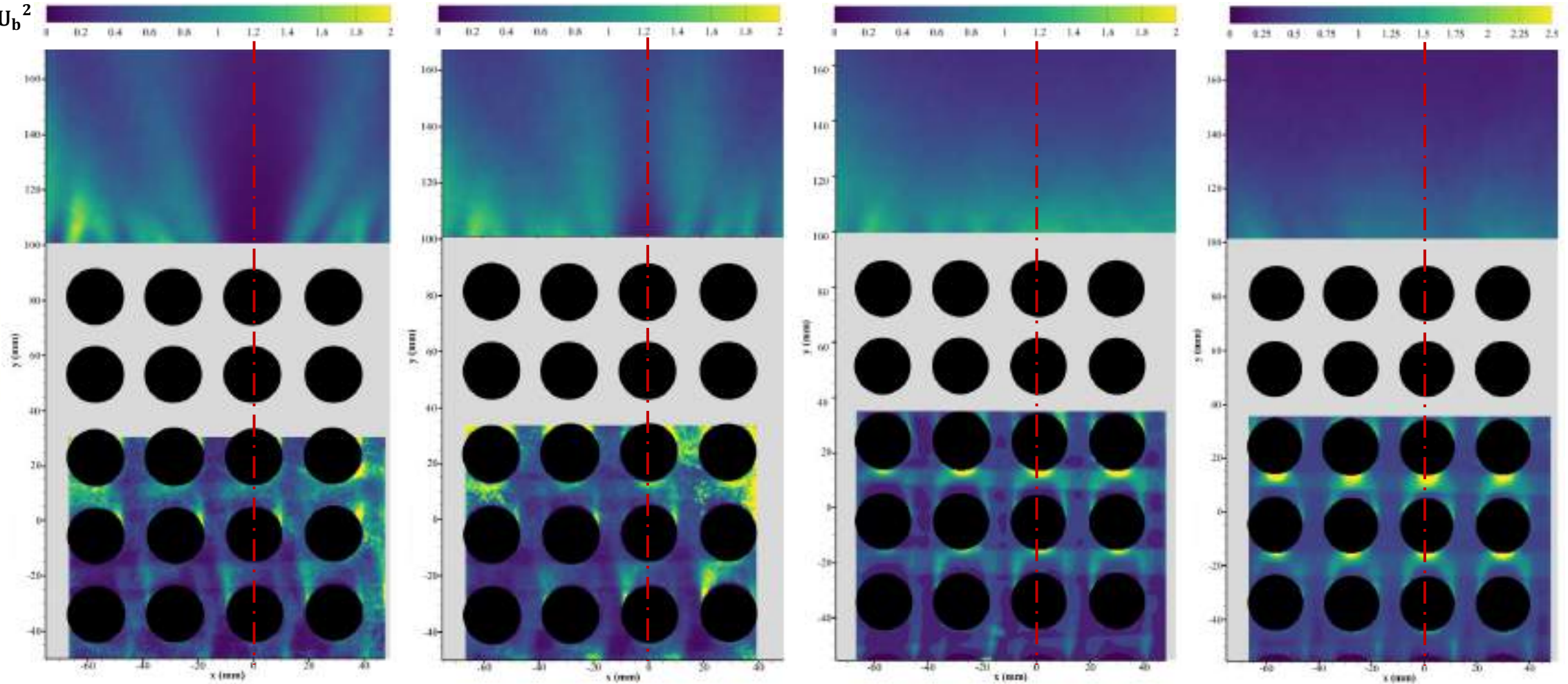
$U_b = 0.125 \text{ m/s}$

$U_b = 0.25 \text{ m/s}$

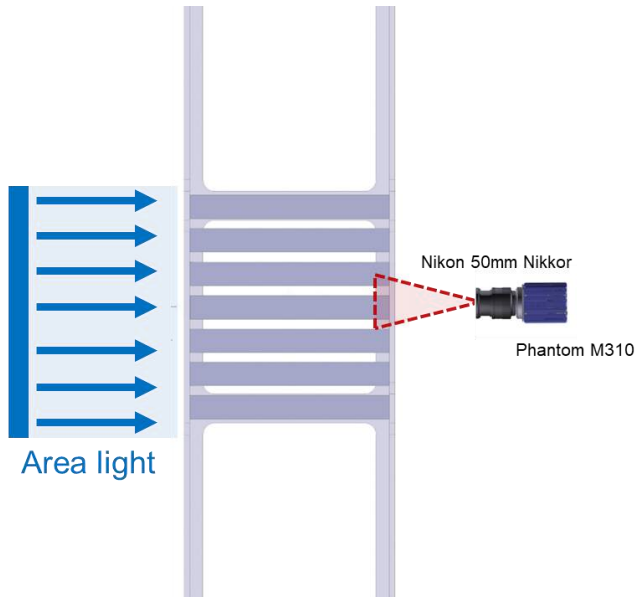
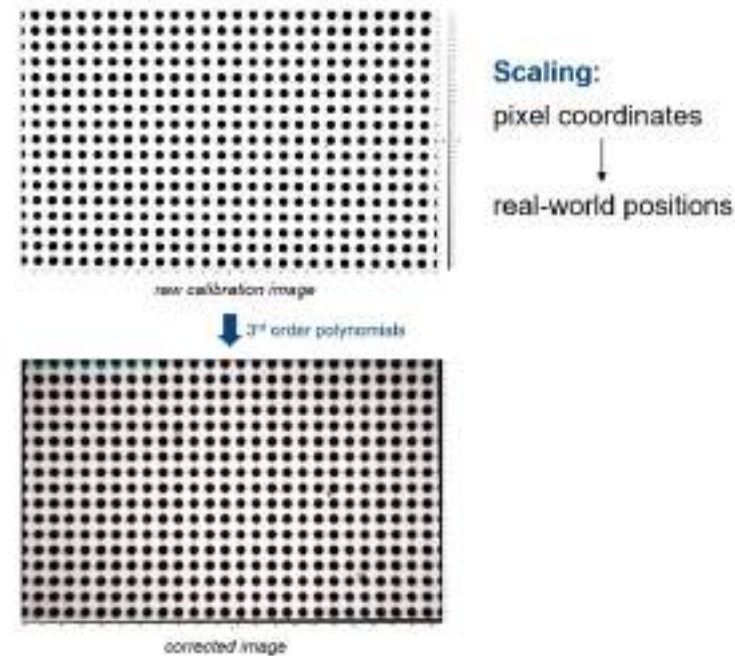
$U_b = 0.49 \text{ m/s}$

$U_b = 0.73 \text{ m/s}$

TKE/U_b^2



Experiments in GOKSTAD



Resolution: 0.1 pixel (%0.015D)

System properties

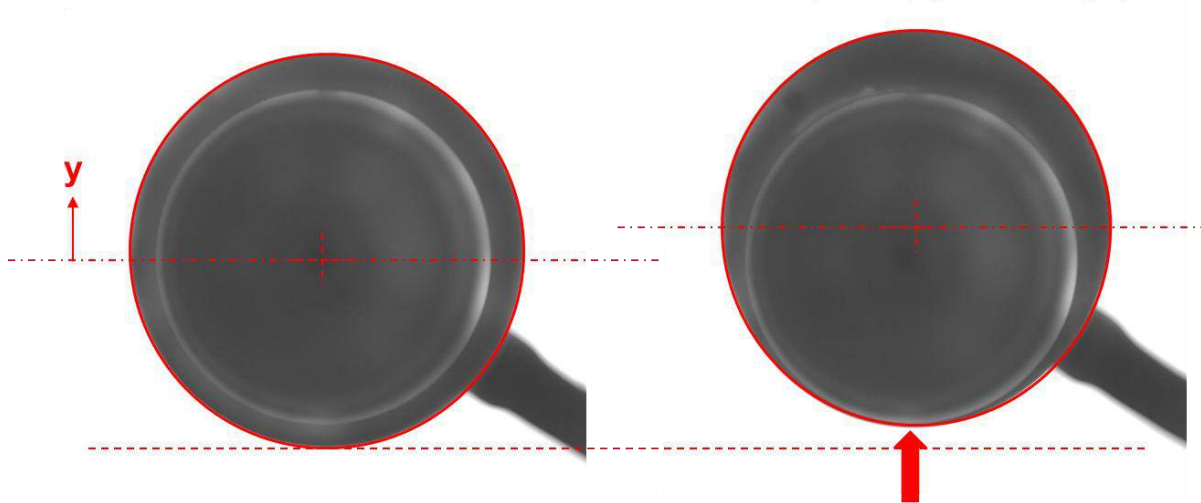
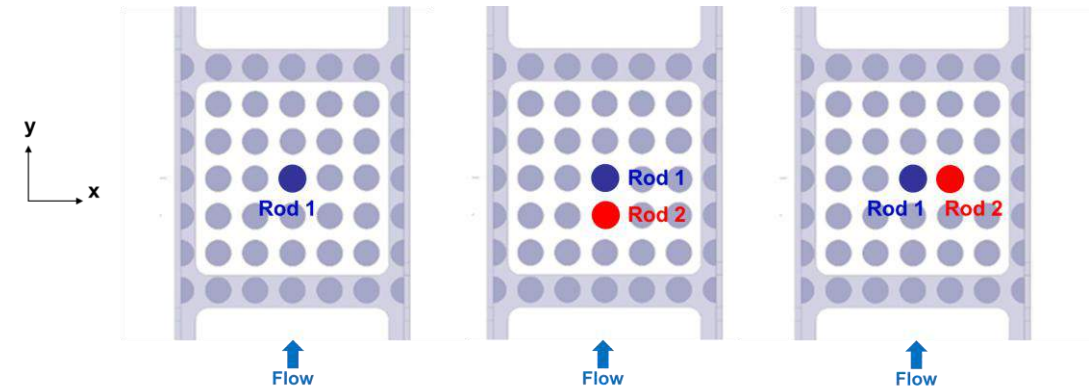
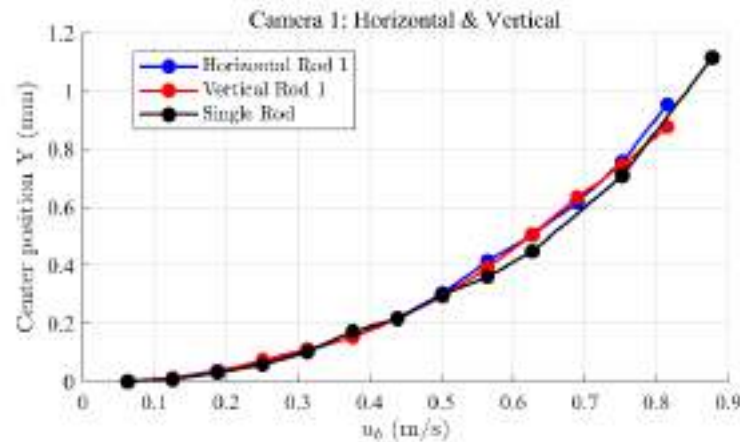
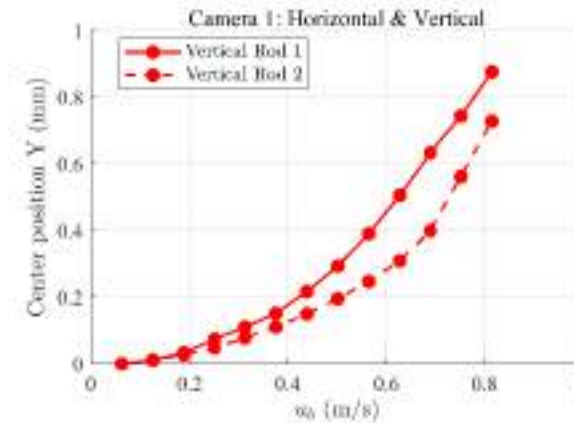
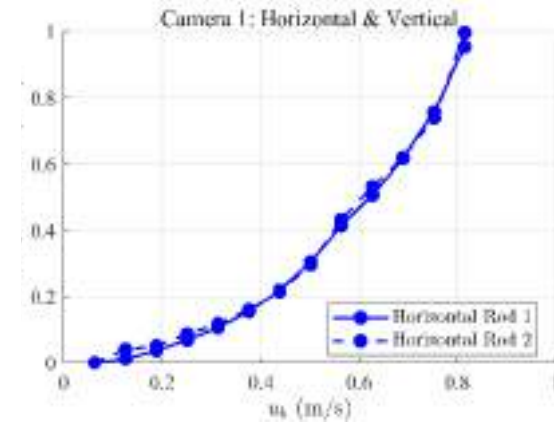
Parameter	Value	Unit
Mass	0.100	kg
Stiffness	7000	N/m
Damping coefficient	1.14	Ns/m
Natural frequency (in air)	42	Hz

- Two configurations:
 - Single vibrating rod
 - Two vibrating rods (vertically and horizontally placed)
- 14 flow speeds:
 - $u_b = 0.063 - 0.878 \frac{\text{m}}{\text{s}}$
 - 500 Hz

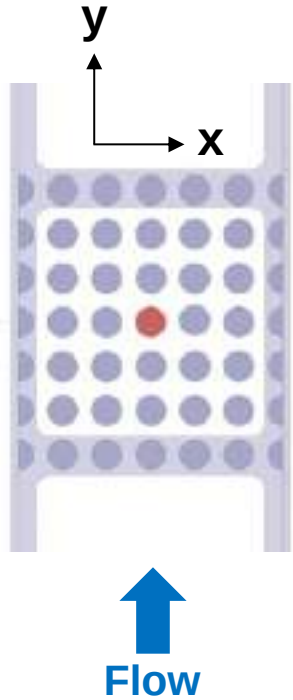
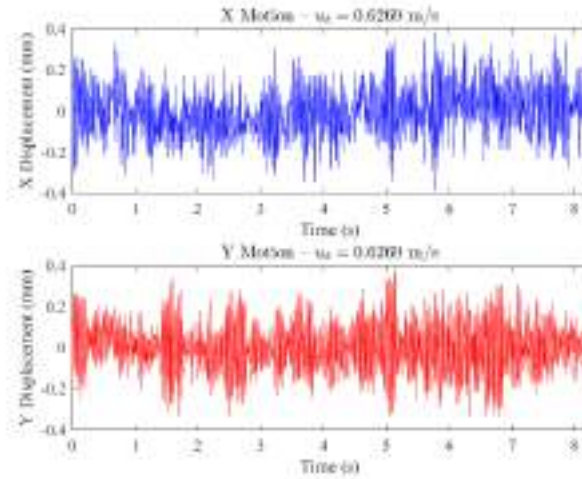
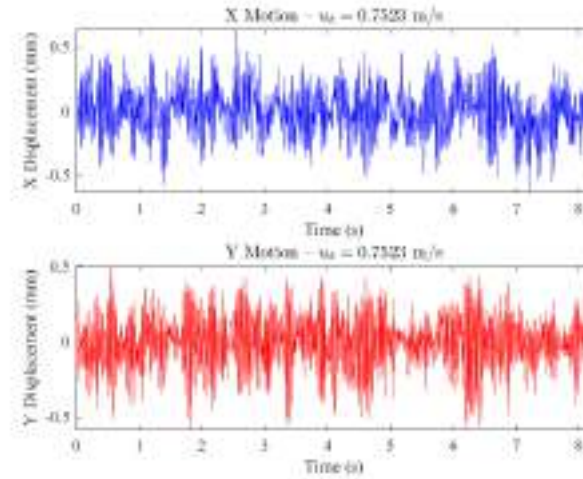
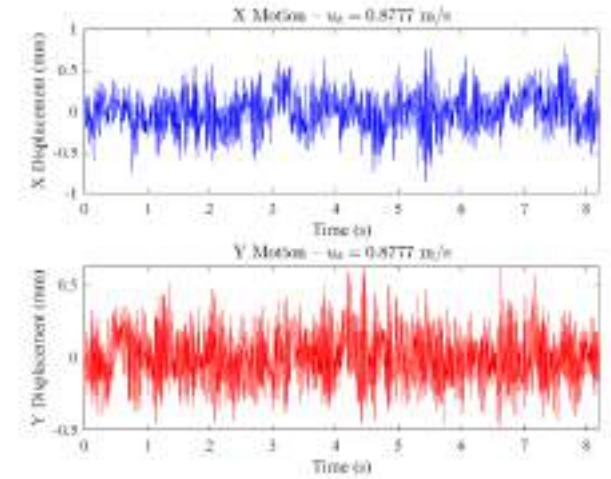
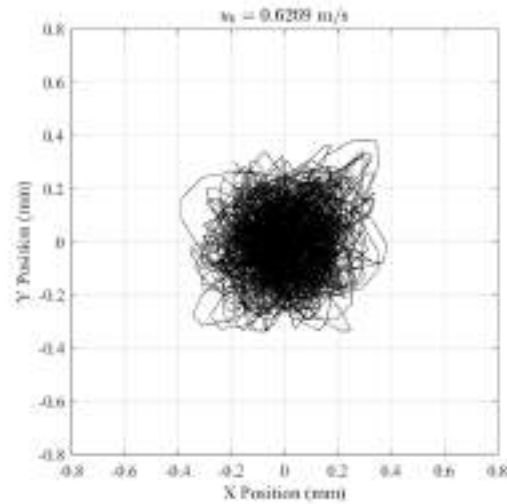
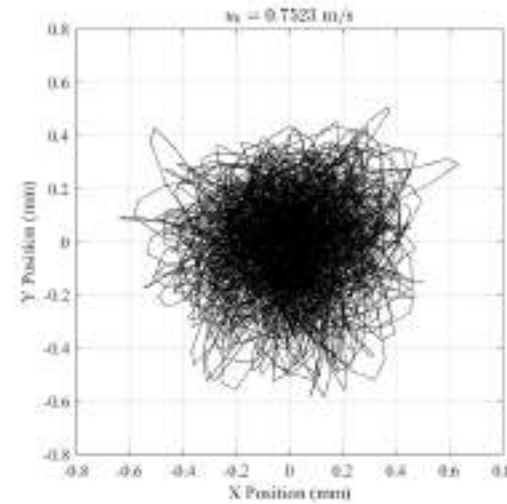
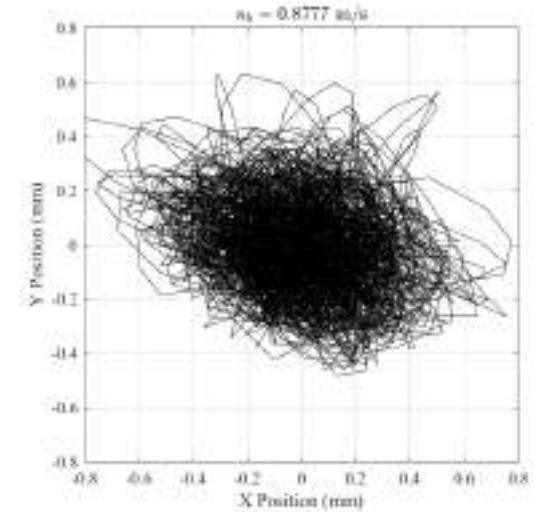
Results: Average center position

Without flow

Max speed ($u_b = 0.87\text{m/s}$)



Single rod vibrating

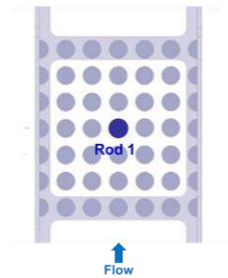
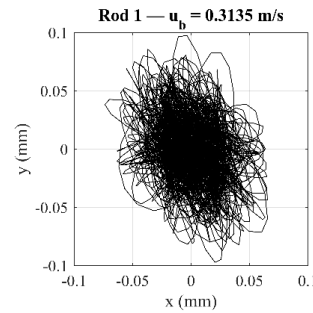
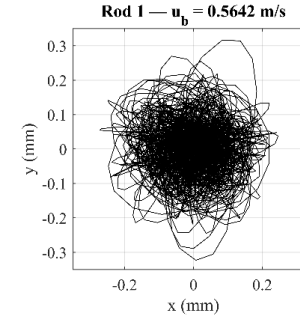
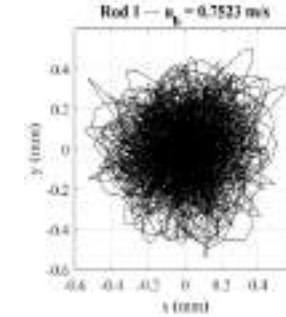
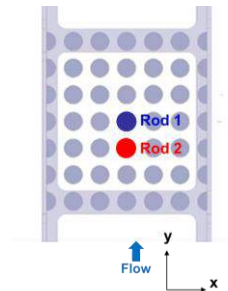
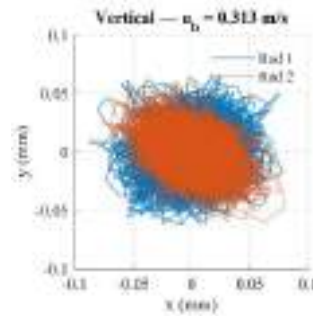
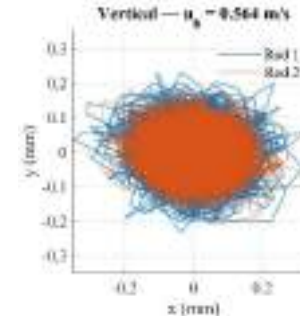
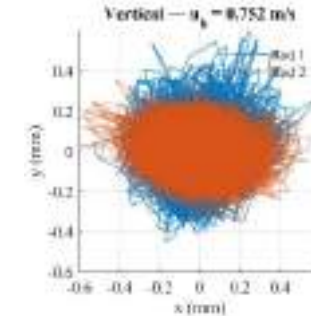
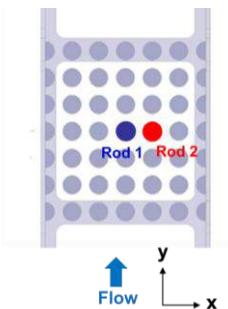
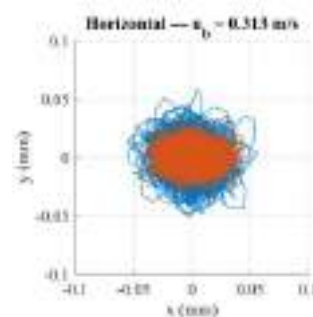
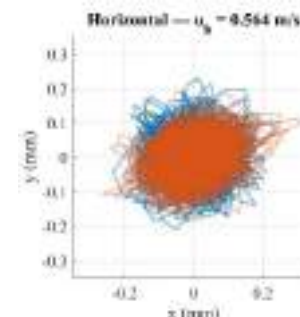
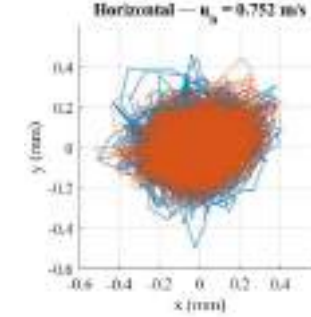


Results: Orbit plots

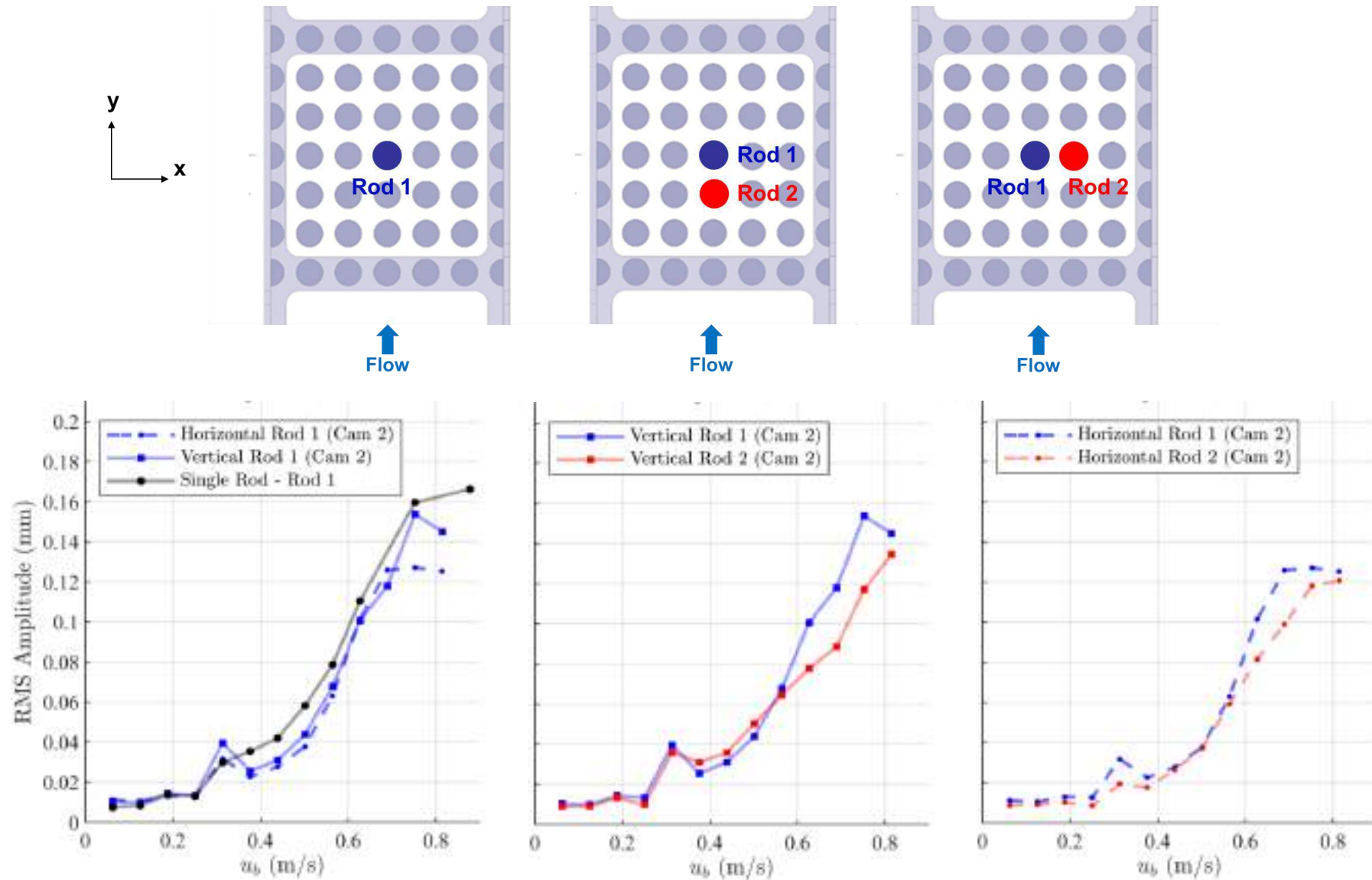
$$u_b = 0.31 \text{ m/s}$$

$$u_b = 0.56 \text{ m/s}$$

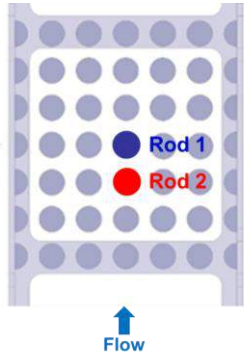
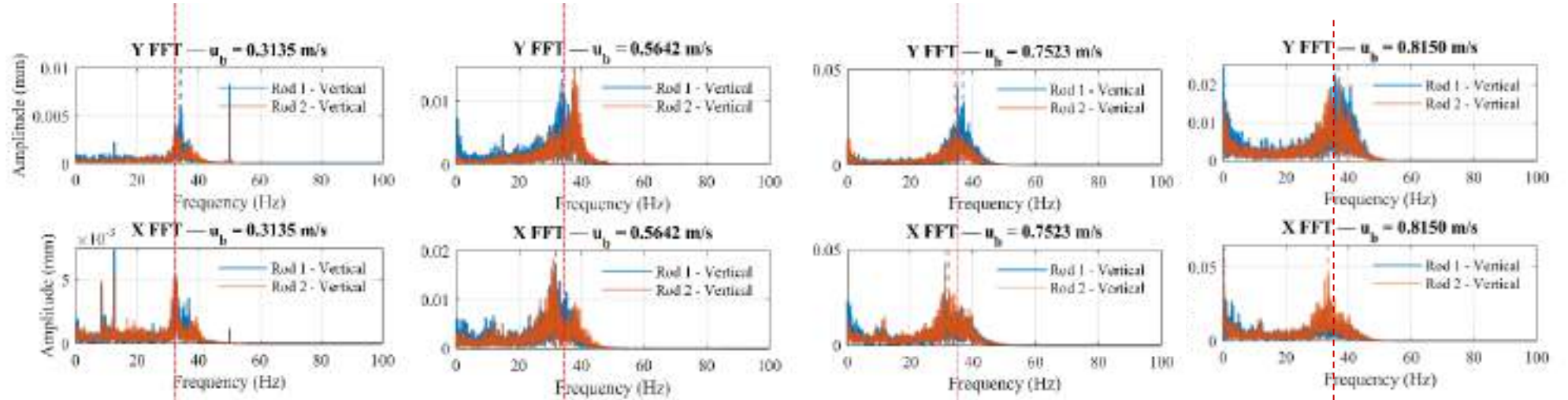
$$u_b = 0.72 \text{ m/s}$$



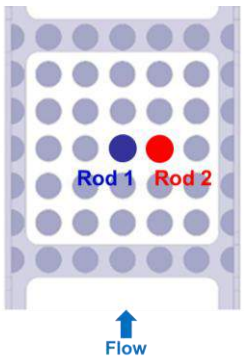
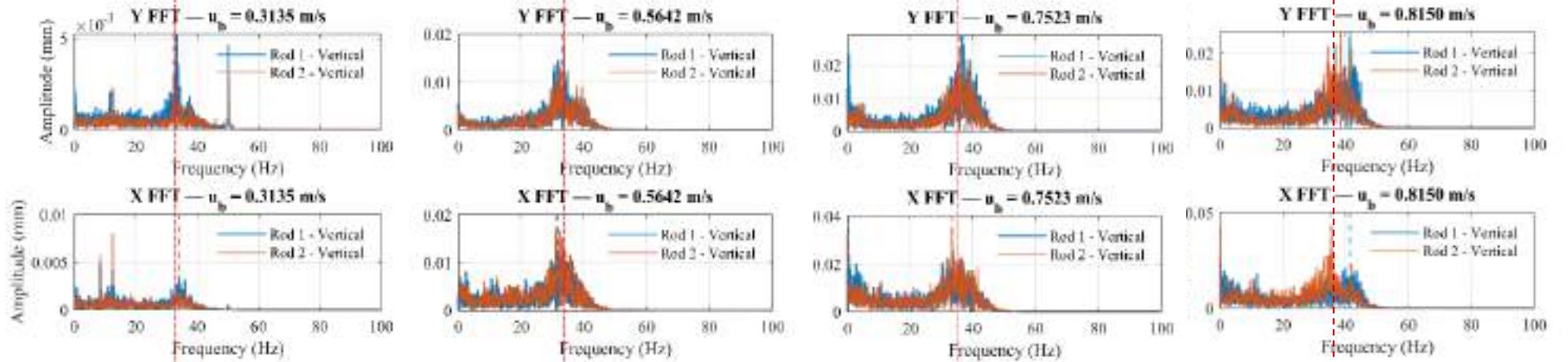
Results: Y RMS vs bulk velocity



Results: FFTs

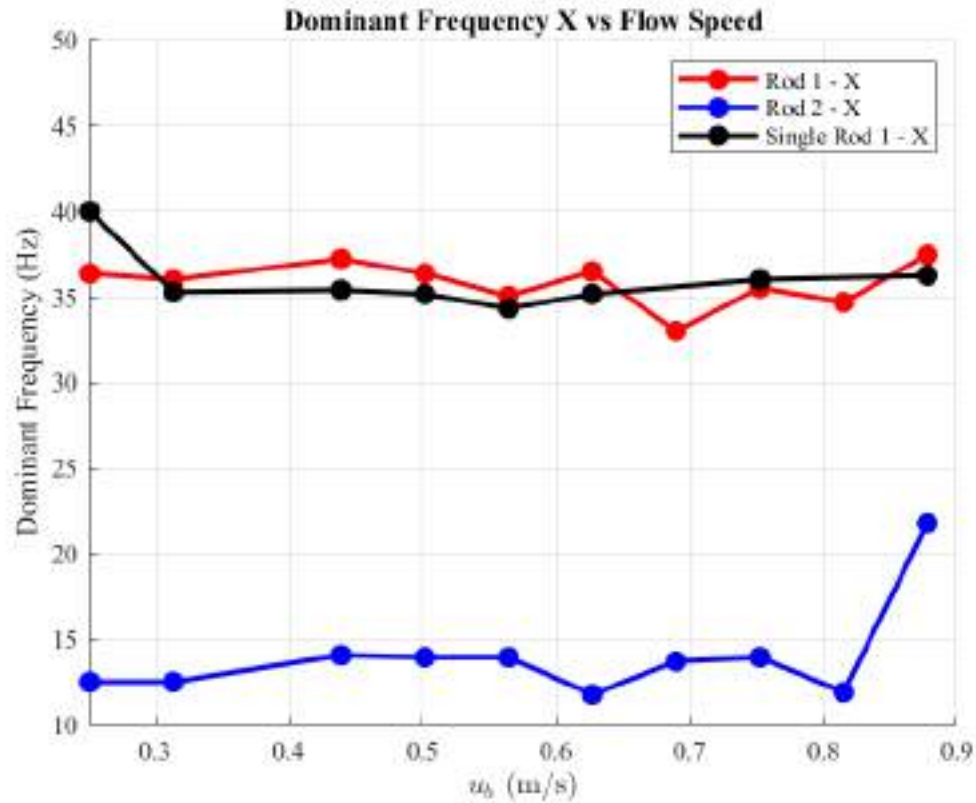
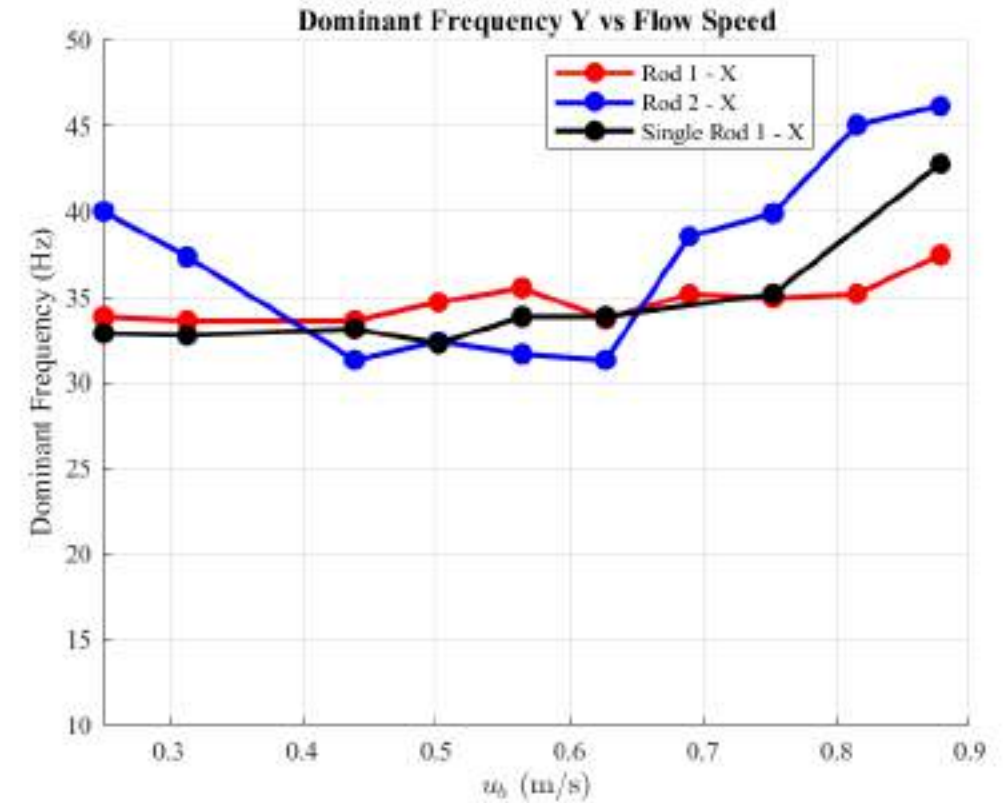


(b)

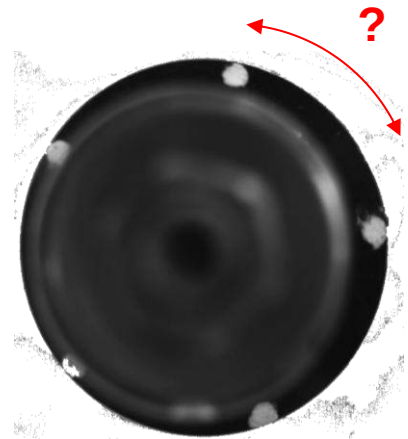


(c)

Two rods vibrating

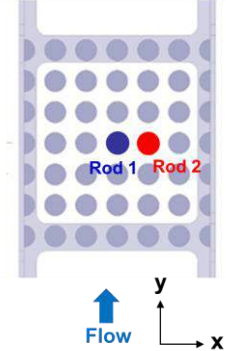


Results: Rotational motion

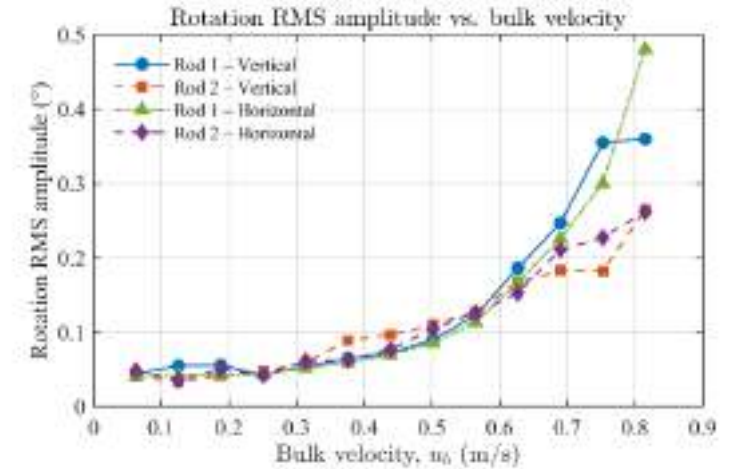


Motion Comparison — $u_b = 0.5642$ m/s

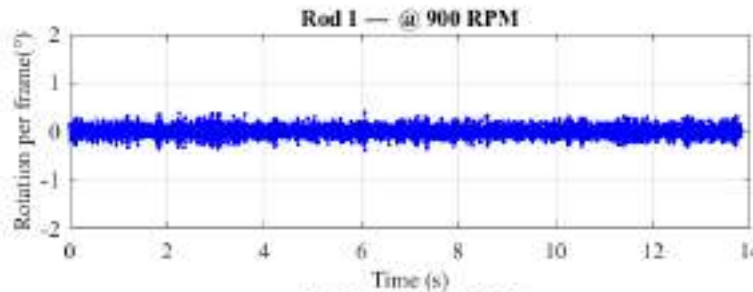
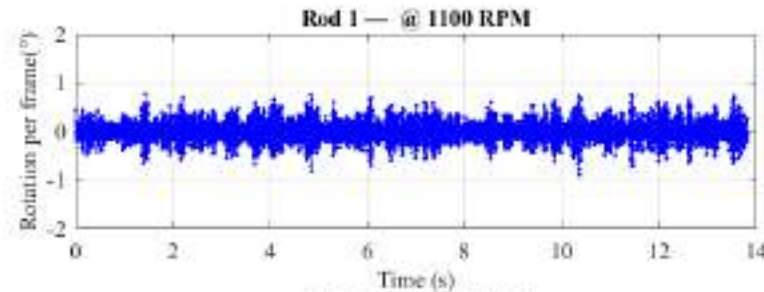
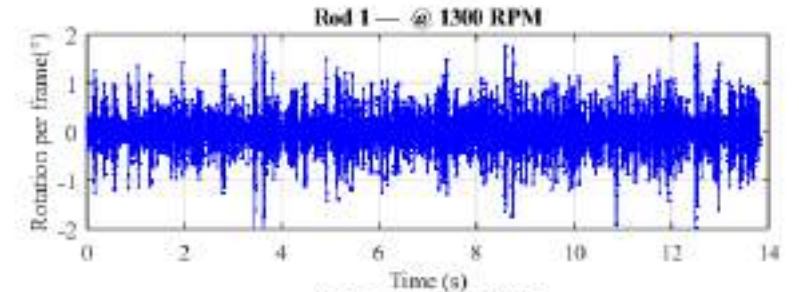
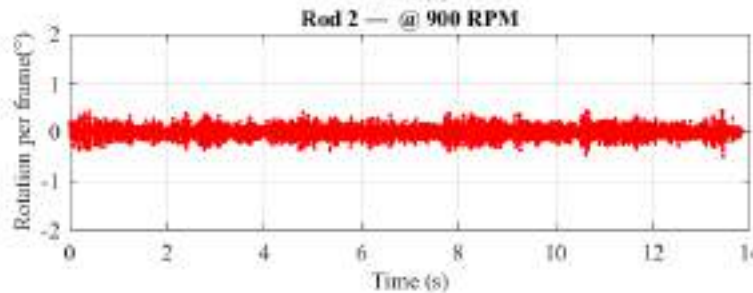
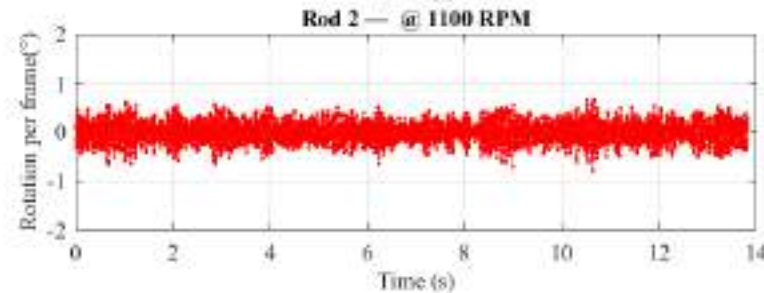
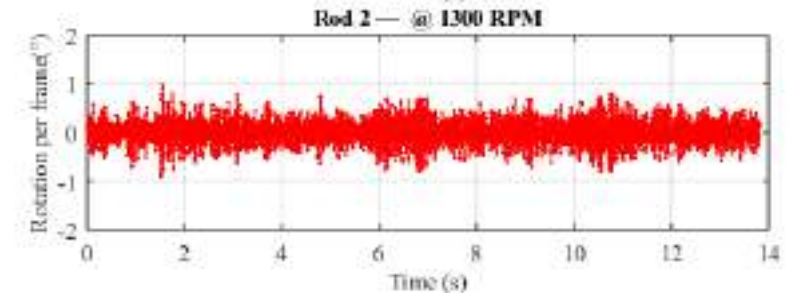
Two rods vibrating
(Horizontal)



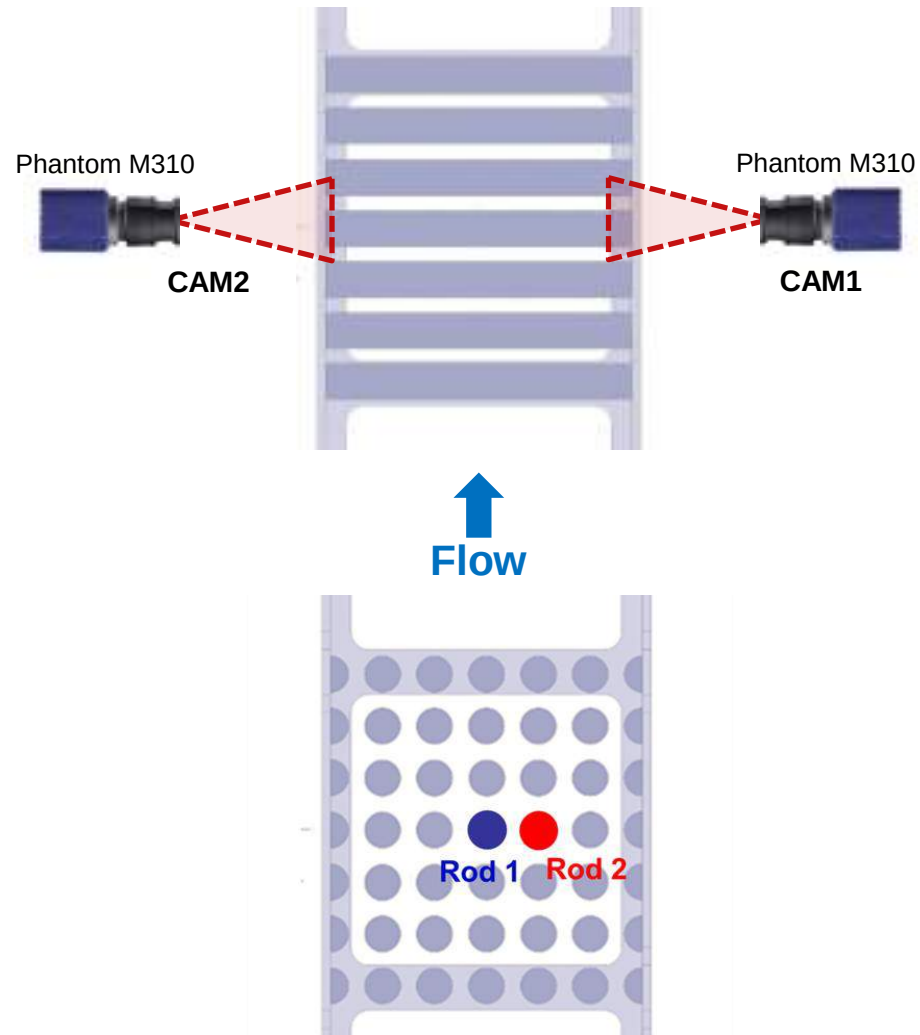
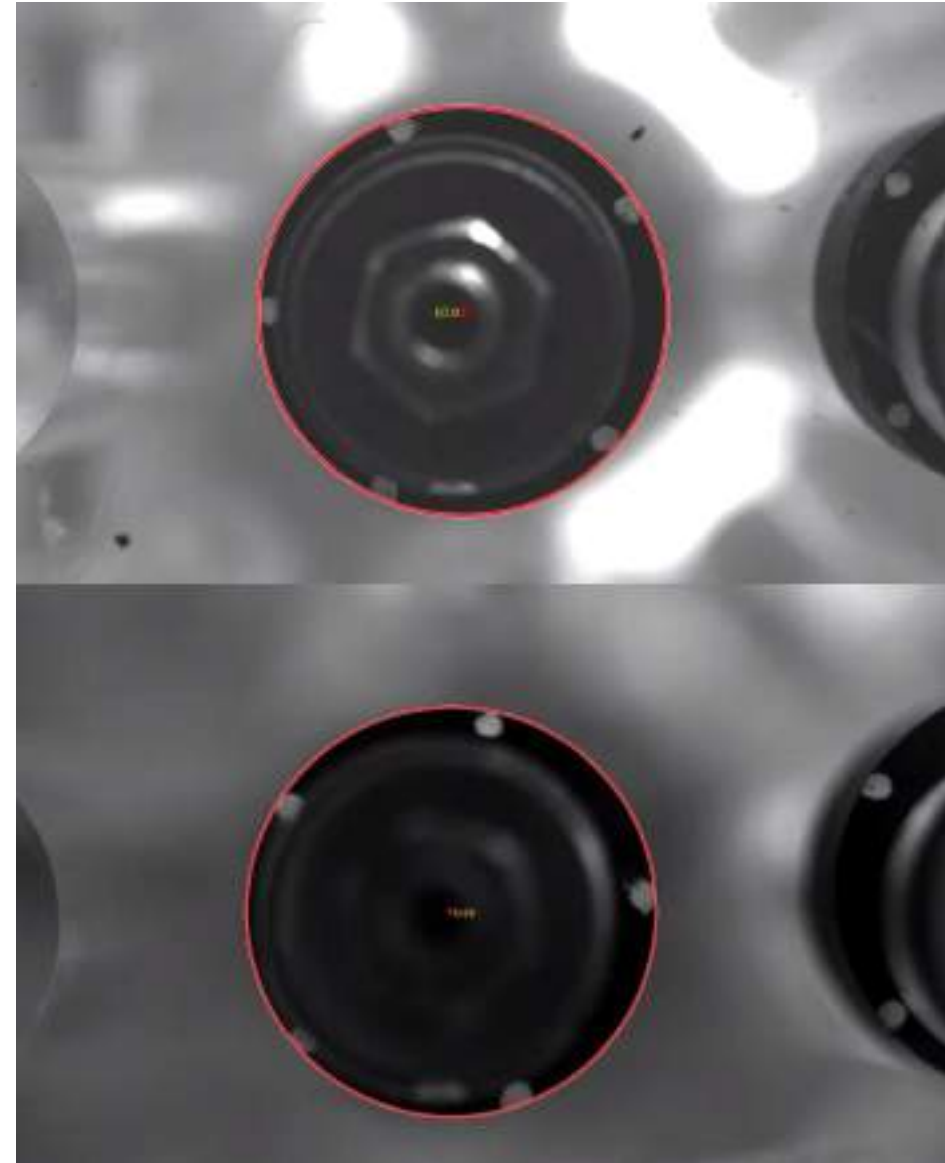
Motion Comparison — $u_b = 0.6896$ m/s



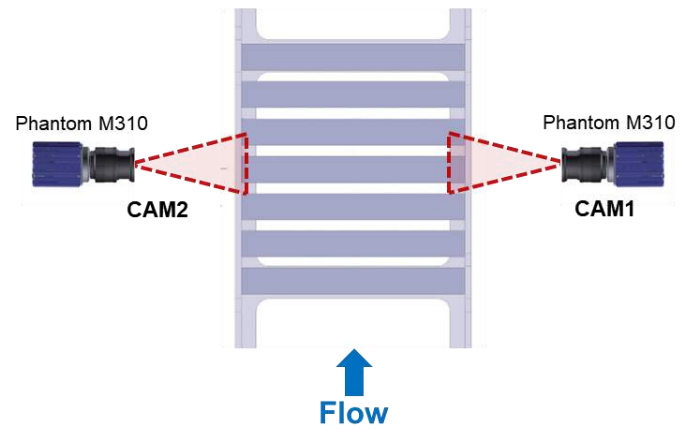
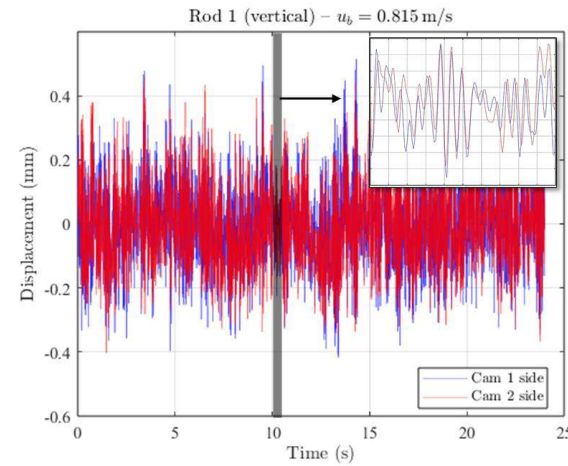
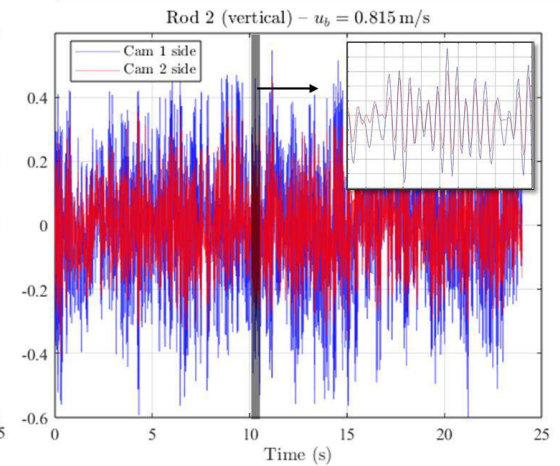
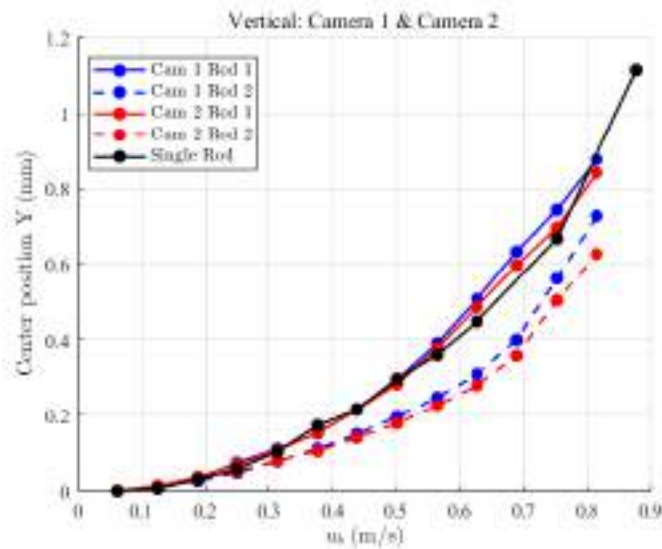
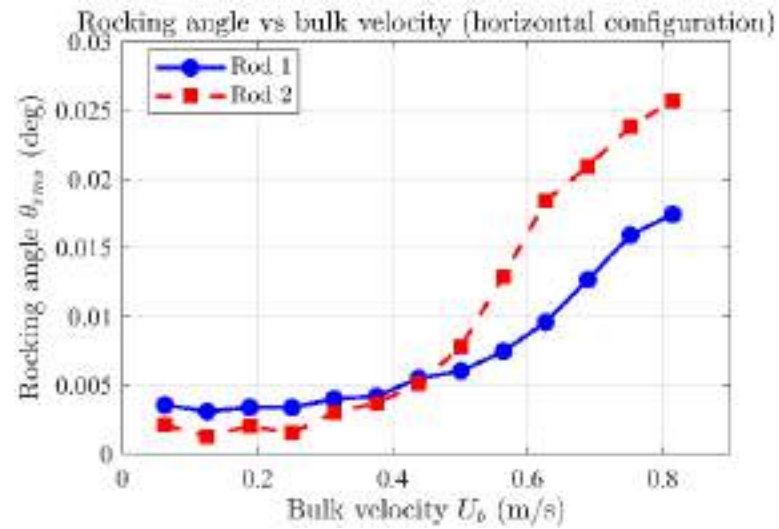
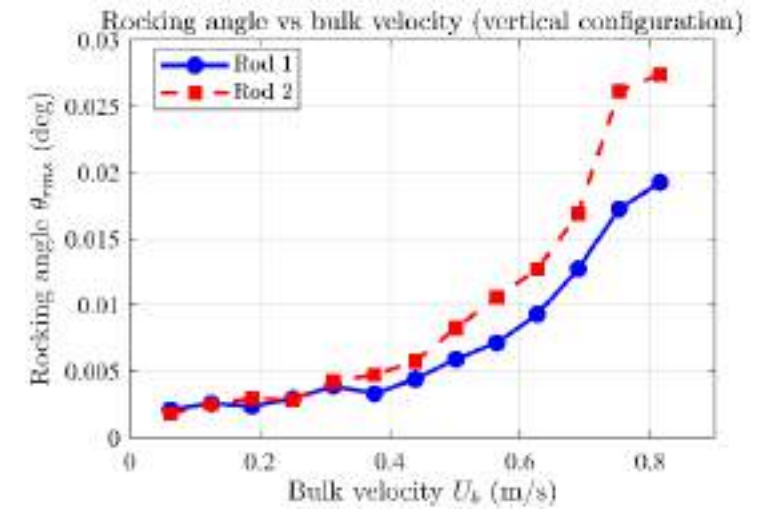
Motion Comparison — $u_b = 0.8150$ m/s



Results: Rocking Effect



Results: Rocking Effect



Achievements

- GOKSTAD facility
- Flow field measurements with fixed rods
 - different Reynolds,
 - different places,
 - inlet flow included
- Vibrating rods
- Vibration measurements – high-speed camera
 - Single rod movement,
 - Two rods in vertical and horizontal
 - Rocking effect
 - Rotation study
- Data collected and arranged for validation.

Challenges faced

- The loop will leak if the pressure is high, the pump will cavitate if the pressure is too high, so know your facilities operational range.
- When interesting 'physics' pop-up, check the integrity of your springs.
- Obtaining developed inlet condition: 11D was not sufficient to have symmetric profile. The solution for the future would be to use flow straightener placed.
- Delivering laser sheet through the plexi rods. Light refractions through the laser were very strong leading nonuniform illumination of the particles. POD based background subtraction worked very well retrieving sufficient particle images/signals
- Usage of the accelerometer are quite challenging inside the water. Sealing or selection of the components are to be updated in the future.
- Perspective distortion has an big effect on the visibility of the channel at the midspan planes. Using telecentric lens successfully solved the problem.

The way forward

We have built a beautiful facility

- Time-resolved PIV together with the vibration measurements
- More rods to vibrate
- Force measurements
- Pressure tap measurements on the fixed rods
- Better inlet control and better characterization. Cross-flow planes. Maybe at the outlet, too?
- Different P/D ratios
- Let one of the rod side “free”, representing the CR in cross-flow.



High-resolution GOKSTAD computations at VCU

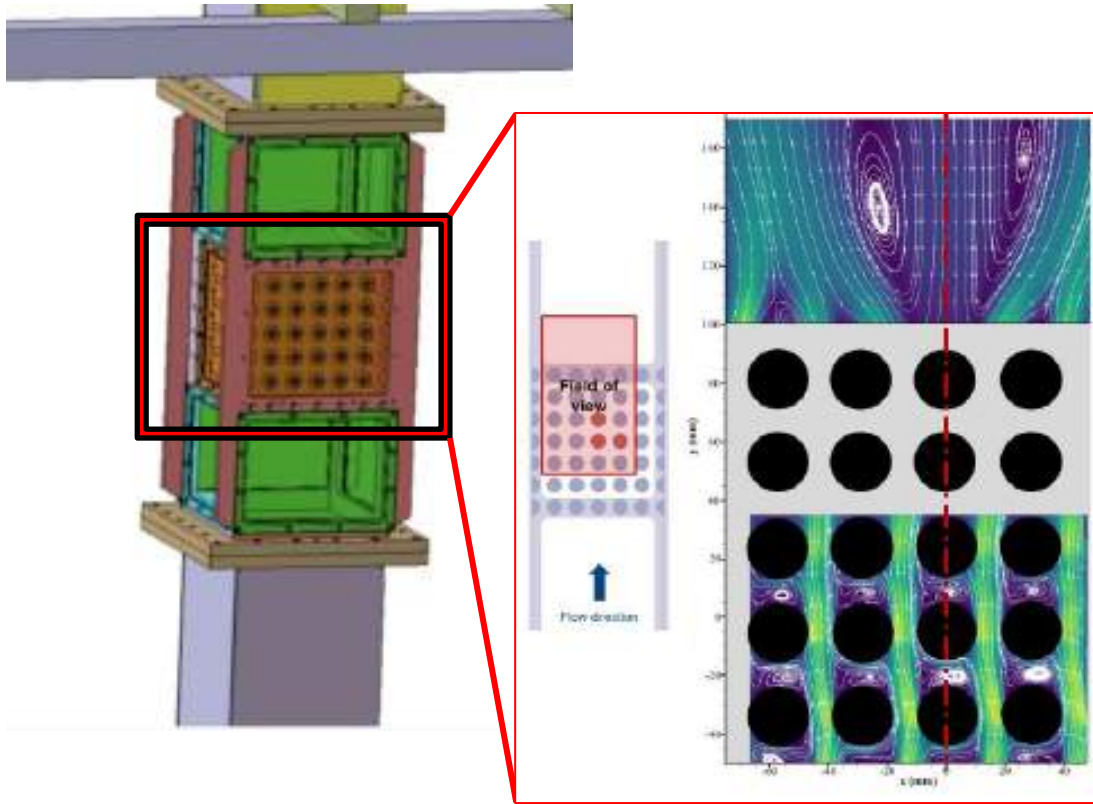
Reduced Domain and Full Experimental Domain Studies

Trevor Franklin and Lane Carasik, Ph.D.

Fluids in Advanced Systems and Technology (FAST) Research Group, Virginia Commonwealth University

Email: lbcarasik@vcu.edu – Carasik Website: <https://eqr.vcu.edu/directory/lane.carasik/> & Google Scholar: [Link](#)

Motivation – Simulating the GOKSTAD experiment



Schematic and velocity measurements of the GOKSTAD experiment¹

- The VKI GOKSTAD facility (7 by 7 tube bundle) involved both flow and displacement measurements.
 - Time-resolved particle image velocimetry (TR-PIV) - flow field (mean, fluctuating components, Reynolds stresses)
 - Strain gages (forces) and accelerometers (displacements)

Motivation – Simulating the GOKSTAD experiment

- The VKI GOKSTAD facility (7 by 7 tube bundle) involved both flow and displacement

VCU's GO-VIKING Efforts:

High-resolution Large Eddy Simulations and if possible, Direct Numerical Simulations, of IV - flow GOKSTAD experiments using the CFD tool, Nek5000/NekRS to generate relevant data sets.

stresses)

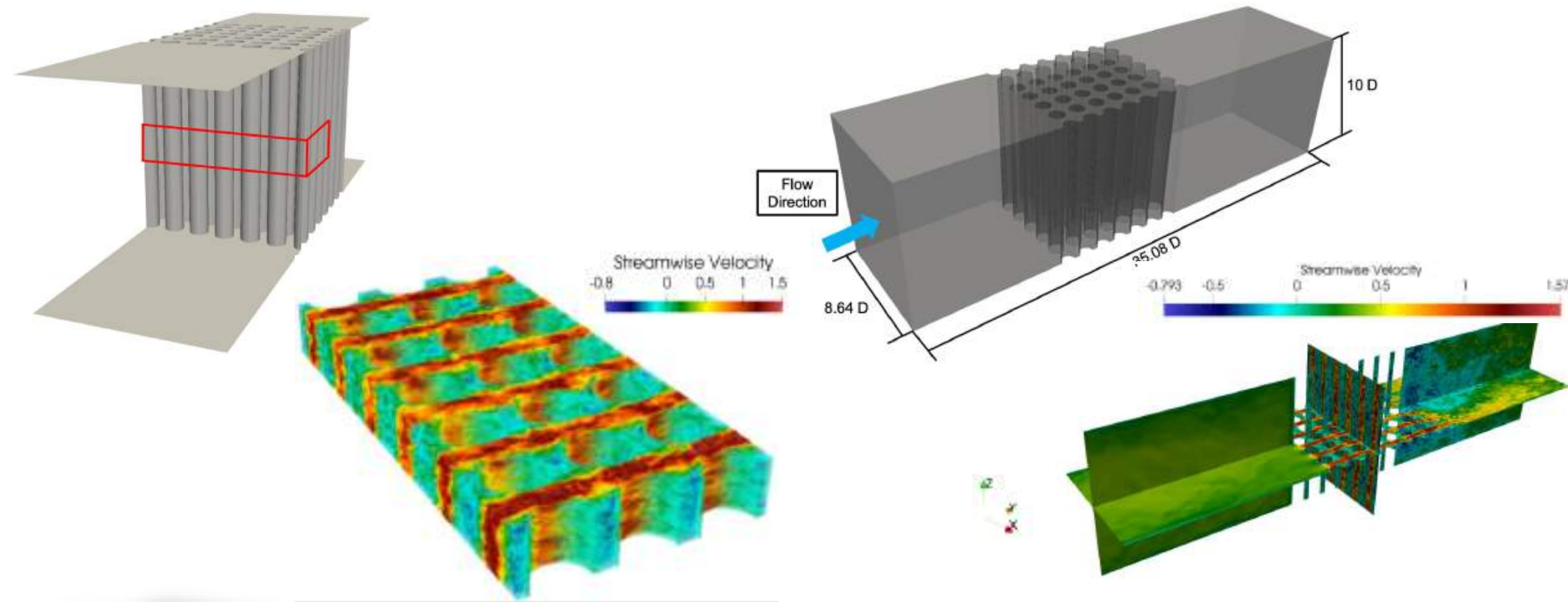
- Strain gages (forces) and accelerometers (displacements)

Schematic and velocity measurements of the GOKSTAD experiment¹

¹A. Papukchiev, et al., 2024, <https://hal.science/hal-04664785v1>

What are the two studies at VCU for GO-VIKING?

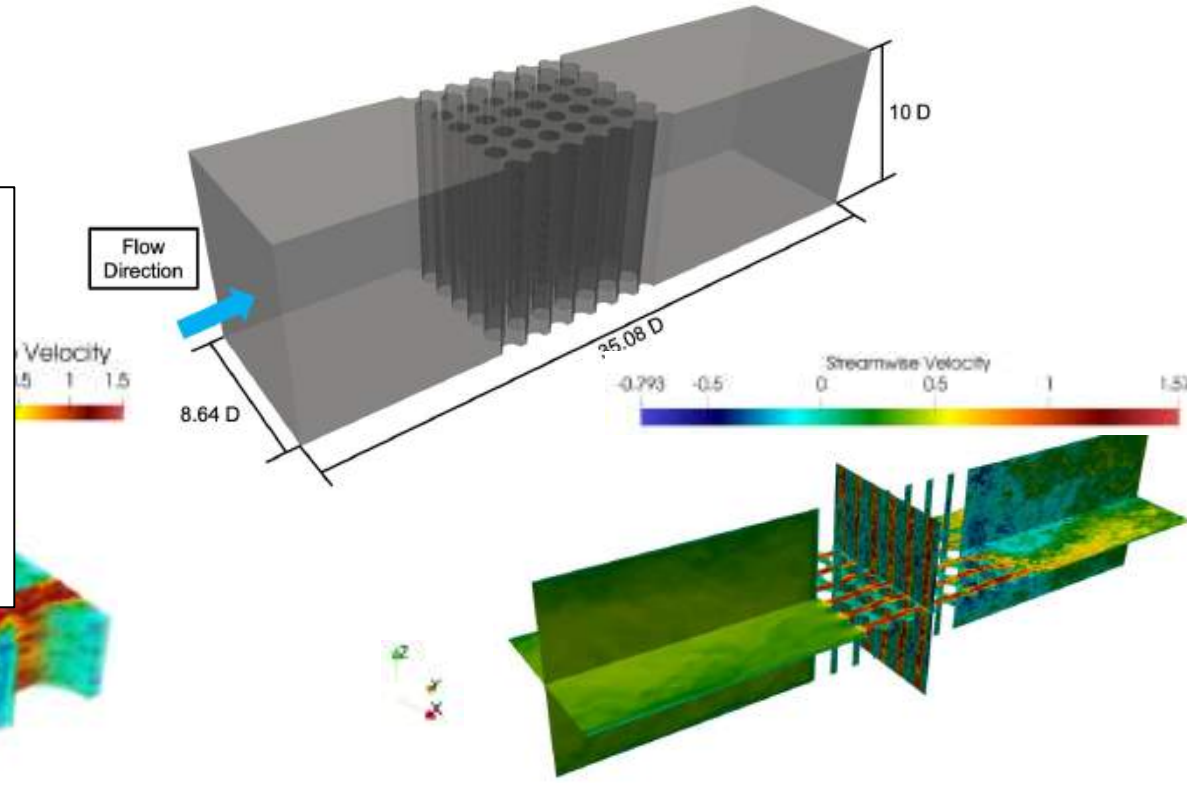
- Reduced GOKSTAD bundle
- Full GOKSTAD Bundle



What are the two studies at VCU for GO-VIKING?

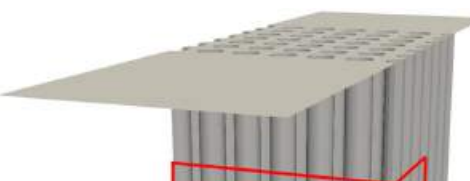
- Reduced GOKSTAD bundle
- Full GOKSTAD Bundle

Determine the minimum bundle sizes needed for future LES and DNS studies to compare to experiments (i.e. lift and drag force).

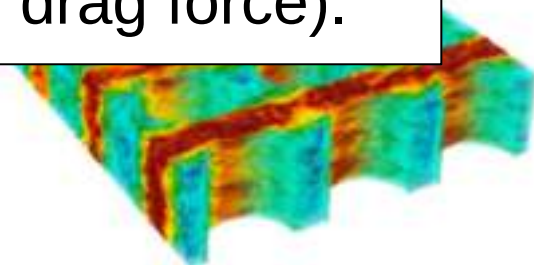
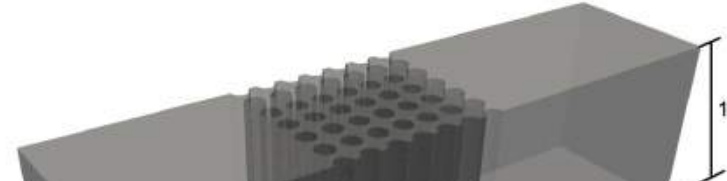


What are the two studies at VCU for GO-VIKING?

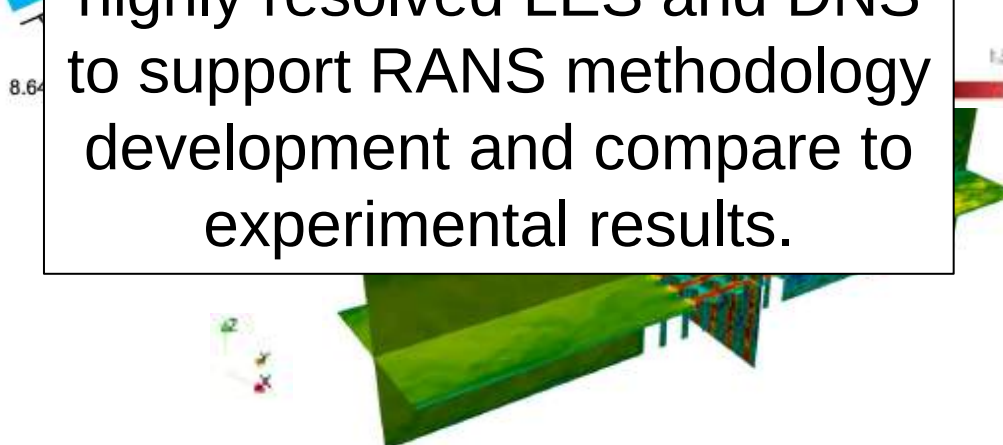
- Reduced GOKSTAD bundle
- Full GOKSTAD Bundle



Determine the minimum bundle sizes needed for future LES and DNS studies to compare to experiments (i.e. lift and drag force).

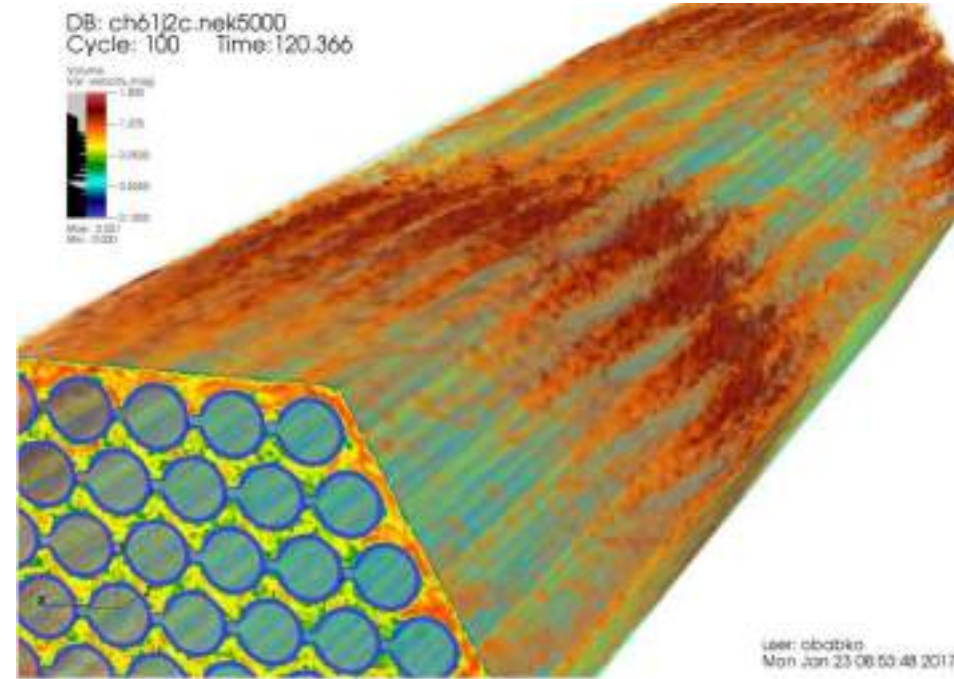


Run full bundle to generate highly resolved LES and DNS to support RANS methodology development and compare to experimental results.



Methodology – Nek5000/NekRS CFD code

- Developed at Argonne National Laboratory¹
- Open source CFD code using the spectral element method
- Solves the unsteady Navier-Stokes equations with either DNS, LES, or RANS turbulence modeling
- Highly scalable for high performance computing systems:
 - CPU version: Nek5000
 - GPU & CPU version: NekRS



Instantaneous axial velocity in 61-Pin wire-wrapped SFR fuel bundle at $Re=15,000^2$

Methodology – Governing equations for both studies

- The non-dimensional Continuity and Navier-Stokes equations were used for Large Eddy Simulations.

Nondimensional Continuity Equation:

$$\nabla^* \cdot \mathbf{u}^* = 0$$

Nondimensional Navier-Stokes Equations:

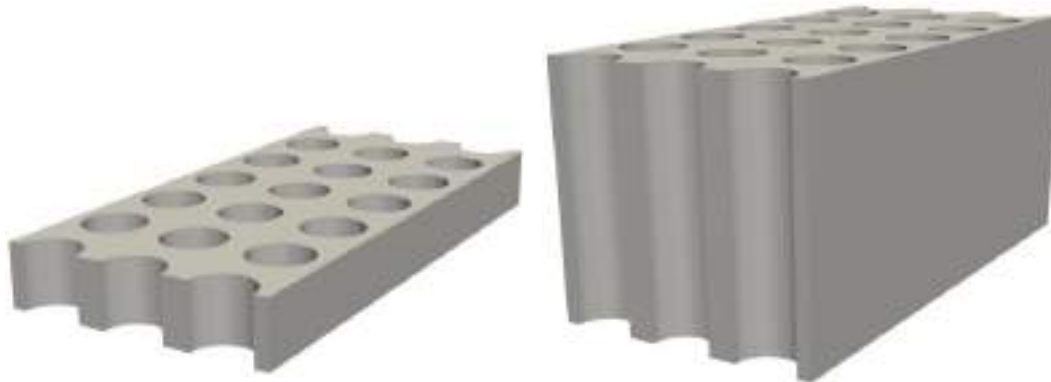
$$\frac{\partial \mathbf{u}^*}{\partial t^*} + \mathbf{u}^* \cdot \nabla^* \mathbf{u}^* = -\nabla^* p^* + \frac{1}{Re} \nabla^{*2} \mathbf{u}^*$$

- Explicit low-pass filter used to remove the smallest unresolved scales (eddies)
- Constant thermophysical fluid properties

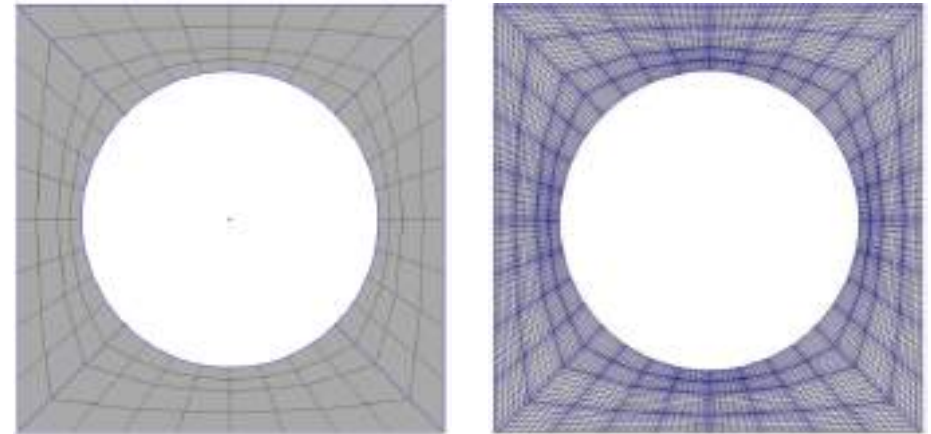
Methodology – Reduced Bundle – Geometry and Mesh

Case	Gap Area [nd]	Mean Vol. Flow Rate [nd]	Hydraulic Diameter [m]	Reynolds Number	Elements	Grid points (9 th order)	Ranks
1 Dia	3.72	3.72	0.01686	6650	28,800	2.88e7	600
2 Dia	7.43	7.43	0.01686	6650	57,600	5.76e7	1,280
3 Dia	11.14	11.14	0.01686	6650	86,400	8.64e7	1,800
4 Dia	14.86	14.86	0.01686	6650	115,200	1.152e8	2,560

All cases run for ~6 days in compute time

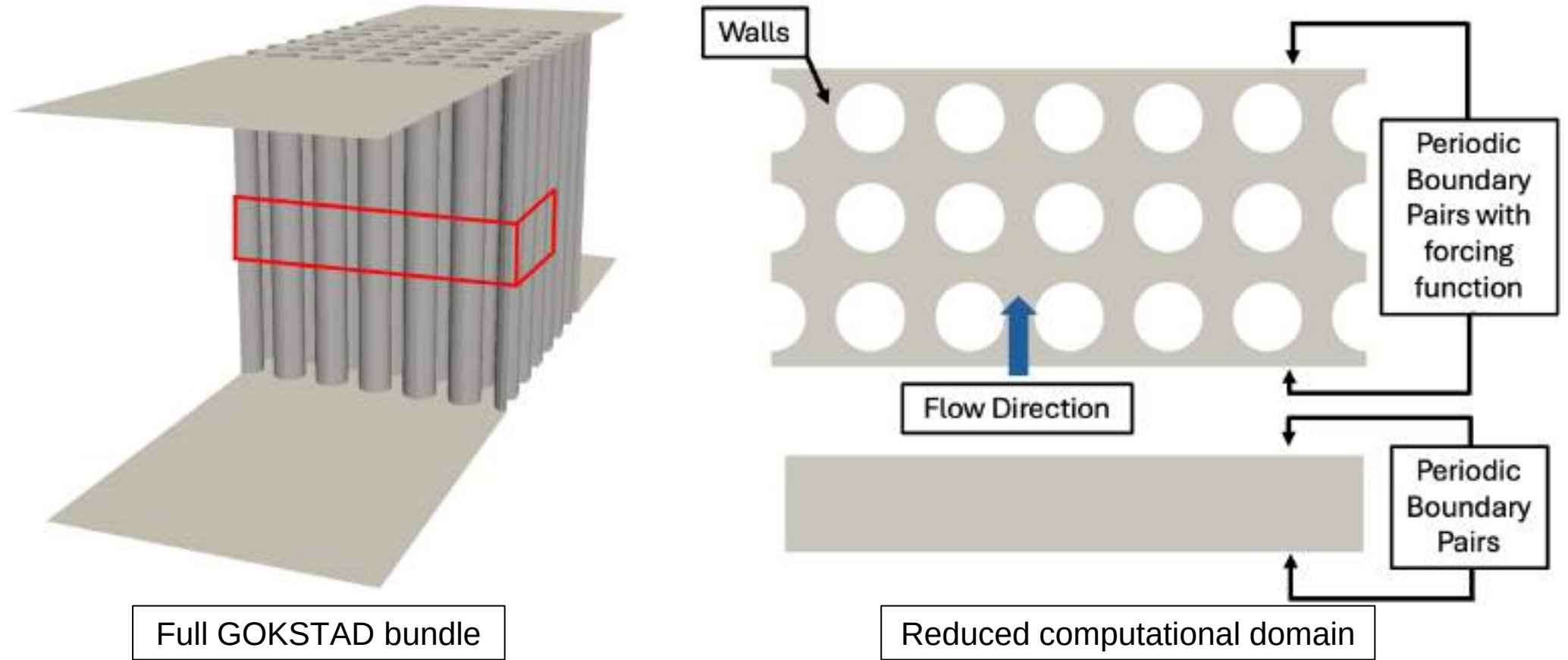


1D (left) and 4D (right) spanwise domains



GMSH (left) and Nek5000 (right) mesh

Methodology – Reduced Bundle – Boundary Conditions



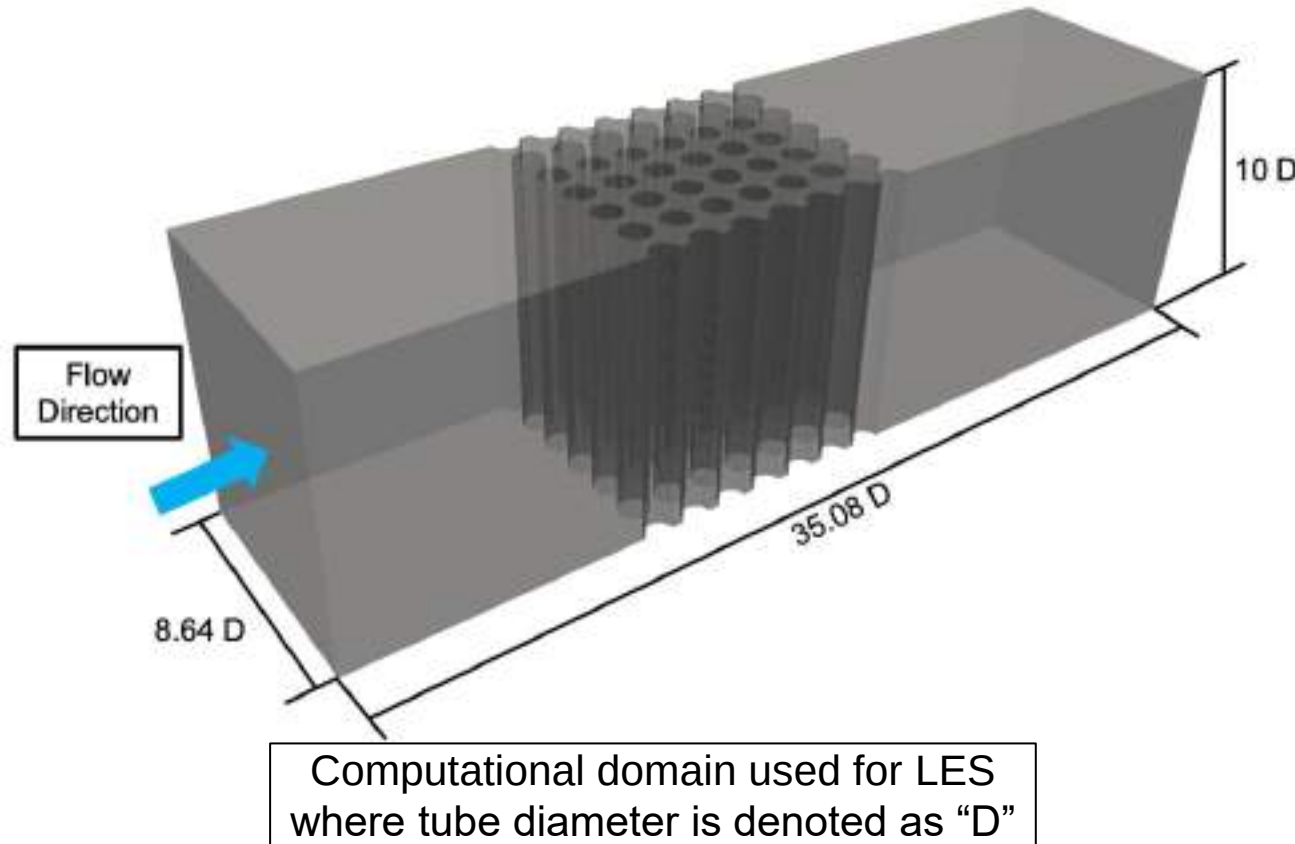
Methodology – Reduced Bundle – Mesh Study

- The mean drag force for each case was compared.

Case	Mean	% Difference
1 Diameter		
5 th Order	0.262	9.30
7 th Order	0.259	8.51
9 th Order	0.238	-
2 Diameter		
5 th Order	0.568	13.53
7 th Order	0.539	8.32
9 th Order	0.496	-

Case	Mean	% Difference
3 Diameter		
5 th Order	0.902	9.903
7 th Order	0.850	4.02
9 th Order	0.817	-
4 Diameter		
5 th Order	1.214	4.715
7 th Order	1.159	0.048
9 th Order	1.158	-

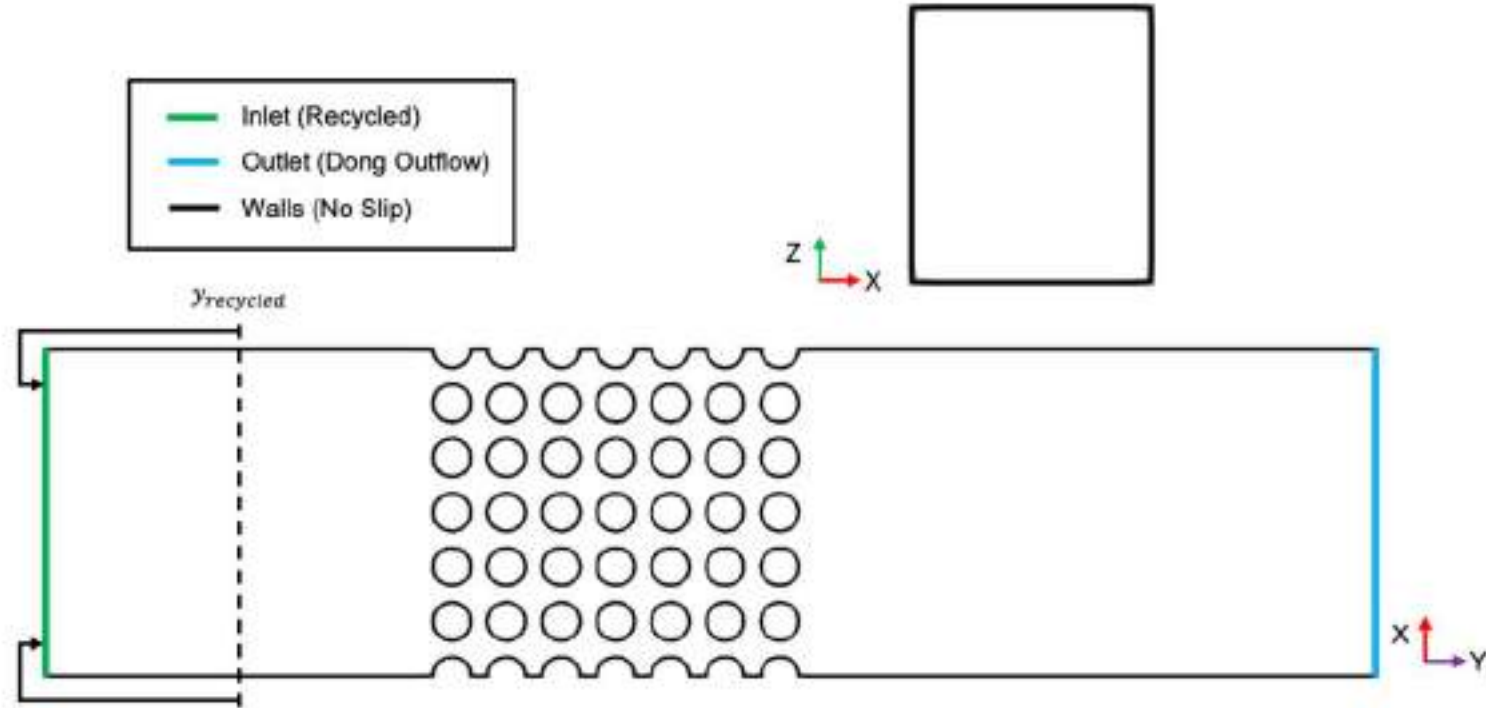
Methodology – Full Bundle – Geometry



- 5 rows of 7 full tubes and 2 rows of 7 half tubes.
- $10 D_{\text{tube}}$ inlet and $15 D_{\text{tube}}$ outlet region
- Flow rate: $15 \text{ m}^3/\text{hr}$
($U_b = 0.125 \text{ m/s}$)

Methodology – Full Bundle – Boundary Conditions

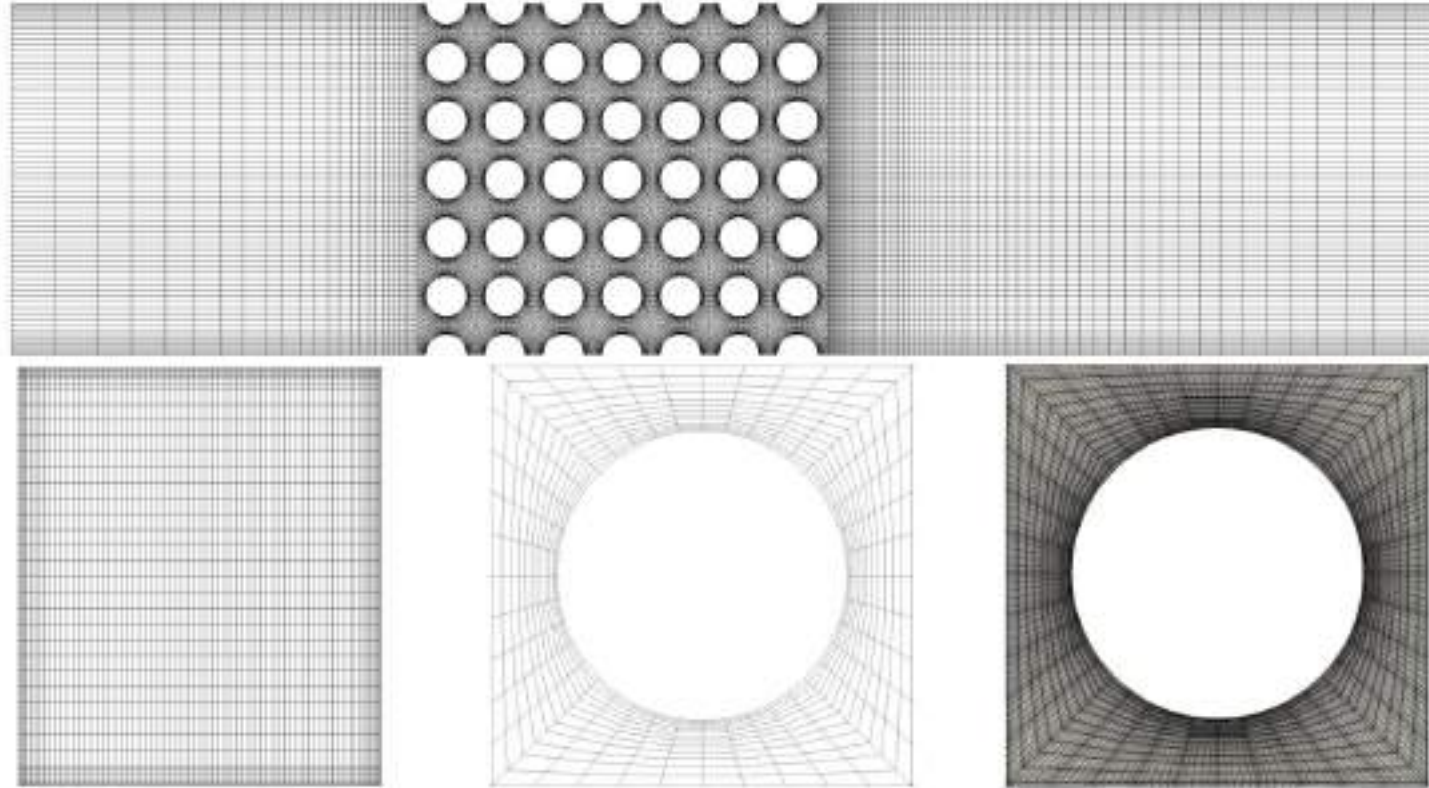
- Recycling is applied $5D_{\text{tube}}$ to the inlet boundary
- Ave. velocity of 0.306 is enforced at the inlet
- $Re_{\text{gap}} = 6873.5$
- Dong outflow¹ applied to the outlet boundary
- No slip BC used for all walls



17280 ranks with a wall time of a little ~15 days in compute time

Methodology – Full Bundle – Mesh

- Base mesh is generated in GMSH¹
- Consist of 1.02 Millon fluid elements
- Nek5000 further discretizes using a 7th order polynomial
- Final mesh uses **524 Millon** collocation points



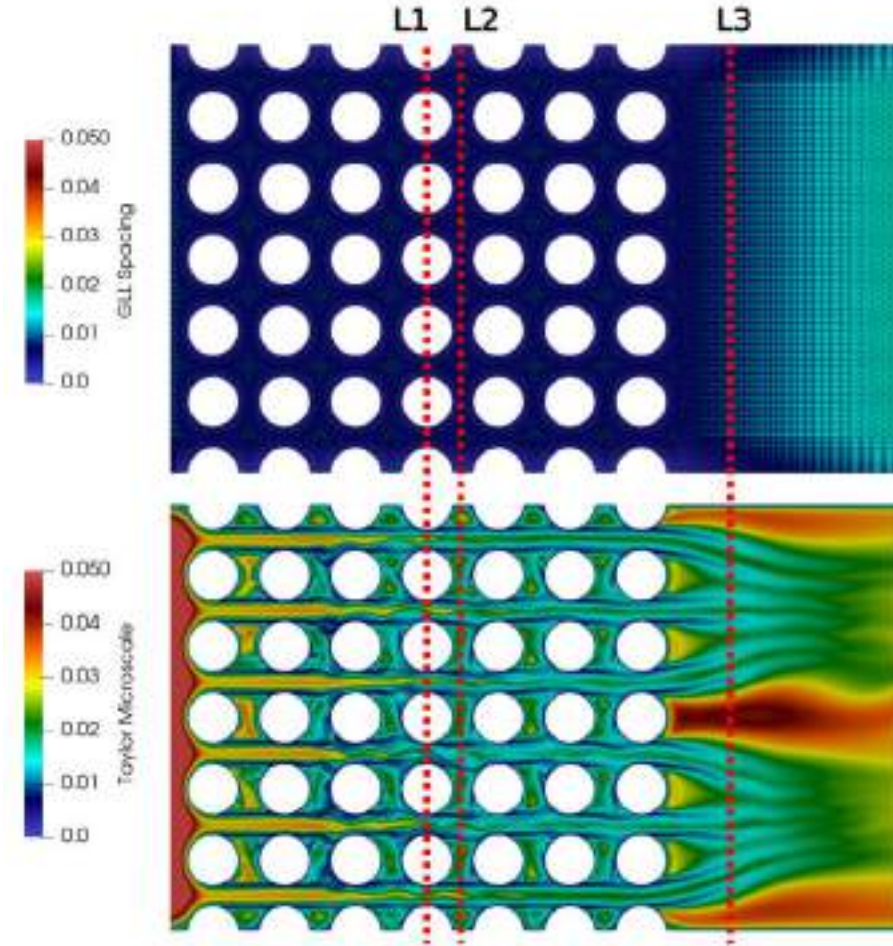
Mesh generated in GMSH along with the spectral element mesh generated by Nek5000

Methodology – Full Bundle – Taylor Microscale Calc.

- Prior to LES, a precursor RANS simulation was used to estimate the Taylor Microscale
- The same boundary conditions used for LES was used for RANS.

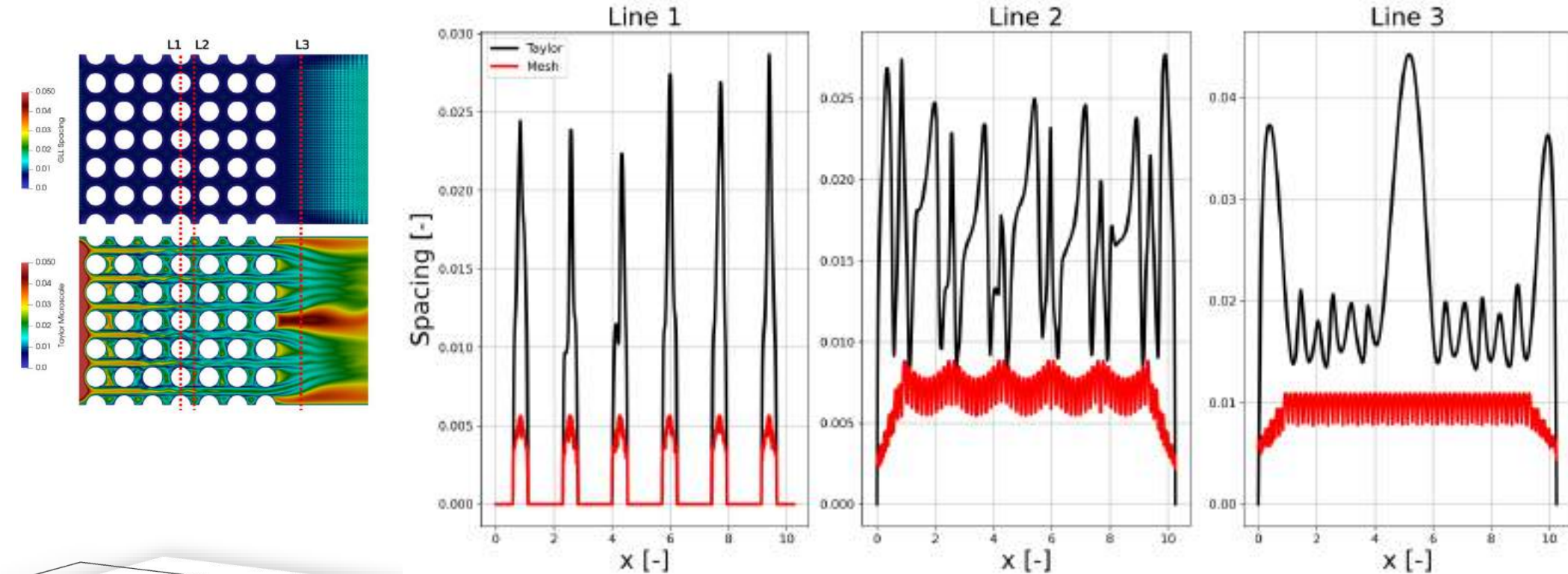
RANS Parameters

Turbulence model	$k - \tau$
Collocation points	128 Million
k_{init}	0.01
τ_{init}	0.2
dt	$1E - 3$
Ave. CFL	0.370
Ave. y^+	0.9



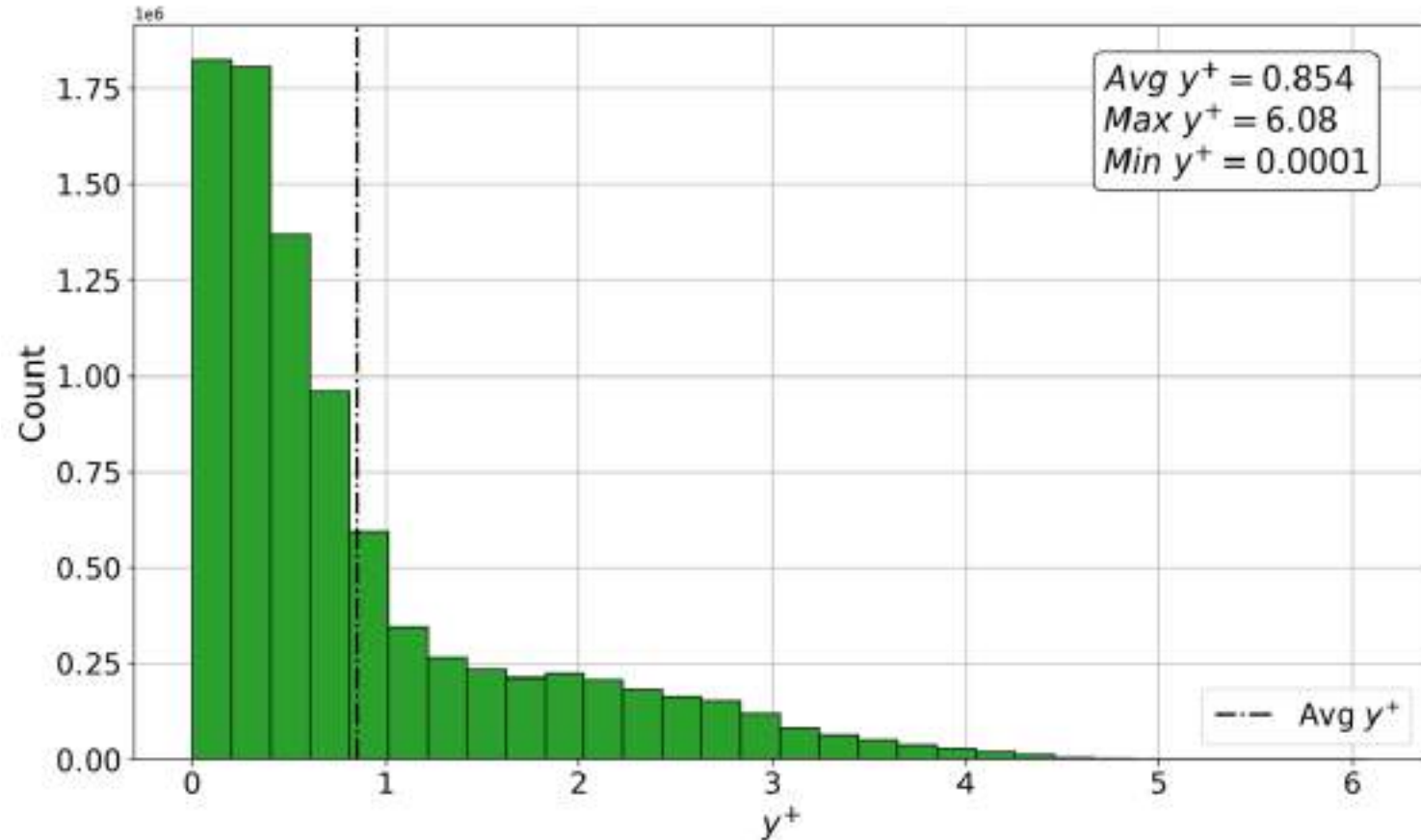
Methodology – Full Bundle – Taylor Microscale Calc.

- The LES mesh is sufficient to resolve the Taylor microscale from GLL spacing $<$ Taylor microscale



Methodology – Full Bundle – LES Wall Distance

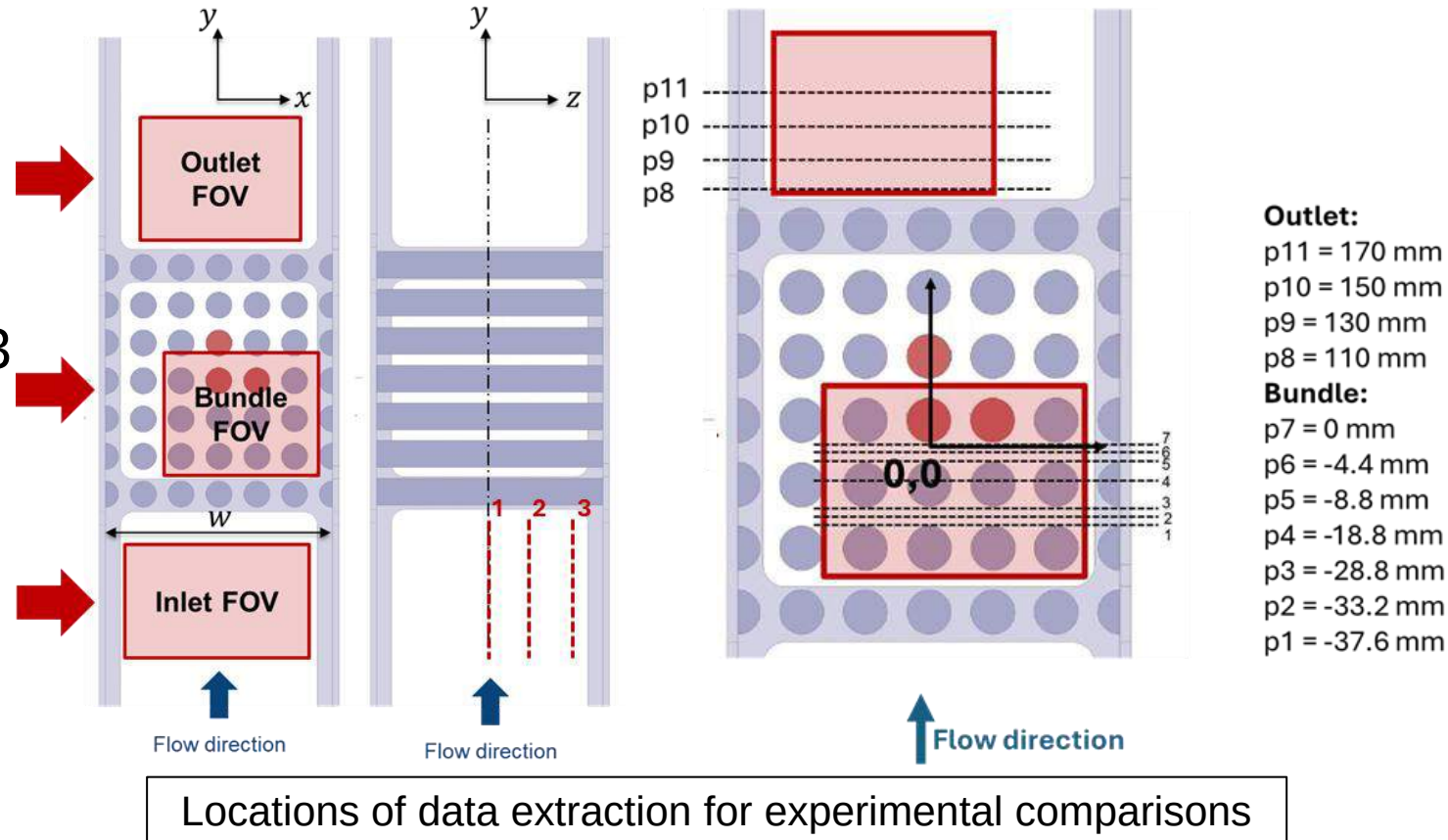
- y^+ was computed for the LES full bundle domain
- 73% of all y^+ values are less than 1 with 87% being less than 2



y^+ Histogram for the full bundle LES domain

Methodology – Full Bundle – Post Processing

- Velocity & TKE Comparisons:
 - Fields (Inlet, Bundle, Outlet)
 - Inlet Profiles at 3 different planes
 - Profiles through bundle and outlet (P1-P11)
- Tube Displacements



Methodology – Full Bundles – Displacement Calc.

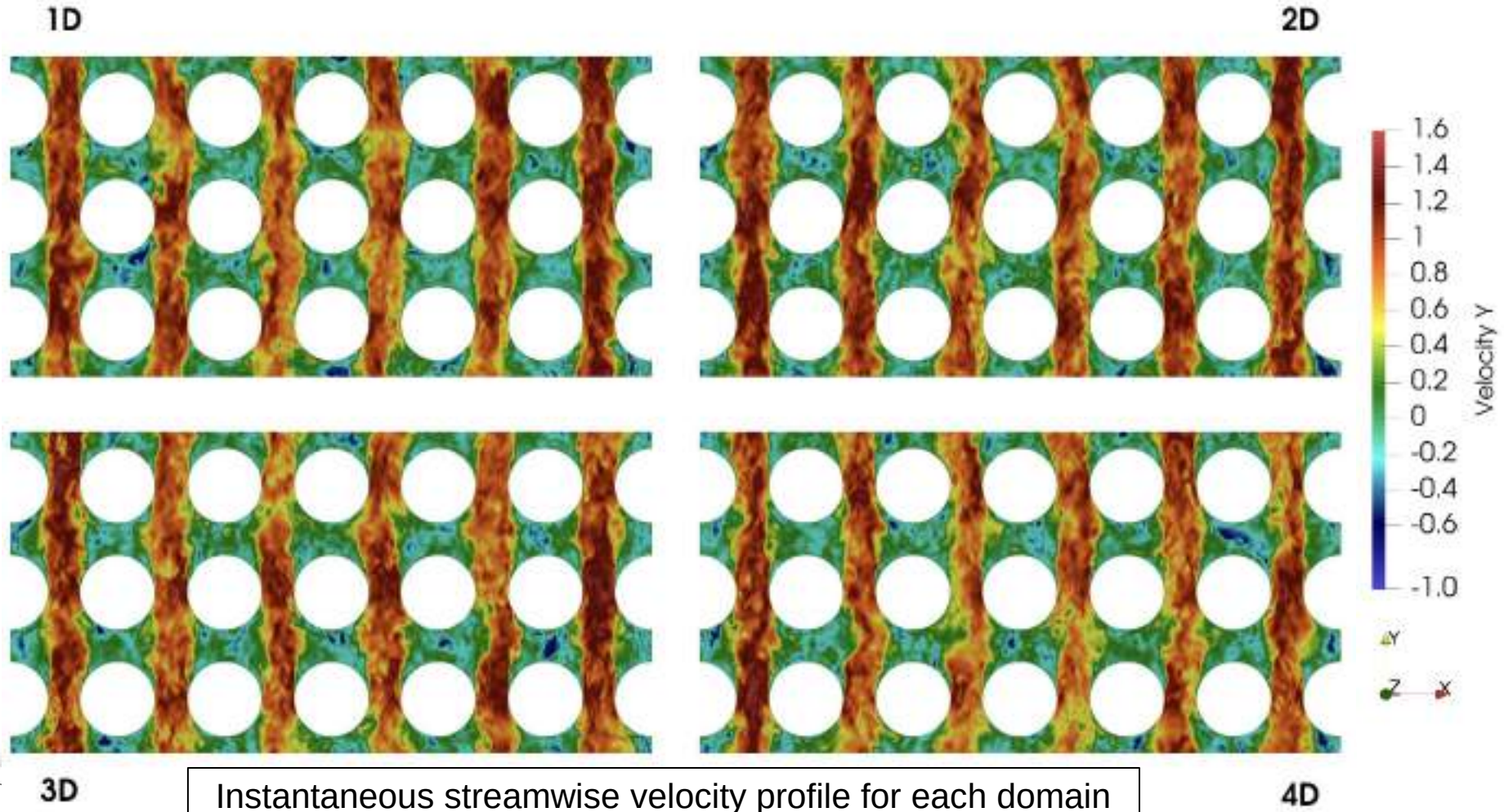
- A one-way FSI approach was used to estimate the response of the center tube using the fluid forces obtained from the LES.

$$m\ddot{x}(t) + c\dot{x}(t) + kx(t) = F_x(t)$$

$$m\ddot{y}(t) + c\dot{y}(t) + ky(t) = F_y(t)$$

- m , c , and k are the Mass, dampening, and stiffness coefficient respectively.
- For this work, $m = 0.1 \text{ kg}$, $c = 1.14 \text{ n} \cdot \text{s}/\text{m}$, $k = 7000 \text{ N}/\text{m}$.

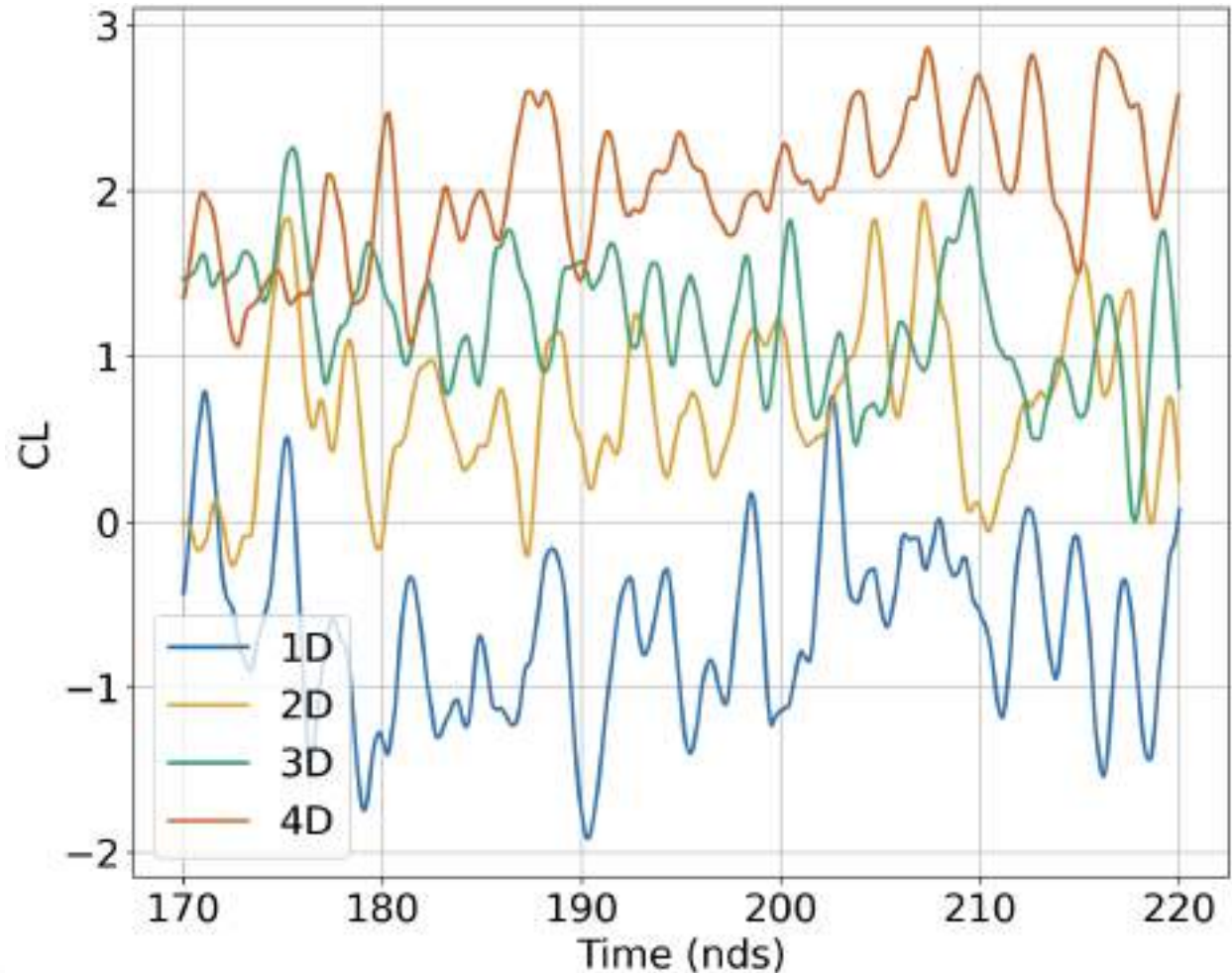
Results – Reduced Bundle – Flow Fields



Results – Reduced Bundle – Force (C_L) Response

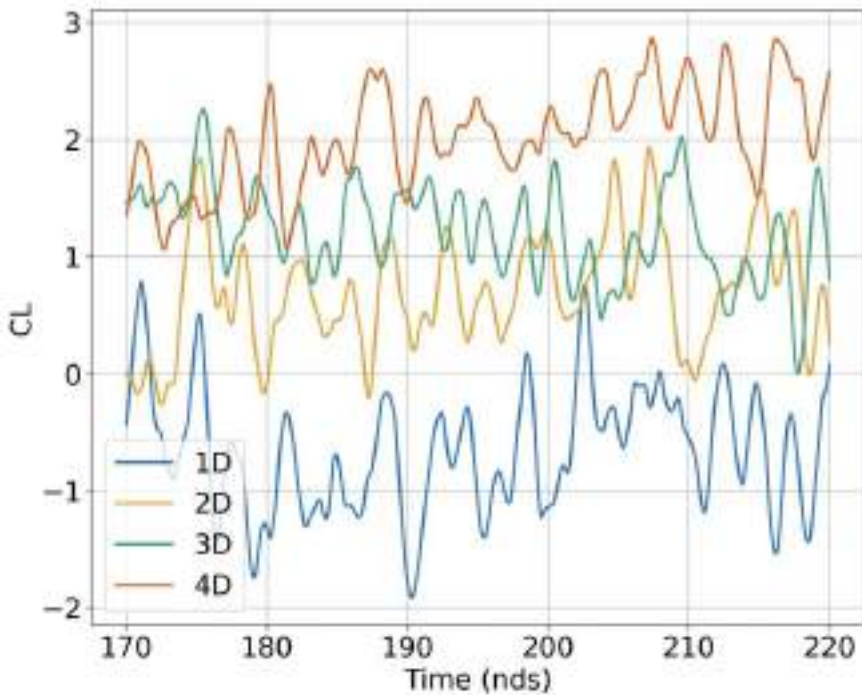
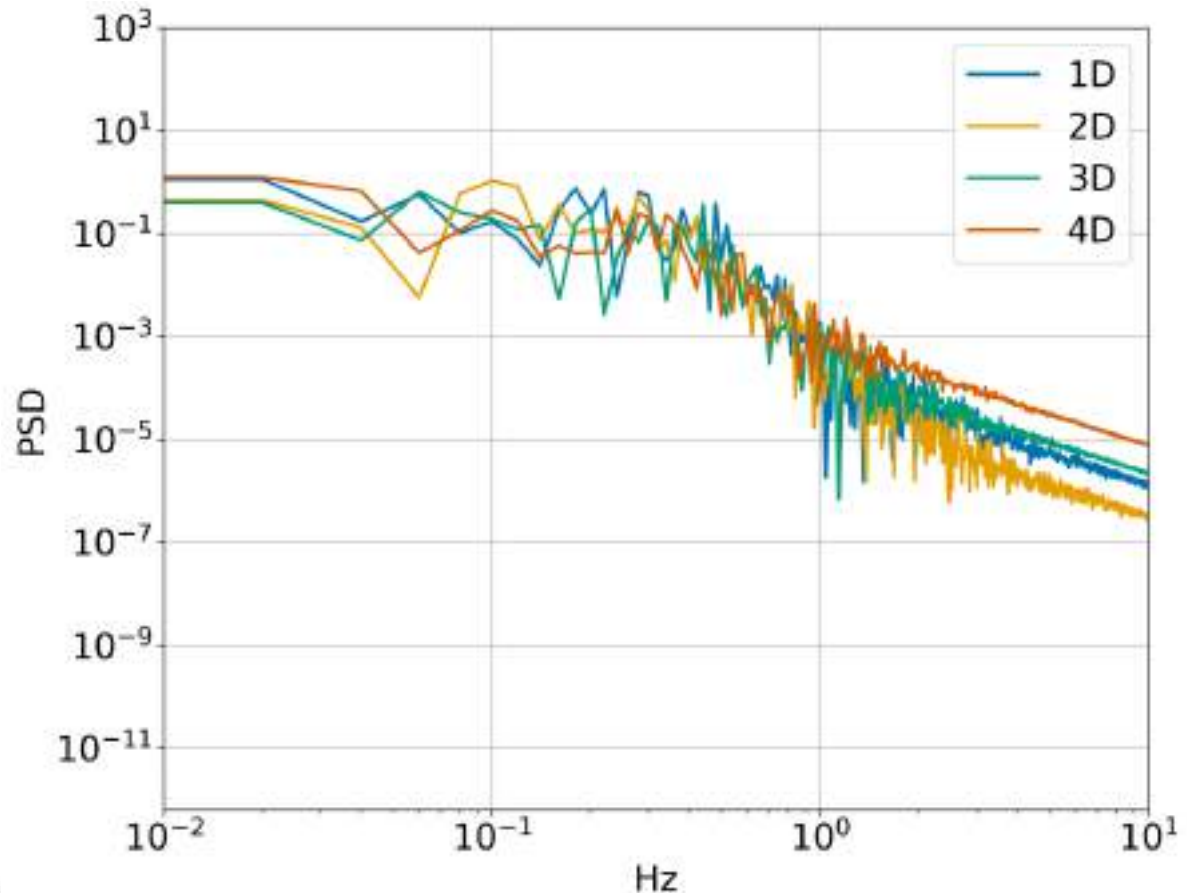
- Lift coefficient is shown for each spanwise height case through run time

Spanwise height	Mean C_L	RMS C_L
1 Diameter	-0.656	0.529
2 Diameter	0.699	0.486
3 Diameter	1.218	0.392
4 Diameter	2.020	0.419



Results – Reduced Bundle – Force Response

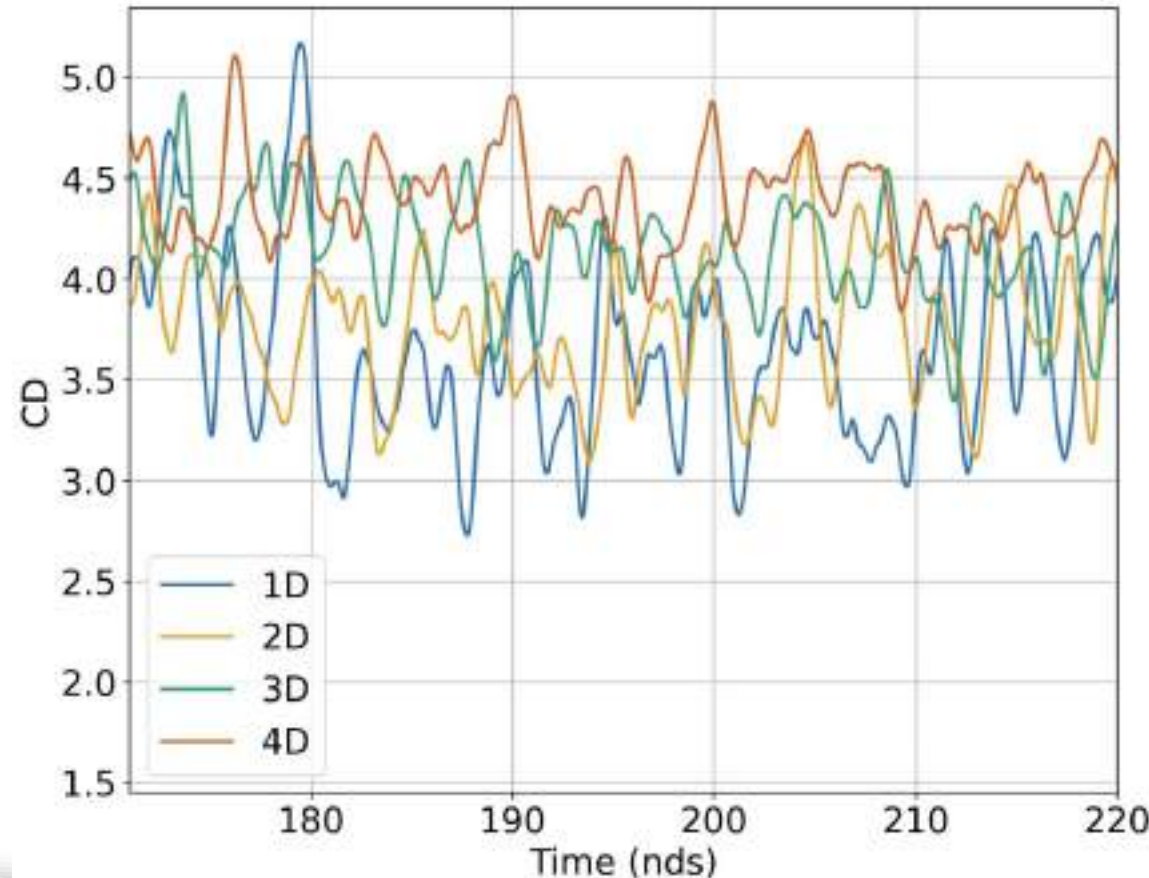
- A power spectral density (PSD) analysis was performed on the lift coefficient.



Results – Reduced Bundle – Force Response

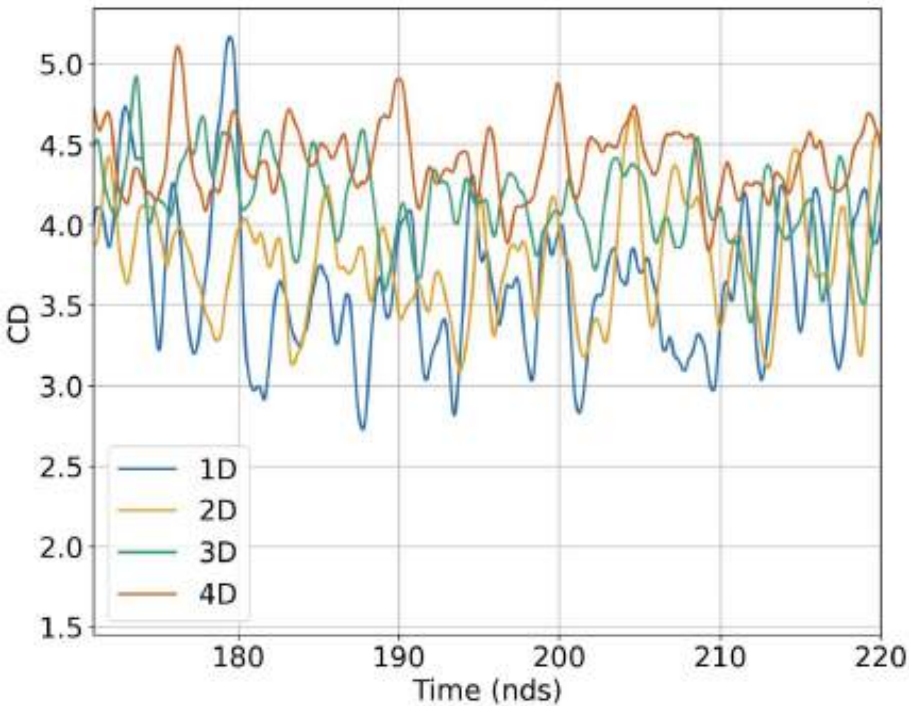
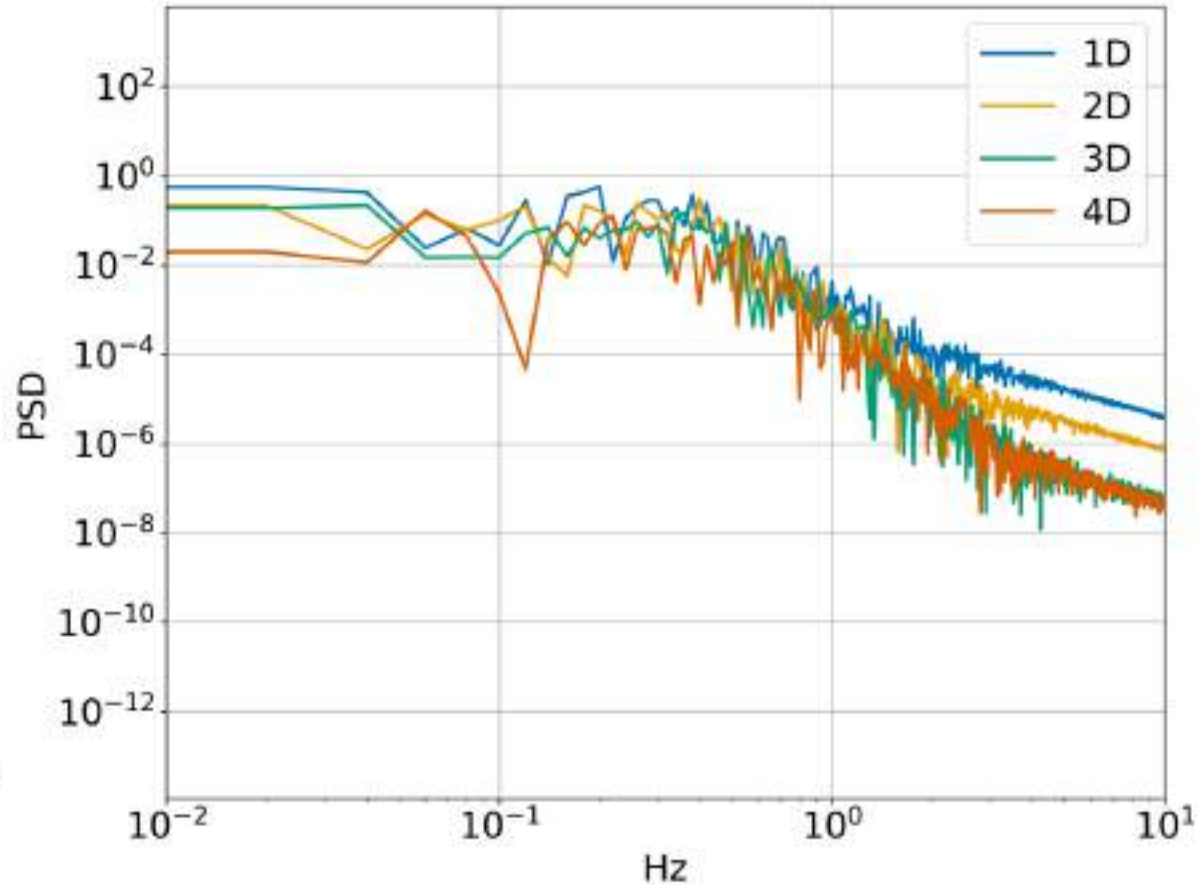
- Drag coefficient is shown for each spanwise height case through run time.

Spanwise height	Mean C_D	RMS C_D
1 Diameter	3.63	0.460
2 Diameter	3.775	0.336
3 Diameter	4.144	0.267
4 Diameter	4.408	0.215



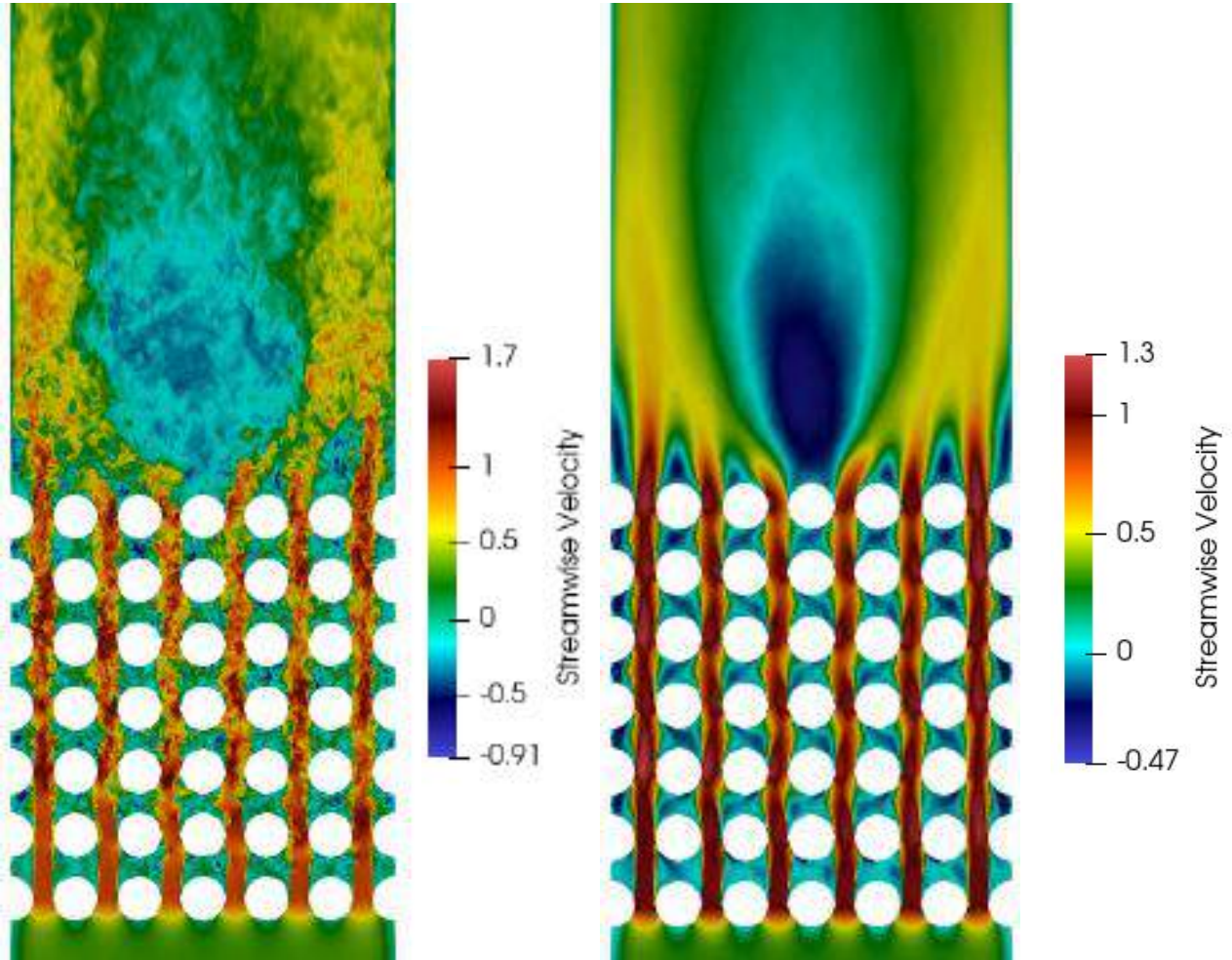
Results – Reduced Bundle – Force Response

- A power spectral density (PSD) analysis was performed on the drag coefficient.



Results – Full Bundle – Flow Fields

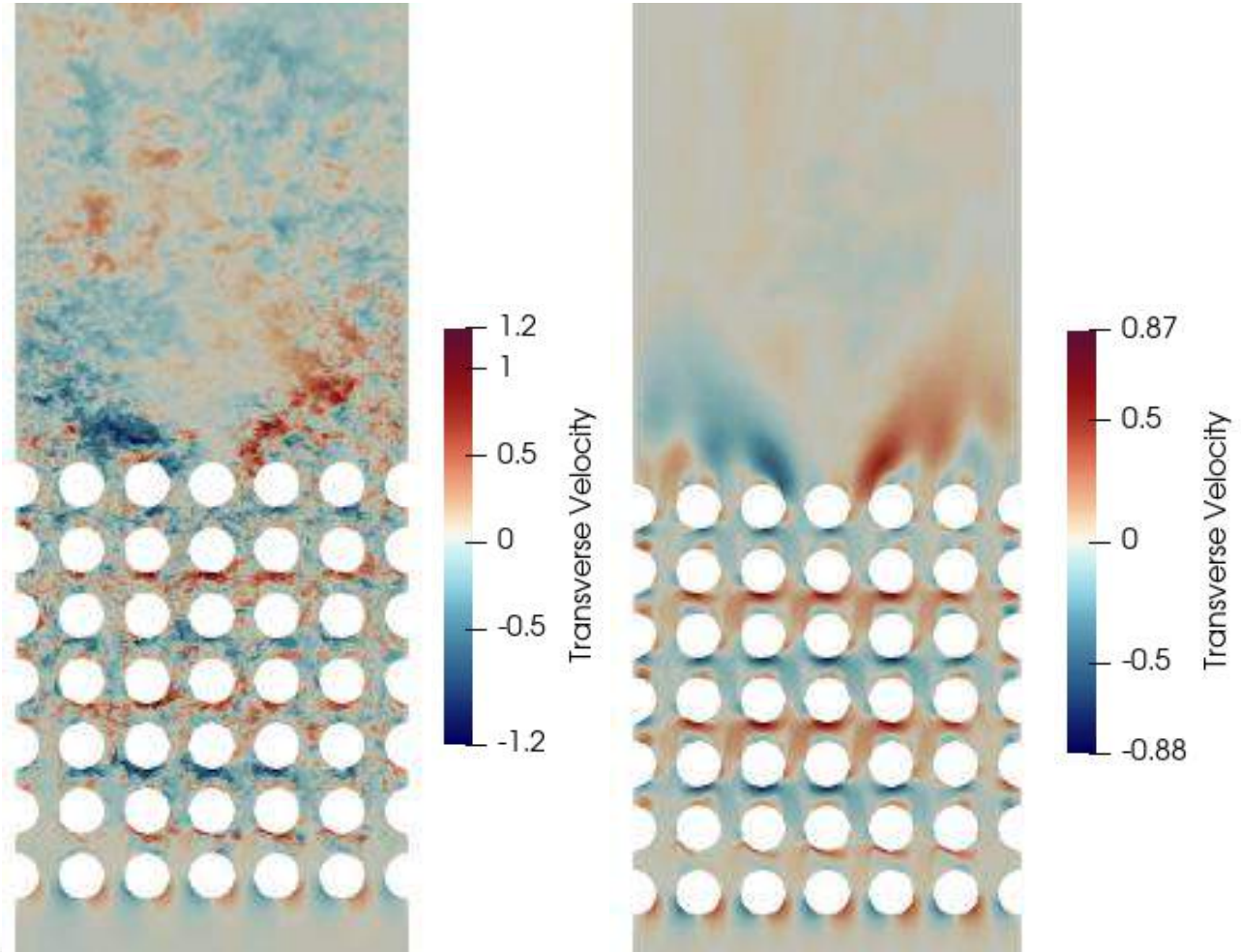
- LES shows high velocity jet like structures form in the tube bundle
- Jet like structures merge towards the walls at the outlet, forming a large recirculation zone in the center.



Instantaneous (left) & ave. (right) streamwise velocity

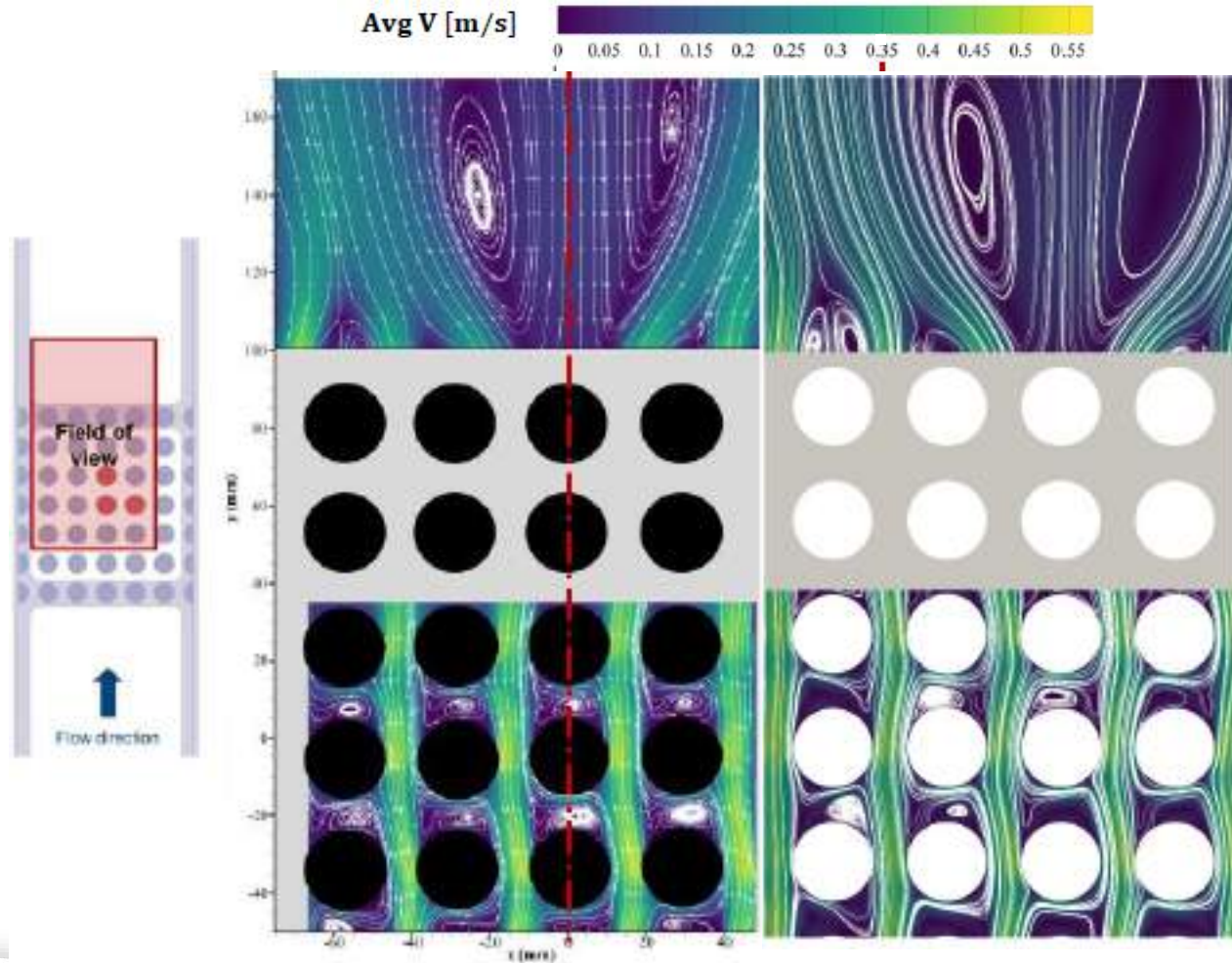
Results – Full Bundle – Flow Fields

- Significant cross flow is observed after the 2nd row of tubes in the tube bundle
- Further cross flow is shown at the outlet going towards the side walls



Results – Full Bundle – Ave. Flow Field Comparison

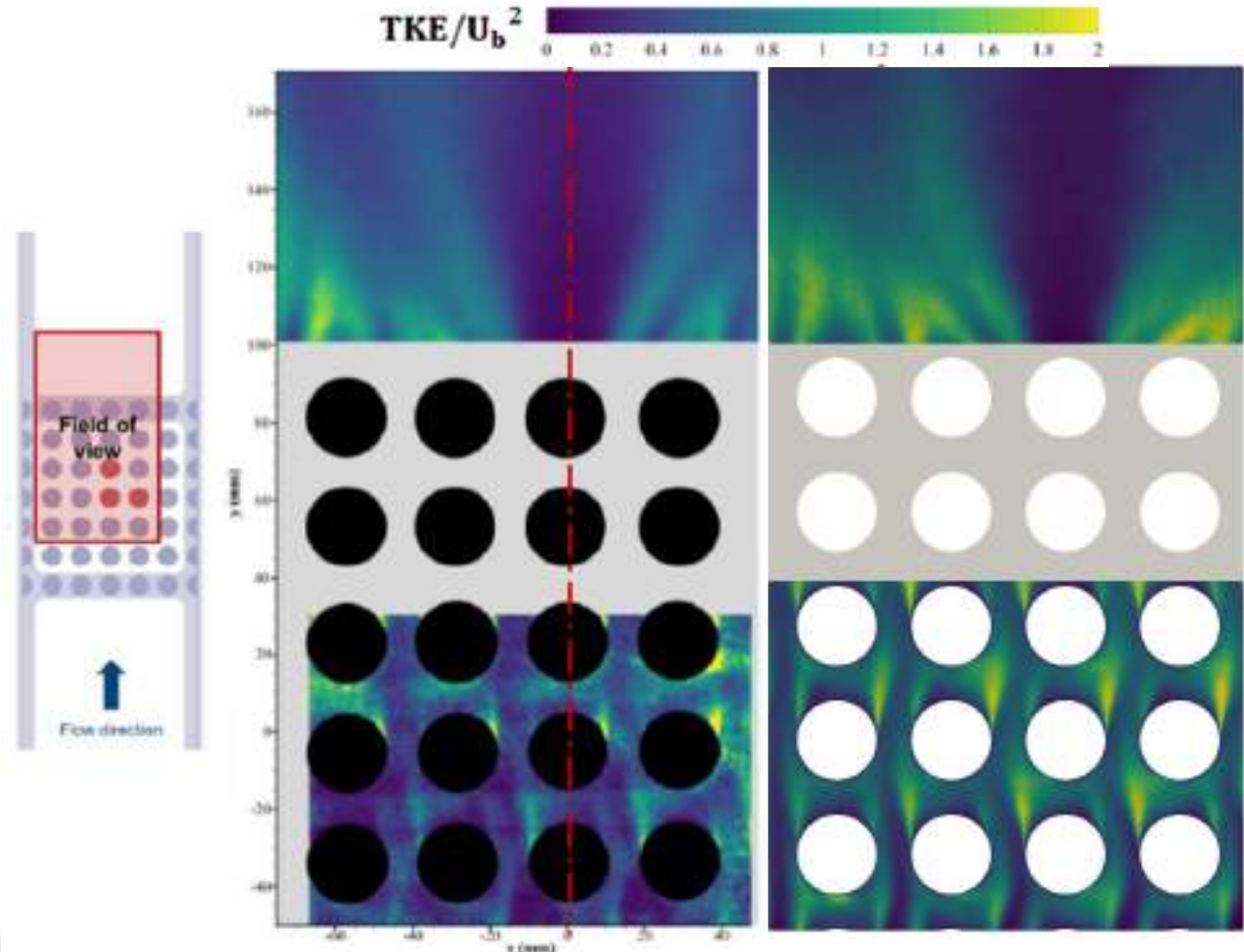
- Measured flow behavior is captured in the simulations.



Average velocity from experiment (left) and LES (right)

Results – Full Bundle – Ave. Flow Field Comparison

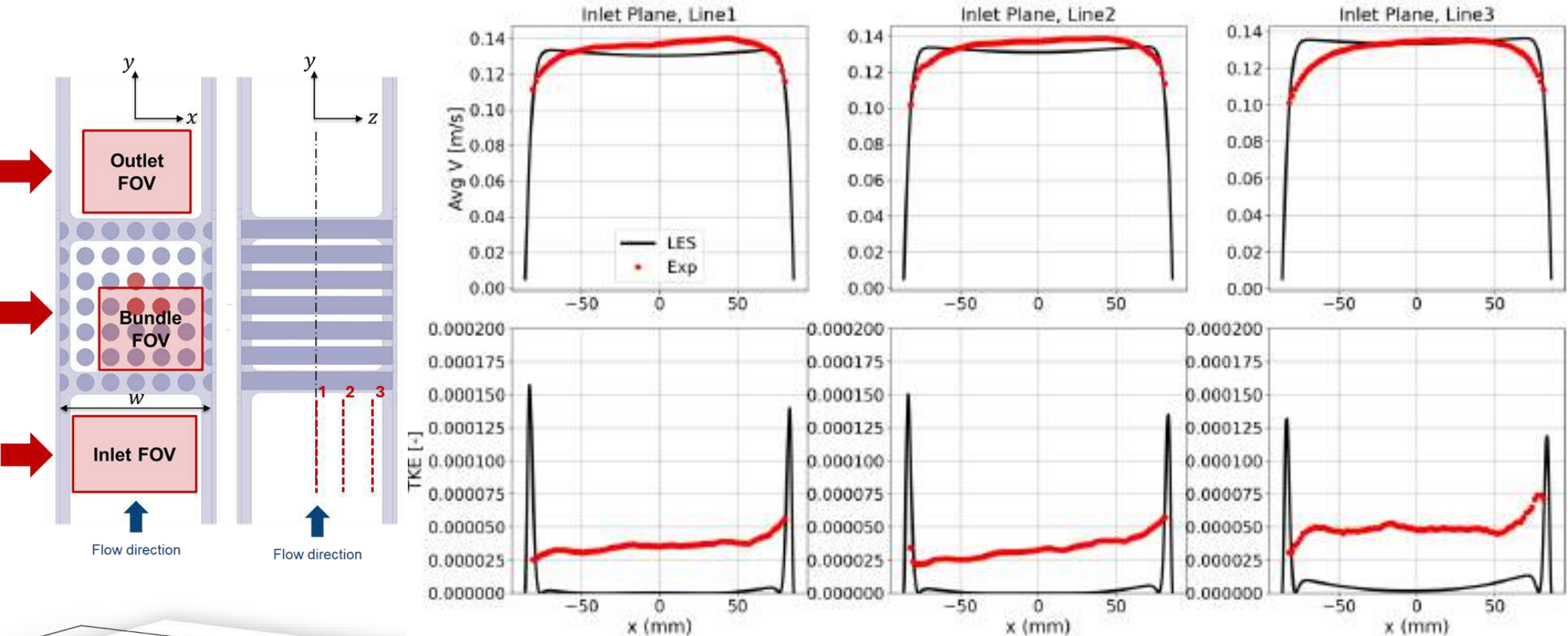
- LES captures the general spatial profile of the experiment, but overpredicts the magnitude compared to the experiment.



Average TKE From experiment (left) and LES (right)

Results – Full Bundle – Inlet Profiles

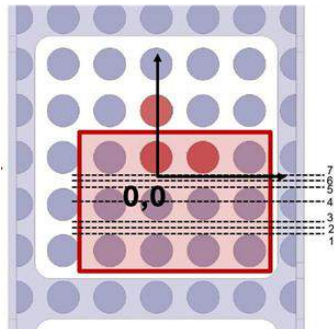
- Minimal TKE from LES indicates more inlet conditioning is needed



Inlet ave. velocity (top) and ave. TKE (bottom) for LES and the experiment

Results – Full Bundle – Bundle Profiles

- The trends and magnitude of average velocity agrees well.
- Slight discrepancy at line 3 and 7 near the tube walls.



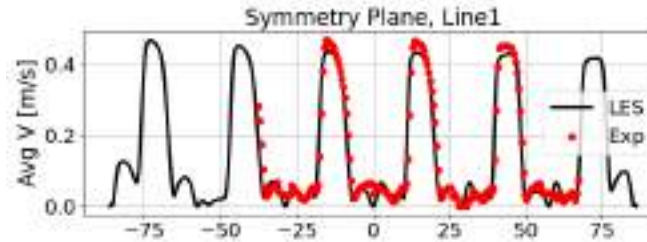
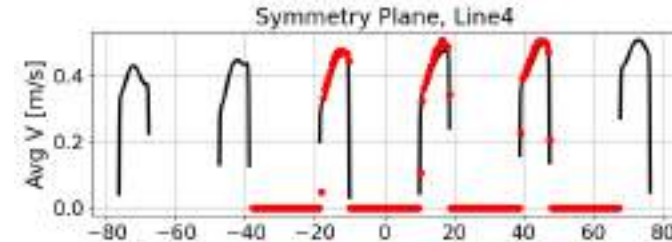
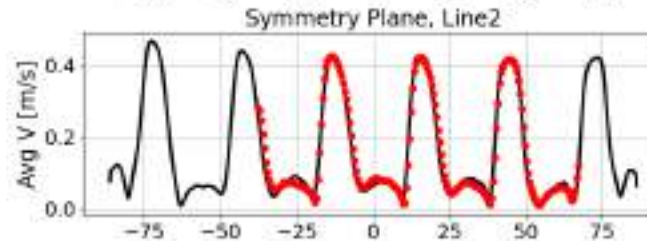
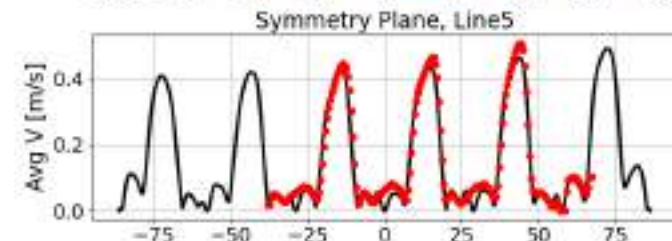
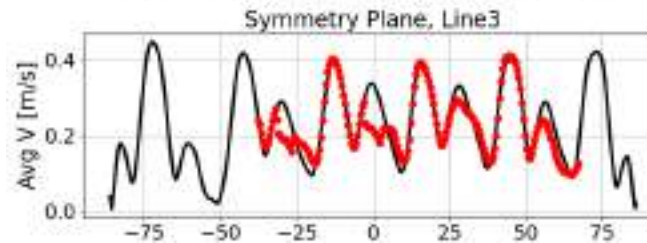
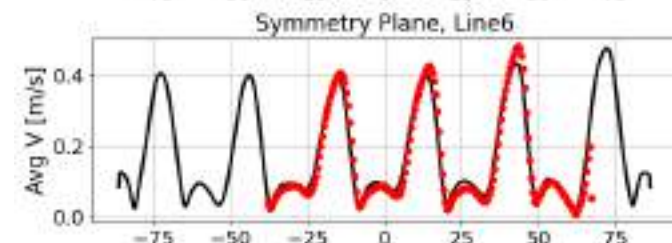
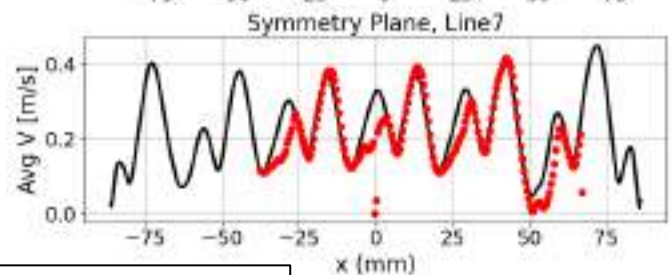
Outlet:

p11 = 170 mm
p10 = 150 mm
p9 = 130 mm
p8 = 110 mm

Bundle:

p7 = 0 mm
p6 = -4.4 mm
p5 = -8.8 mm
p4 = -18.8 mm
p3 = -28.8 mm
p2 = -33.2 mm
p1 = -37.6 mm

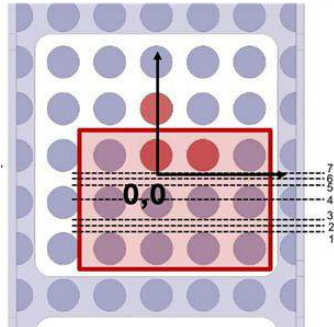
↑ Flow direction



Bundle ave. velocity profiles

Results – Full Bundle – Bundle Profiles

- Good spatial agreement for simulations and experiment.
- Overprediction of experimental TKE magnitude in the bundle



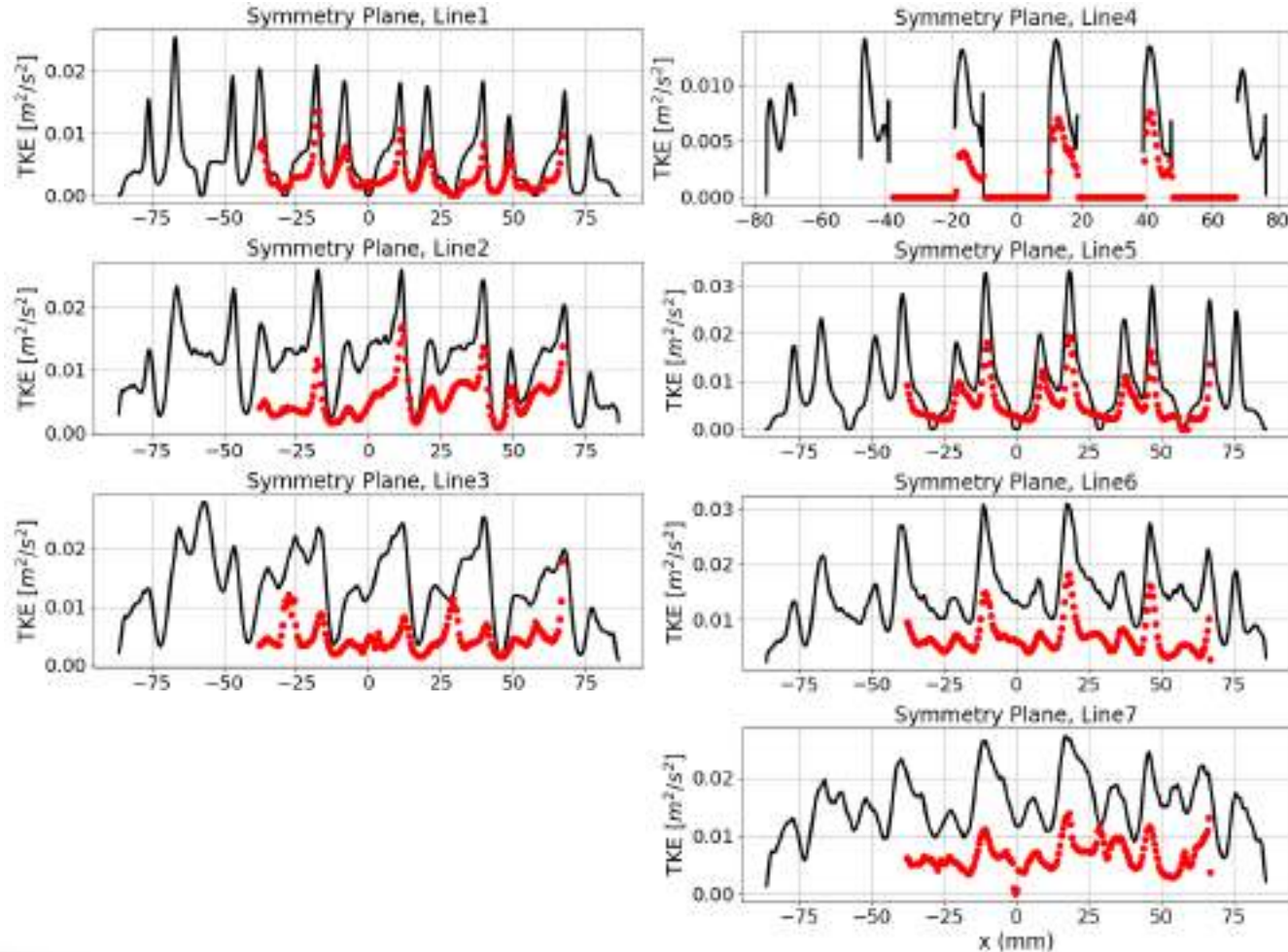
↑ Flow direction

Outlet:

p11 = 170 mm
p10 = 150 mm
p9 = 130 mm
p8 = 110 mm

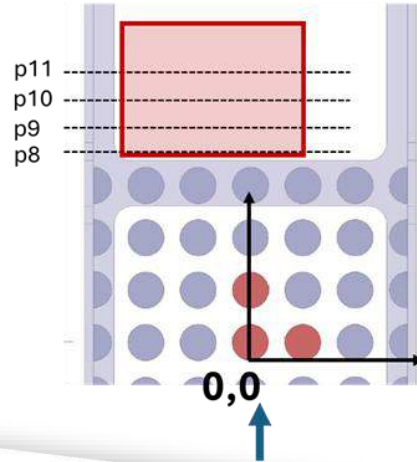
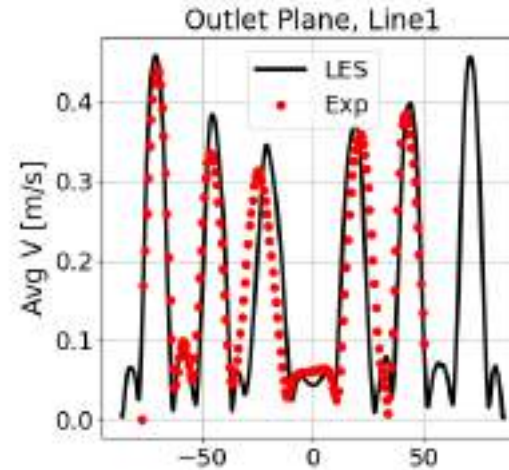
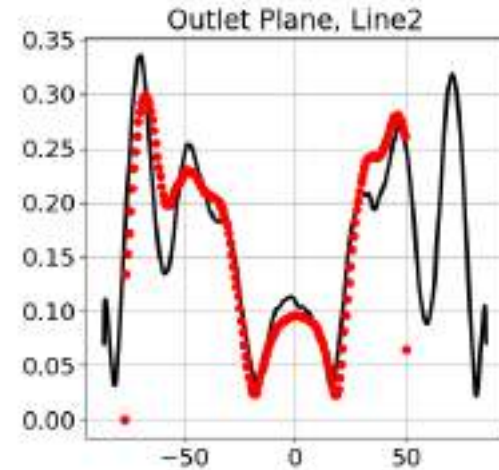
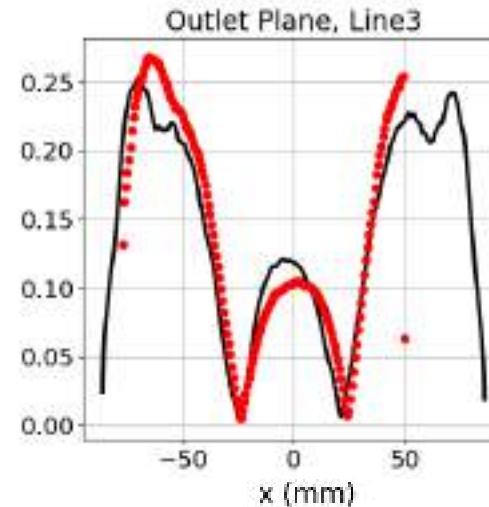
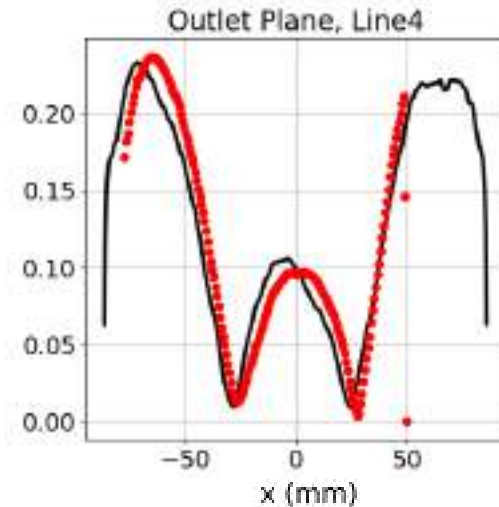
Bundle:

p7 = 0 mm
p6 = -4.4 mm
p5 = -8.8 mm
p4 = -18.8 mm
p3 = -28.8 mm
p2 = -33.2 mm
p1 = -37.6 mm



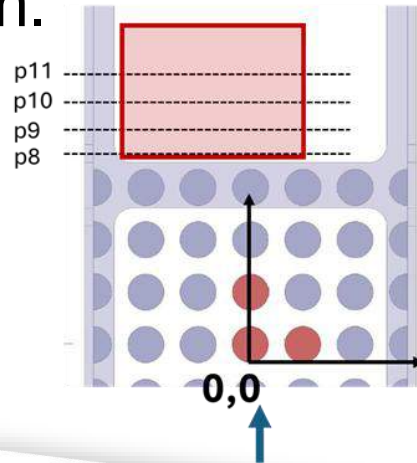
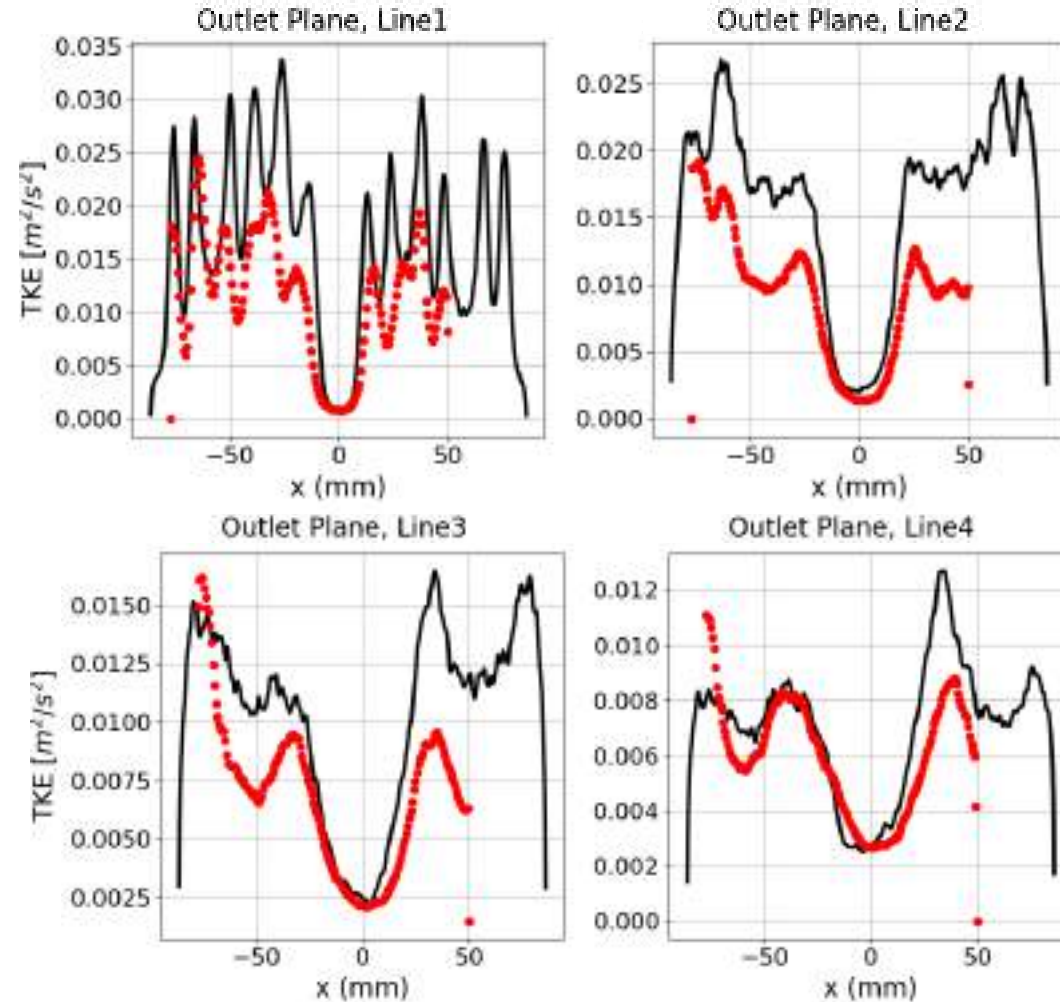
Results – Full Bundle – Outlet Profiles

- The simulations match well for average velocity in the outlet regions.
- There is some differences but we need to include uncertainty for full comparison.



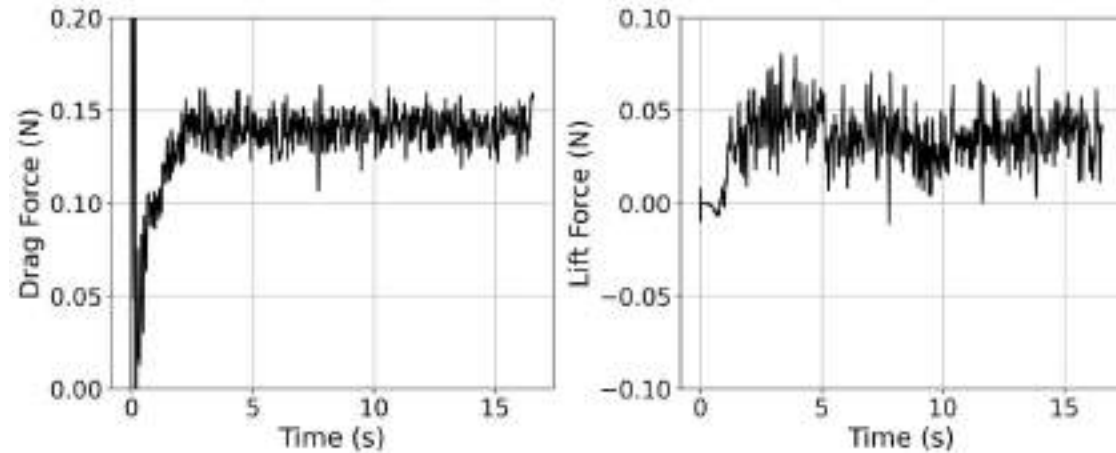
Results – Full Bundle – Outlet Profiles

- The simulations reasonably well for average TKE in the outlet regions with some overpredictions.
- There is some differences but we need to include uncertainty for full comparison.

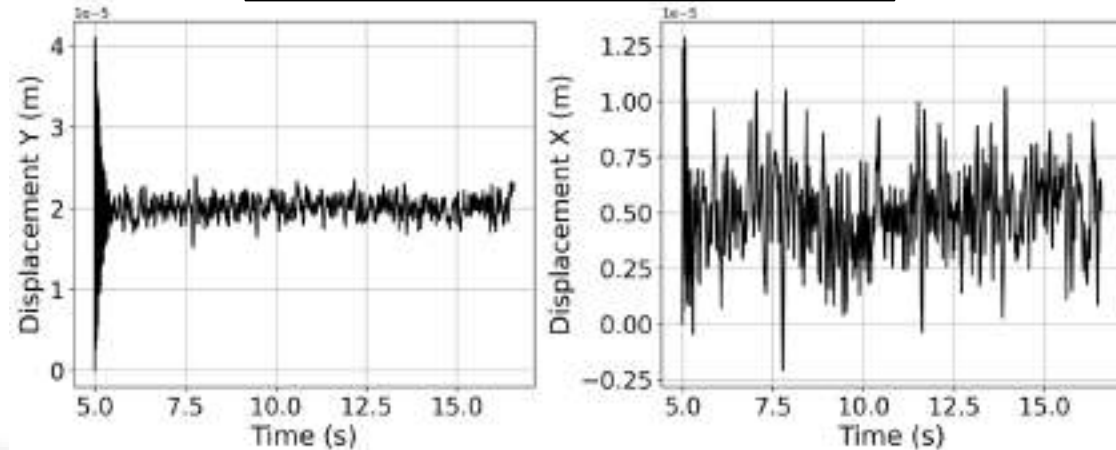


Results – Full Bundle – Tube Displacement

- Displacement of the center tube was calculated using one-way coupled spring-mass solve
- First 5s was neglected from the force signal to flush out initial transient



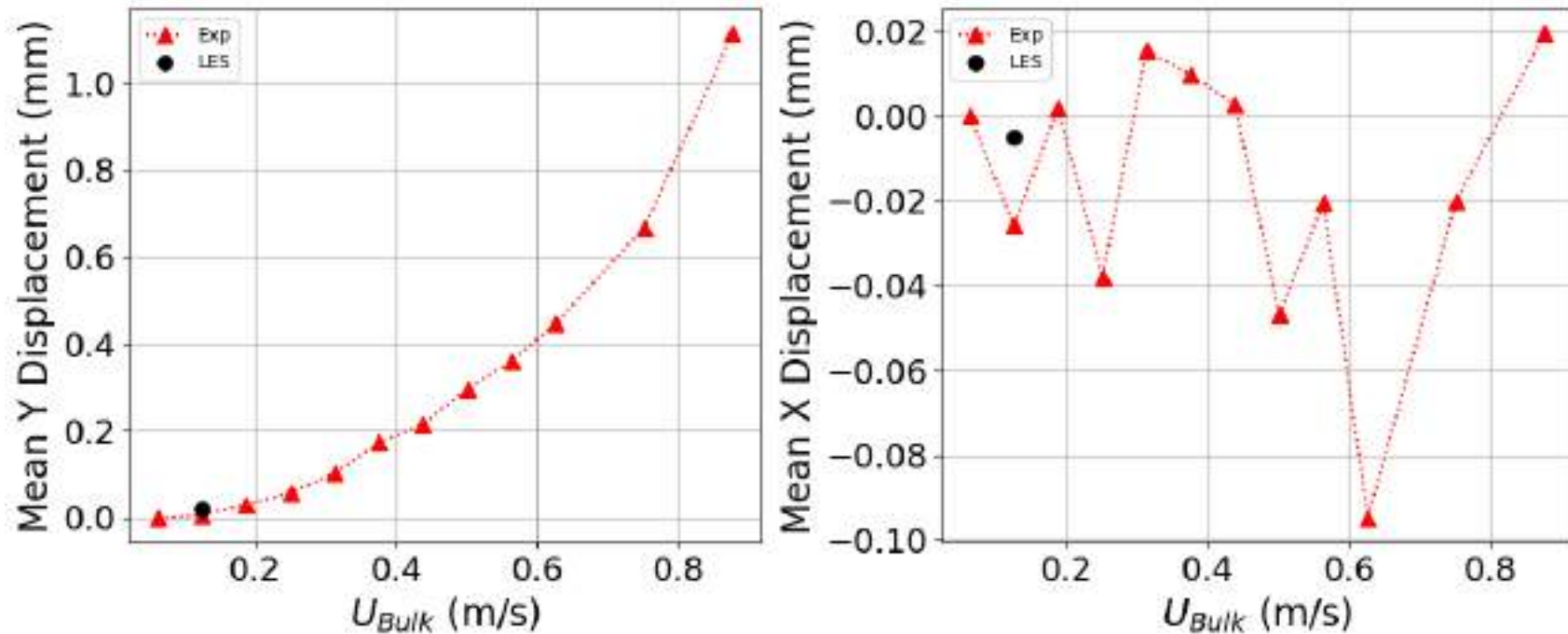
Drag and lift force from LES



Displacement from one-way FSI

Results – Full Bundle – Tube Displacement

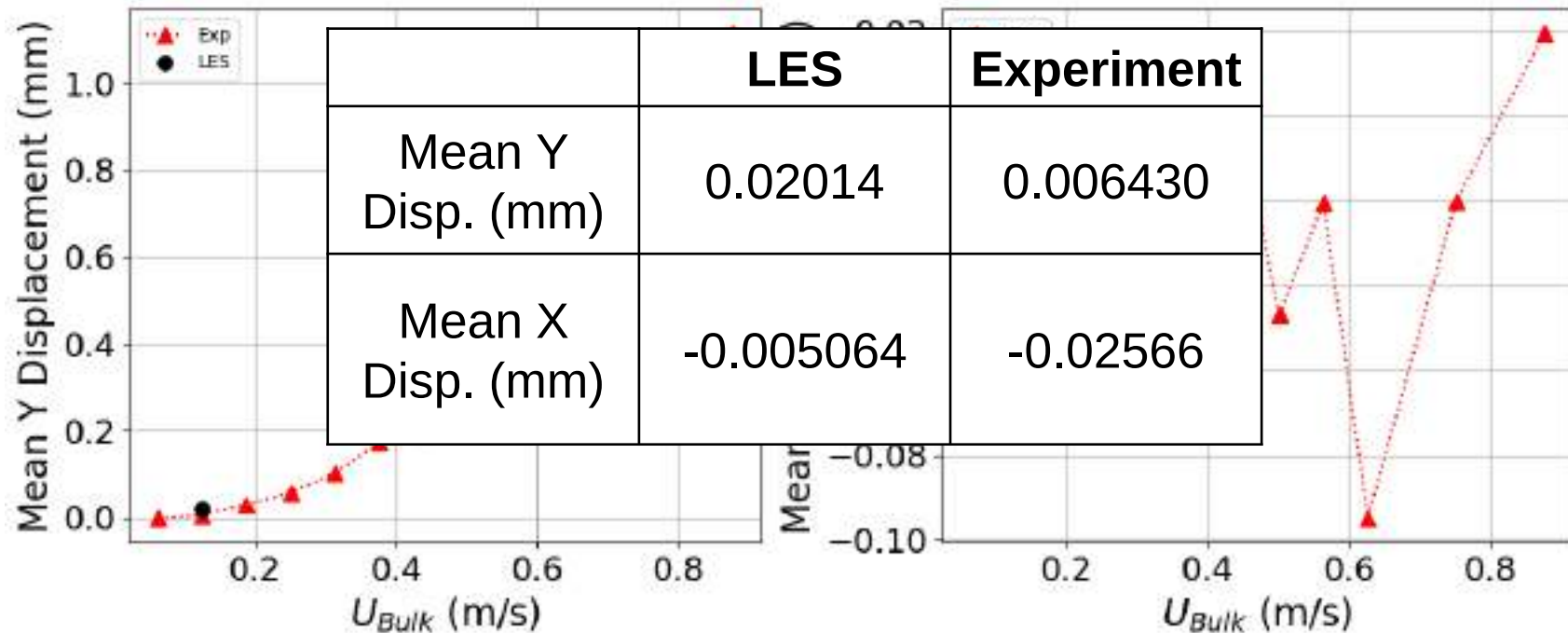
- One-way coupled approach follows the general trend for the mean displacement of the center rod when compared to the experiment.



Mean displacement in the X and Y direction
for all flow rates compared to LES

Results – Full Bundle – Tube Displacement

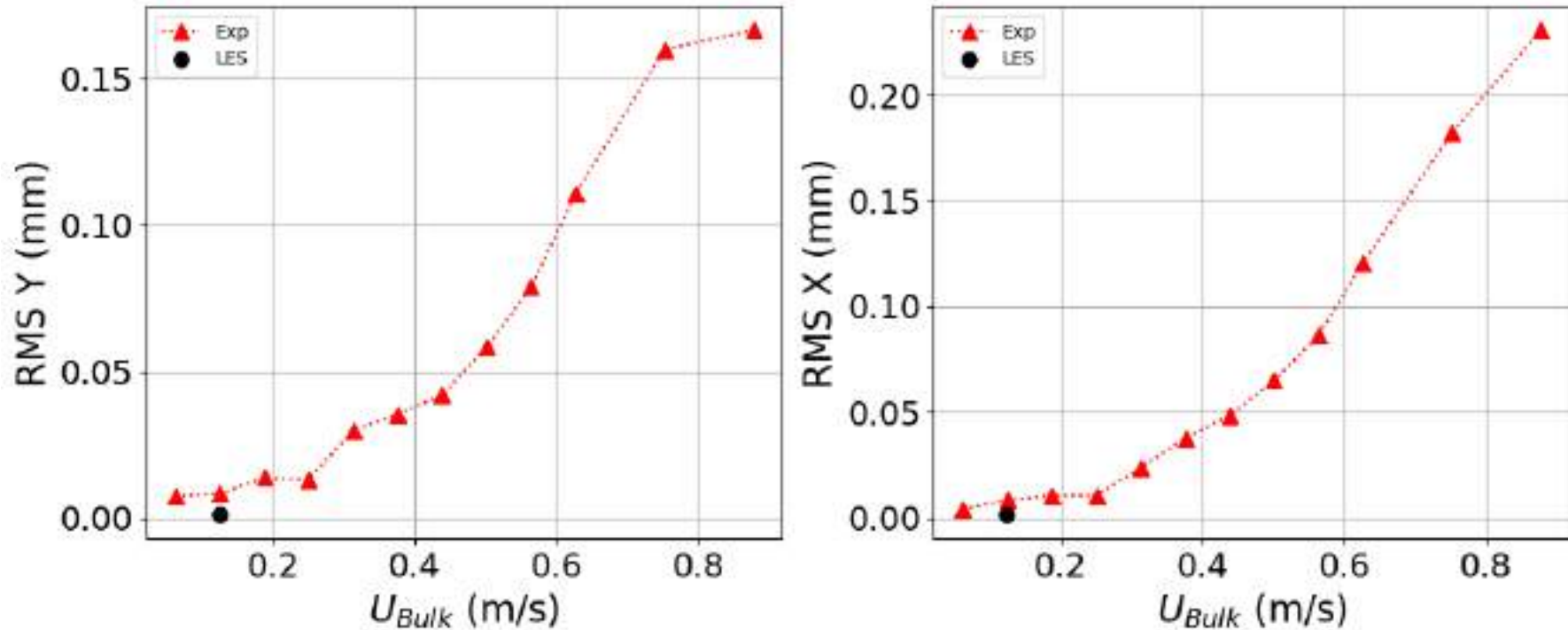
- One-way coupled approach follows the general trend for the mean displacement of the center rod when compared to the experiment.



Mean displacement in the X and Y direction
for all flow rates compared to LES

Results – Reduced Bundle – Tube Displacement

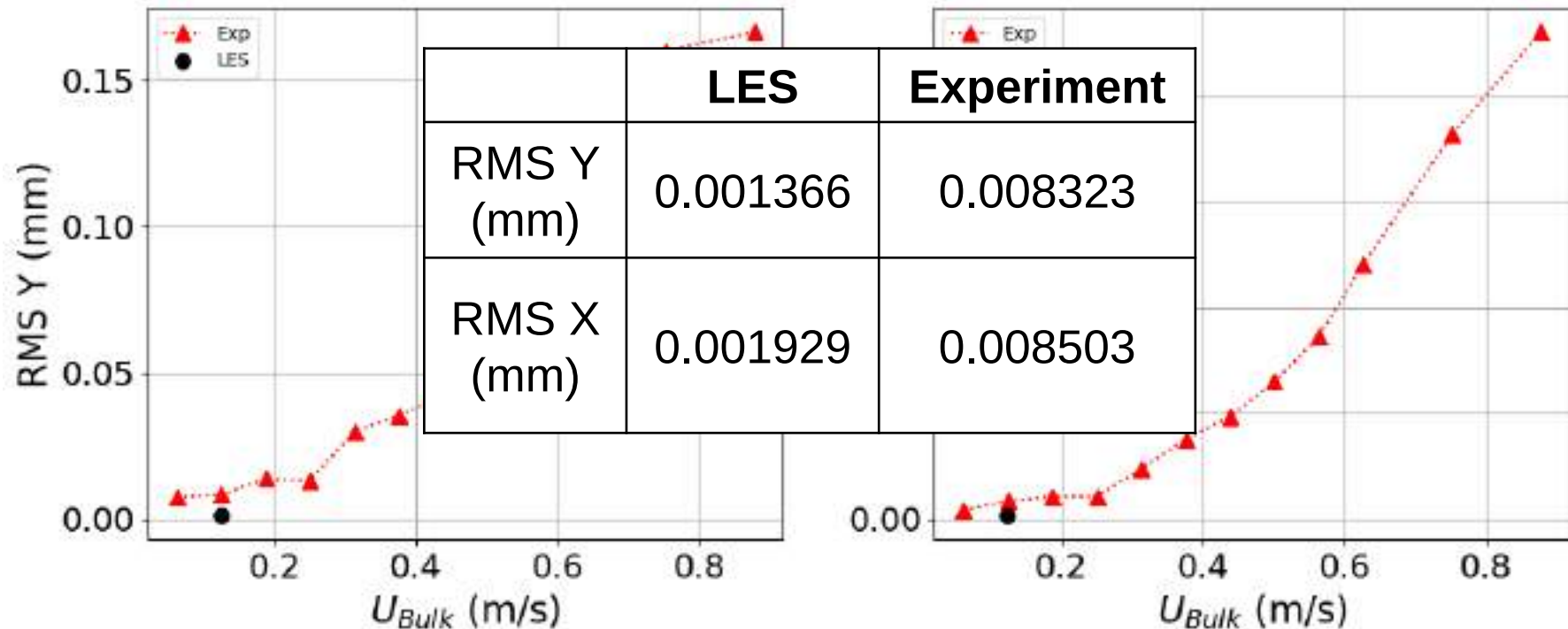
- RMS from the one-way coupled approach in both directions is underpredicted when compared to the experiment.



X and Y RMS for all flow rates compared to LES

Results – Reduced Bundle – Tube Displacement

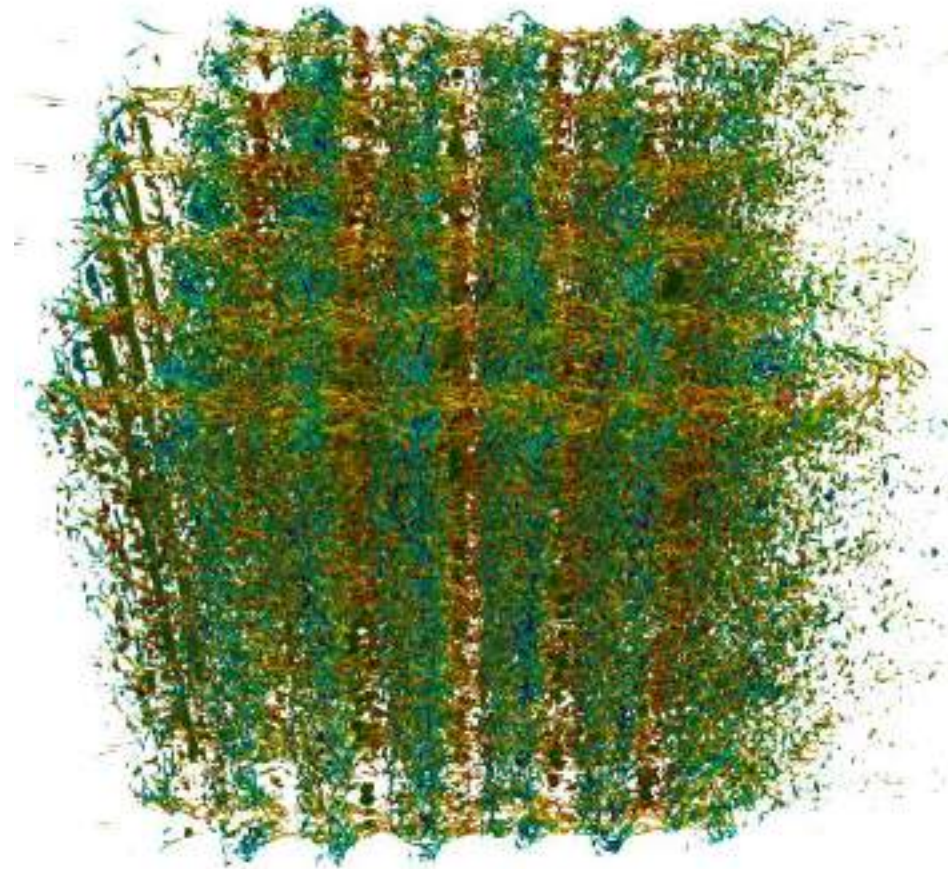
- RMS from the one-way coupled approach in both directions is underpredicted when compared to the experiment.



X and Y RMS for all flow rates compared to LES

Conclusions for both studies

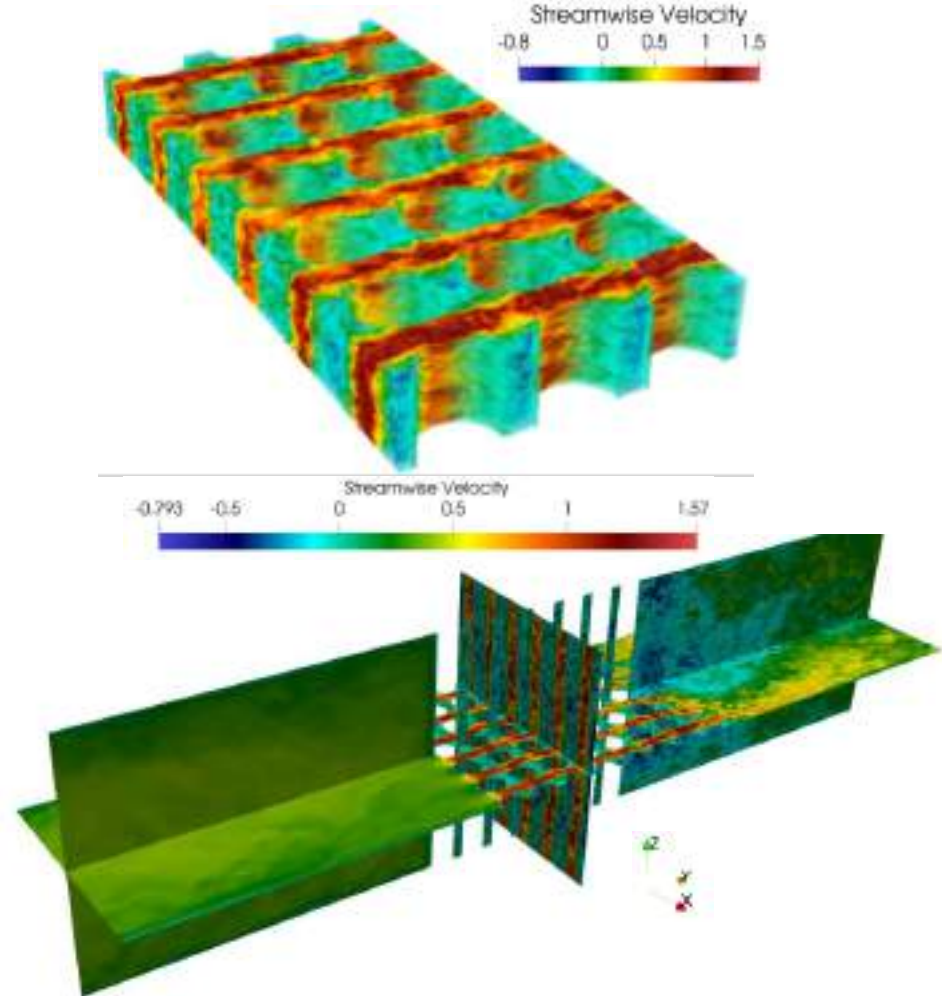
- Overall, good agreement is observed in the mean velocity field between the LES and the experiment results
- LES overpredicts the magnitude of TKE throughout the bundle when compared to the experiment.
- Mean displacements follow the general trend of the experiment.
- Underprediction when comparing RMS displacements.



Isosurface contours of Q-criterion ($Q=80$)
colored by streamwise velocity

Future work for both studies

- Finalize the full bundle results using the turbulent inlet conditions.
- Redo the reduced domain bundle with appropriate meshing requirements.
- Finalize comparison of results between reduced, full bundles and experiment.



Acknowledgements

This material is based on work supported by the Gathering expertise On Vibration ImpaKt In Nuclear power Generation (GO-VIKING) project funded by the European Union under grant proposal ID [101060826](#). This material is also based upon work supported by the **National Science Foundation**, United States under Award No. [2130169](#). This material is based upon work supported under a Department of Energy, Office of Nuclear Energy **University Nuclear Leadership Program Graduate Fellowship**.

Any opinions, findings and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the European Commission, Department of Energy Office of Nuclear Energy or the National Science Foundation.

This research made use of Idaho National Laboratory's High Performance Computing systems located at the Collaborative Computing Center and supported by the Office of Nuclear Energy of the U.S. Department of Energy and the Nuclear Science User Facilities under Contract No. DE-AC07-05ID14517.



High-resolution GOKSTAD computations at VCU

Reduced Domain and Full Experimental Domain Studies

Trevor Franklin and Lane Carasik, Ph.D.

Fluids in Advanced Systems and Technology (FAST) Research Group, Virginia Commonwealth University

Email: lbcarasik@vcu.edu – Carasik Website: <https://eqr.vcu.edu/directory/lane.carasik/> & Google Scholar: [Link](#)



The GOKSTAD benchmark: comparison of numerical results and data

GO-VIKING Workshop

Kevin Zwijsen (NRG PALLAS), Trevor Franklin (VCU), Lane Carasik (VCU), Sofiane Benhamadouche (EDF)
GRS, Garching, Germany, 20 May 2026



Funded by
the European Union

NRG ref. 2.6206.20/26.3/
EU DuC = 0E001

Outline

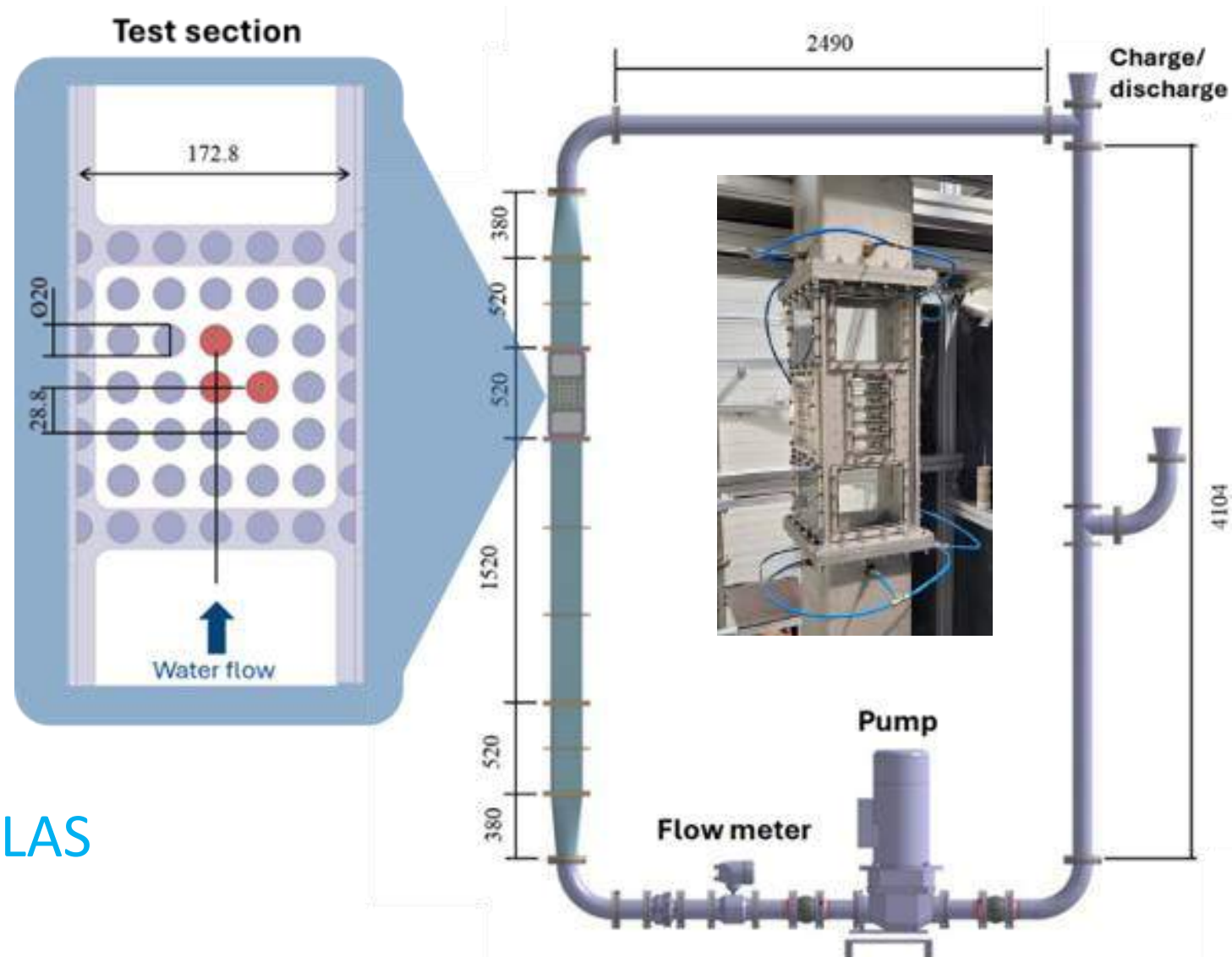


- Introduction
- Benchmark Case
- Results

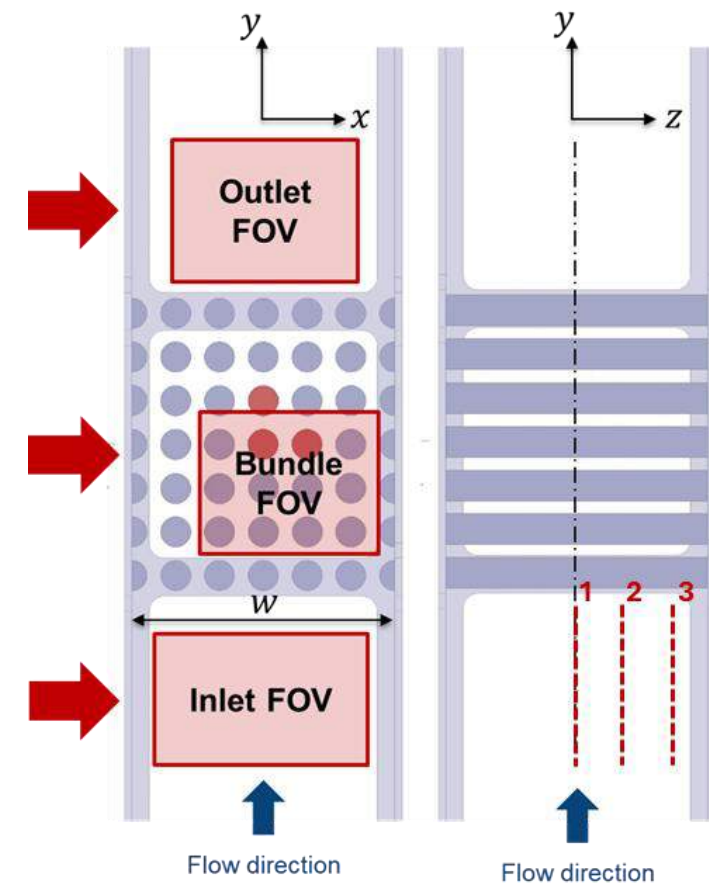
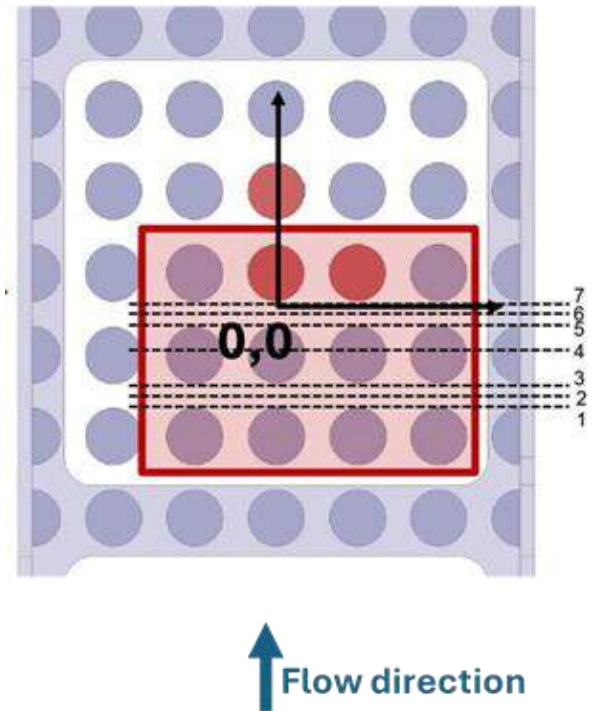


Introduction

- GOKSTAD
- 7x7 tube bundle in cross flow
- PIV + vibration data
- Central vibrating tube
- Inlet velocity = 0.125 m/s
 - $Re_{U_{bulk}} \approx 23.125$
- Working Fluid → Water
- Temperature Range → 20-21 °C
- 3 Participants: VCU, EDF, NRG PALLAS



Introduction: PIV measurements

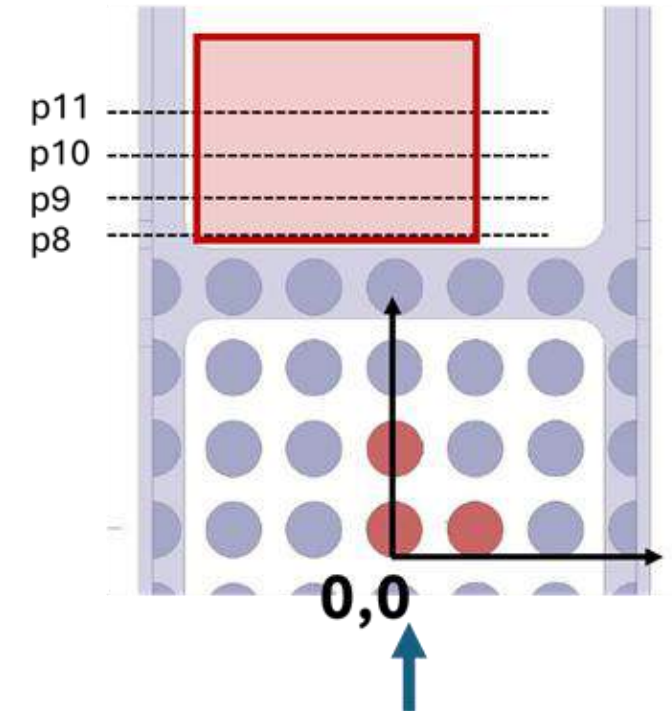


Outlet:

p11 = 170 mm
p10 = 150 mm
p9 = 130 mm
p8 = 110 mm

Bundle:

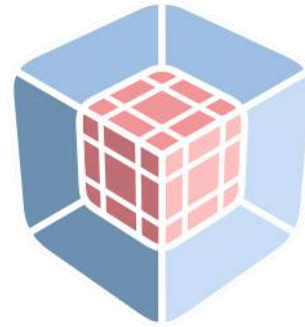
p7 = 0 mm
p6 = -4.4 mm
p5 = -8.8 mm
p4 = -18.8 mm
p3 = -28.8 mm
p2 = -33.2 mm
p1 = -37.6 mm



Different Approaches



Institution	Approach	Turbulence Modelling	Solid Modelling	Software(s)
VCU	One-way FSI	Resolved LES	Rigid-body approach	Nek5000
EDF	Two-way FSI	Elliptic-Blending RSM	Rigid-body approach	Code_Saturne V8
NRG PALLAS	Two-way FSI	WR-LES WALE	Rigid-body approach	Simcenter STAR-CCM+ V2510



Solid Domain



- Moving cylinder shows no rocking or rotation motion.
- Only translation in lift and drag direction
- All partners modeled it as 2DoF mass-spring-damper system:

$$m\ddot{x}(t) + c\dot{x}(t) + kx(t) = F_x(t)$$

$$m\ddot{y}(t) + c\dot{y}(t) + ky(t) = F_y(t)$$

- From experiment: $m = 0.1 \text{ kg}$, $c = 1.14 \text{ n} \cdot \text{s/m}$, $k = 7000 \text{ N/m}$.
- Hence not explicitly meshed, simple rigid body

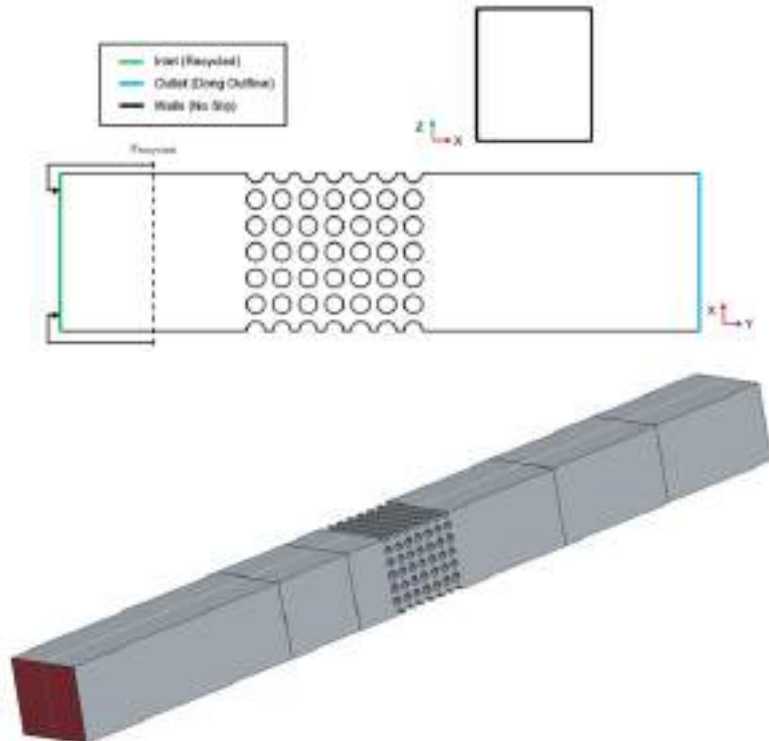
Solid Boundary Conditions



Institution	Moving cylinder	All other walls	Remarks
VCU	No-slip wall	No-slip	LES forces fed to cylinder in stand-alone simulation.
EDF	No-slip wall	No-slip	-
NRG PALLAS	No - slip + FSI interface	No-slip	Sides of moving cylinder slide over side walls.

Fluid Domain

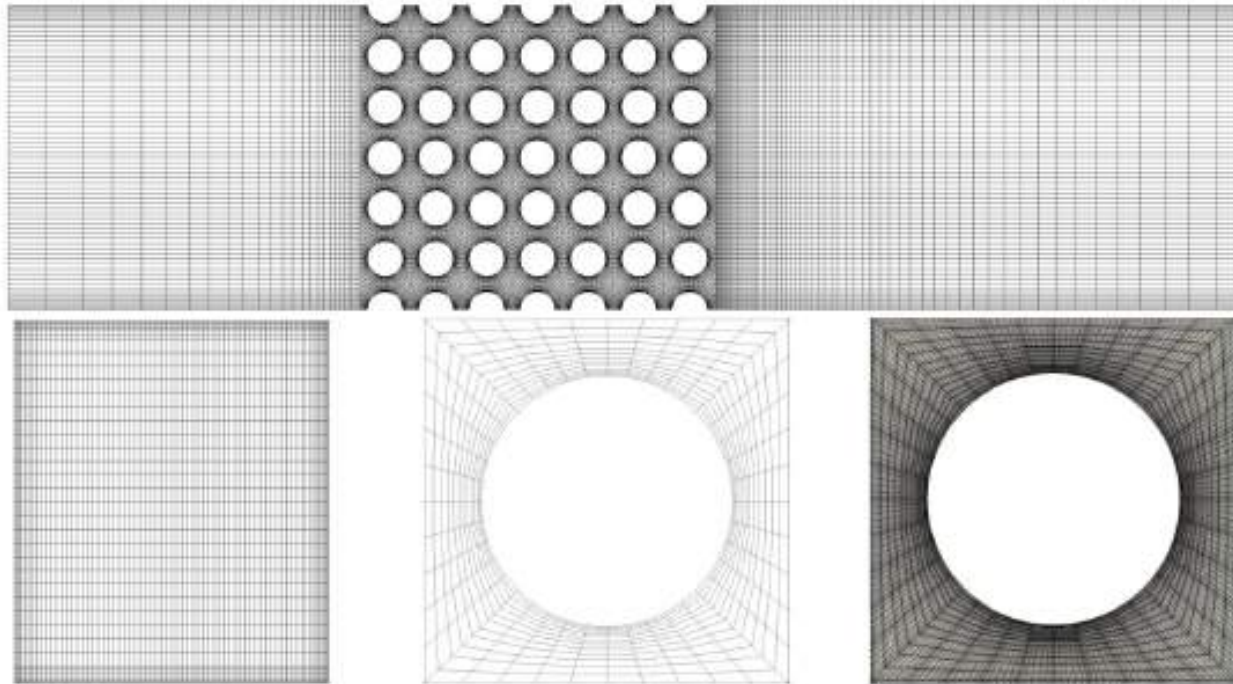
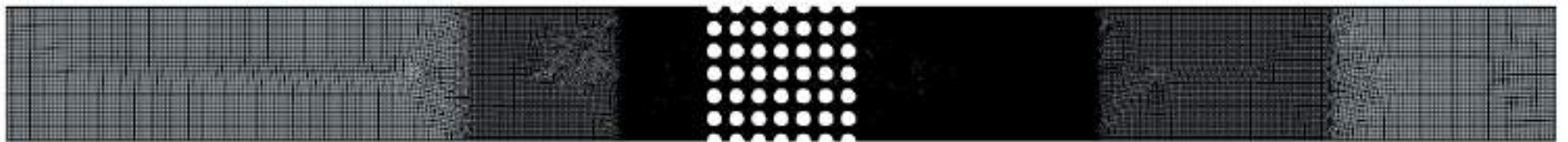
- In the experiment, have bend upstream. Not included in simulations



Institution	Geometry Model	Mesh Cell Type	Mesh Cell Count (M Cells)	Wall Y+	Courant Number
VCU	Full bundle included. Inlet region is 10D, and outlet region 15D.	Hexahedral	1.02 (Polynomial Order=7)	Max. = 6.08 Avg. = 0.854	Avg. = 0.36
EDF	Full bundle included. Inlet region is 10D, and outlet region 50D.	Hexahedral	10	Max. = 2.0 Surf. Avg. = 0.26	Max. = 0.97 Vol. Avg. = 0.06
NRG PALLAS	Full bundle included. Upstream and downstream regions $\approx$ 0.9m.	Hexahedral	10.7	Max. = 0.207 Surf. Avg. = 0.0386 (Excl. front and back)	Max. = 2.99 Vol. Avg. = 0.0821

Fluid Domain - Meshes

- VCU: base mesh + 7th order polynomial

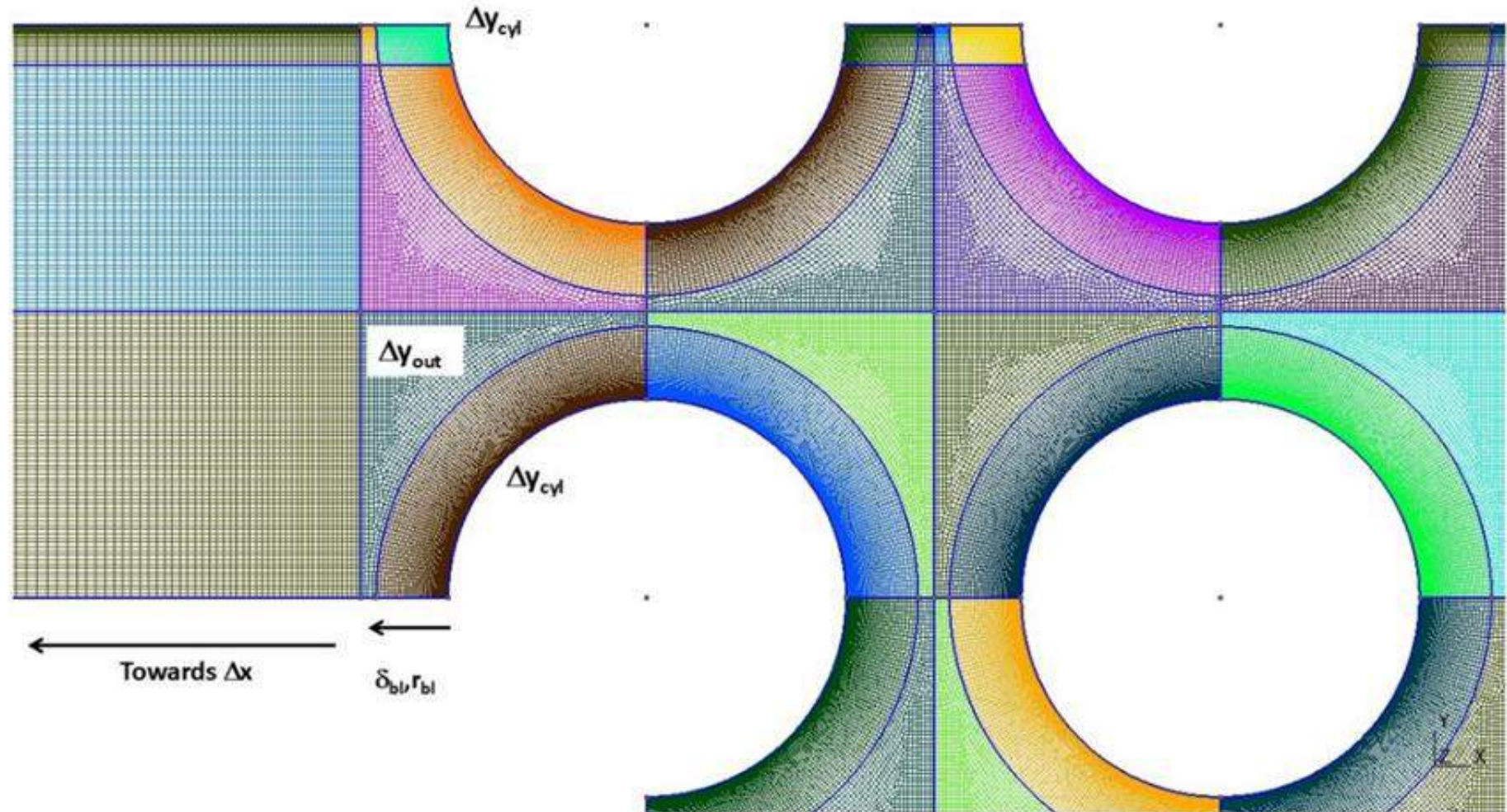


- NRG PALLAS



Fluid Domain - Meshes

- EDF: similar meshing strategy as used for AMOVI.



Fluid Material Properties



Institution	Density (kg/m ³)	Dynamic Viscosity (mPa·s)	Remarks
VCU	1000.0	1.0	Experiment at 20-21°C, which means $\rho \approx 998 \text{ kg/m}^3$ and $\mu \approx 1.0 \text{ mPa}$
EDF	1000.0	1.0	
NRG PALLAS	1000.0	1.0	

Fluid Boundary Conditions



Institution	Inlet BC	Outlet(s) BC	Remarks
VCU	Inlet velocity of 0.125 m/s	Dong outflow condition	Inflow velocity profile obtained by recycling over inlet section of length $5D_{\text{pipe}}$.
EDF	Inlet velocity of 0.125 m/s	Free outlet	Fully developed profile using an embedded 2D simulation
NRG PALLAS	Uniform value of 0.125 m/s	0 Pa Gauge Pressure Outlet	At inlet and initial conditions, 5% turbulence intensity set with turbulent length scale of 0.13m

Numerical Settings



Fully developed flow before starting FSI simulations and gathering statistics

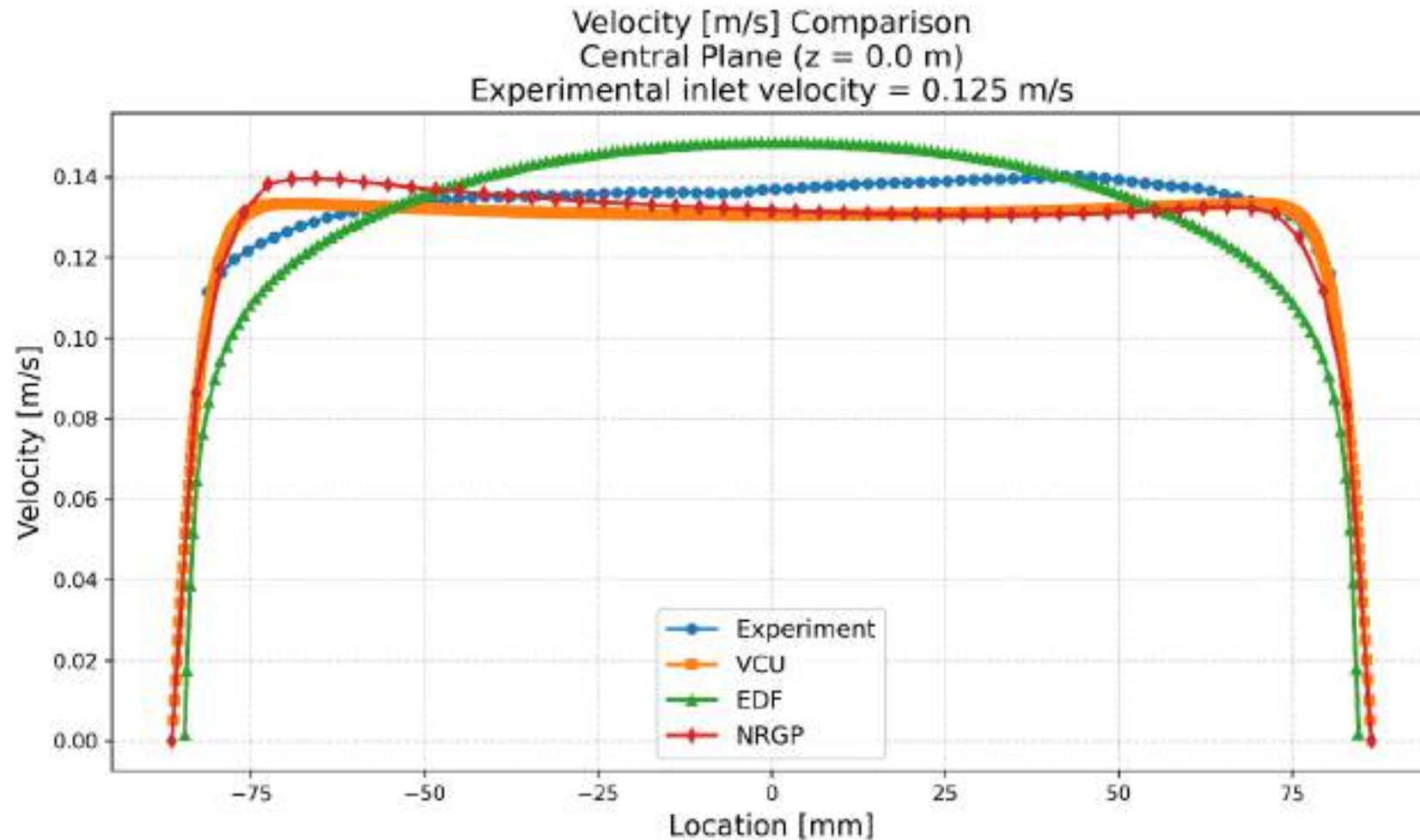
Institution	Time Step Size (ms)	Total Simulation Time (s)	Order of Discretization	Other Provided Data
VCU	0.25	9 FP for flow-only averaging and force data	Temporal = 2 nd (BDF2) Spatial = 7 th	For LES, only the explicit high-pass filter is used. No additional sub-grid scale model is included.
EDF	0.25	>361s	Temporal = 2 nd (Crank-Nicolson) Spatial = 2 nd	Maximum number of ALE outer iterations = 10.
NRG PALLAS	1.0	74.0 (40.0 s flow + 34.0s FSI)	Temporal = 2 nd (BDF2) Spatial = 2 nd	Number of inner iterations per time step = 25 SIMPLEC pressure-velocity coupling scheme

Computational Resources

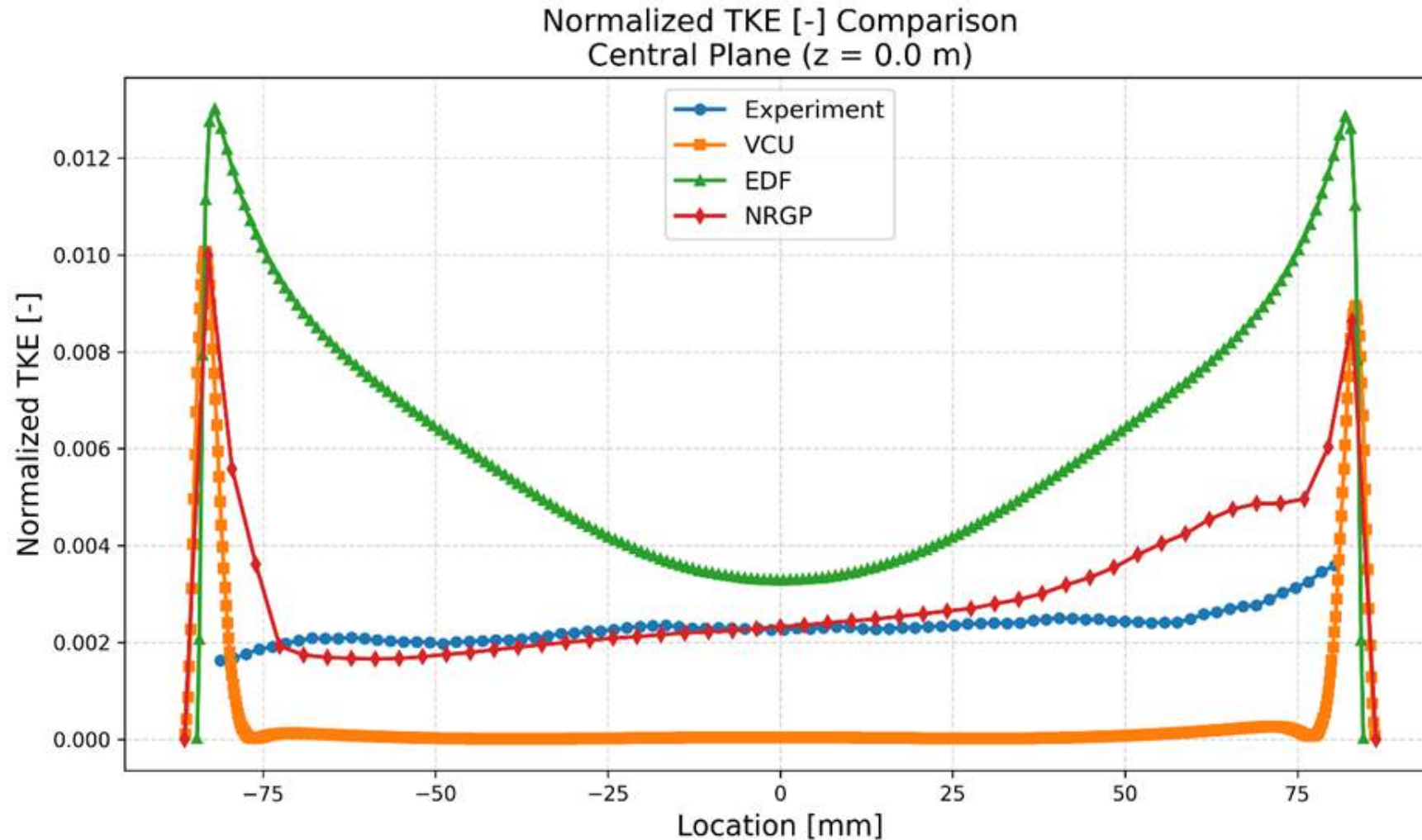


Institution	Wall Clock Time	Equivalent CPU/Node hours	RAM	No. of (CPU/GPU)Cores	Other Provided Data
VCU	14d 22h 5m	61.9E5 node h	8.5 TB	17280 CPU cores	Number of nodes used = 90
EDF	44d 3h	6.77E5 CPU h	6 TB	640 CPU cores	10
NRG PALLAS	12d 5h 45m	4.22E5 CPU h	768 GB	144 CPU cores	Number of nodes used = 3

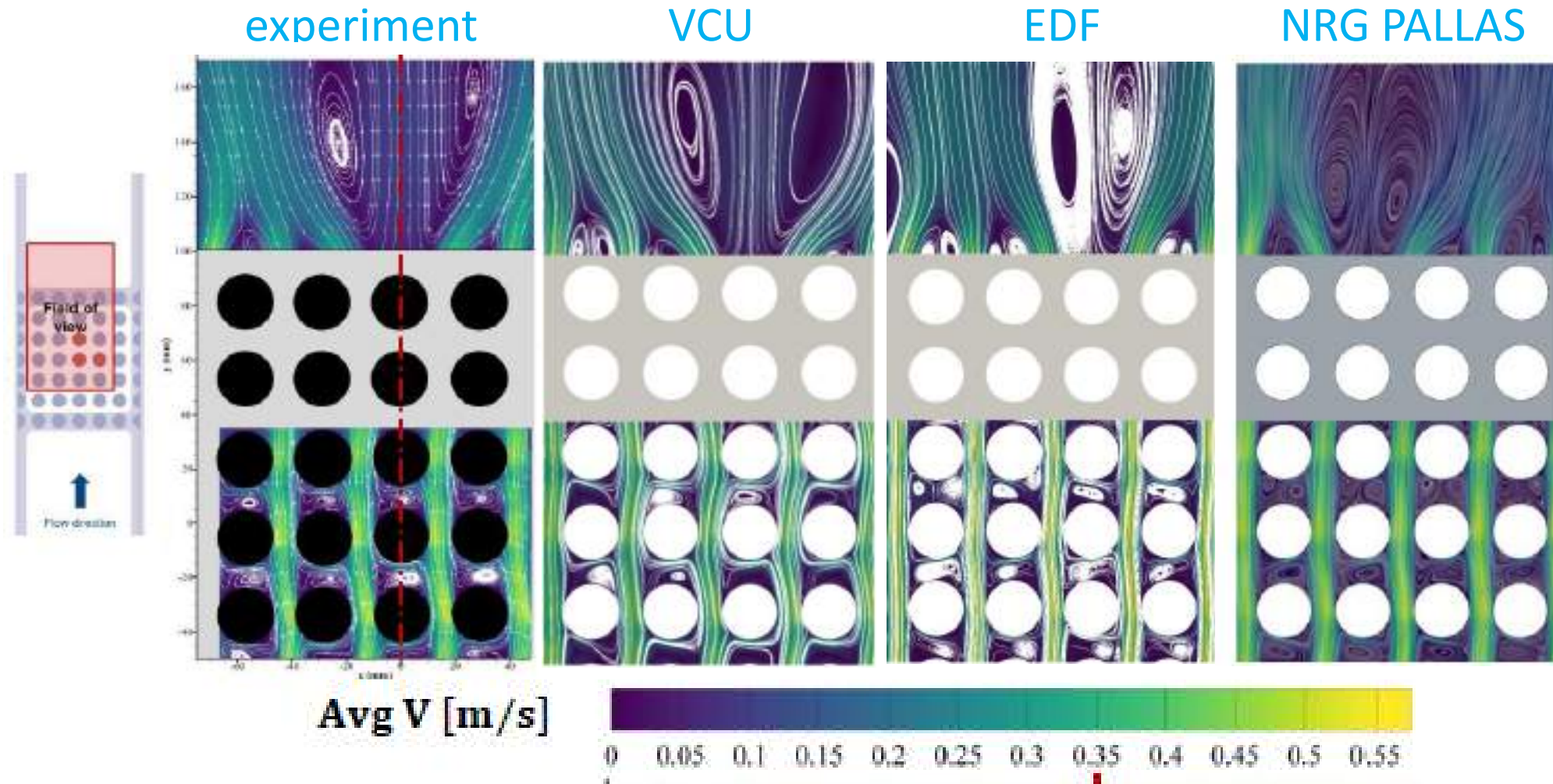
Results: inlet velocity



Results: inlet normalized TKE



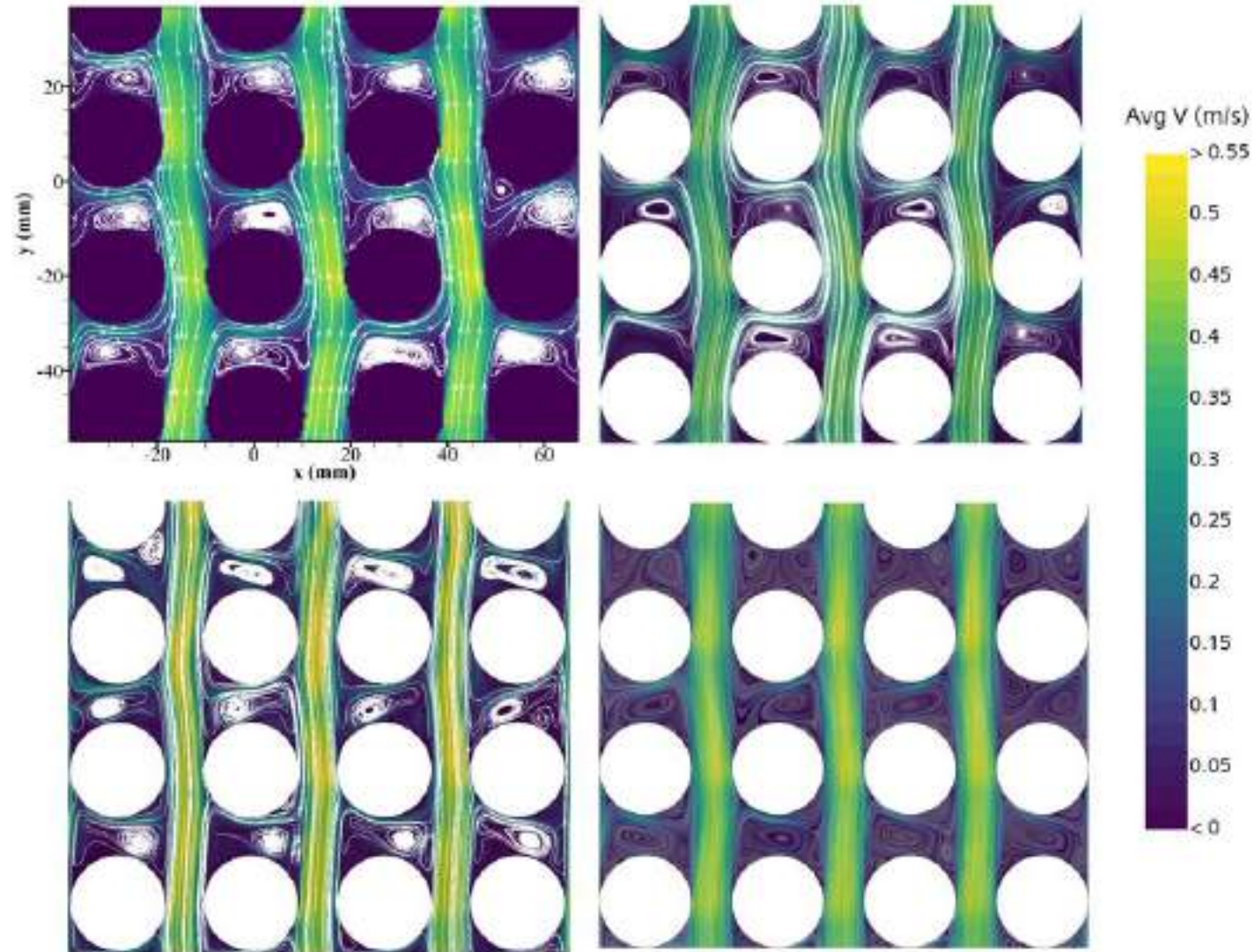
Results: bundle + outlet velocity



Results: bundle velocity



Experiment

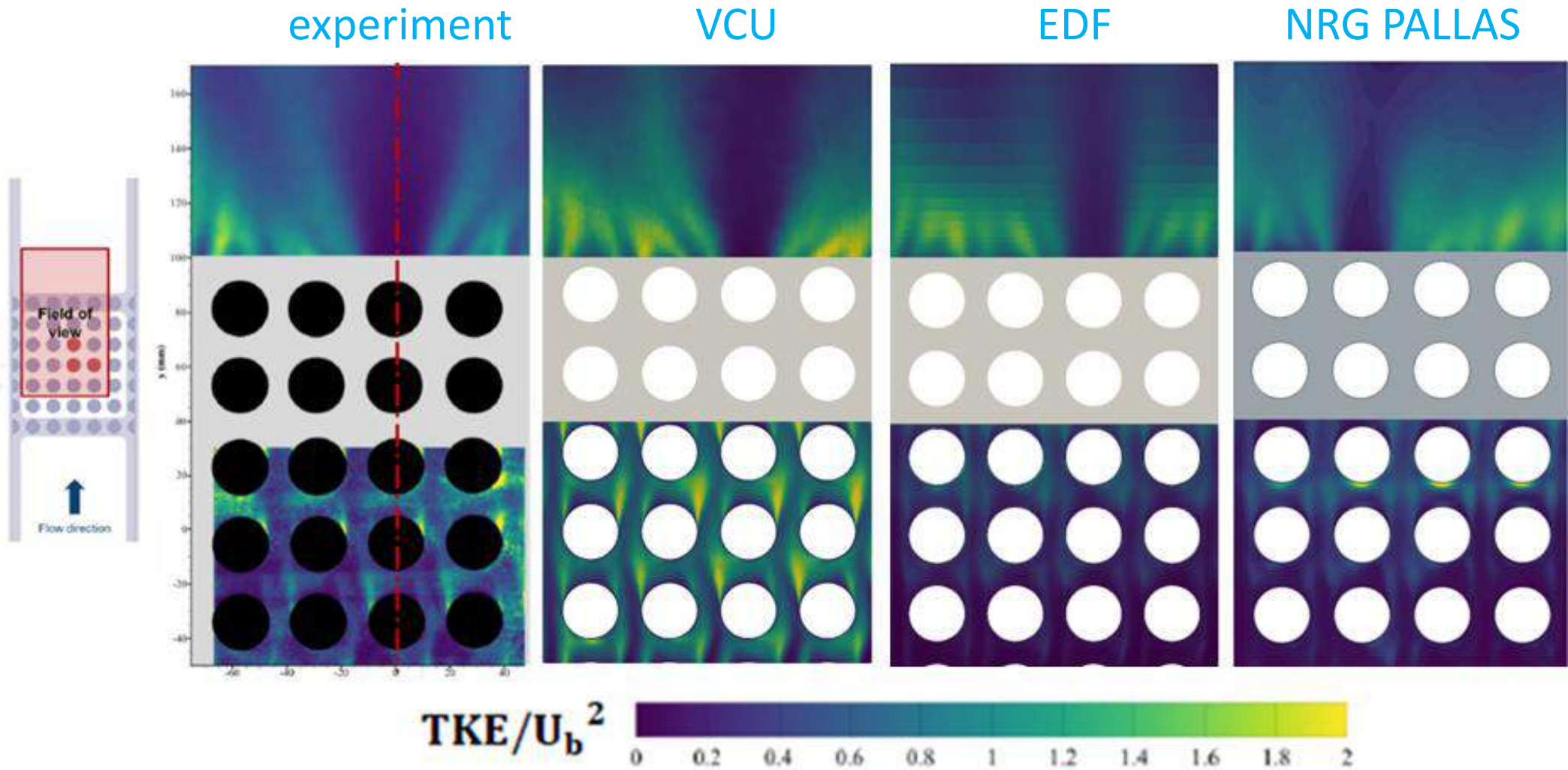


VCU

EDF

NRG PALLAS

Results: bundle + outlet norm. TKE



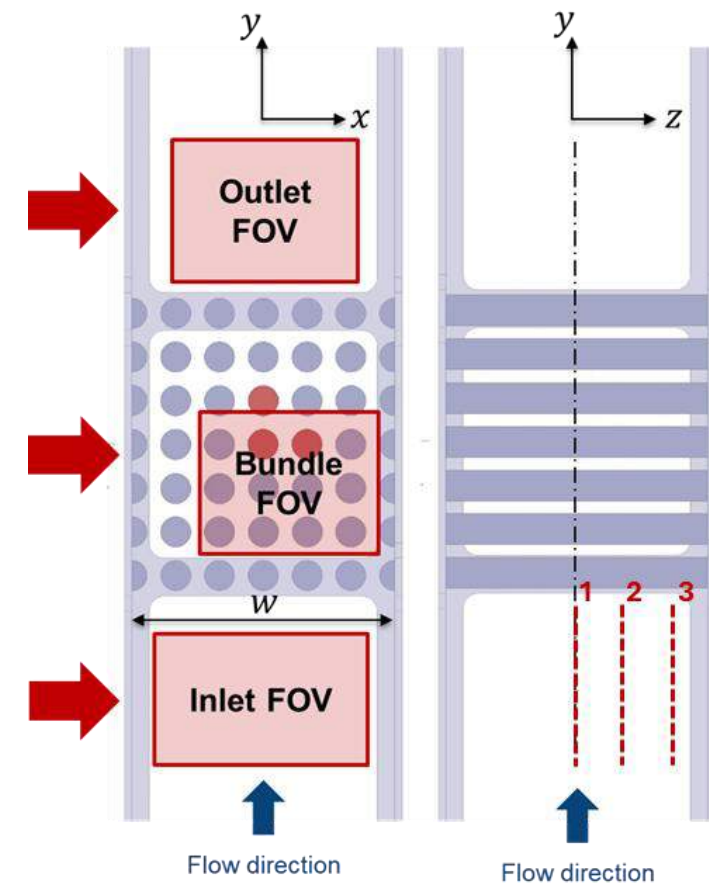
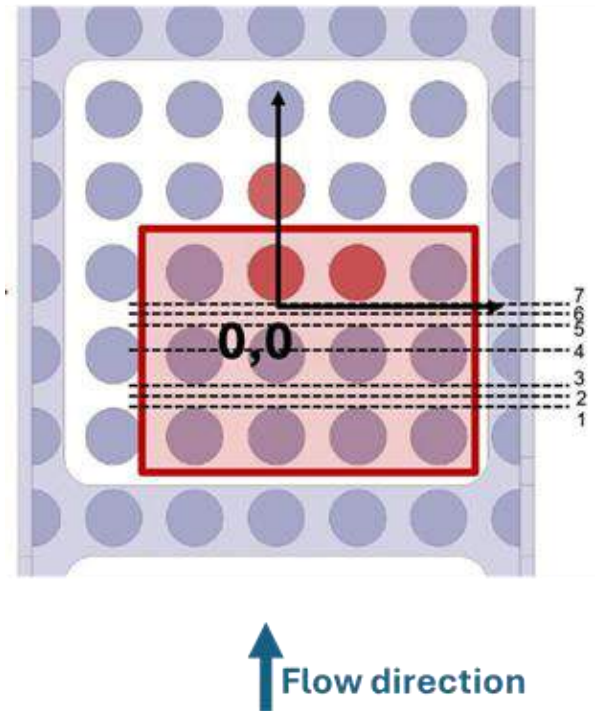


VCU

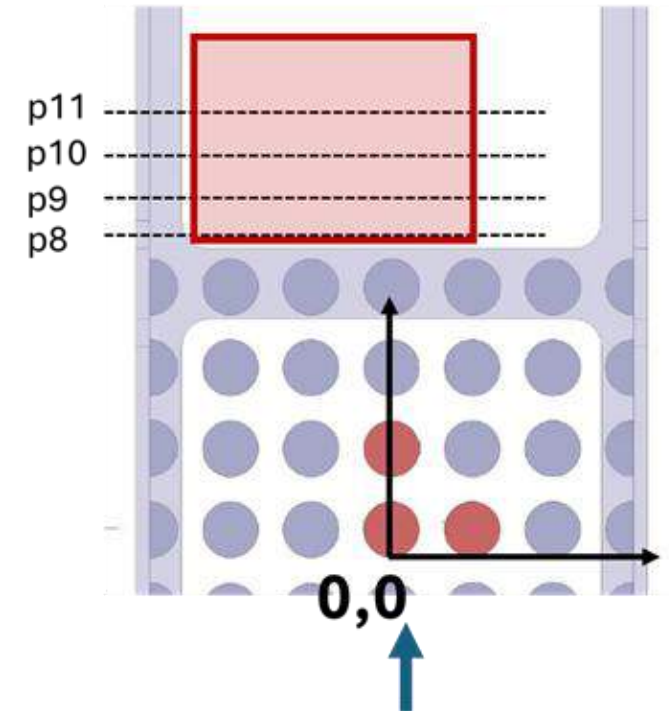


EDF

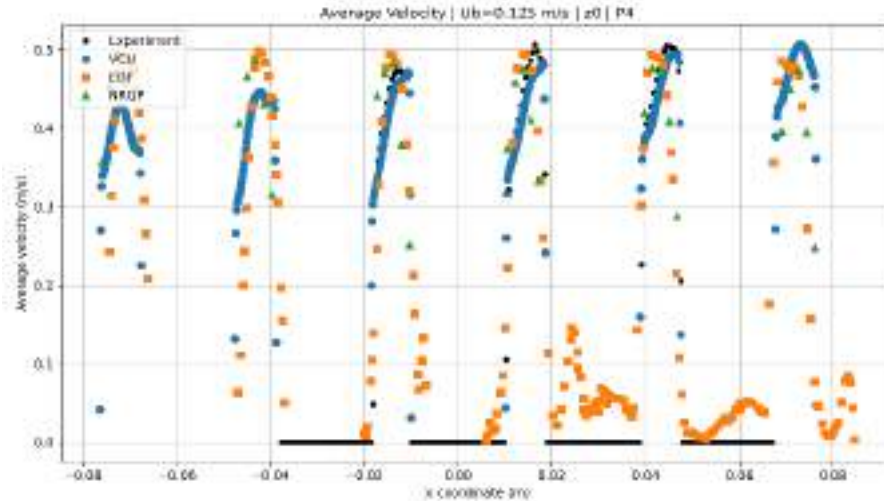
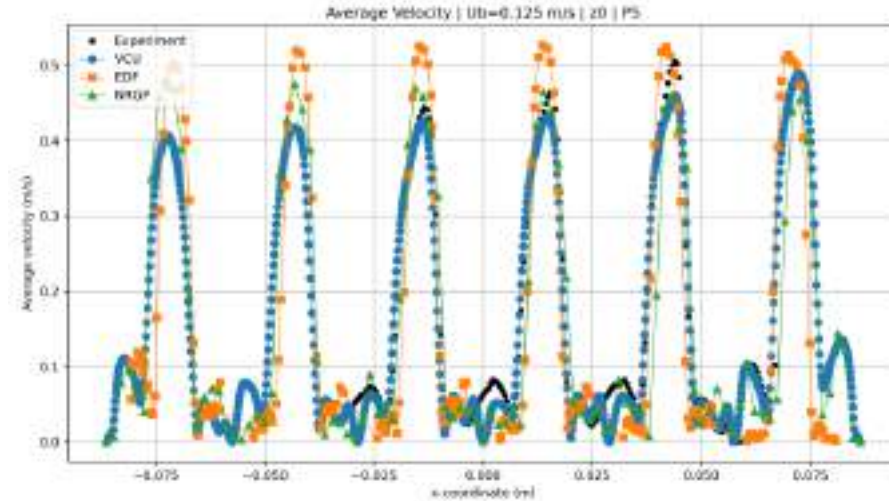
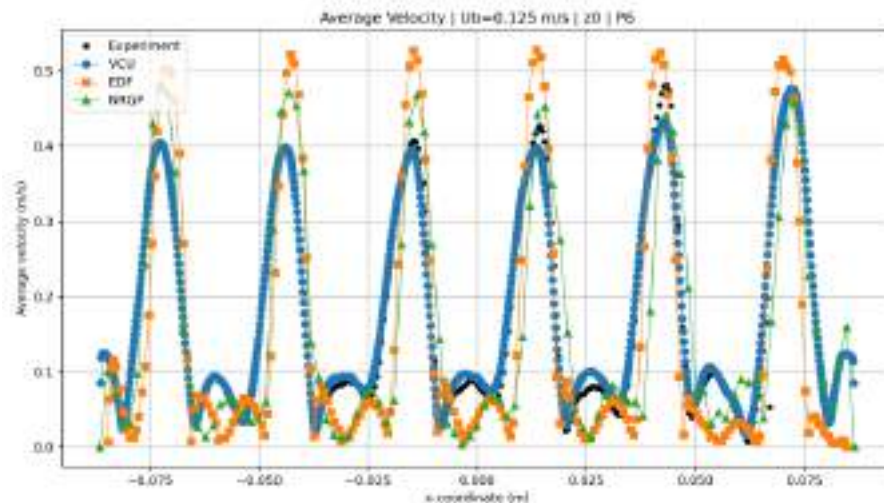
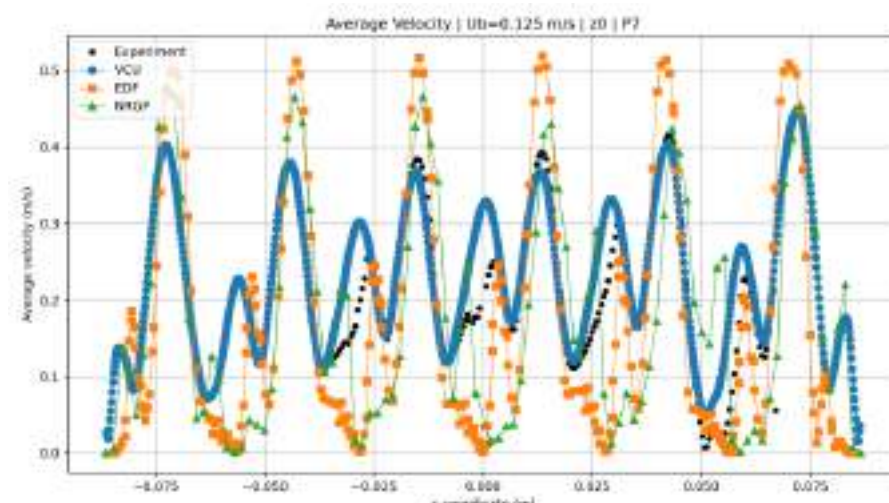
Reminder: PIV measurements



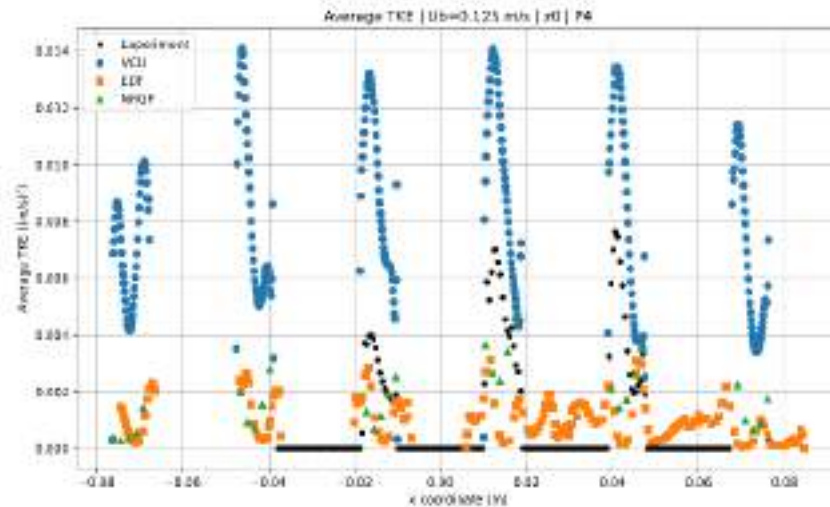
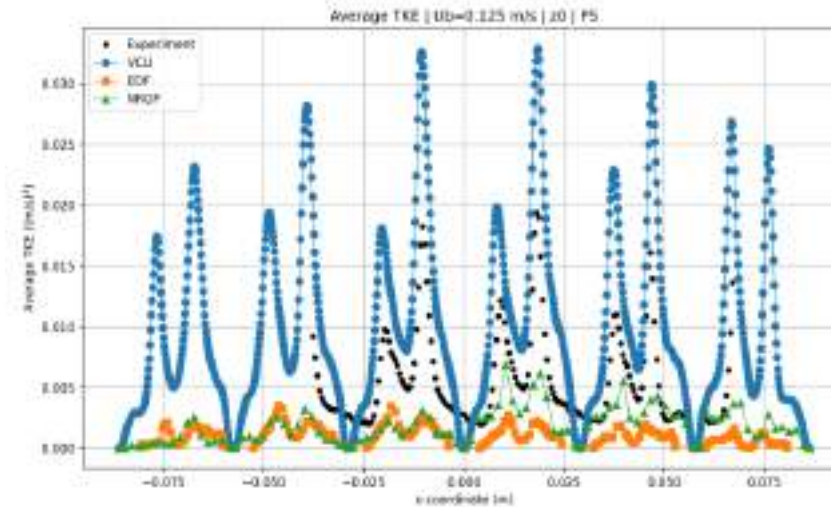
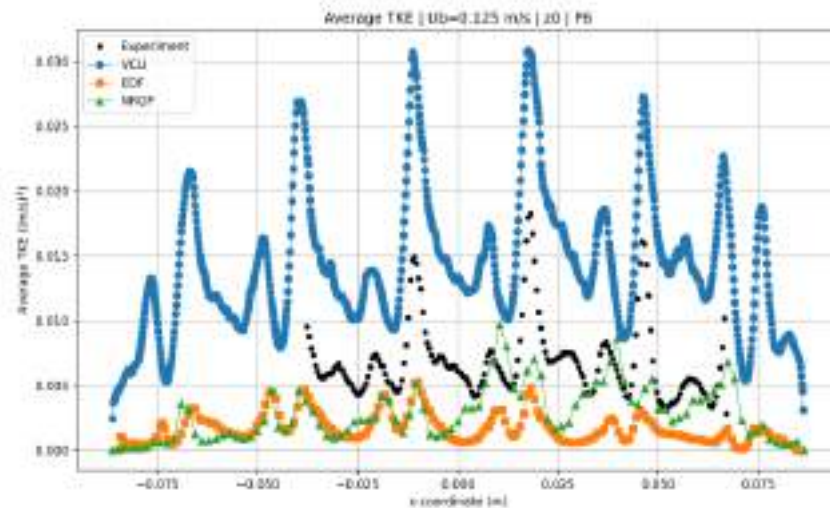
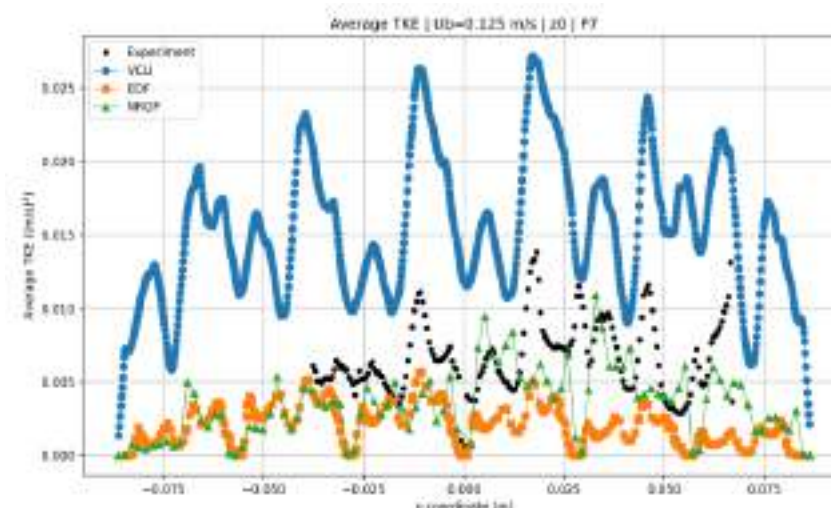
Outlet:
p11 = 170 mm
p10 = 150 mm
p9 = 130 mm
p8 = 110 mm
Bundle:
p7 = 0 mm
p6 = -4.4 mm
p5 = -8.8 mm
p4 = -18.8 mm
p3 = -28.8 mm
p2 = -33.2 mm
p1 = -37.6 mm



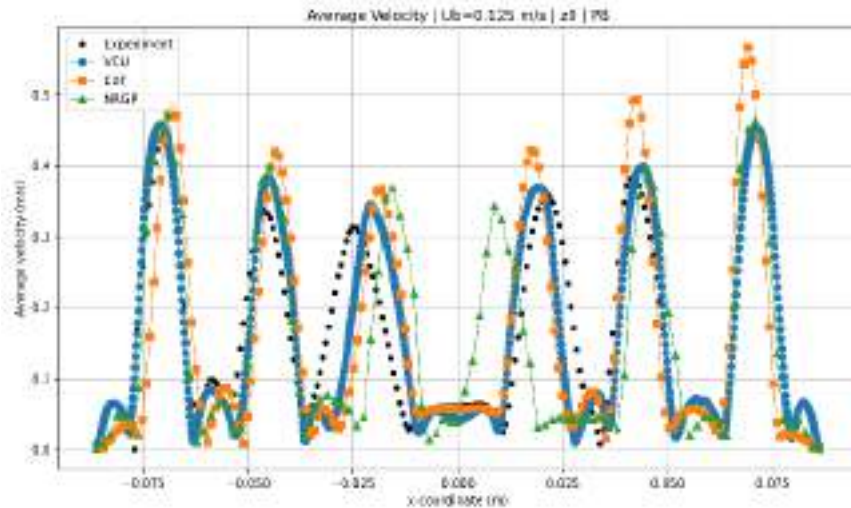
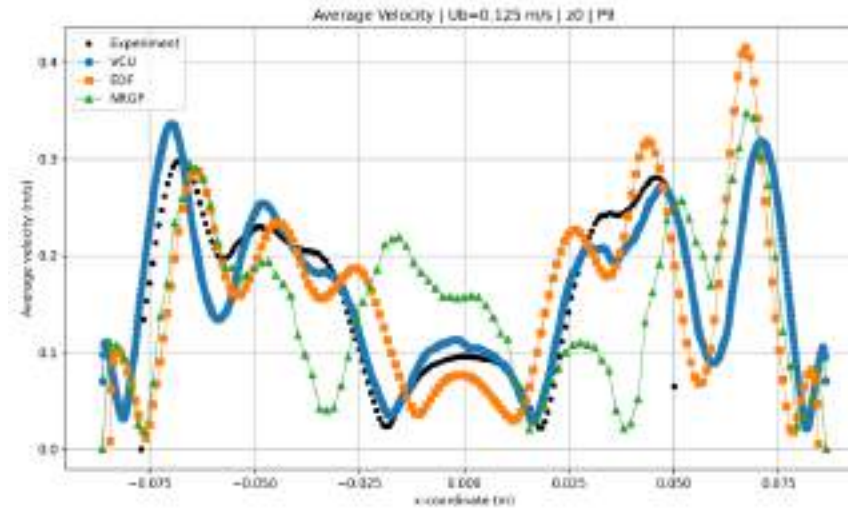
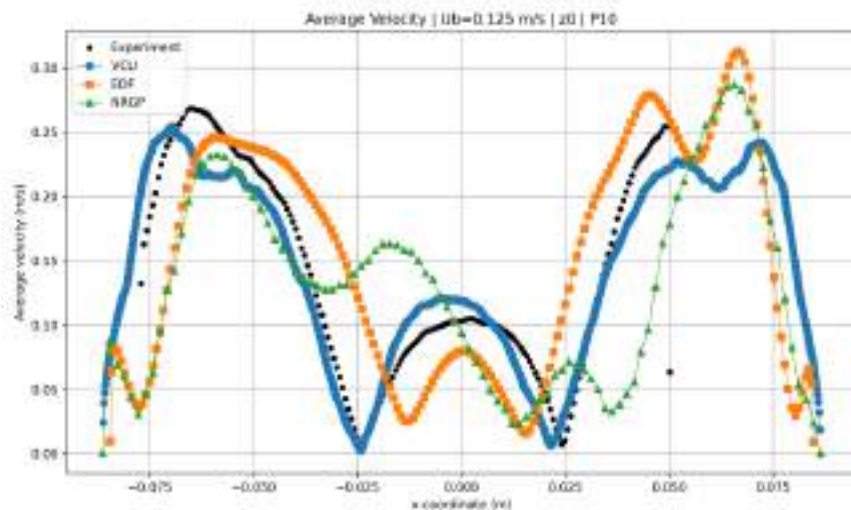
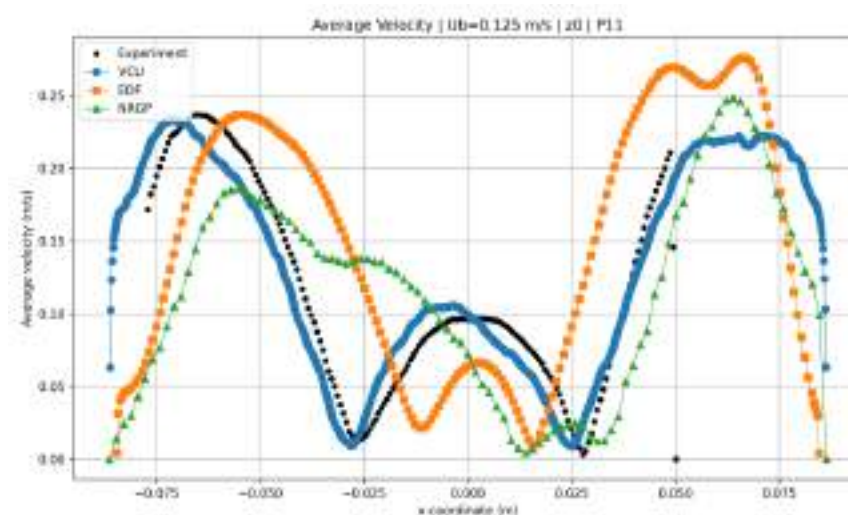
Results: bundle velocity profiles



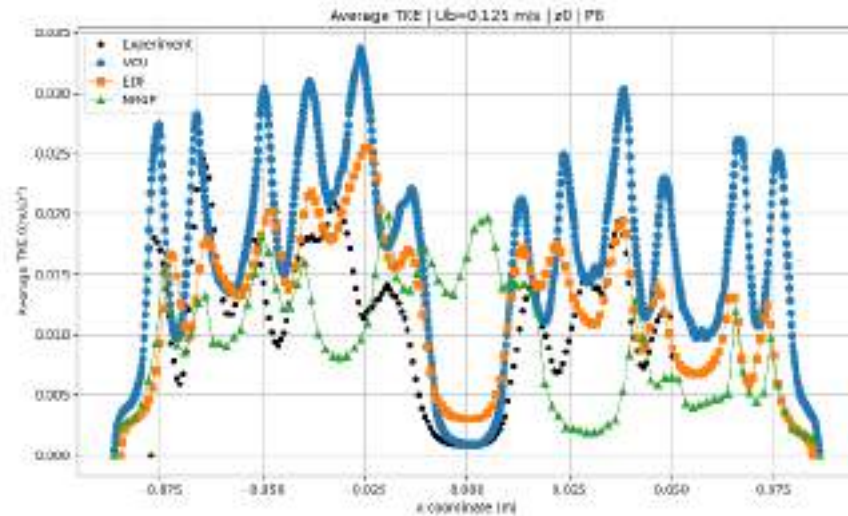
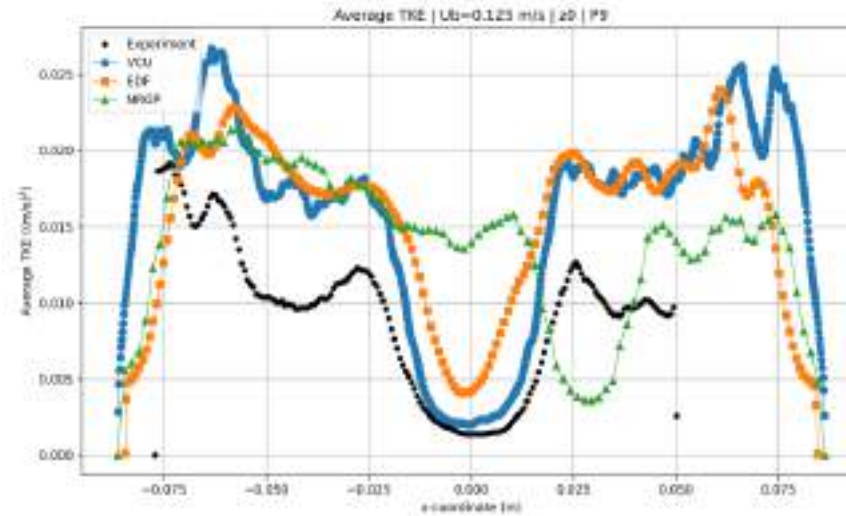
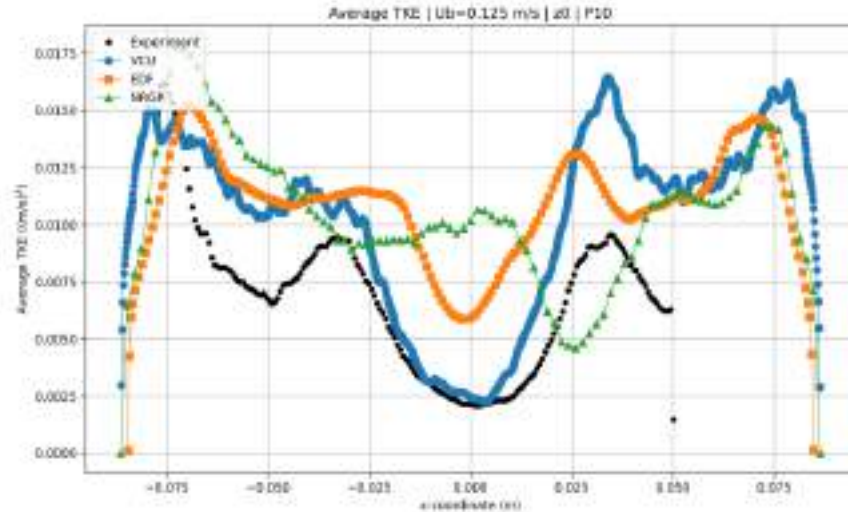
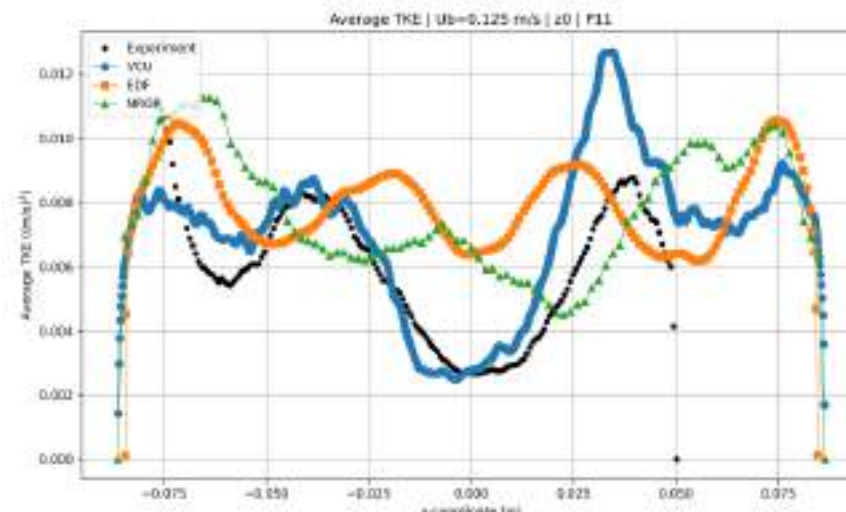
Results: bundle TKE profiles



Results: outlet velocity profiles



Results: outlet TKE profiles

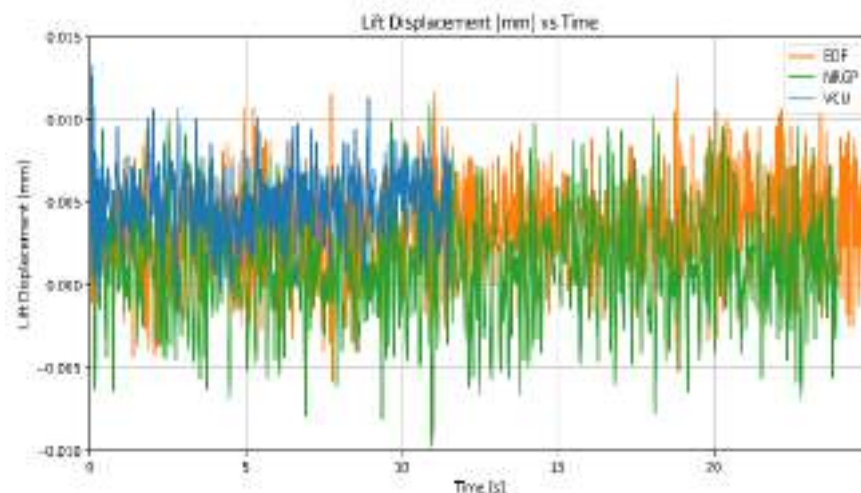
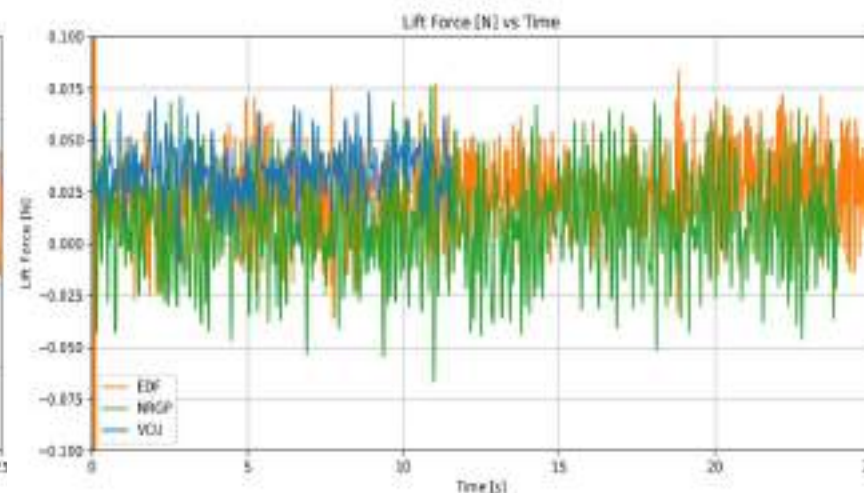


Results: FSI time series

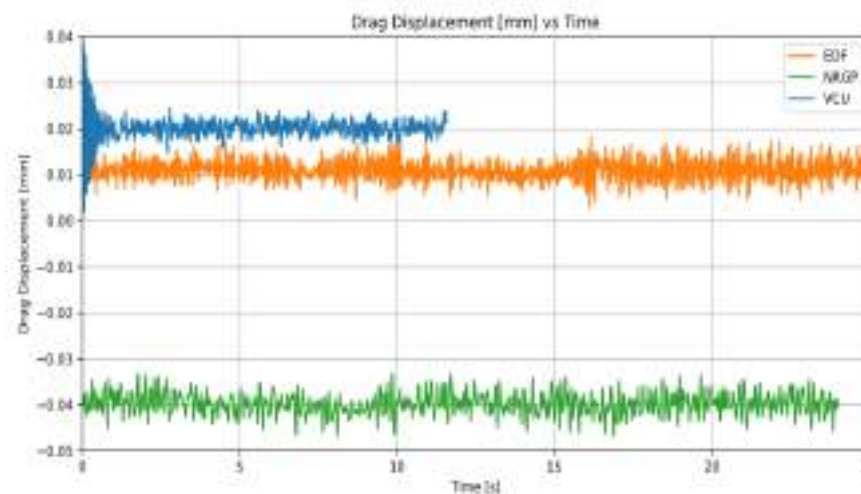
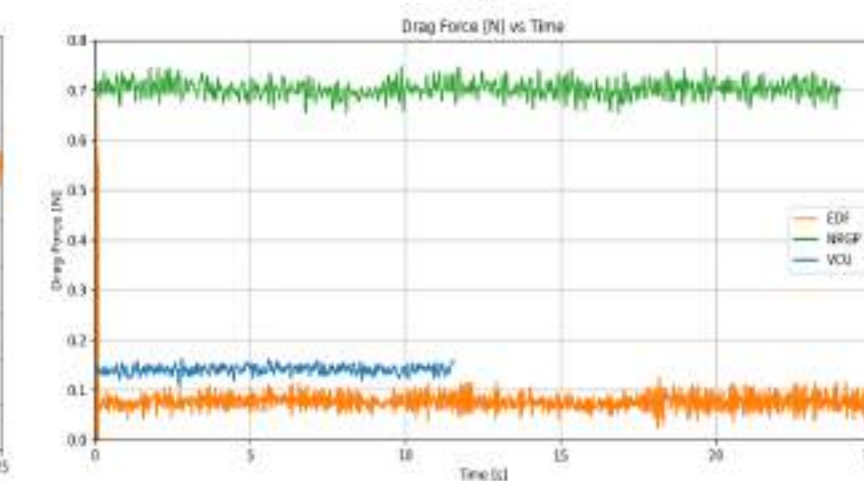
Displacement

Forces

Lift



Drag



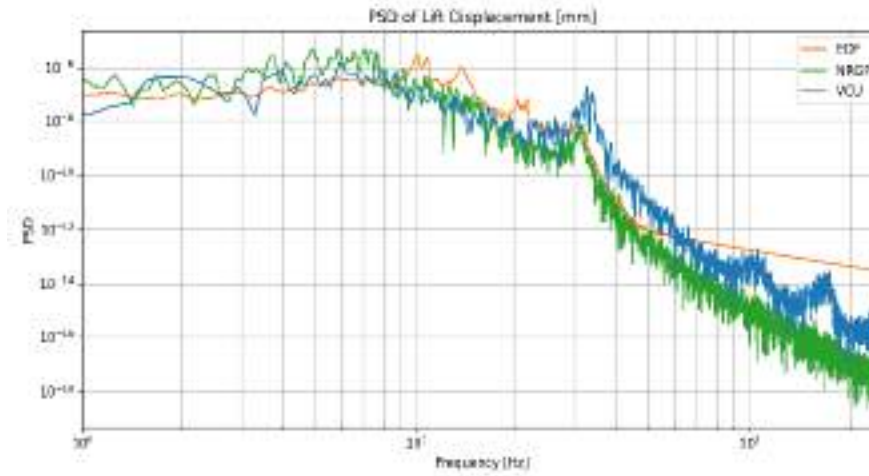
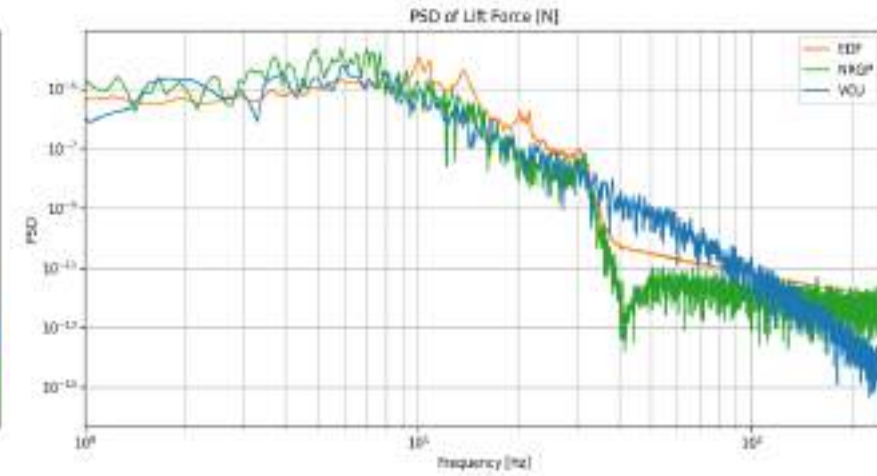
Results: FSI PSDs



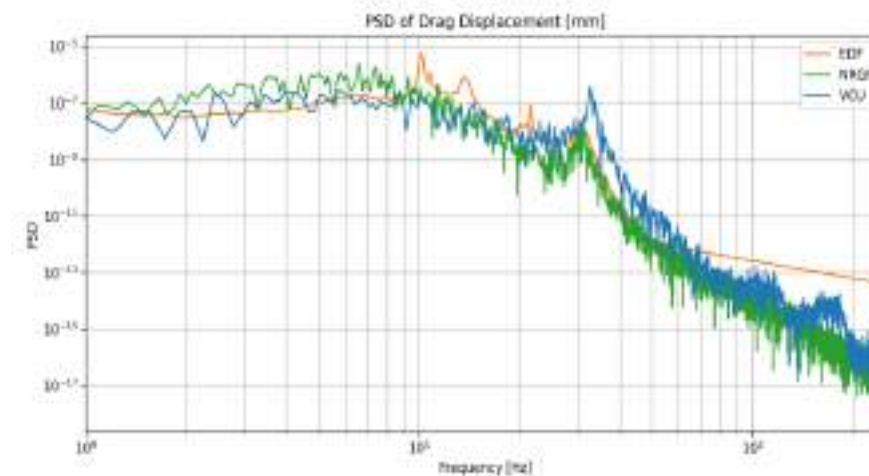
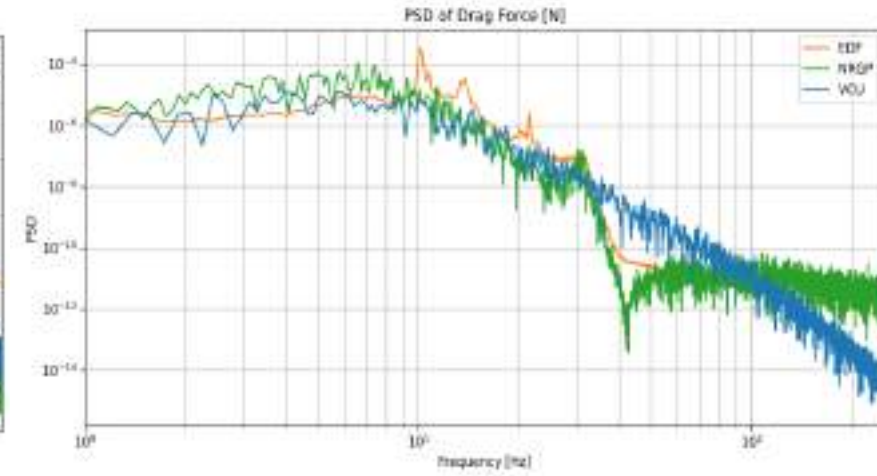
Displacement

Forces

Lift



Drag



Results: FSI qualitative comparison



U = 0.125 m/s	Frequency [Hz]		Mean displ. [mm]		RMS displ. [mm]	
	Lift	Drag	Lift	Drag	Lift	Drag
Experiment	N/A	N/A	-2.57E-2	6.43E-3	8.50E-3	8.32E-3
VCU	32	32	5.05E-3	2.01E-2	1.96E-3	2.18E-3
EDF	31	31	3.79E-3	1.10E-2	2.57E-3	2.34E-3
NRG PALLAS	31	31	1.25E-3	-3.98E-2	3.33E-3	2.30E-3

Conclusions



- Flow Field:
 - Using a high-resolution approach does improve predictions of the flow field, especially in regions of intense, locally-generated, turbulence such as that inside a tube bundle. This comes at a significantly increased computation cost, however.
 - For high-resolution approaches, it is important to have a properly generated inlet turbulent velocity field.
 - Medium-resolution approaches can give quite accurate predictions, as long as the inlet turbulence is properly prescribed based on known, experimental, data. Especially if anisotropic turbulence models are used. Predictions are further improved if the relevant turbulent scales are resolved.

Conclusions



- Vibration data:
 - Using a simple 2DoF rigid-body approach seems to be sufficient when the motion of the structure is largely in a 2D-plane without any significant deformation.
 - For cases with relatively low amplitudes, one-way approaches give similar predictions as two-way approaches, thereby avoiding the challenges of mesh morphing.
- General considerations:
 - For high-quality validation of numerical simulations, high-resolution, and CFD-grade, experimental data is needed. This means PIV measurements of the inlet region, to properly set the inlet conditions in the models, as well as in the regions of interest, to allow accurate comparison of the obtained results.
 - To properly assess FIV predictions, it is important to also have measurements of the forces on the moving cylinder.

Thank You!



- VKI: Experiment
 - Muhsin Can Akkurt, Mathieu Delsipée, Lilla Koloszar, Philippe Planquart
- VCU:
 - Trevor Franklin, Lane Carasik
- EDF:
 - Sofiane Benhamadouche

Partners



Associated Partners





GO-VIKING

Thank you!



K. Zwijsen



NRG PALLAS



kevin.zwijsen@nrgpallas.com

www.go-viking.eu



Funded by the
European Union



How GPUs Change Industrial CFD Simulation Strategies

Florian Menter
Synopsys Fellow

GPU – Power – The Driving Factor in CFD

- Impact on CFD code development
- Impact on Turbulence modeling
 - Stress-Blended Eddy Simulation (SBES)
 - Moving LES into the boundary layer
 - Options for handling boundary layers in LES and meshing requirements
- Alternative meshing strategies
 - Bottom-up versus Top-Down meshing
- Applications
 - Automotive
 - Aeronautics
 - A look back at RANS mesh Adaptation

New Ansys GPU solver

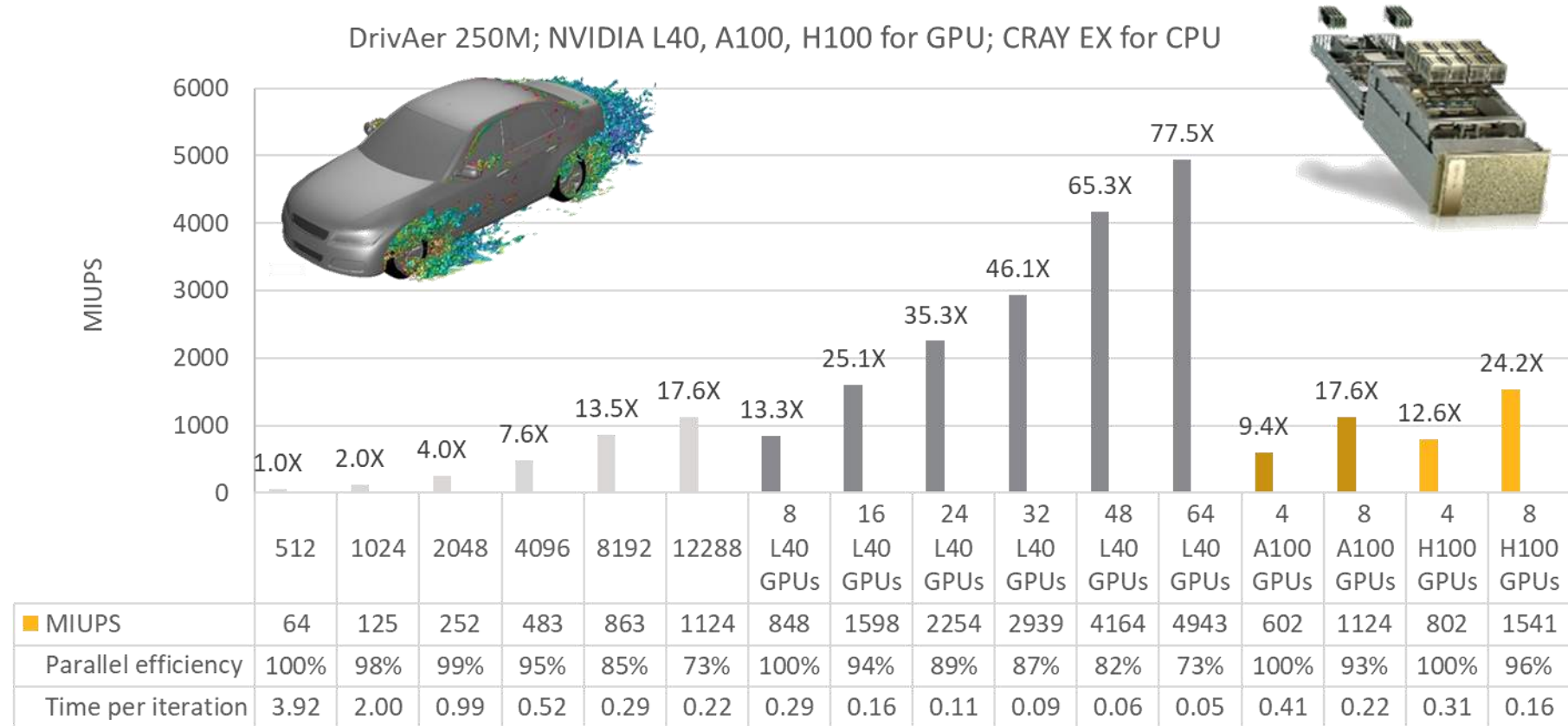
- General:
 - New solver (no off-loading of existing solver loops)
 - 2nd order cell centred FV
 - 2nd order implicit time stepping (Gear's method)
 - Incompressible discretization using SIMPLEC
- LES runs:
 - **Skew symmetric formulation** (central reconstruction)
 - Wall function uses 2nd cell off-wall as input
- SBES runs:
 - Apr. 4th Order dissipation (optional), WENO-type smoothing
 - Face flow rate: central
- Density Based Solver in progress



Native GPU implementation shows astounding performance gains



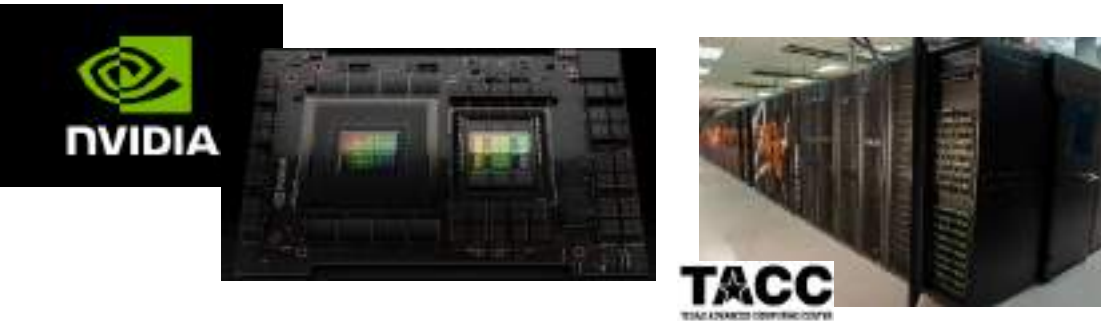
- CPU
 - CRAY EX
 - AMD EPYC 7763 64-Core
 - 128 cores per node
- GPU
 - Cluster 1
 - NVIDIA L40, 4 GPUs per node, Infiniband
 - Cluster 2
 - NVIDIA A100 80GB, 8 GPUs per node
 - Cluster 3
 - NVIDIA H100 80GB, 8 GPUs per node
- With 64 L40 GPUs
 - 0.05 seconds per iteration
 - 2 iterations per step
 - 250 million cells
 - 36,000 time steps in 60 minutes



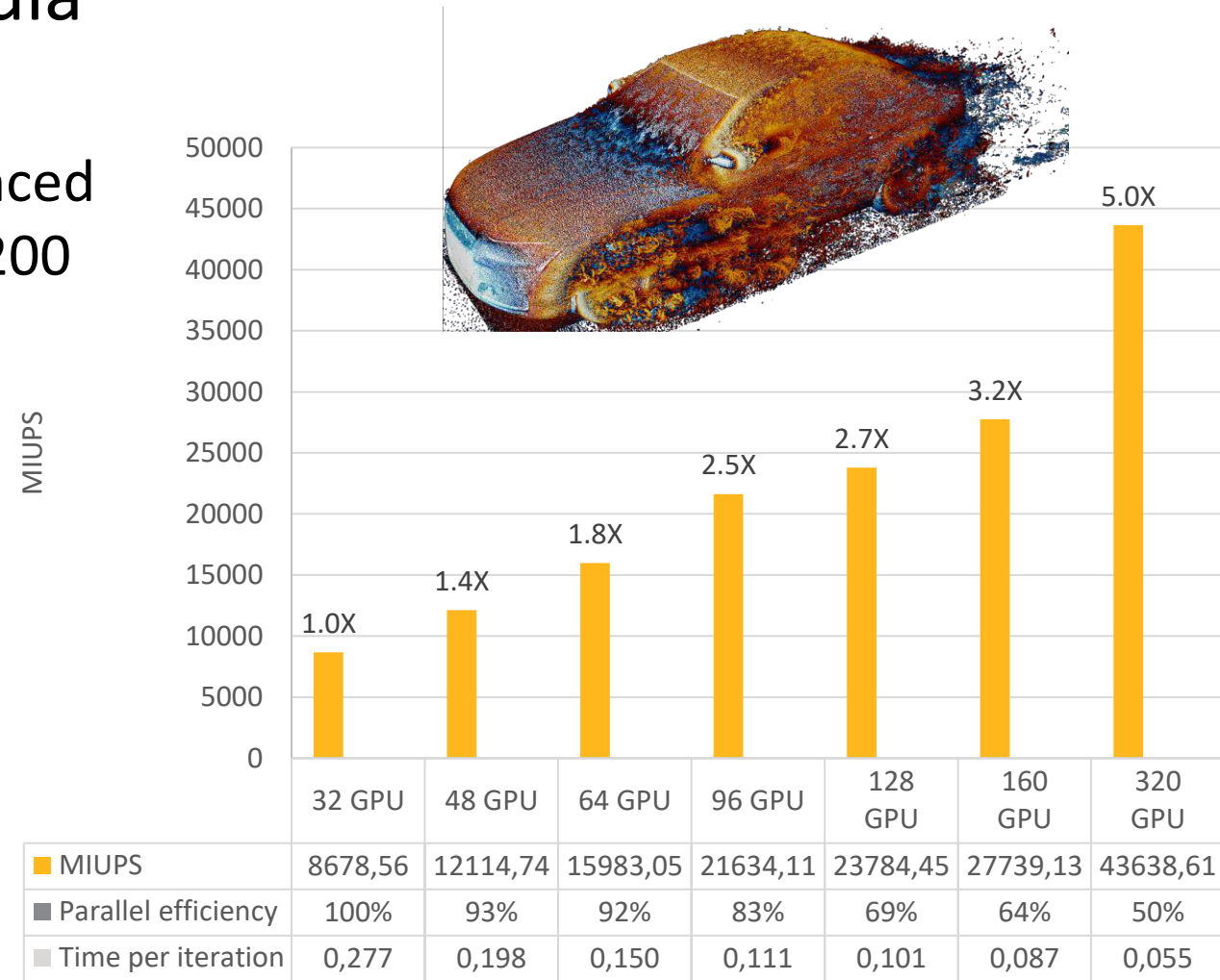
A single H100 GPU ≈ 1,500 CPU cores

Compute Capacity @ TACC on Nvidia

- Simulation on Vista system at Texas Advanced Computing Center (TACC) with Nvidia GH200



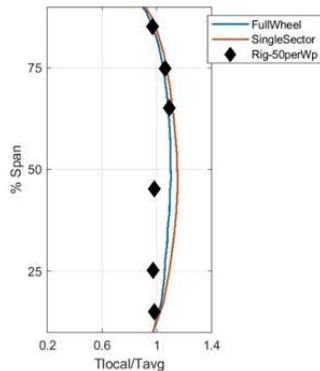
- Astounding performance
- Still scaling to at least 320 GPU
 - DrivAer case with **2.4B elements**
 - 180.000 Δt**
 - Becoming 'too small' $\Rightarrow$ efficiency drops



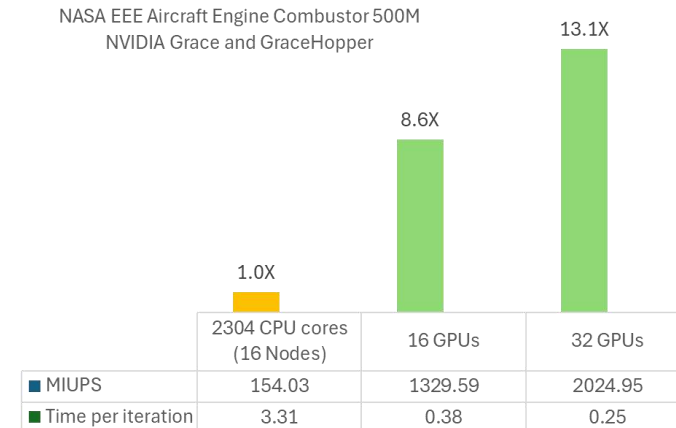
From 29 days (2048 cores) $\rightarrow$ 6 hrs (320 GPUs)

Full Annular Combustor Study with 500M Cells

- Energy Efficient Engine (EEE) Combustor
- 500M Rapid Octree Mesh
- Large Eddy Simulation with optimized numerics
- Flamelet generated manifold (FGM) combustion model
- Partition tracking for liquid fuel spray with SSD Break-up model

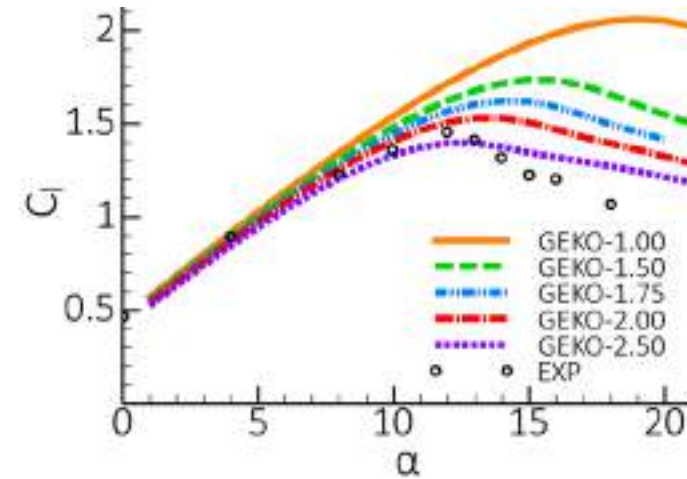
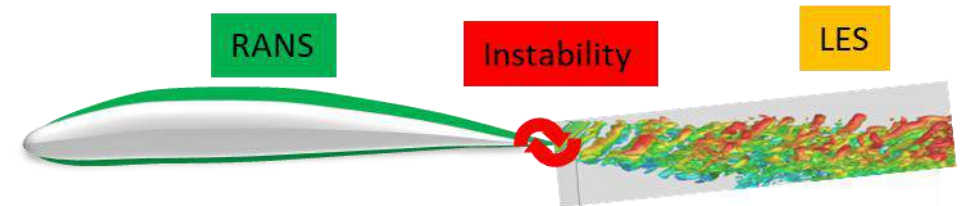


- Ansys Fluent ARM64 @ TACC
 - NVIDIA Grace CPU Superchip
 - NVIDIA GH200 Grace Hopper Superchip
- Performance
 - 92 hours to solve on 2304 CPU cores (16 nodes) ~ 1GPU=1240CPUs
 - **10.5 hours to solve on 16 Grace Hopper GPUs for 50,000 time steps**



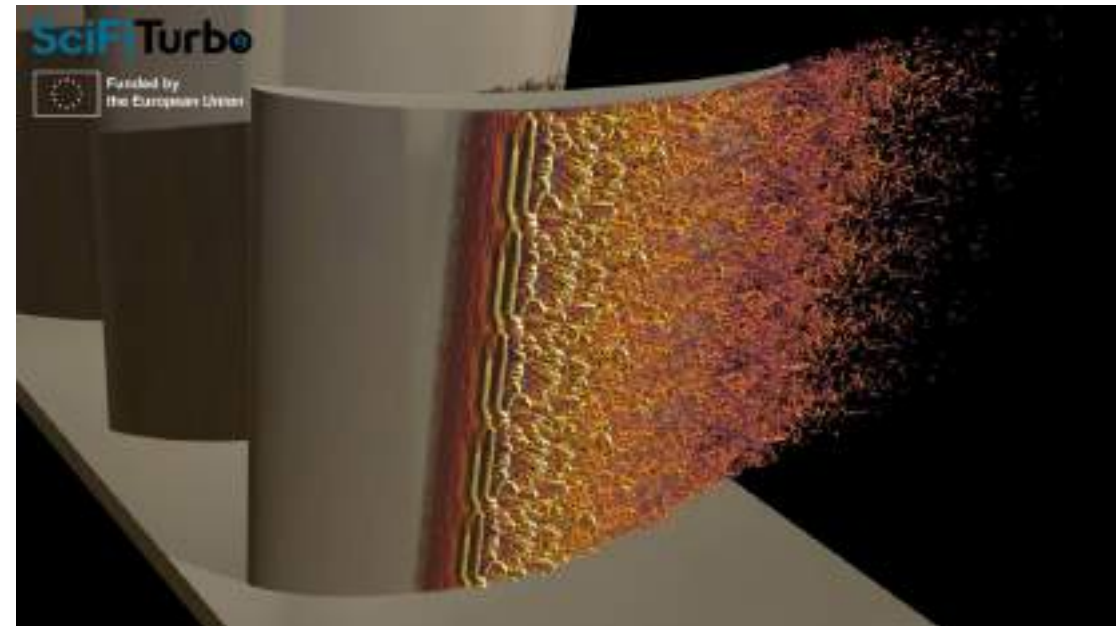
Turbulence Models

- RANS Modeling
 - Generalized k- ω (GEKO) model
 - Algebraic laminar turbulent transition model
- Hybrid RANS-LES model
 - Stress-Blended Eddy Simulation (SBES) model
 - Can be combined with GEKO
- LES Models
 - Wall Function LES (WFLES)
 - Wall-Modeled LES (WMLES)



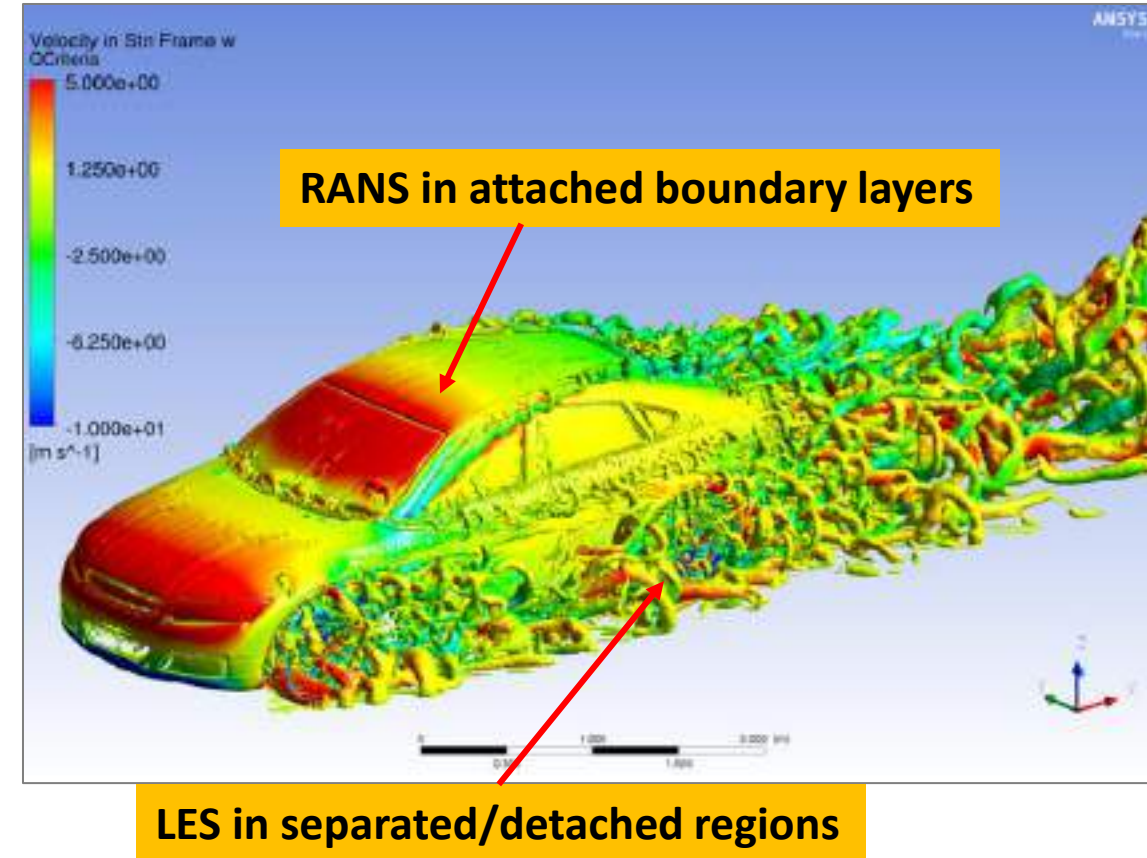
Scale Resolving Simulations (SRS) of Boundary Layer Flows

- LES in industrial flows was mostly used for free shear flows
 - Jets, Mixers, Combustion chambers...
- LES of boundary layers only industrially feasibly with GPU clusters
- However Wall-Resolved LES (WRLES) still too expensive
- Different approaches to bridge the resolution gap near the wall needed



Motivation for Hybrid RANS-LES Methods

- Wall-bounded Flows:
 - The cost of LES for wall-bounded flows is excessive
 - The accuracy of RANS models for pure wall boundary layers is often good / acceptable
- Free shear flows
 - LES for free shear flows is much less costly than for wall-boundary layers
 - For complex free shear flows, RANS models often fail dramatically and can even result in incorrect flow topology
- Spalart 1997 Detached Eddy Simulation (DES)
 - Combine RANS and LES into hybrid RANS-LES models



Geometry: Courtesy Inst. of Aerodynamics and Fluid Dynamics, TU Munich, T. Frank, B. Gerlicher, J. Abanto, ASWC 2013

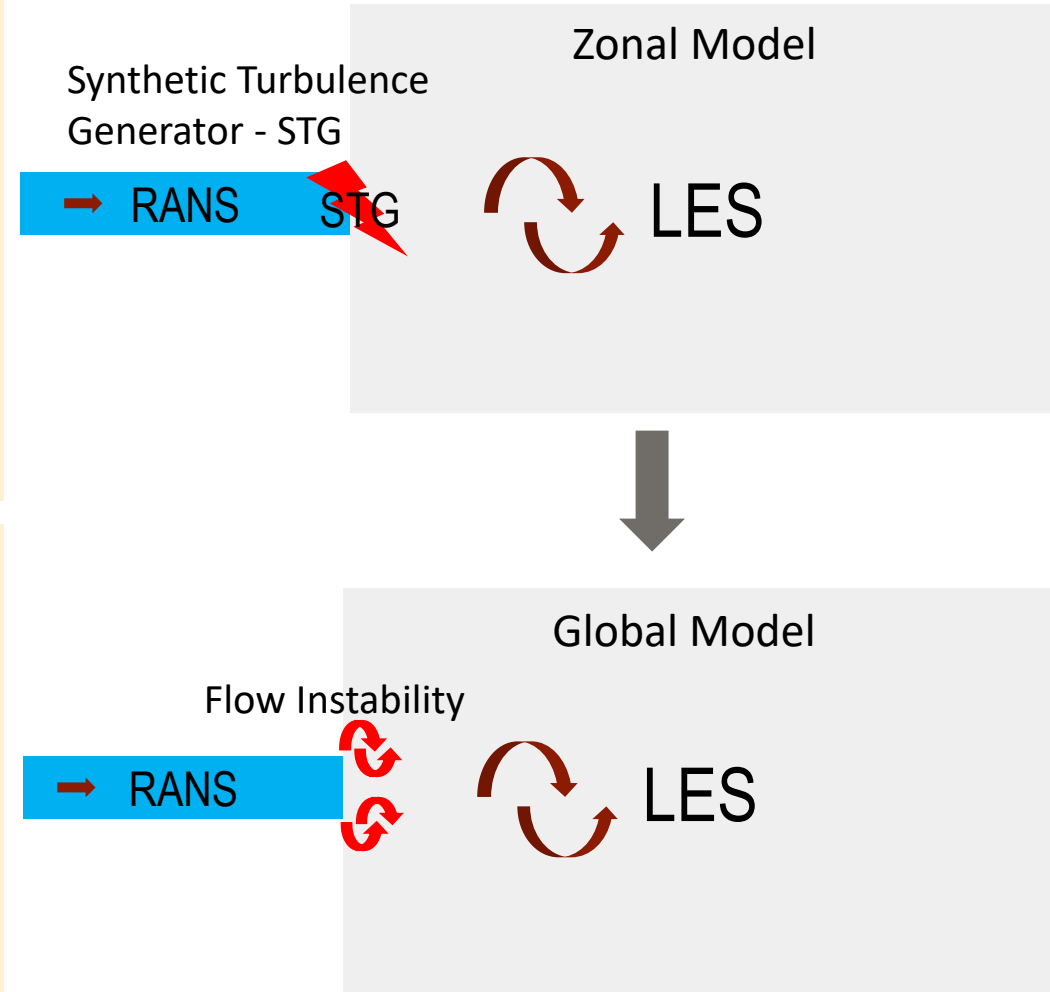
Zonal Versus Global RANS-LES Hybrid Methods

Zonal RANS-LES:

- User selects RANS and LES models
- User selects RANS and LES zones
- User defines interface where Synthetic Turbulence Generator (STG) transfers modelled RANS turbulence into resolved LES turbulence
- Examples
 - Embedded LES (ELES)
- Complex set-up – error prone

Global Hybrid Models:

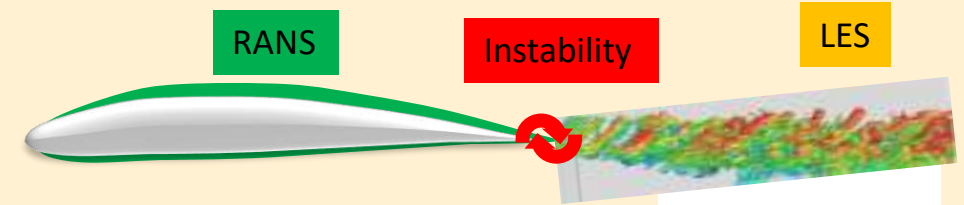
- User selects global hybrid model (SAS, DES, SBES, ...)
- Global hybrid model selects RANS and LES zones during run-time
- Resolved turbulence is generated by flow instability in separating shear layer
- User set-up much simpler (similar to RANS)
- Examples
 - SAS, DES, DDES, SDES, SBES



Global Hybrid RANS-LES Model Scenarios

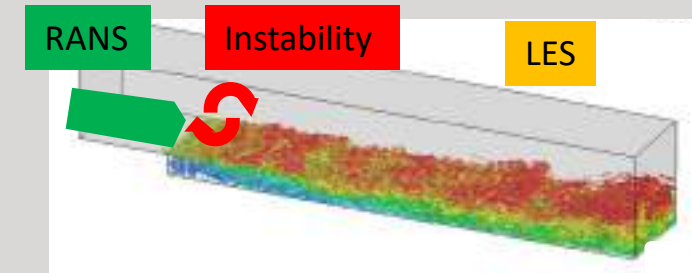
- Flow detaches at corner

- Boundary layer in RANS mode
- Free shear layers in LES



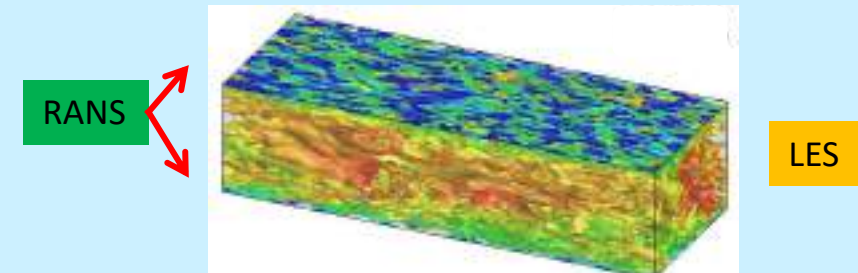
- Flow Separates and reattaches

- Flow should remain in LES downstream of step
- Recover to RANS on coarse grids downstream



- Flow is Wall bounded

- Should hybrid model do WMLES?
- Yes – if there is sufficient resolution



Detached Eddy Simulation (DES) for SST – Strelets (2000)

- k-equation RANS

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{U}_j k)}{\partial x_j} = P_k - \rho \frac{k^{3/2}}{L_t} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\tilde{\sigma}_k} \right) \frac{\partial k}{\partial x_j} \right] \quad L_t = \frac{\sqrt{k}}{\beta^* \omega}$$

- k-equation LES

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{U}_j k)}{\partial x_j} = P_k - \rho \frac{k^{3/2}}{C_{DES} \Delta} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\tilde{\sigma}_k} \right) \frac{\partial k}{\partial x_j} \right] \quad \Delta = \max(\Delta x, \Delta y, \Delta z)$$

- k-equation DES

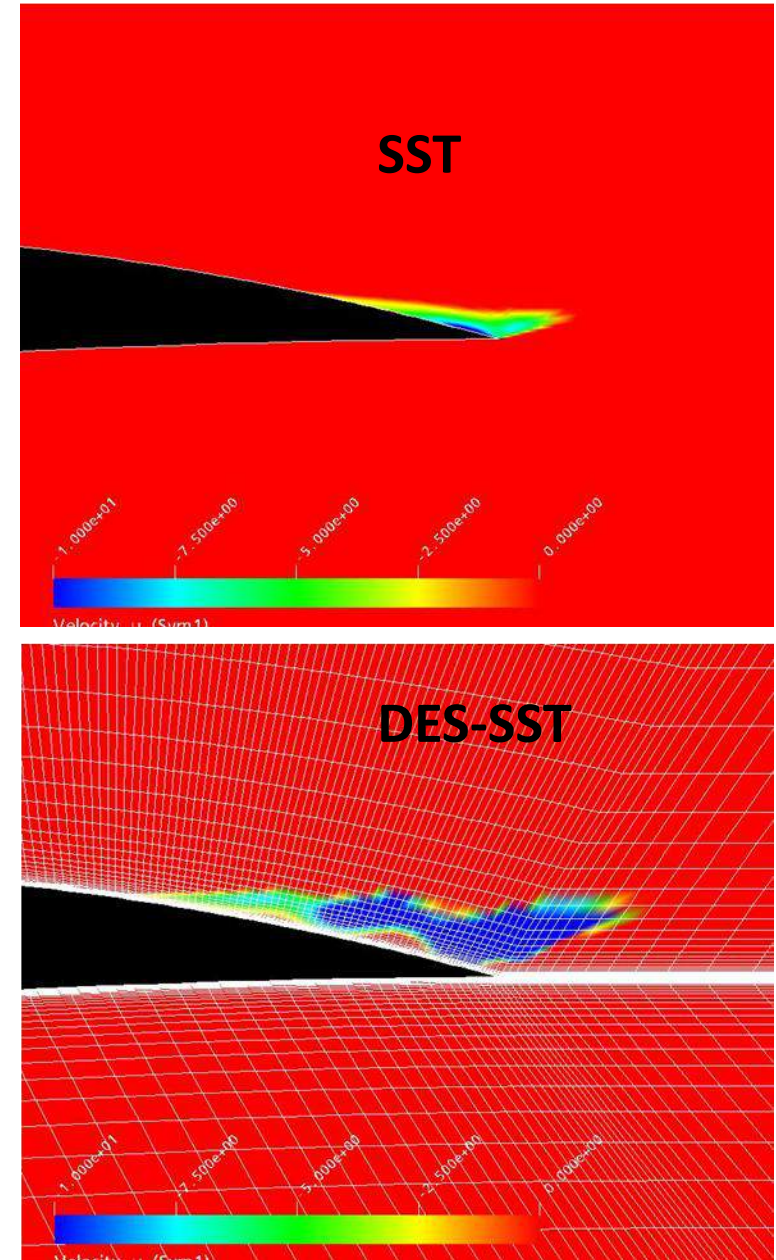
$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho \bar{U}_j k)}{\partial x_j} = P_k - \rho \frac{k^{3/2}}{\min(L_t; C_{DES} \Delta)} + \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\tilde{\sigma}_k} \right) \frac{\partial k}{\partial x_j} \right]$$

$$\varepsilon_{DES} = \frac{k^{3/2}}{\min(L_t, C_{DES} \Delta_{\max})} = \varepsilon_{RANS} F_{DES}; \quad F_{DES} = \max\left(\frac{L_t}{C_{DES} \Delta_{\max}}, 1\right)$$

- DES formulation is very simple. It introduces grid spacing Δ into k-equation. In case $L_t > C_{DES} \Delta$ the model switches to an LES formulation
- Grid dependency is dangerous, as it can negatively affect the RANS model

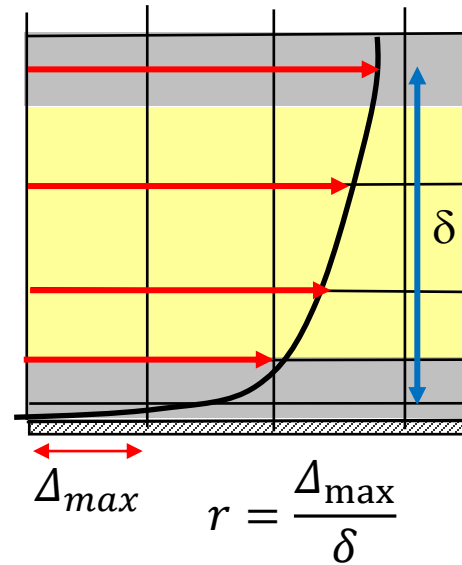
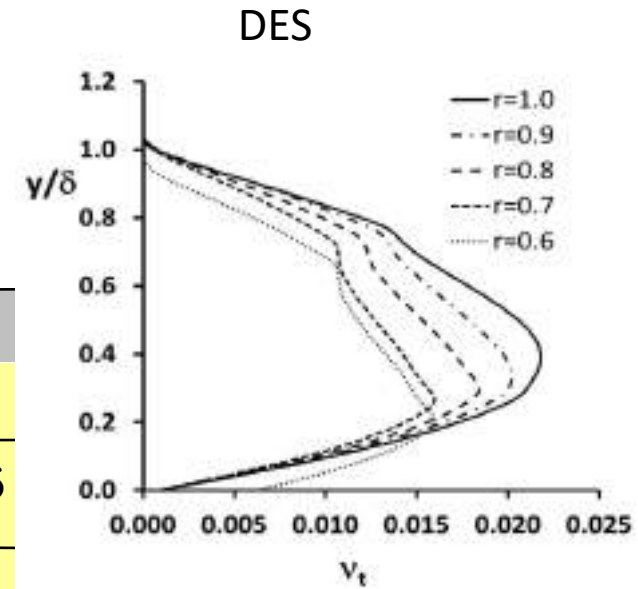
Grid Induced Separation for NACA 4412 Airfoil

- SST model gives good estimate for separation point
- DES-SST model moves separation point to upstream location due to grid impact on RANS solution – Grid-Induced Separation (GIS)
- GIS depends entirely on the grid – not acceptable!
- One of the reasons for further developments of DES model:
 - Menter and Kuntz proposed boundary layer shielding to reduce impact of D in boundary layer. Shielding was based on SST functions
 - Spalart and Strelets proposed similar function for Delayed Detached Eddy Simulation (DDES)

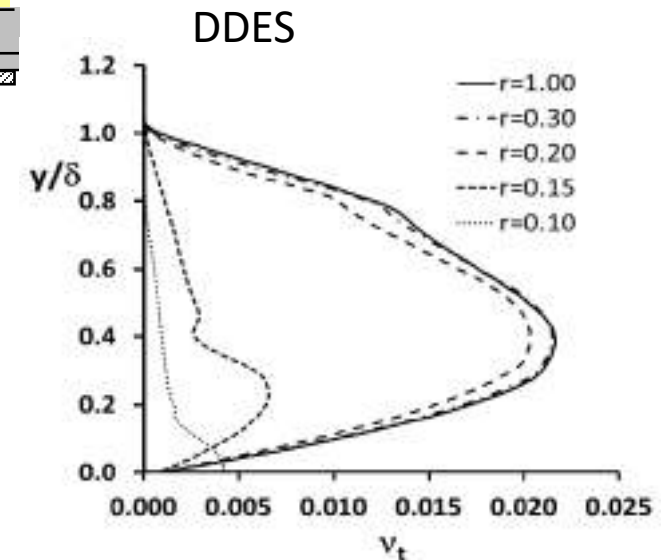


DDES Shielding

- Combination of natural shielding and shielding function
- Impact of grid refinement is “delayed” to about $\Delta_{max} < 0.2 \delta$ (δ – boundary layer thickness)
- However, impact of grid is much more drastic
 - Marginal change of refinement leads to strong change in eddy-viscosity
 - Non-linear feedback from DES term on shielding function
 - Exaggerated by self-similar boundary layer equations – assuming infinite running length of boundary layer



DES
deteriorates
for $r \sim 1$



DDES
deteriorates
for $r \sim 0.25$

Goals for Optimal Global Hybrid RANS-LES Model

- Perfect Shielding
 - Boundary layer unaffected by LES grid impact for any grid refinement
- Building block choice for RANS and LES models
 - User can select RANS and LES model separately – different from DES/DDES where they are embedded into the RANS model
- Clear indication of RANS and LES regions
 - Blending function which shows user which mode of the model is used in which region of the flow
- Fast “Transition” from RANS to LES
 - As “RANS-LES Transition” is non-physical, optimal model quickly transitions between modes (from steady RANS to 3D LES turbulence)

Formulation of SBES

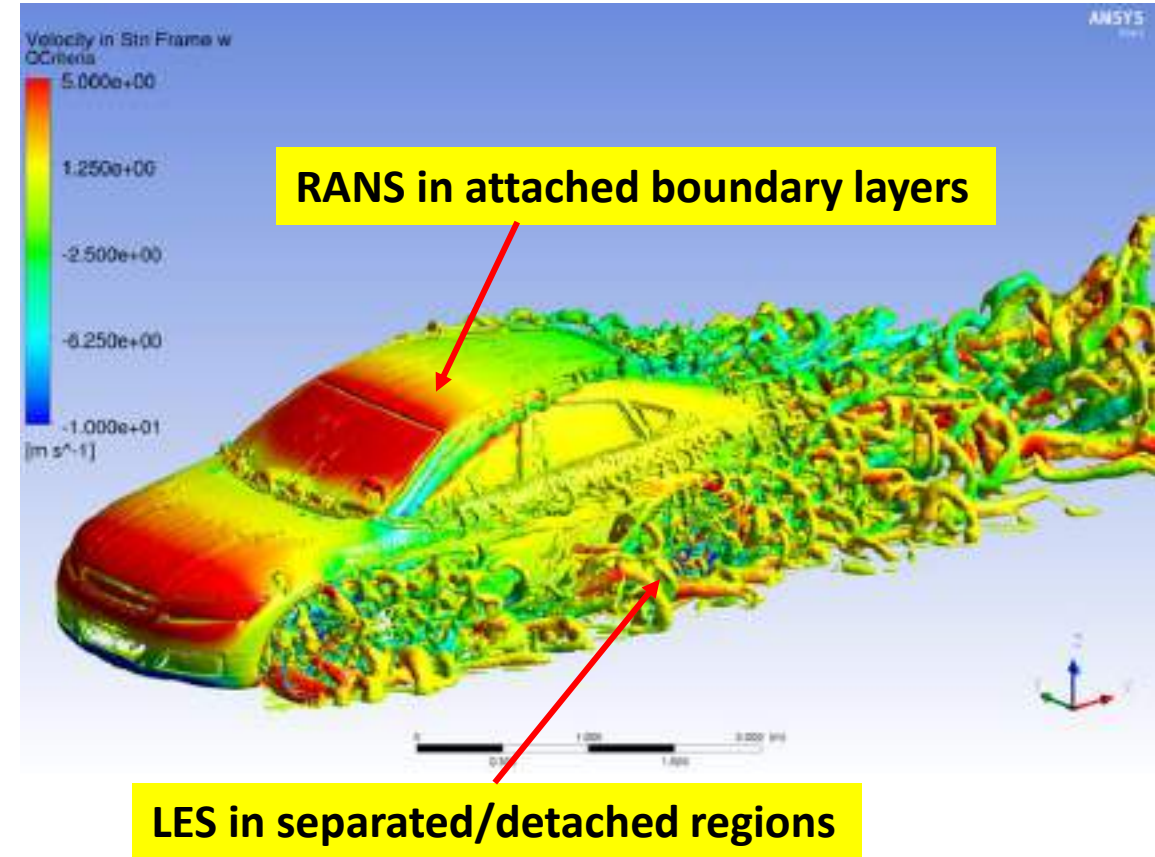
- Blending between RANS and LES:

$$\tau_{ij}^{SBES} = \tau_{ij}^{RANS} \cdot f_{SBES} + \tau_{ij}^{LES} (1 - f_{SBES})$$

- For Eddy Viscosity models that translates to:

$$\nu_t^{SBES} = \nu_t^{RANS} \cdot f_{SBES} + \nu_t^{LES} (1 - f_{SBES})$$

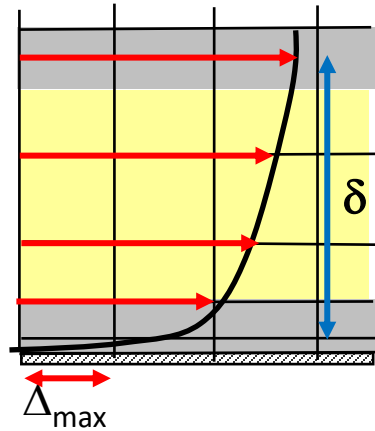
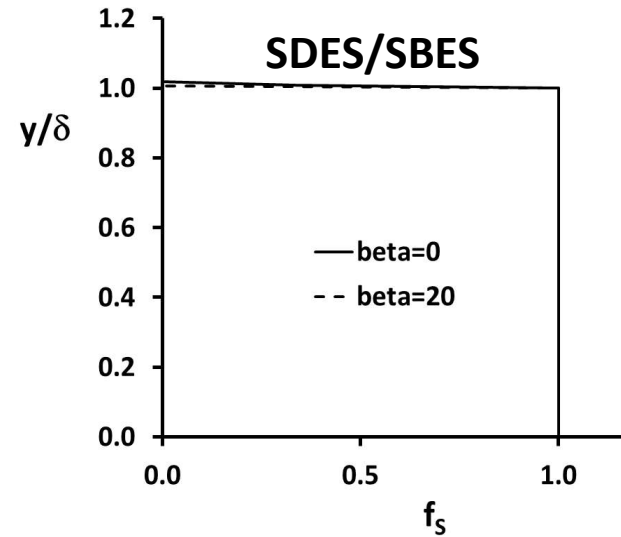
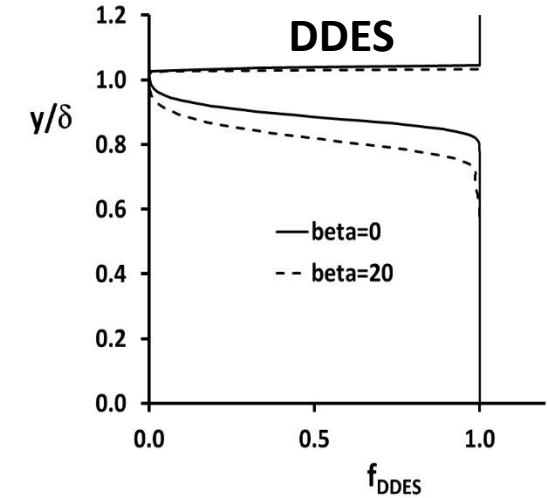
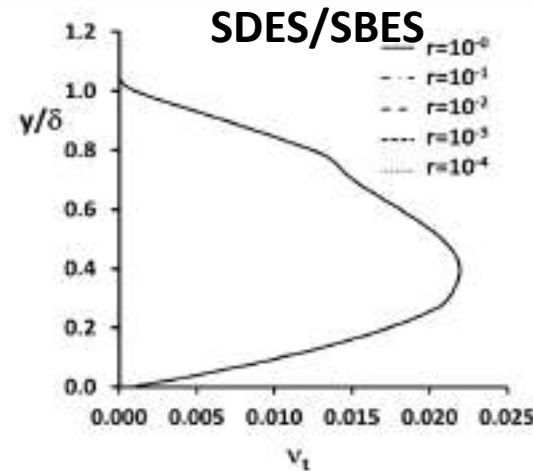
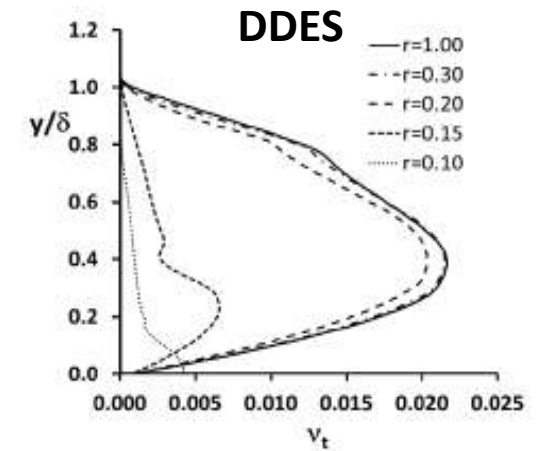
- The underlying RANS and LES models are unaffected
- In principle, and RANS model can be combined with any alg. LES model - modular
- All complexity is in the blending function f_{SBES}



Improved Shielding Function for SDES/SBES Models

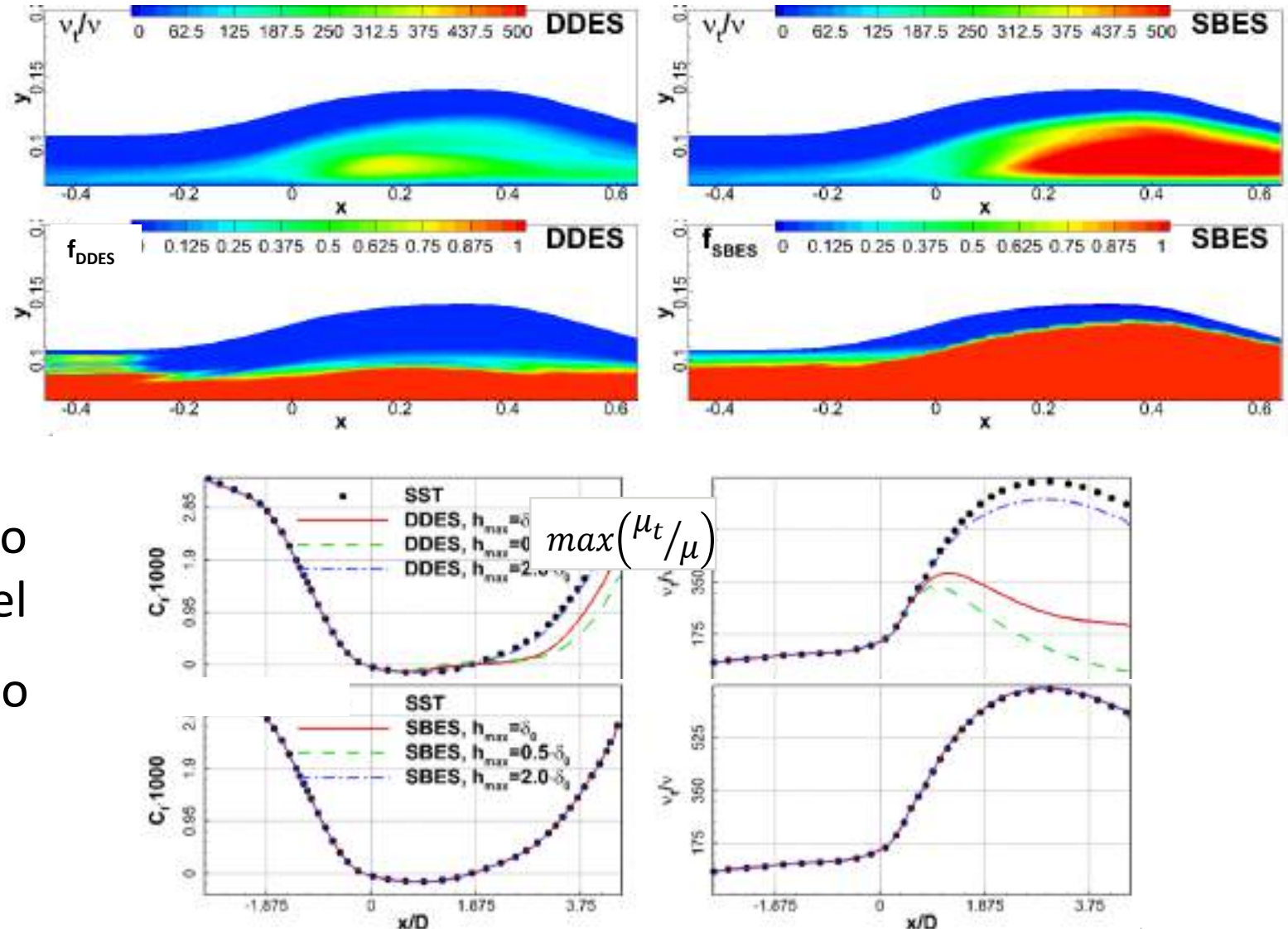
- New shielding function covers entire boundary layer
- Shielding down to very fine mesh spacing
- No GIS in new SBES/SDES models

$$r = \frac{\Delta_{\max}}{\delta}$$



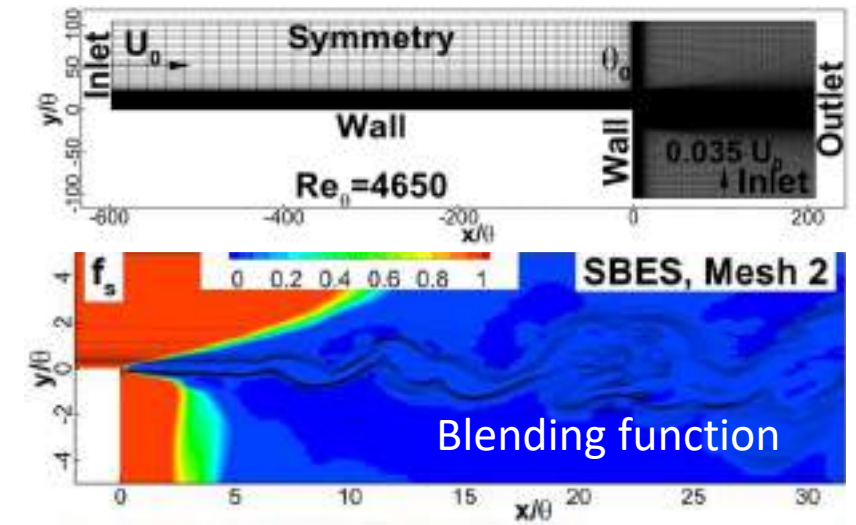
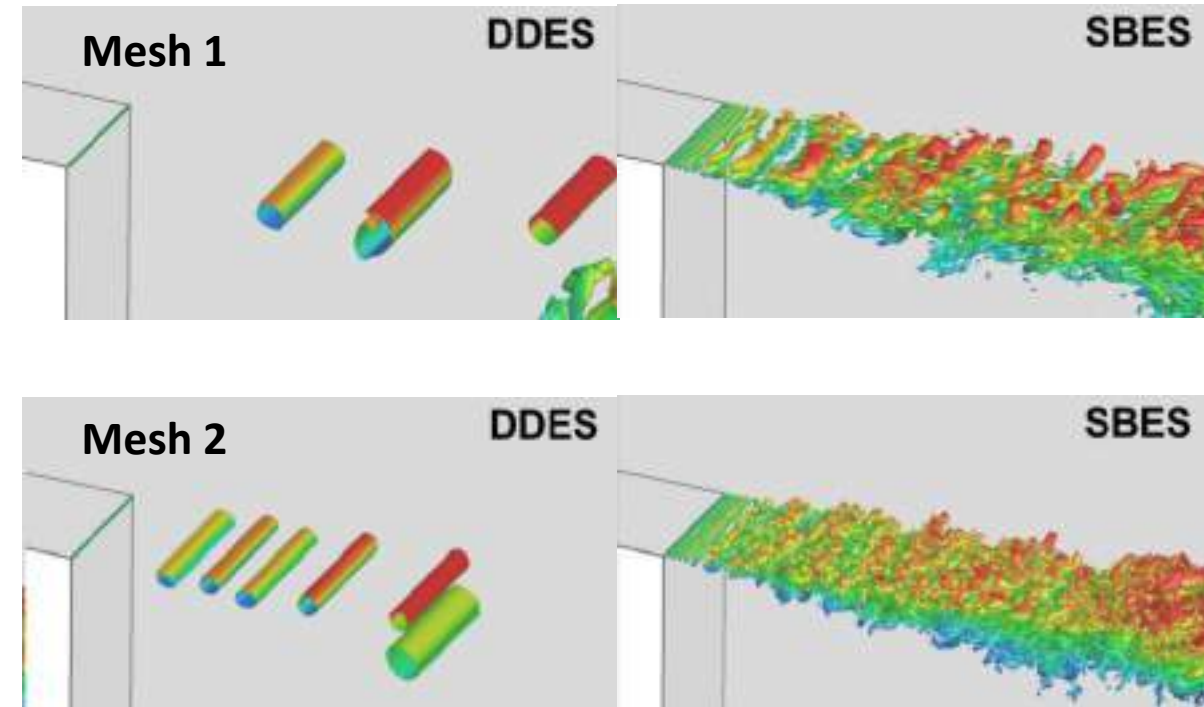
Shielding Test Cases: CS0 Diffuser

- For diffuser the DDES model fails pronouncedly. The shielding function does not grow with the boundary layer
- The SBES function follows the growth of the boundary layer
- The SBES solution is identical to the underlying SST RANS model
- Most new “DES”-like models do not pay enough attention to shielding



Mixing Layer from Corner

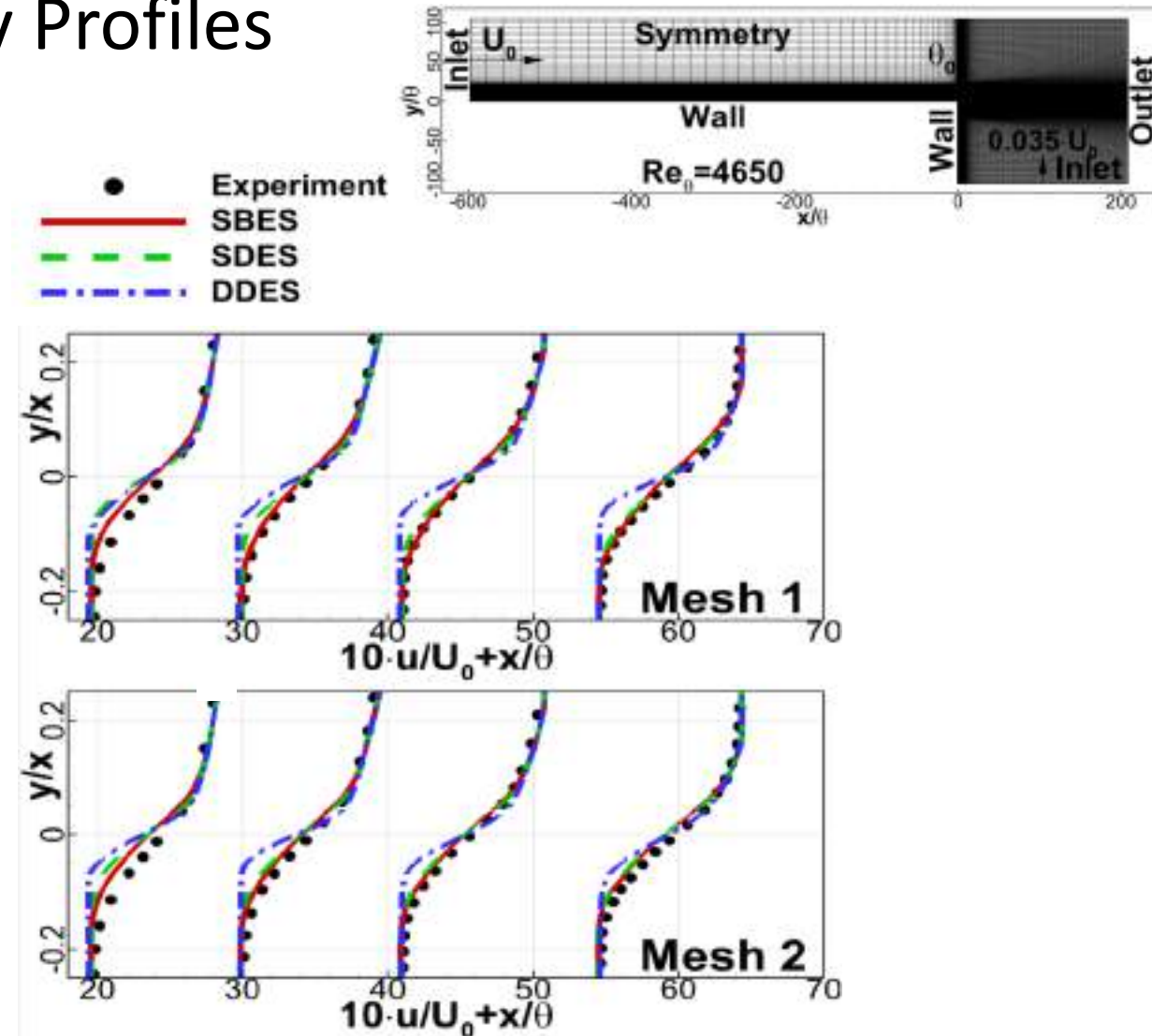
- Set-Up:
 - Mesh 1: $2.1 \cdot 10^6$ cells
 - Mesh 2: $7.6 \cdot 10^6$ cells
 - Reynolds number based on momentum thickness at separation point $Re_\theta = 4650$.
 - Domain size of mixing layer zone $(208 \times 104 \times 41)\theta$
- The SBES blending function covers the upstream boundary layer in RANS mode (SST-red) and switches quickly to LES mode (WALE-blue) at the separation point
- RANS-LES “Transition” much faster with SBES compared to DDES
- SBES can switch to LES on a much coarser mesh than DDES



S. C. Morris and J. F. Foss, “Turbulent boundary layer to single-stream shear layer: the transition region”, Journal of Fluid Mechanics Fluid Mech., vol. 494, pp. 187–221, 2003.

Single Stream Mixing Layer: Velocity Profiles

- Mixing of the velocity profiles much better with SBES than DDES
- SDES – is an improved DDES model using the SBES shielding function and lower eddy-viscosity

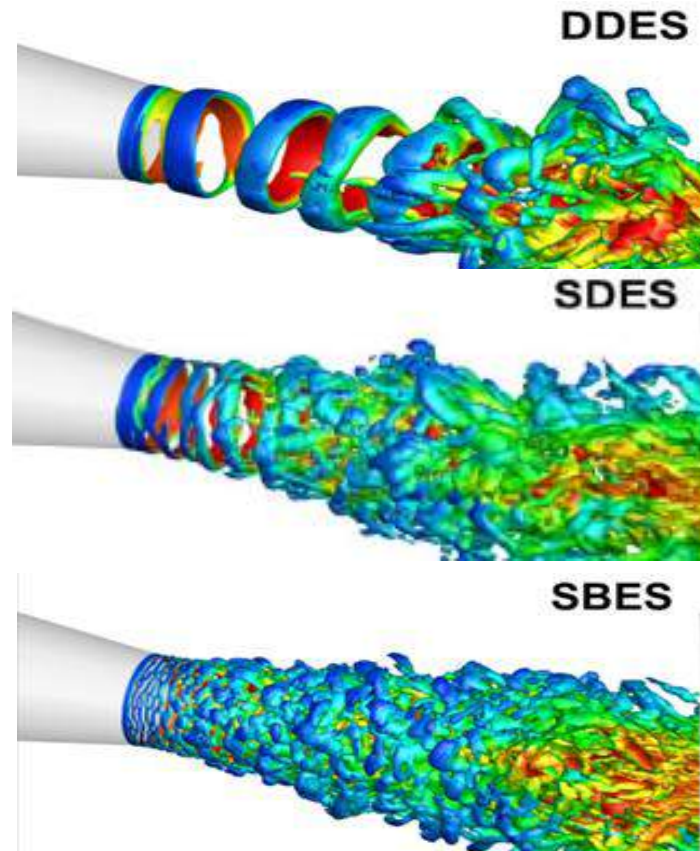
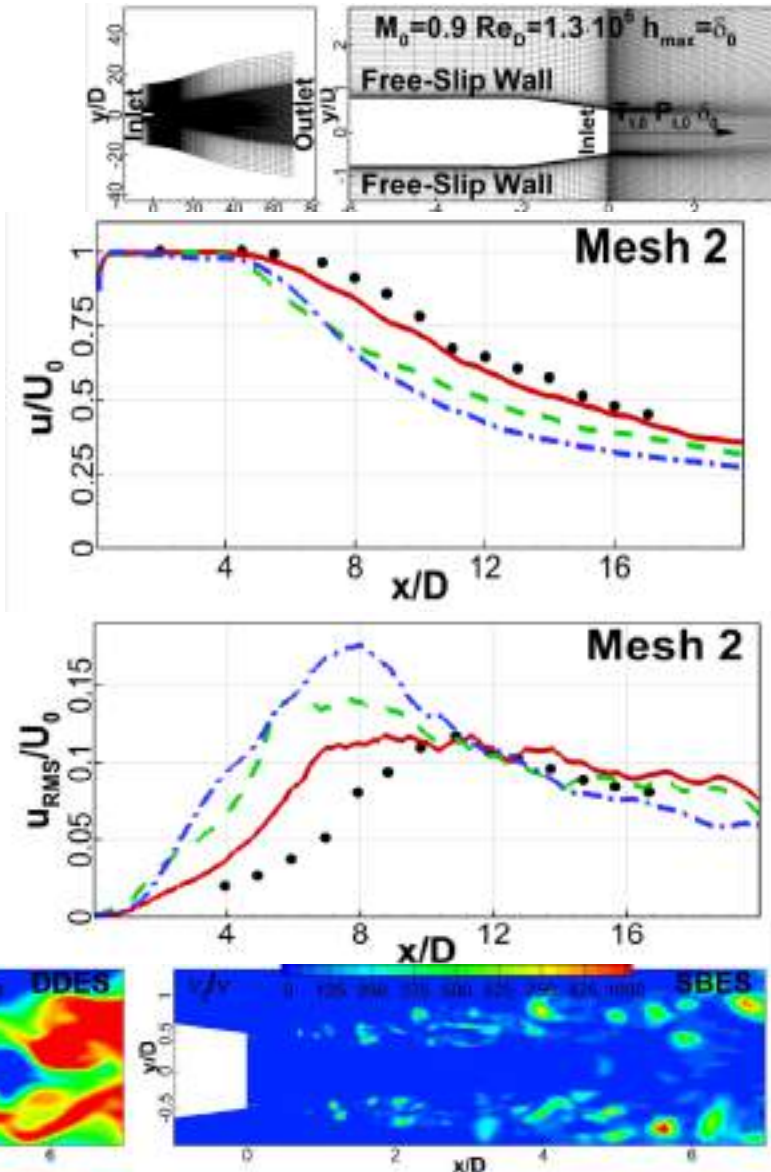


S. C. Morris and J. F. Foss, "Turbulent boundary layer to single-stream shear layer: the transition region", Journal of Fluid Mechanics Fluid Mech., vol. 494, pp. 187–221, 2003.

Testing for SRS Test Cases: Subsonic Round Jet

K. Viswanathan, "Aeroacoustics of Hot Jets", Journal of Fluid Mechanics, vol. 516, pp. 39–82, 2004.

- Jet experiment typically used for aeroacoustics tests
- Mesh 1: $1.5 \cdot 10^6$ cells
- Mesh 2: $4.1 \cdot 10^6$ cells (results shown for Mesh 2)
- Again, much faster "transition" for SBES than for SDES (SDES in the middle)
- Velocity distribution along axis of jet in much better agreement with experimental data for SBES
- U_{RMS} fluctuations much closer to experiments (albeit still over-predicted)
- Eddy-viscosity ratio much lower in LES region for SBES (SST+WALE)

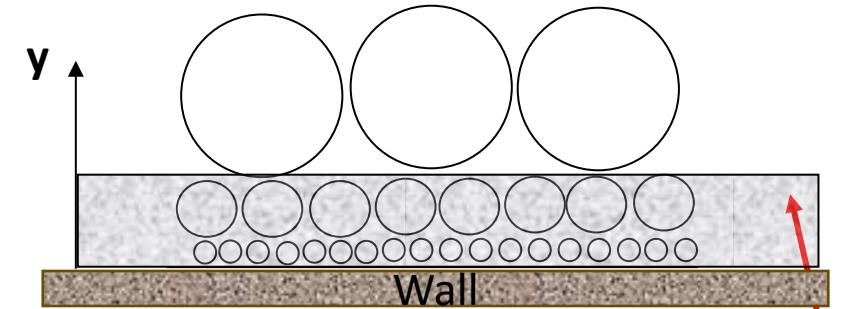


Near Wall Scaling – What’s the Problem

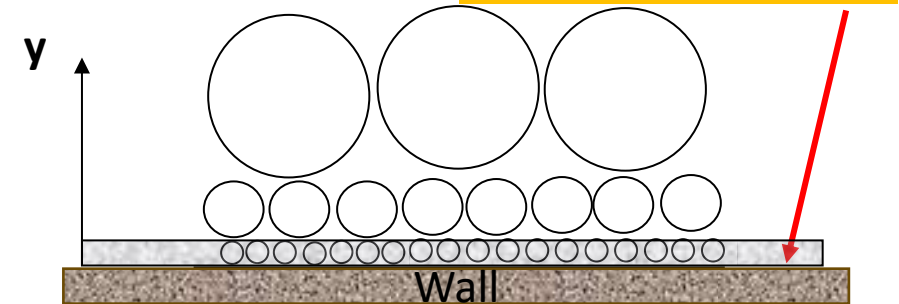
- Turbulent length, L_t , scale is independent of Re number
- Turbulent structures inside sublayer (grey box) are damped out and only structures above the sublayer need to be resolved
- However, the thickness of the viscous sublayer decreases with increasing Re number
- Smaller turbulence structures near the wall get “exposed” as Re increases – these structures need to be resolved in 3D space and time
- Strong Re number scaling for wall-resolved LES costs

$$L_t = \kappa y$$

Low Re

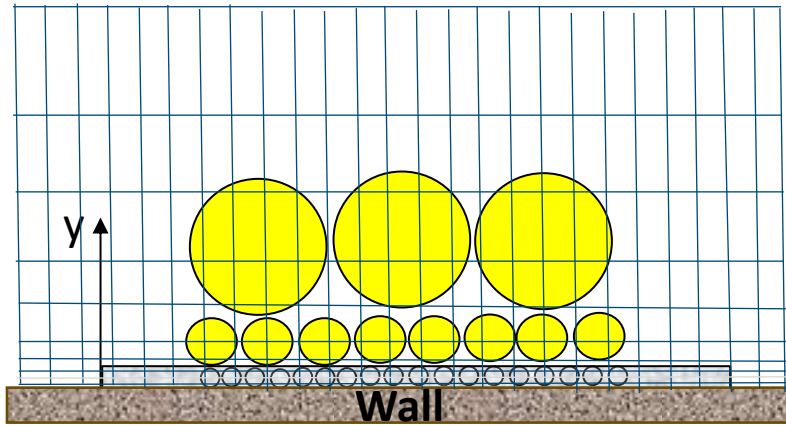


High Re



Near Wall Resolution – What are the Alternatives?

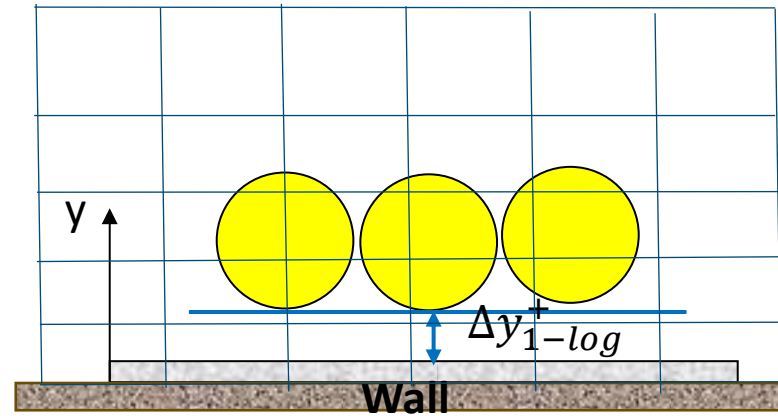
WRLES



- Wall Resolved LES

- Eddies are resolved in all three dimensions
- Fine grid in dx , dy , dz
- $dx^+ < 40$, $dz^+ < 20$, $dy^+ < 1$

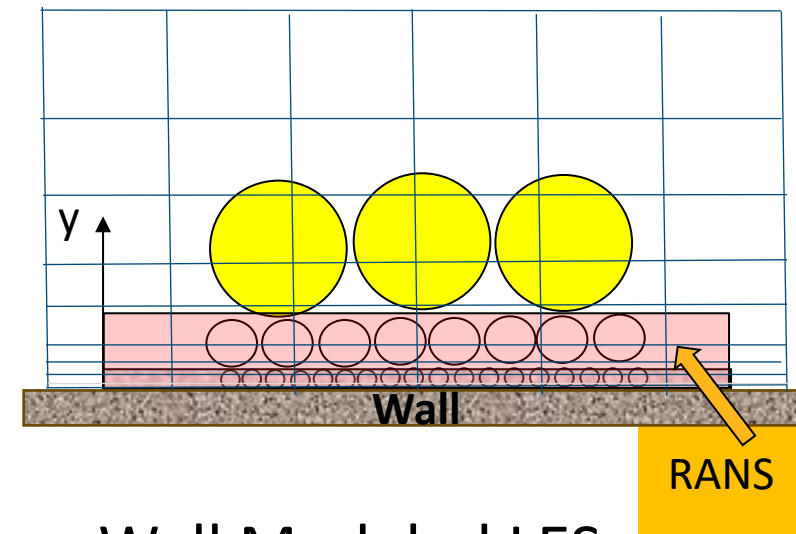
WFLES



- Wall Function LES

- Eddies are resolved only in logarithmic and out layer
- Wall Function for inner layer
- Coarse grid in dx , dy , dz
- dx^+ , dy^+ , dz^+ depend on Re number

WMLES



- Wall Modeled LES

- Eddies are resolved only in logarithmic and out layer
- Wall Function for inner layer
- Coarse grid in dx , dz
- dx^+ , dz^+ depend on Re number
- $dy^+ = 1$

Non-Iterative Wall-Function (Wall-Stress Model)

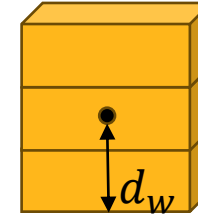
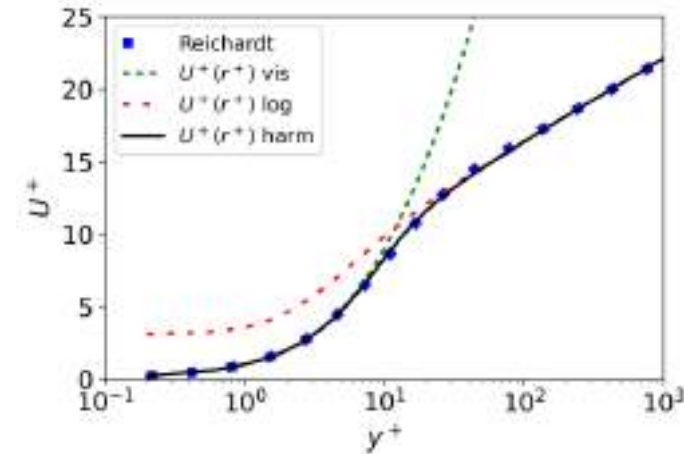
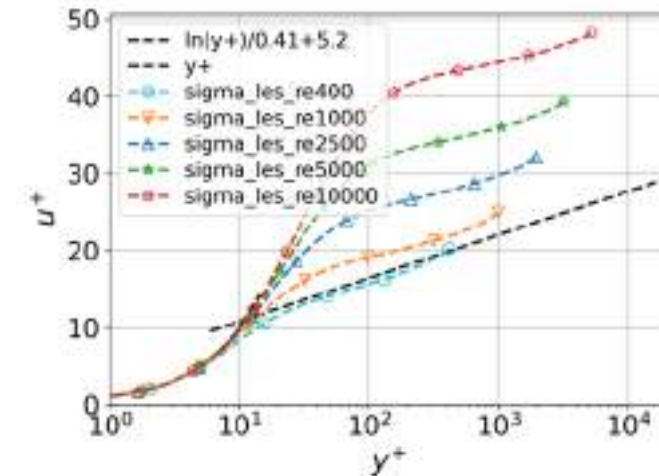
- Based on r^+ not y^+
- Approximation of logarithmic layer
- Harmonic blending of sub- and log-layer
- Is 'equilibrium' good enough?

$$r^+ = y^+ u^+ = \frac{u d_w}{\nu}$$

$$u_{sub}^+ = \sqrt{r^+}$$

$$u_{log}^+ = C_1 \ln(r^+ + C_2)$$

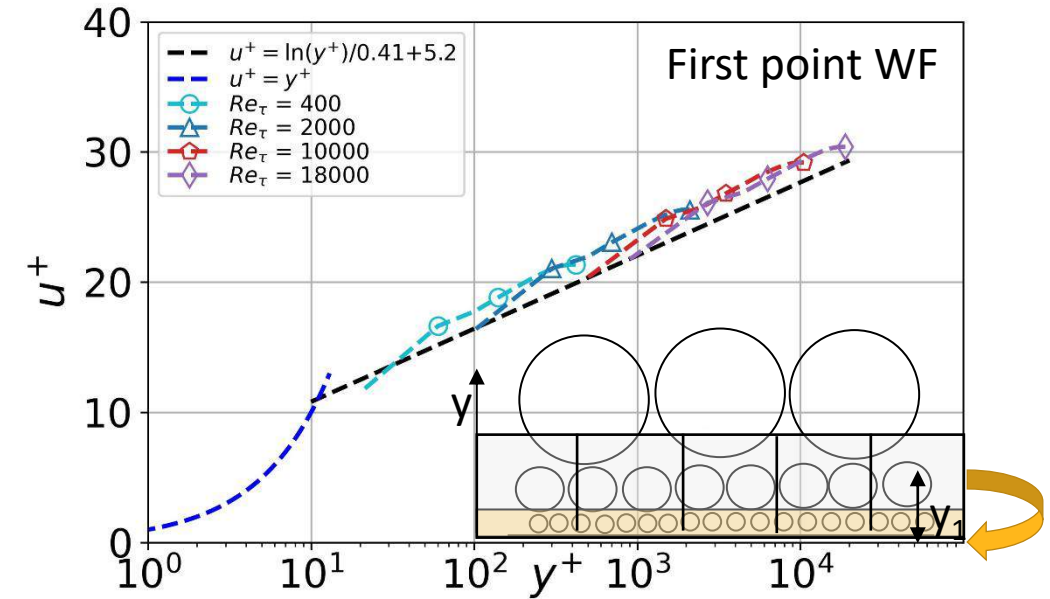
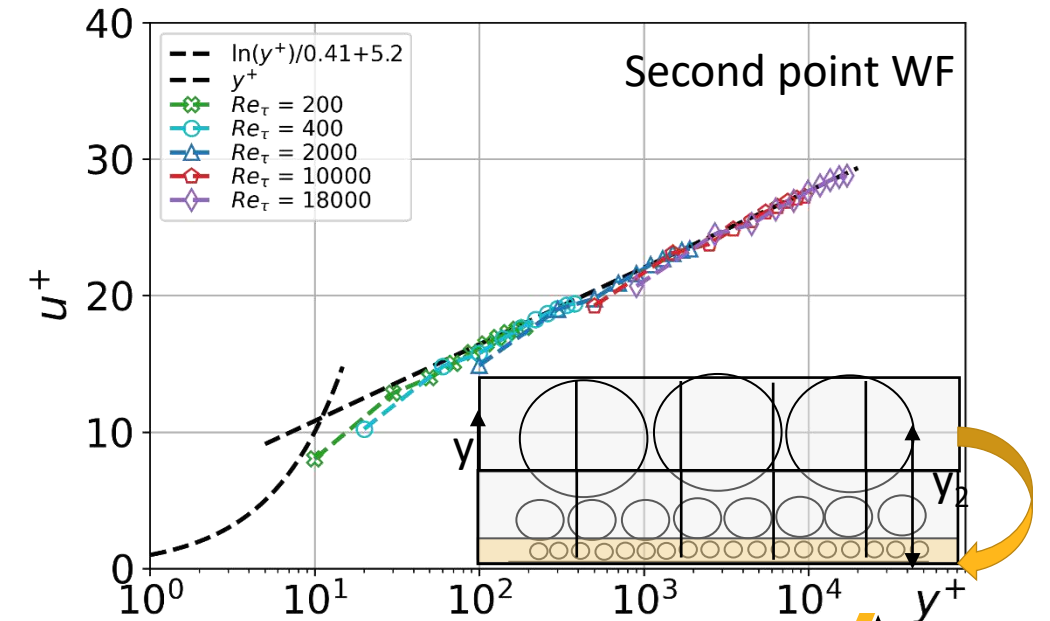
$$u_{eq}^+(r^+) = \left((u_{sub}^+)^{-\alpha} + (u_{log}^+)^{-\alpha} \right)^{-\frac{1}{\alpha}}$$



EVALUATION OF A NEW EXPLICIT WALL-FUNCTION FORMULATION FOR LARGE
EDDY SIMULATION
ETMM15
*E.K. Guseva¹, D.G. Flad¹, A.L. Nogueira², D.P. Graebin² and F.R.
Menter¹*
In Print

Log Layer Mismatch in WFLES

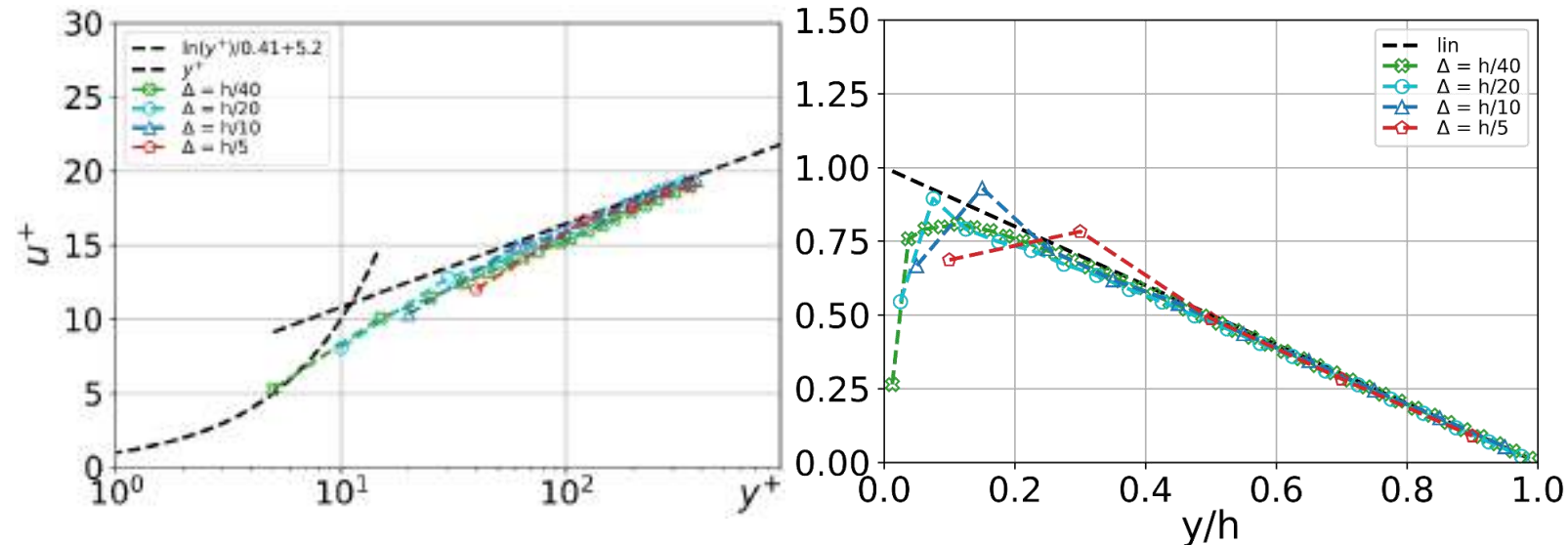
- The input to the wall function is often a first near wall cell
- A known problem in WFLES – a Log Layer Mismatch (LLM)
 - Errors in wall shear estimations
 - Severity depends on model and numerics used
- The known remedies are:
 - time- or spatial filtering of the input for the wall-shear model
 - alternative locations for the velocity input
 - Fixed location (wall distance, fraction of the boundary layer thickness)
 - **Second of-the-wall cell** (or 3rd, 4th ..)



Mesh sensitivity

Isotropic mesh refinement

- Isotropic (cubic) mesh with resolution:
 - 5 to 40 cells per half-channel height h

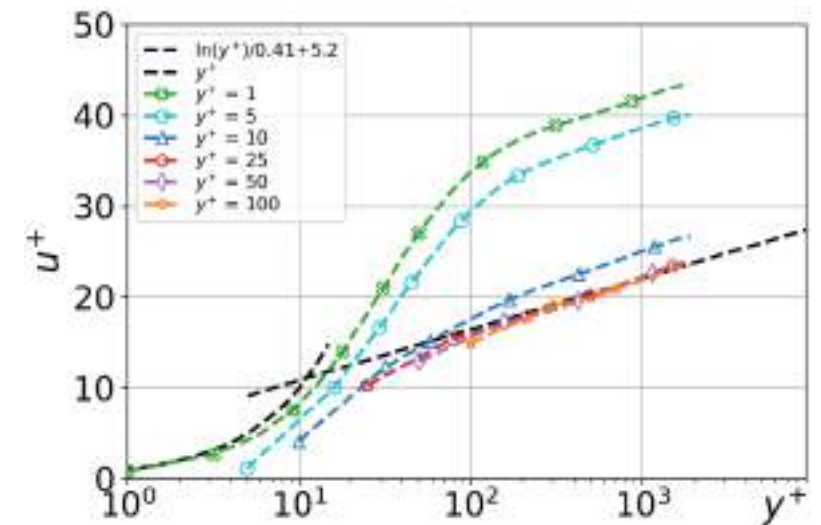


- Velocity profiles and resolved shear stresses are consistent across all mesh resolutions
 - Even on the coarsest mesh!

➤ For isotropic meshes, the position of input to the wall model relative to the boundary thickness appears to be irrelevant

Wall-normal mesh refinement

- Fixed $\Delta x = \Delta z = h/10$ ($\Delta x^+ = \Delta z^+ = 200$)
- The wall-normal mesh is varied



!! Refining the mesh only in the normal direction only results in deteriorating accuracy

Algebraic WMLES Channel

- Blend of Prandtl and any alg. LES model
- Blending based on d_w/h_{max}
- Function f_{sw} has two parameters
 - C_{w1} switching location
 - C_{w2} steepness

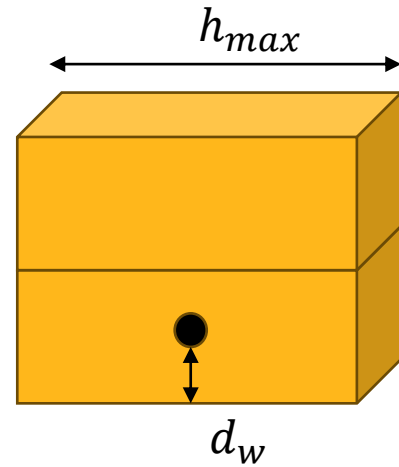
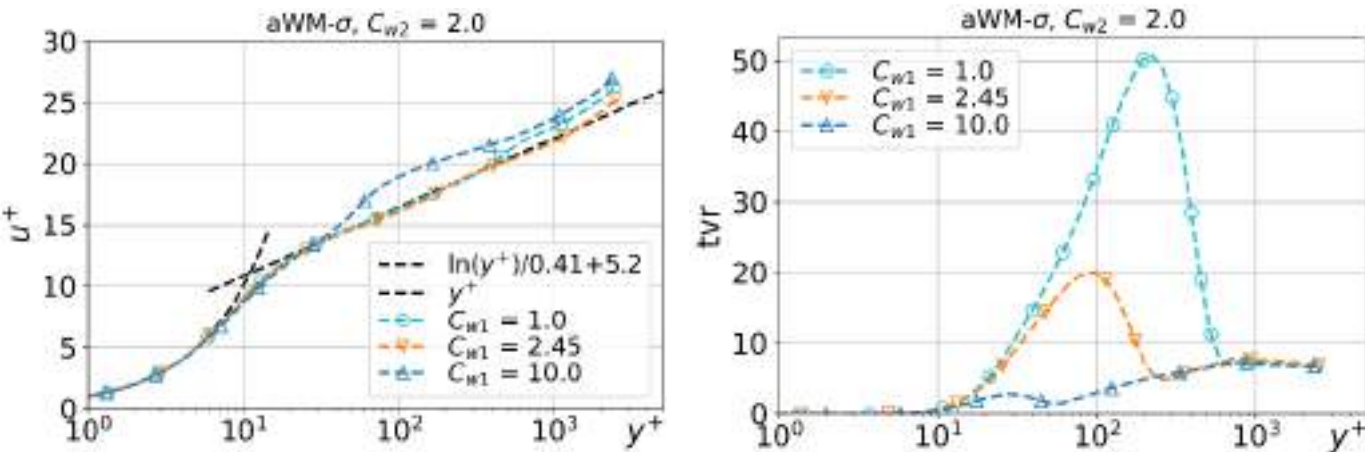
$$\nu_t = f_{sw} \nu_{t,RANS} + (1 - f_{sw}) \nu_{t,LES}$$

$$f_{sw} = e^{-\left(\frac{C_{w1} d_w}{h_{max}}\right)^{C_{w2}}}$$

$$\nu_{t,RANS} = l_{RANS}^2 S$$

$$l_{RANS} = 0.41 d_w \sqrt{f_{wd}}$$

$$f_{wd} = 1 - e^{-\left(\frac{l^+}{25}\right)^3}$$



ETMM14 Conference Barcelona 2023

A Family of Wall-Modeled Large Eddy Simulation Formulations

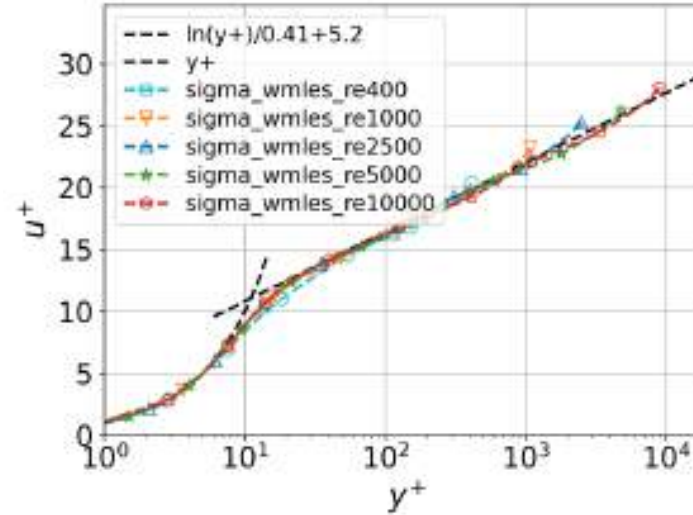
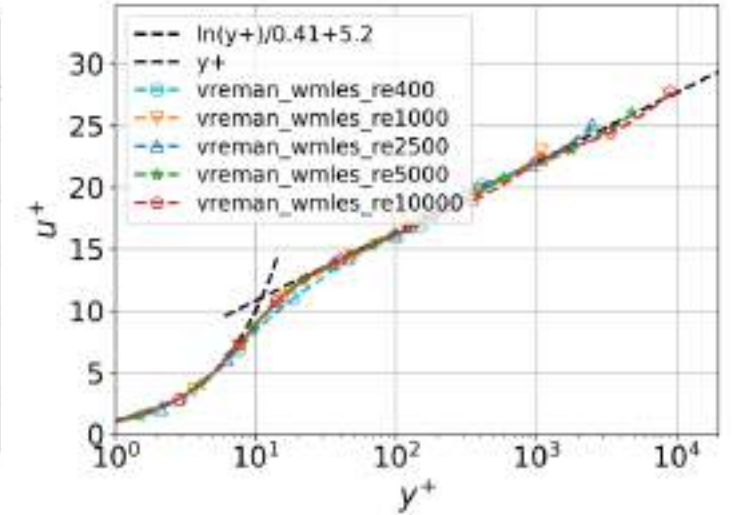
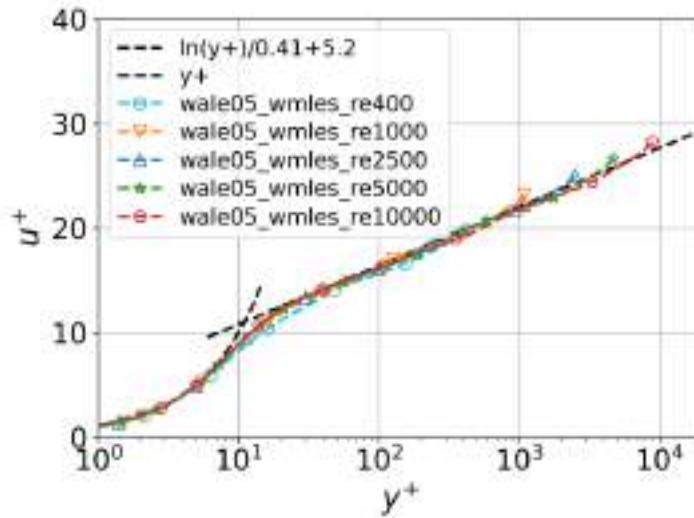
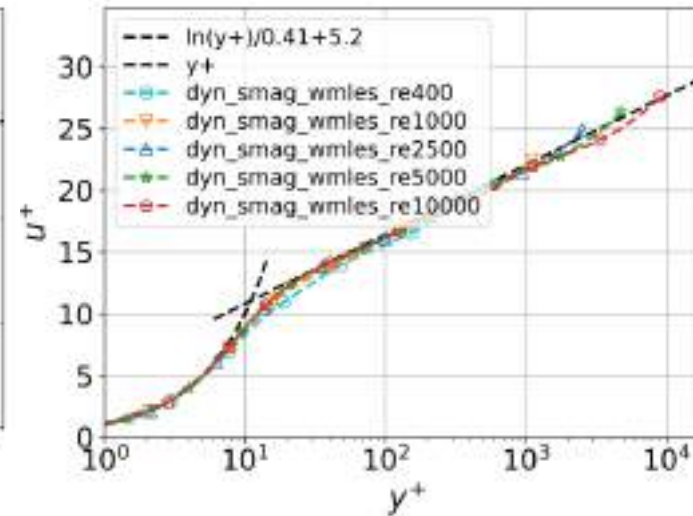
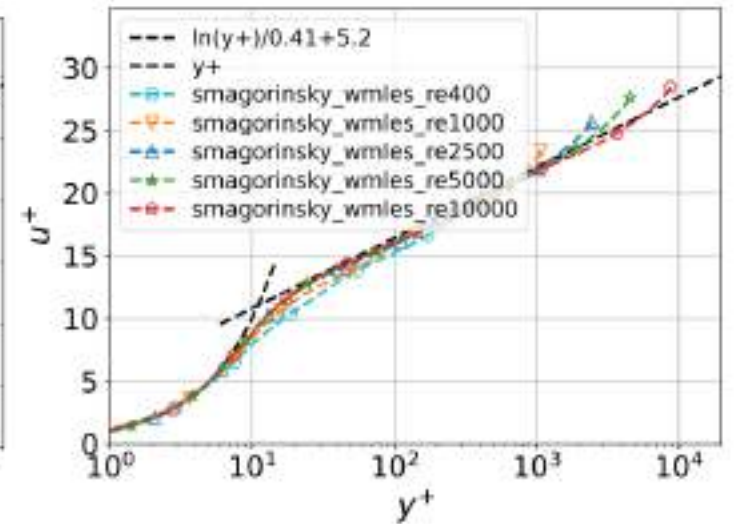
F. R. Menter¹, E. Guseva¹, A. Stabnikov², A. Matyushenko² and A. Garbaruk²

¹ Ansys Germany GmbH

² SPbPU St. Petersburg

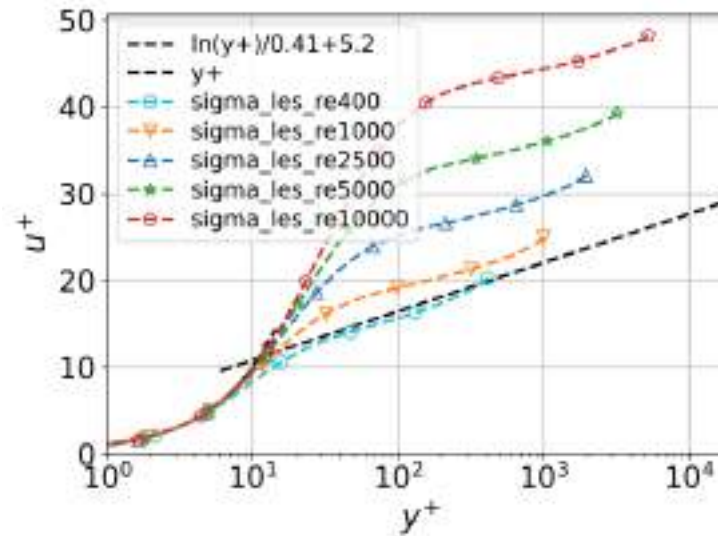
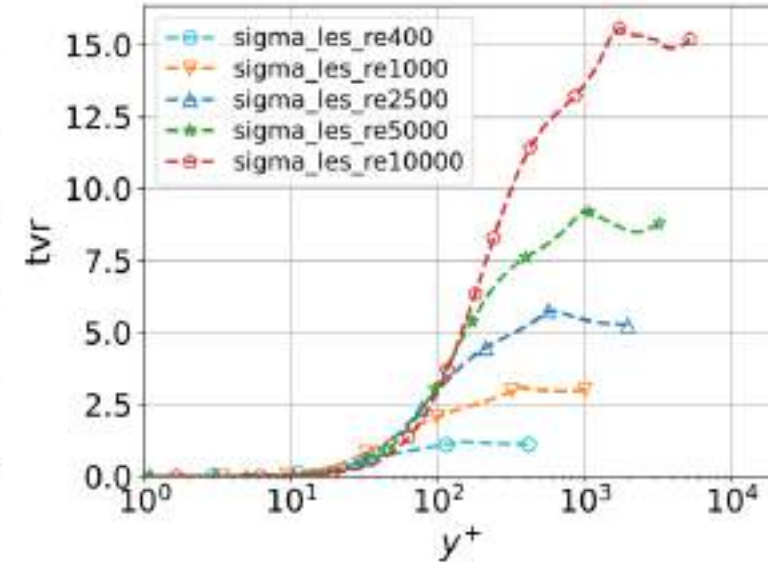
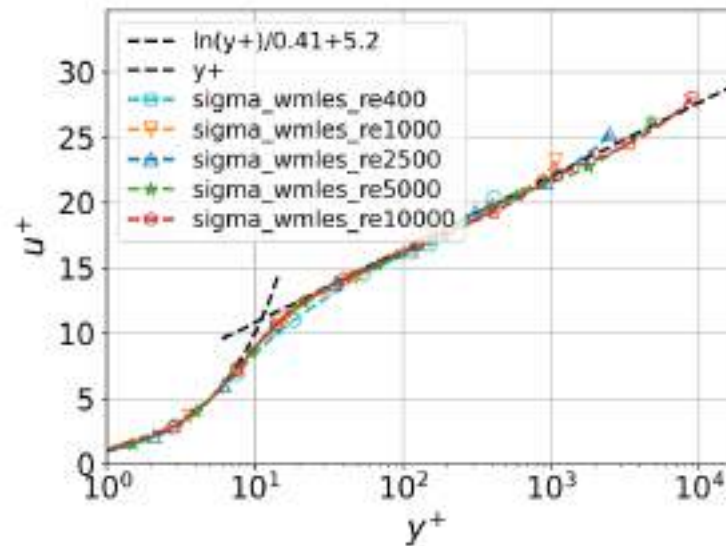
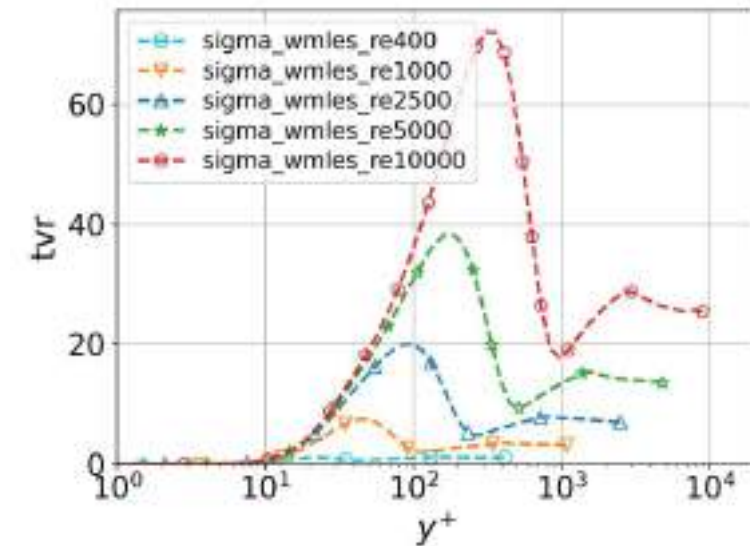
Variation in LES Models

- σ , Vreman, WALE, Smagorinsky, dyn. Smagorinsky
- All LES models work
- No log-layer miss-match only for very low-dissipation numerics
- Grids $\sim 600,000$ cells



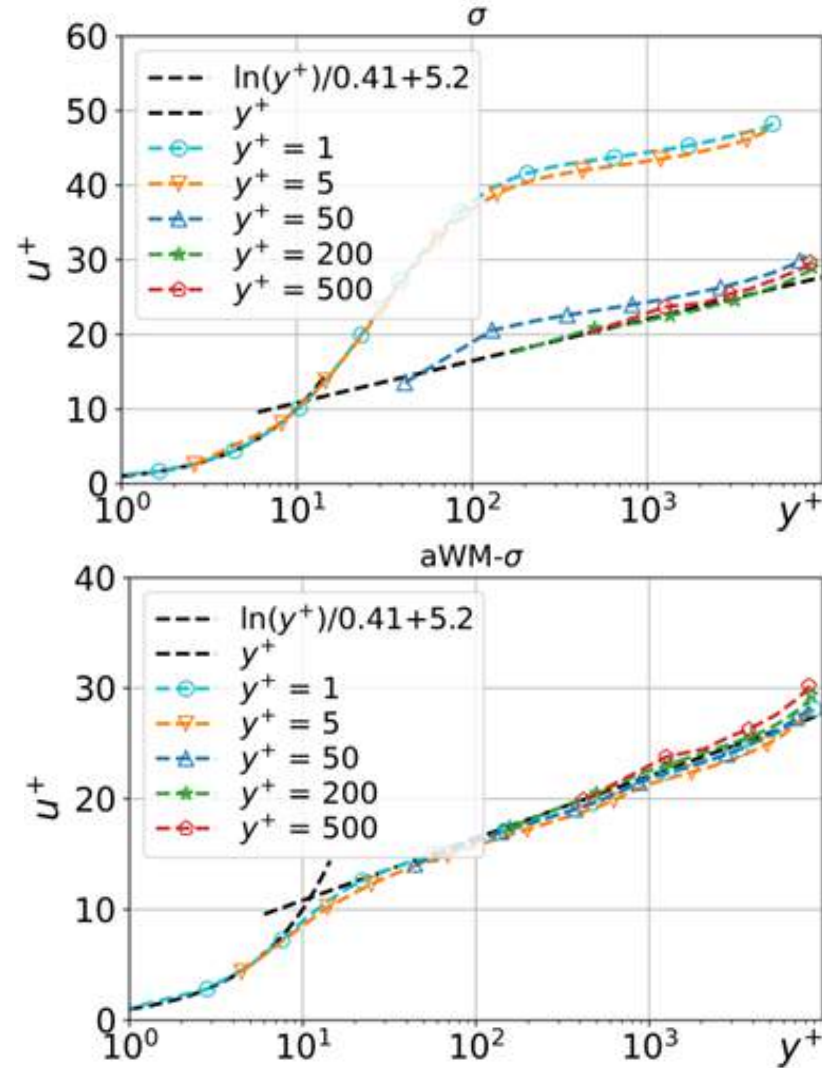
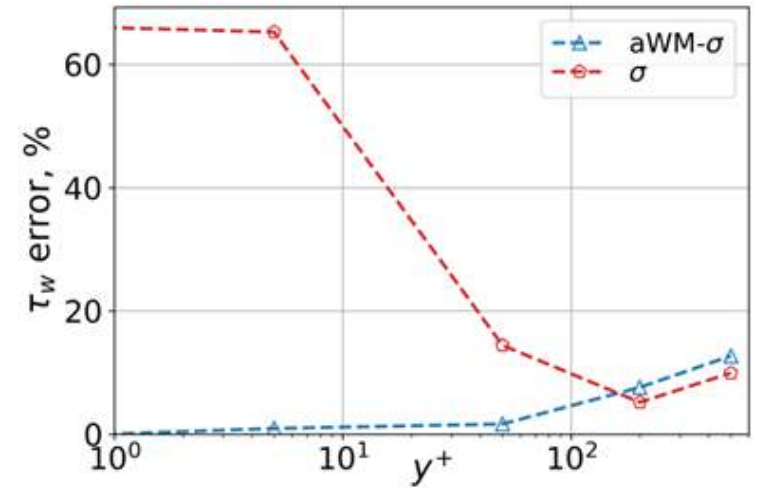
LES vs WMLES

- Fine near wall meshes ($y^+ \sim 1$) and variation in Reynolds number
- On the same mesh, WMLES can handle much higher Reynolds numbers



Changes in y^+ for $Re_\tau = 10,000$

- LES
 - Deteriorates under wall-normal grid refinement
 - Large error in τ_w
- WMLES
 - Maintains log-layer and τ_w
- Penalty under mesh refinement is a no-go in industrial CFD



Summary: Grid Resolution Boundary Layers

X-streamwise, y-wall normal, z-spanwise

- Wall-Resolved LES

$$\Delta x^+ < 40, \quad \Delta z^+ < 20, \quad N_y \approx 30 - 40$$

- Wall Function LES

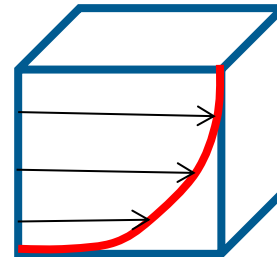
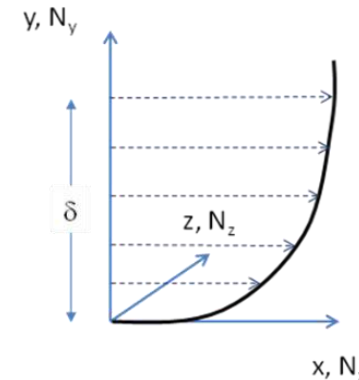
$$N_x \approx N_y \approx N_z \sim \frac{\delta}{\Delta x} > 5 - 20; \quad \Delta x^+, \Delta y^+, \Delta z^+ \text{arbitrary},$$

Keep near wall aspect ratio $AR_{wall} < 4$

- Wall-Modeled LES

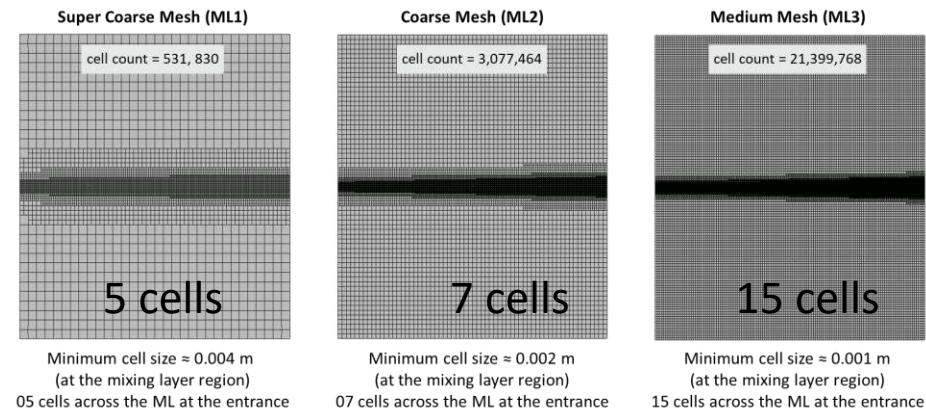
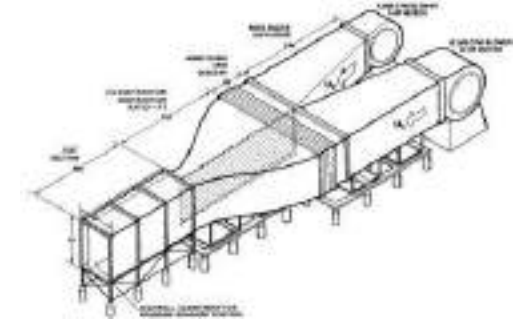
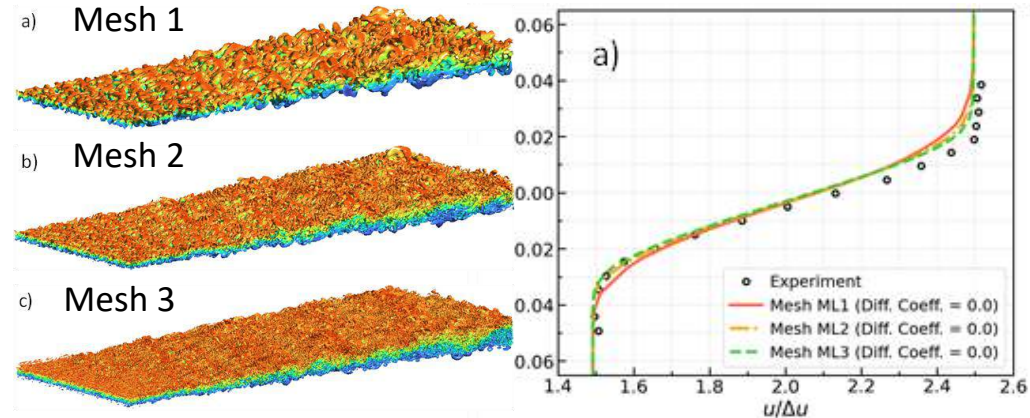
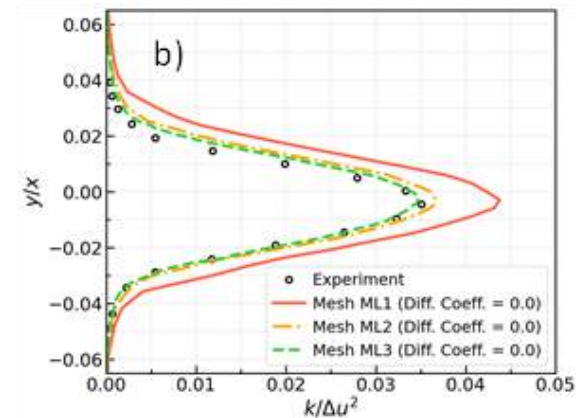
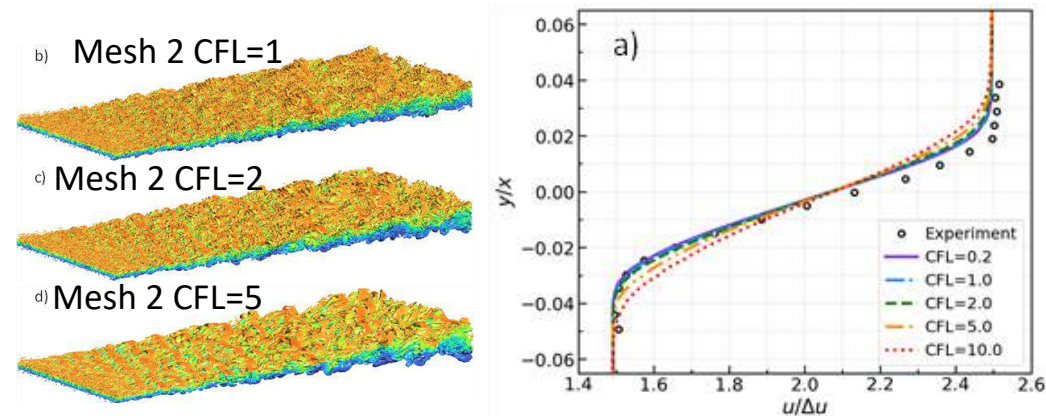
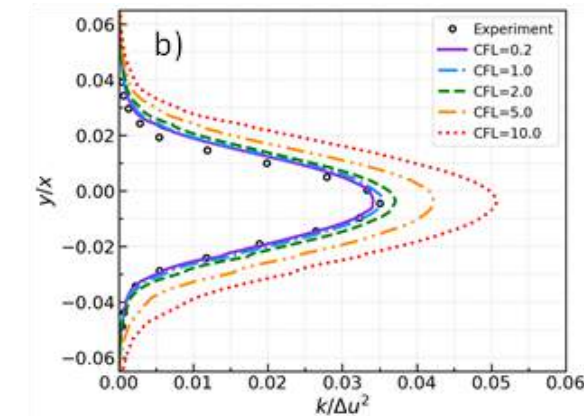
$$N_x \approx N_z \sim \frac{\delta}{\Delta x} > 5 - 20; \quad N_y \approx 20 - 30; \quad \Delta x^+, \Delta z^+ \text{arbitrary}, \Delta y^+ \sim 1$$

Boundary Layer Volume with $\delta x \delta x \delta$



Example: Mixing Layer

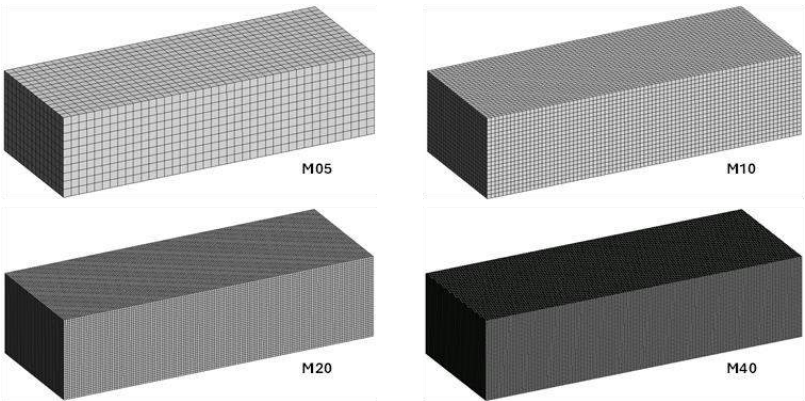
- 3 meshes
- Even with only 5 cells across the inlet mixing layer the velocity profile is captured well
- With increasing cells count the turbulent kinetic energy converges quickly
- The maximum CFL number can be increased
 - CFL=2 very small error increase
 - CFL=5 still reasonable accuracy in velocity profile



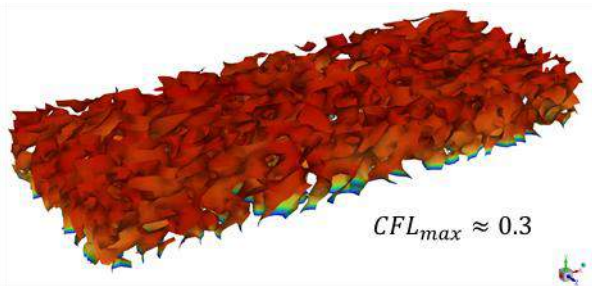
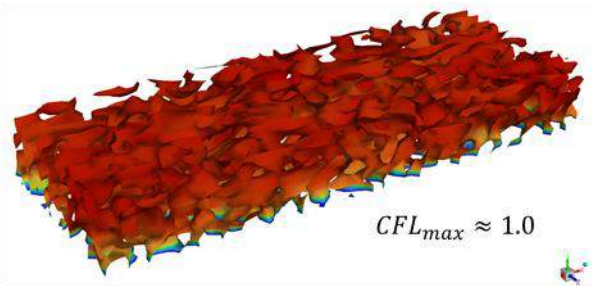
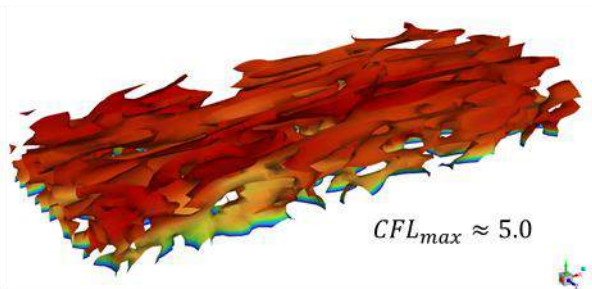
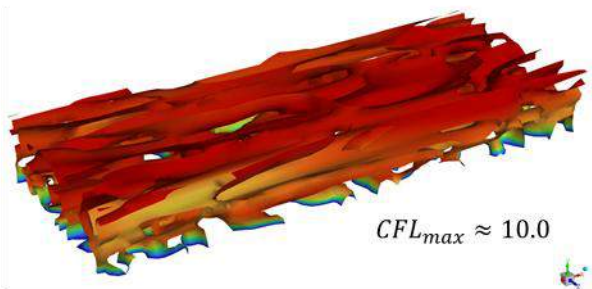
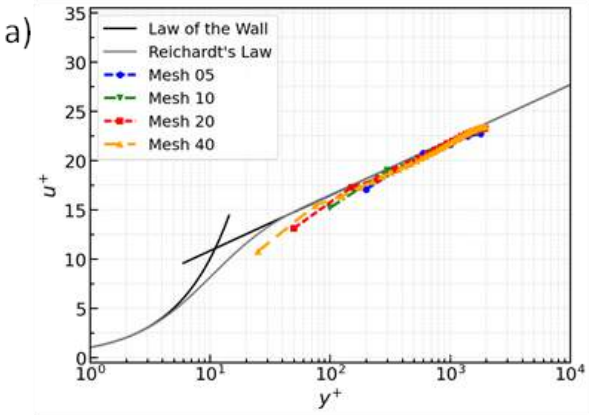
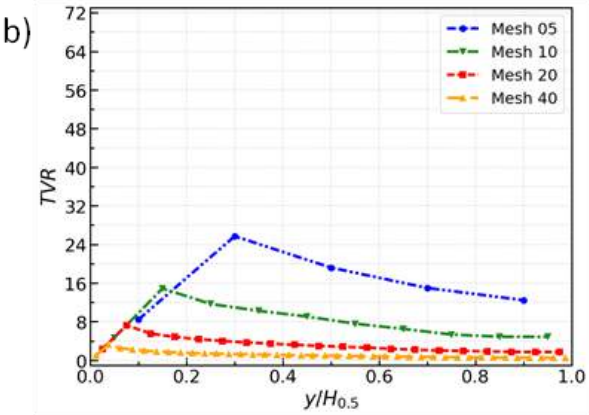
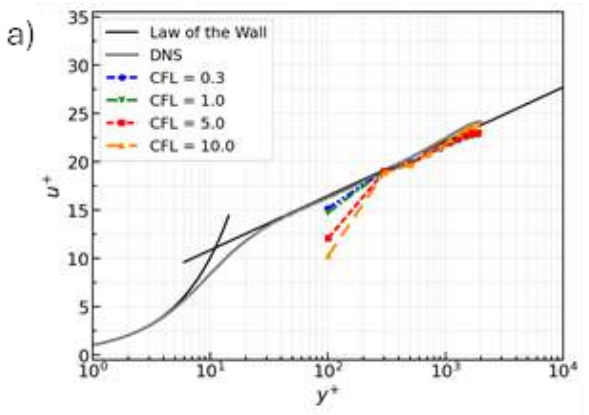
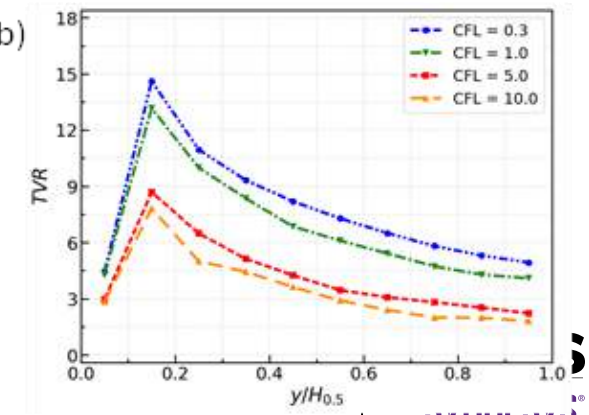
Example: WFLES Channel

- Grid study

- Even with only 5 cells across the half-channel – good match to log-layer
- Sensitivity to CFL number moderate – even though turbulence structures affected



Mesh	Cells per $H/2$	Cell Count
M05	5	6,000
M10	10	48,000
M20	20	384,000
M40	40	3,072,000

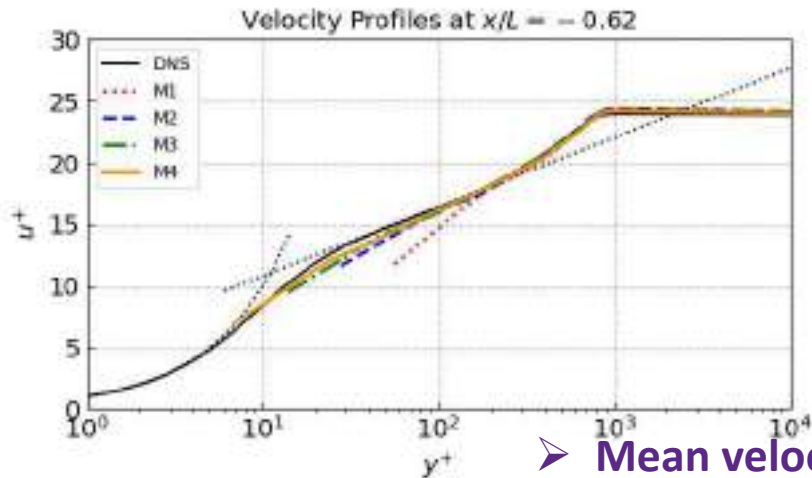


Smooth ramp

- High-Fidelity CFD Verification Workshop 2024
- $Re_L = 6.67 \times 10^5$
- 4 Octree-type meshes with 2 prisms

Mesh	Base cell size Δ , mm	$\delta_{\text{target position}}/\Delta$	Cells, M
M1	8	5	0.57
M2	4	10	3.7
M3	2	20	27

- MANS Velocity profile + unsteady fluctuations
 - Synthetic Turbulence Generator



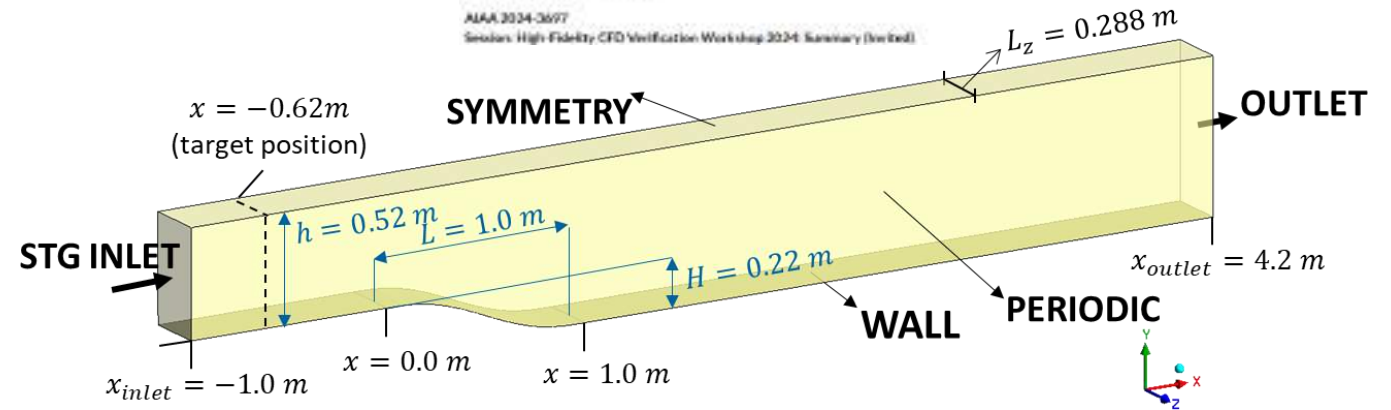
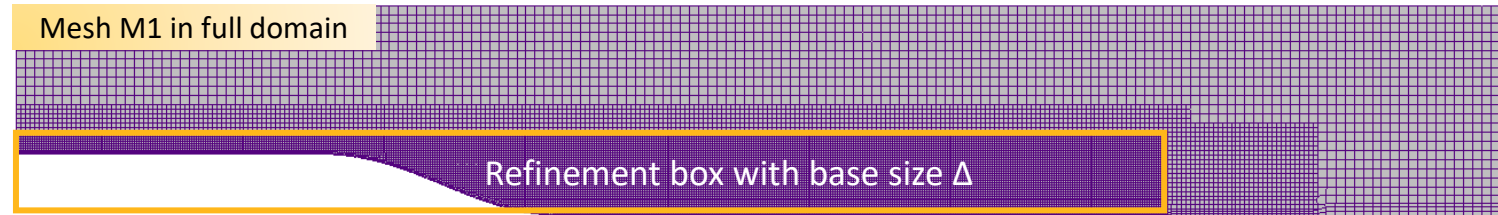
➤ Mean velocity profile predicted well even on the extreme coarse mesh

Summary of the Wall-Modeled LES Test Suite From the 2024 AIAA High-Fidelity CFD Verification Workshop

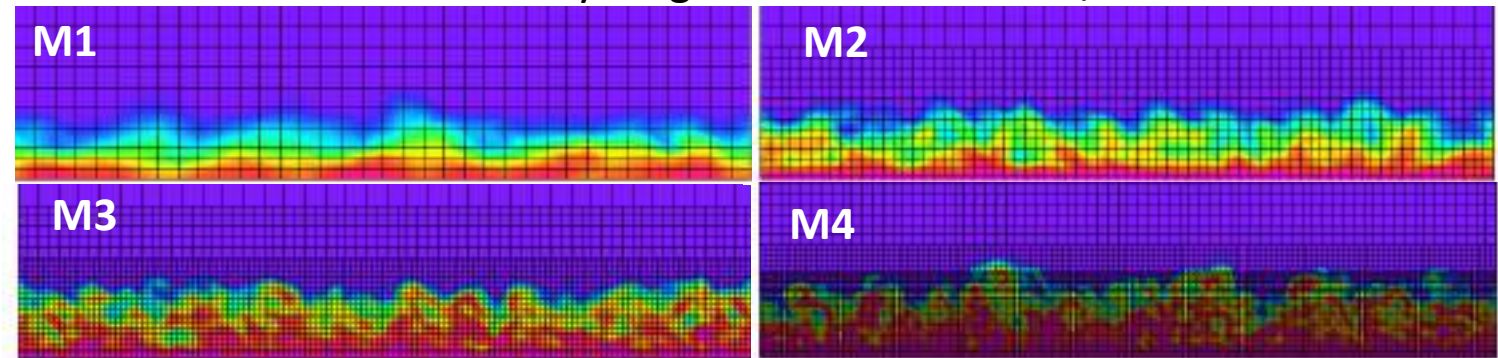
David J. Gersorn, Matthew J. Schwert, John Larsson, Ivan Dumagala-Matras, Ali Ozbun, Sarah Gargu, Christopher Drenth, Timothy Hultin, Matteo Perani, David Pineda and Isaac Schenck

AIAA 2024-3697

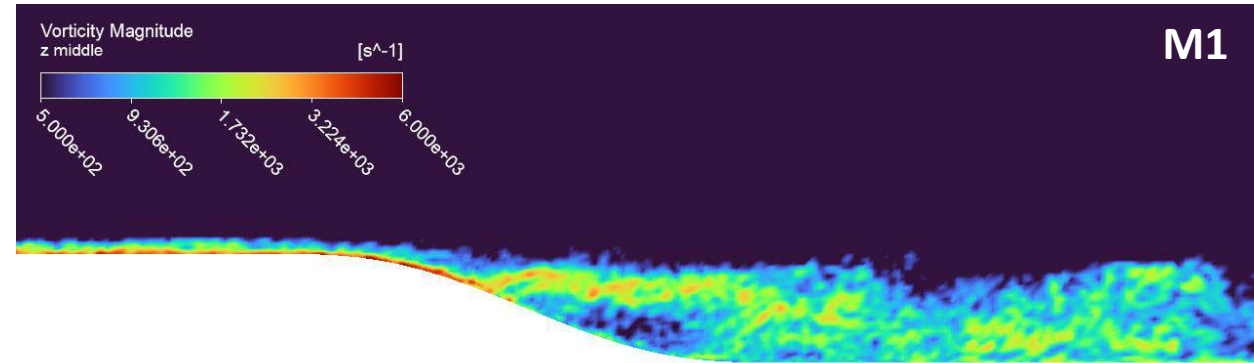
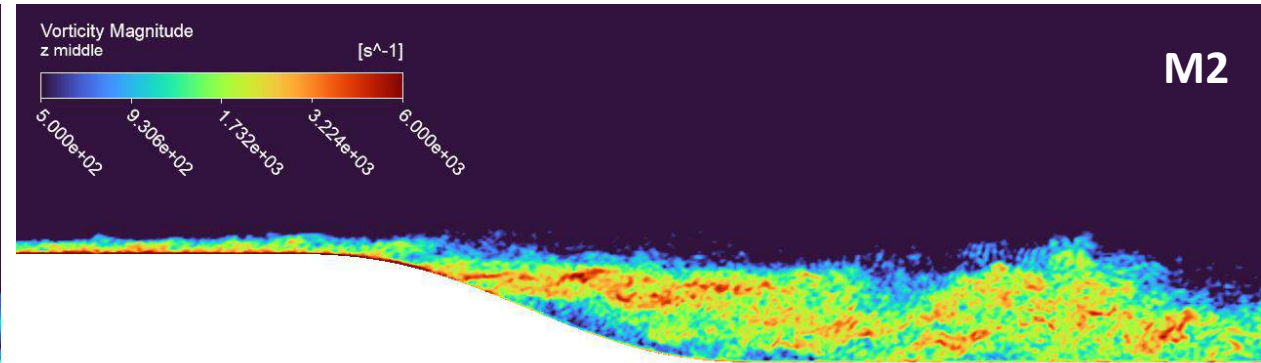
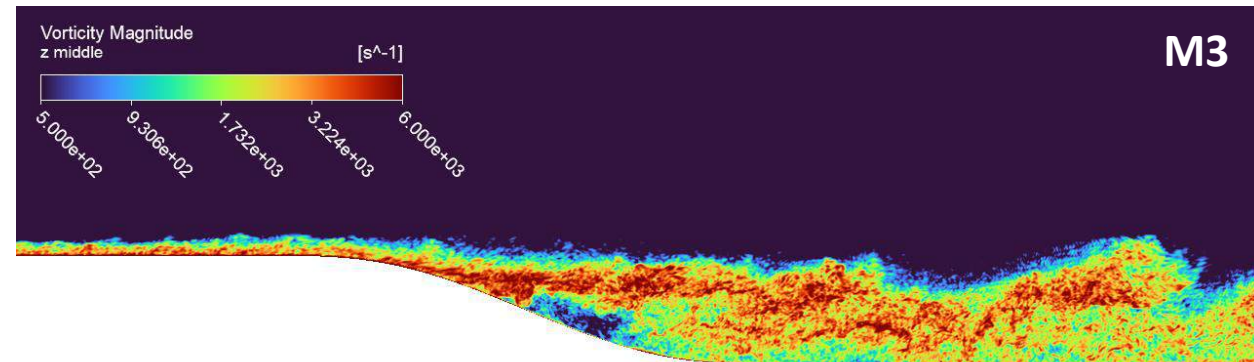
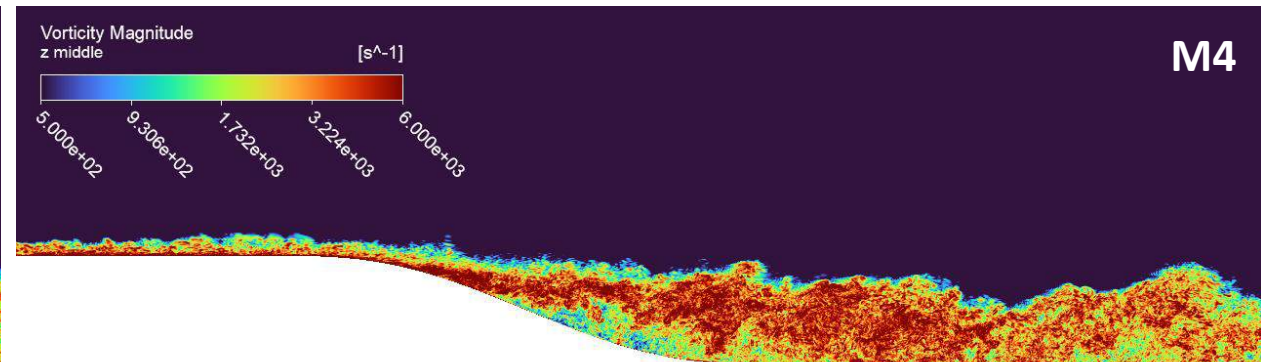
Session: High-Fidelity CFD Verification Workshop 2024: Summary (Invited)



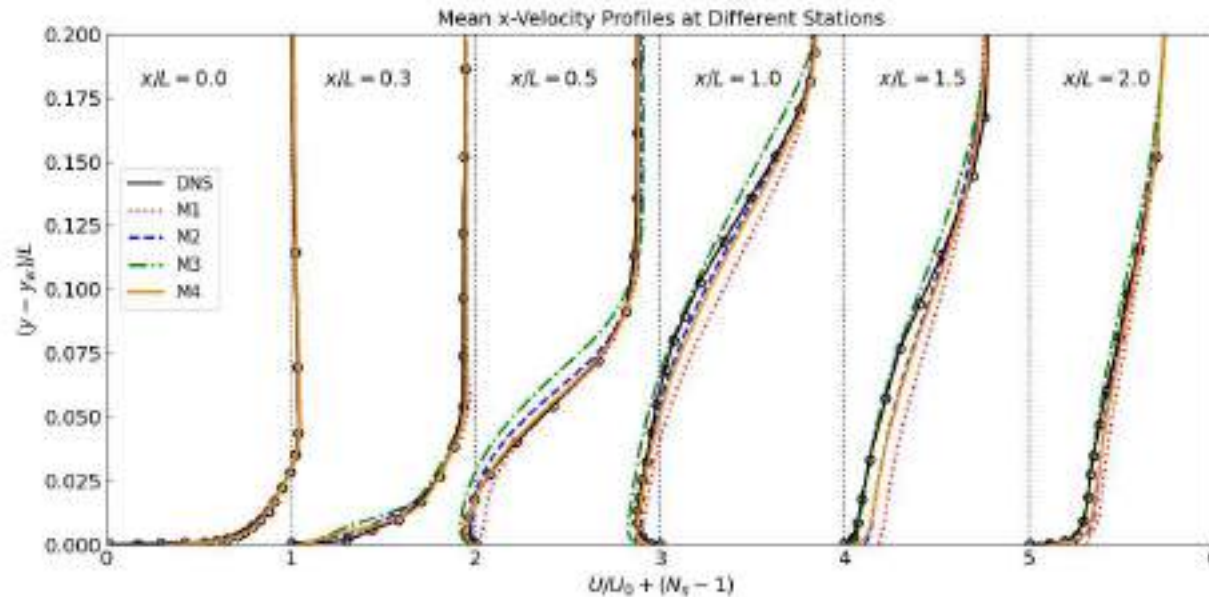
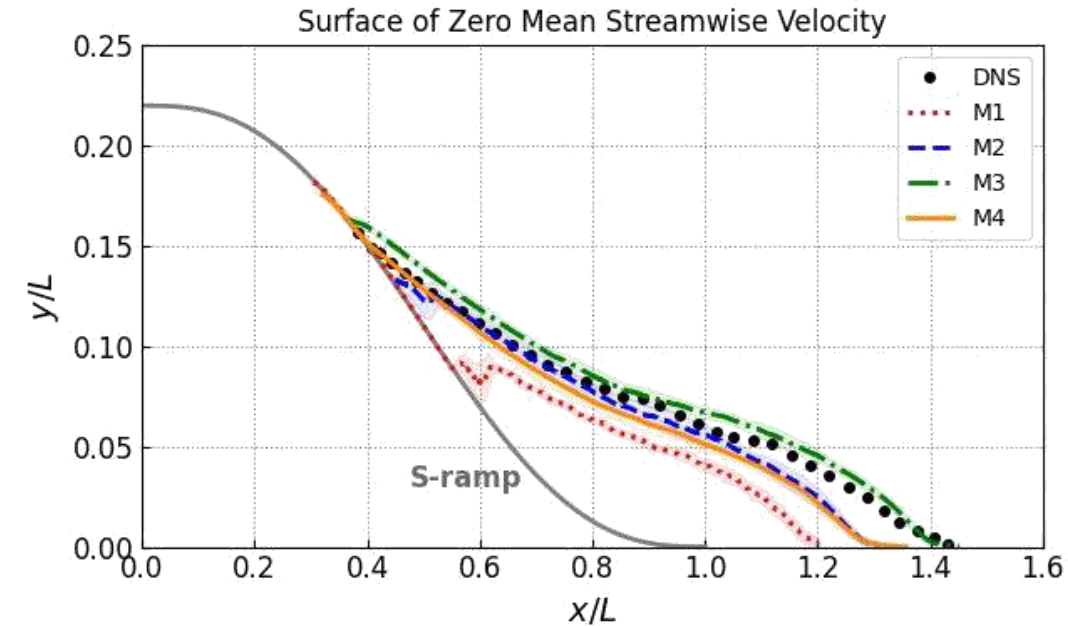
Mesh and vorticity magnitude contours at $x/L = -0.62$



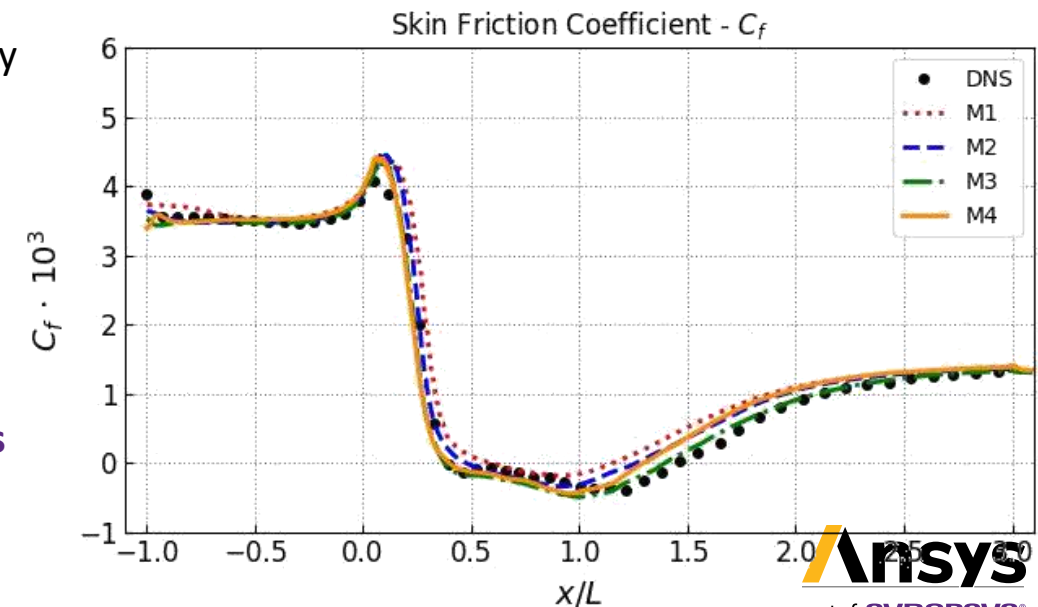
Flow visualization. Vorticity magnitude contours



Smooth ramp

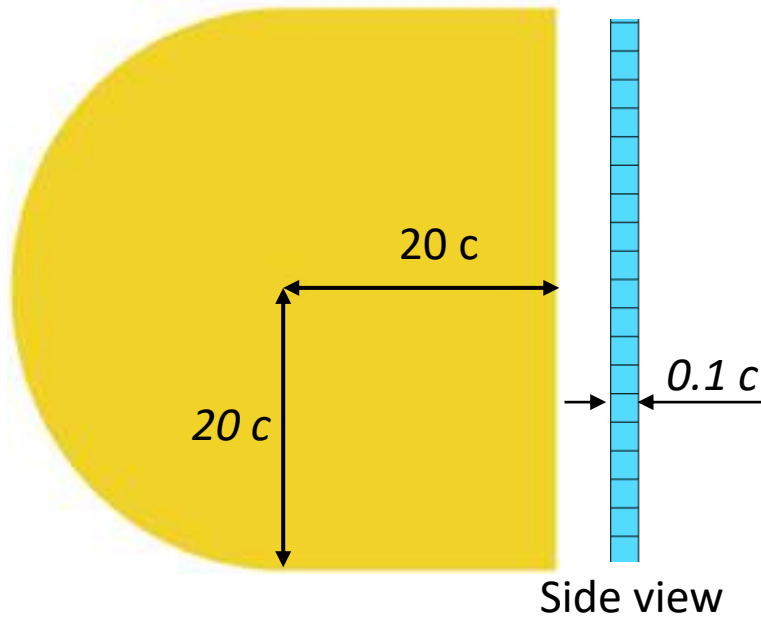


- On all the meshes, except on the coarsest mesh, the separation point is accurately captured
 - The recirculation bubble size is somewhat underpredicted
 - Some mesh dependency still observed between M2-M4
 - Overall, good agreement with the DNS results was achieved
- **No systematic failure of equilibrium wall function in non-equilibrium conditions**

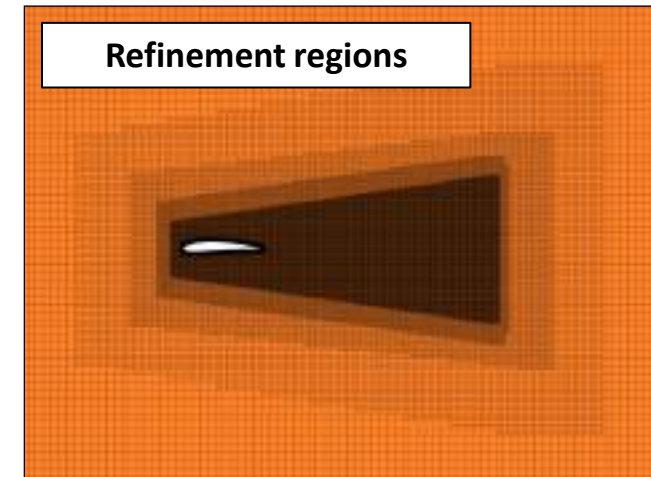
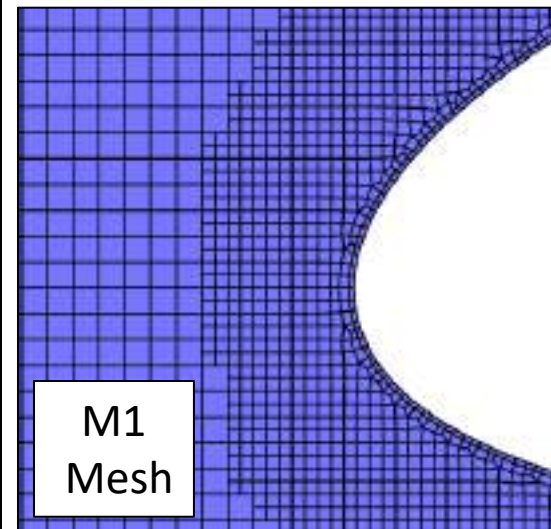
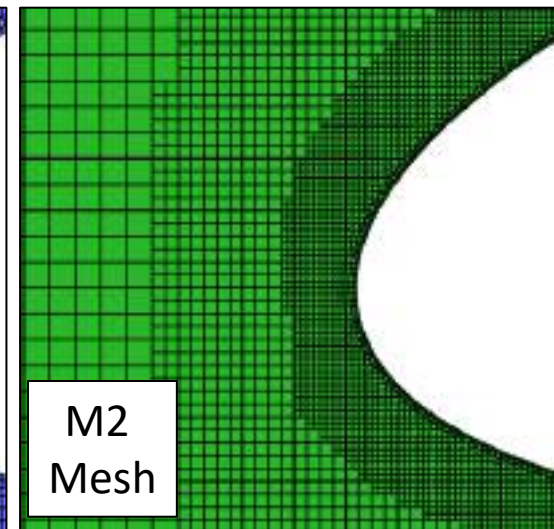
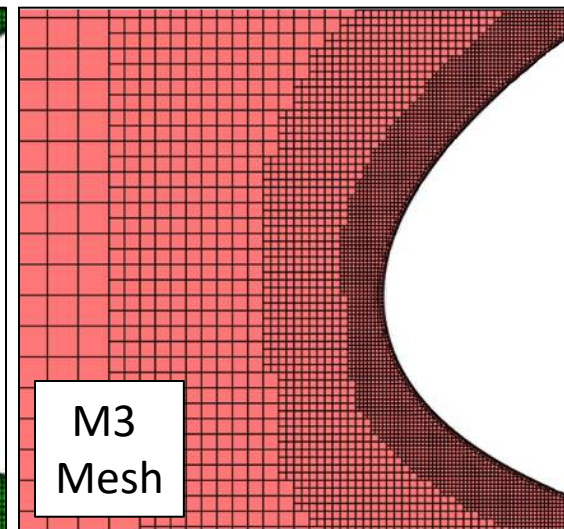


NACA 4412

- NACA-4412 with $c = 1$ m
- Angle of attack: 13.87°
- Experiment - Wadcock (1979)
- $Re_c = 1.5 \cdot 10^6$
- 2 prisms layers near the wall



Mesh	Cell Size at Airfoil	Number of cells	$y^+_{\max}$	Number of cells in BL*
M1	$1.6 \cdot 10^{-3} c$	3,898,080	90	12
M2	$0.8 \cdot 10^{-3} c$	8,117,776	48	19
M3	$0.4 \cdot 10^{-3} c$	24,917,296	24	28

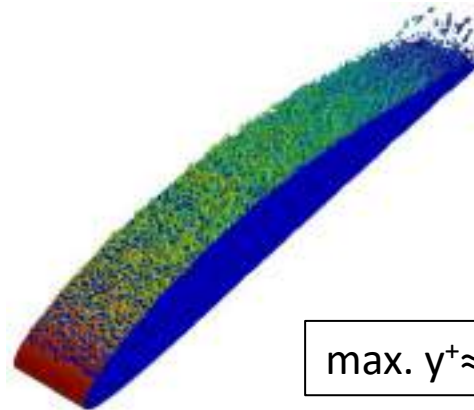


* Measured at $x/c = 0.5$

Results – iPBNS

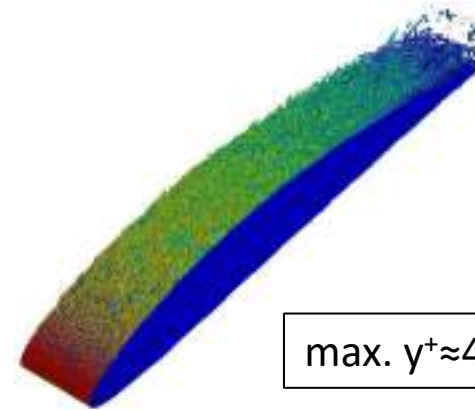
- Boundary Layer resolution
 - 12, 19 and 28 cells per BL at $x/c = 0.5$
 - 6, 9, and 16 cells per BL at $x/c = 0.1$
- Transition is noticeably delayed at meshes M1 and M2
 - (no trip was used in the simulations)
 - Local mesh refinement is an option

M1 mesh ($4 \cdot 10^6$)



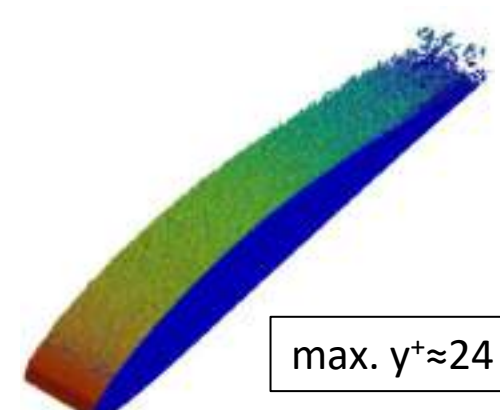
max. $y^+ \approx 90$

M2 mesh ($8 \cdot 10^6$)

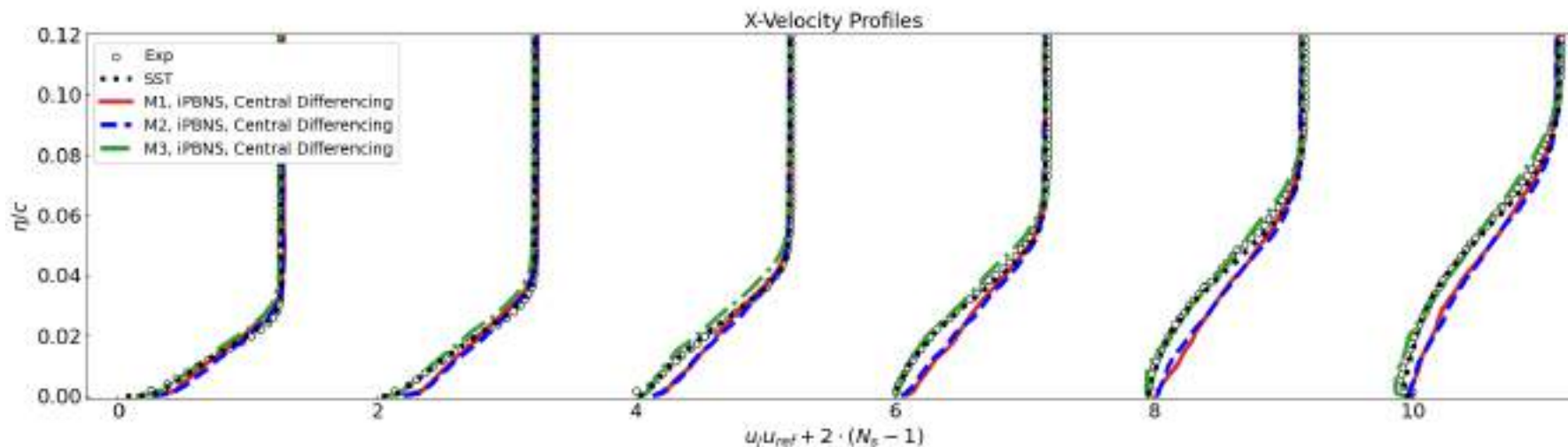
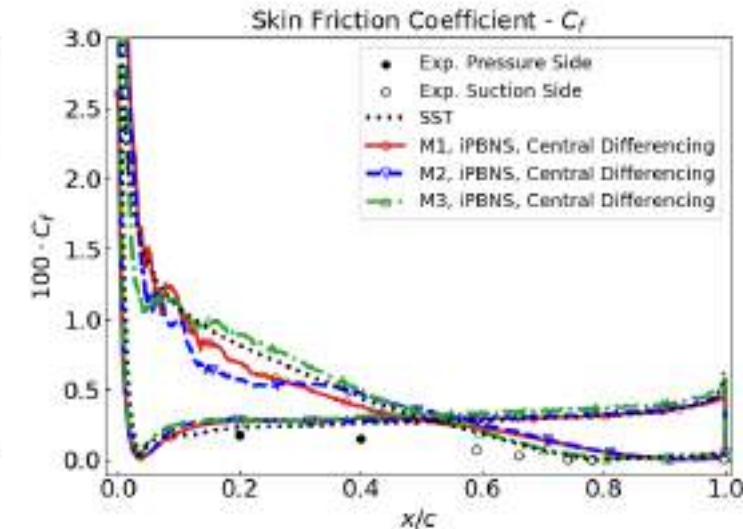


max. $y^+ \approx 48$

M3 mesh ($25 \cdot 10^6$)

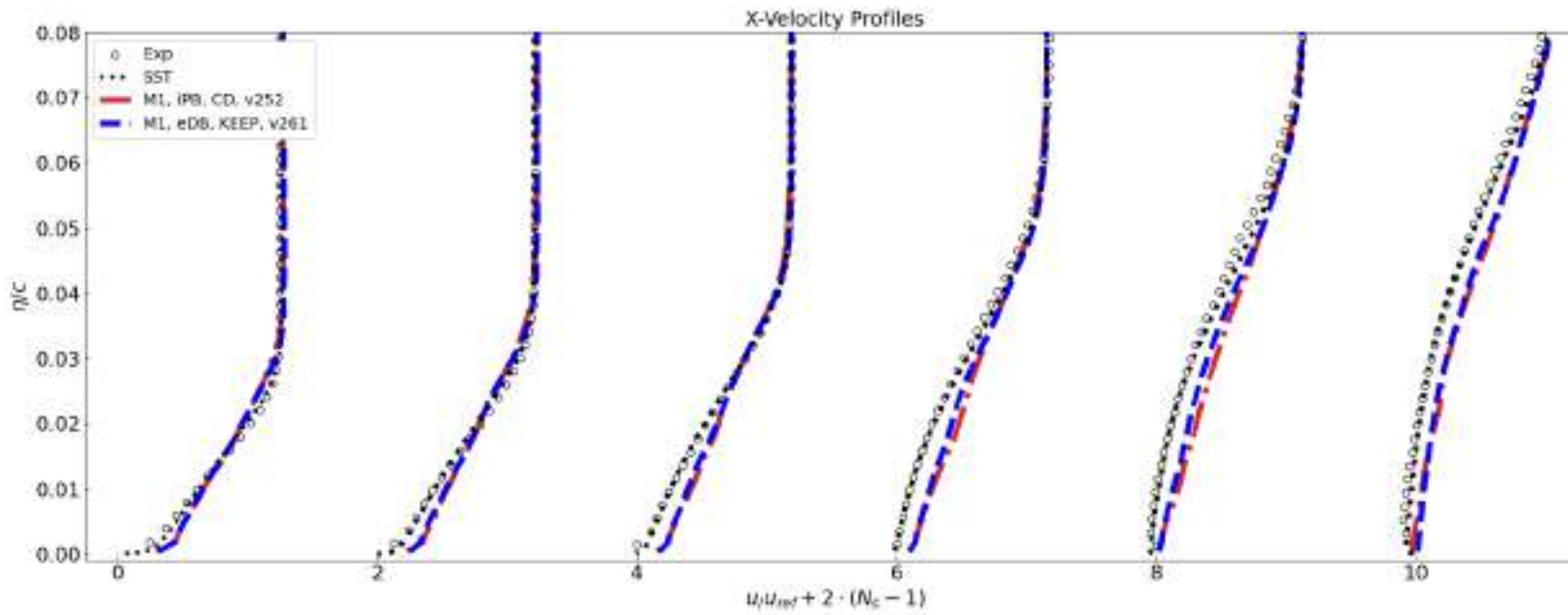
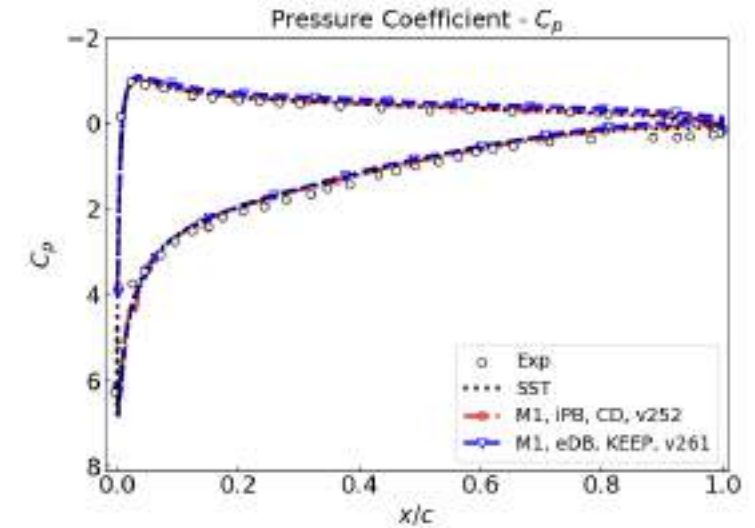
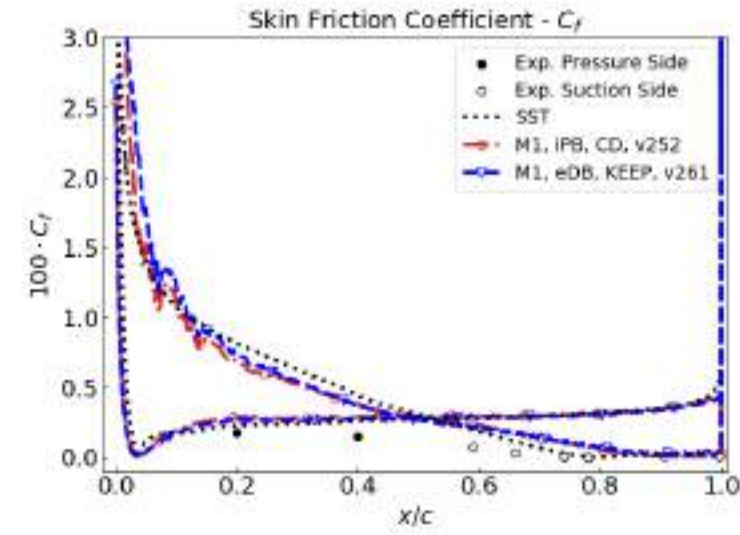


max. $y^+ \approx 24$



Results – M1 mesh (4 million cells)

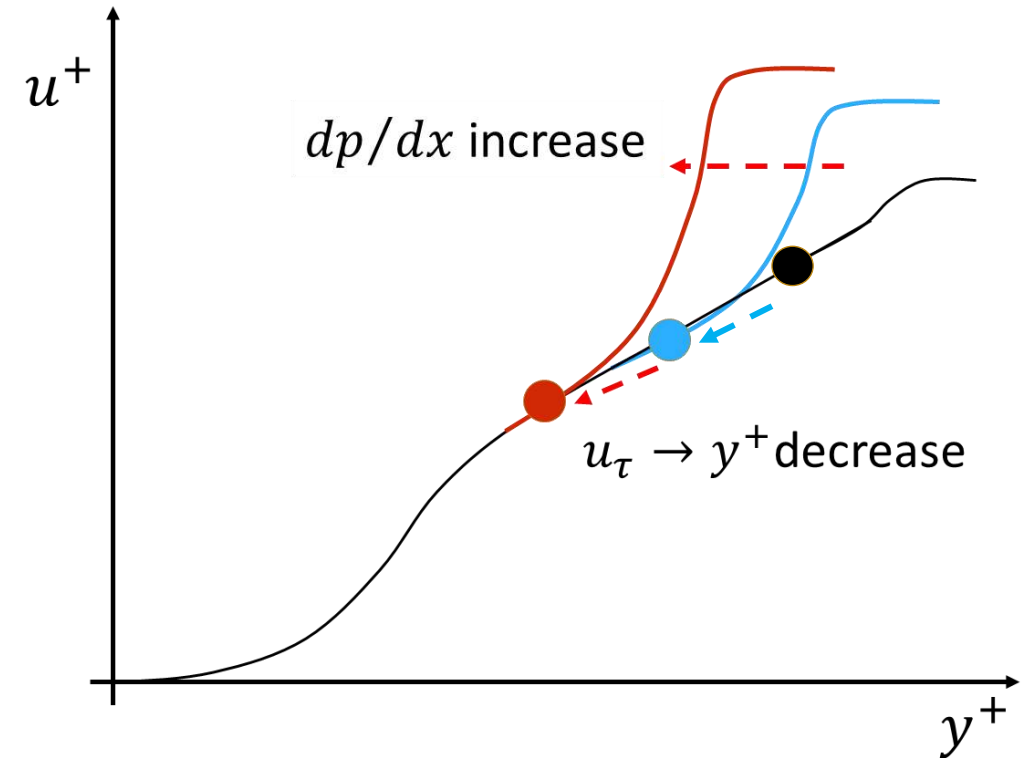
- $\Delta t = 3.3 \cdot 10^{-6}$ s for iPBNS
- $\Delta t = 1 \cdot 10^{-6}$ s for eDBNS
 - Low dissipation KEEP scheme
- Both code central discretization



Using central scheme

Why Do Equilibrium Wall Functions Work under $dp/dx \neq 0$?

- Under strong positive dp/dx the logarithmic layer is lost from the outside
- At the same time u_τ and therefore $y^+ = \frac{u_\tau y}{\nu}$ decreases
- Near separation the wall-function point moves into the sublayer
- Equilibrium wall functions work well even under non-equilibrium conditions



"...combining the experimental uncertainties with the lack of proven grid-convergence in most studies, one must conclude that there has not yet been a truly convincing and conclusive demonstration of any superiority of non-equilibrium over equilibrium wall-stress-models in LES. Performing such a conclusive demonstration is a high priority in the field, regardless of the eventual conclusion from it."

Laminar-Turbulent Transition

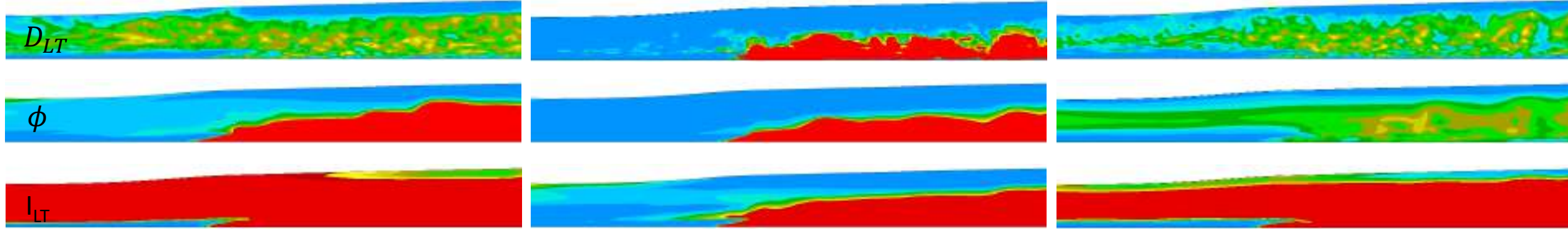
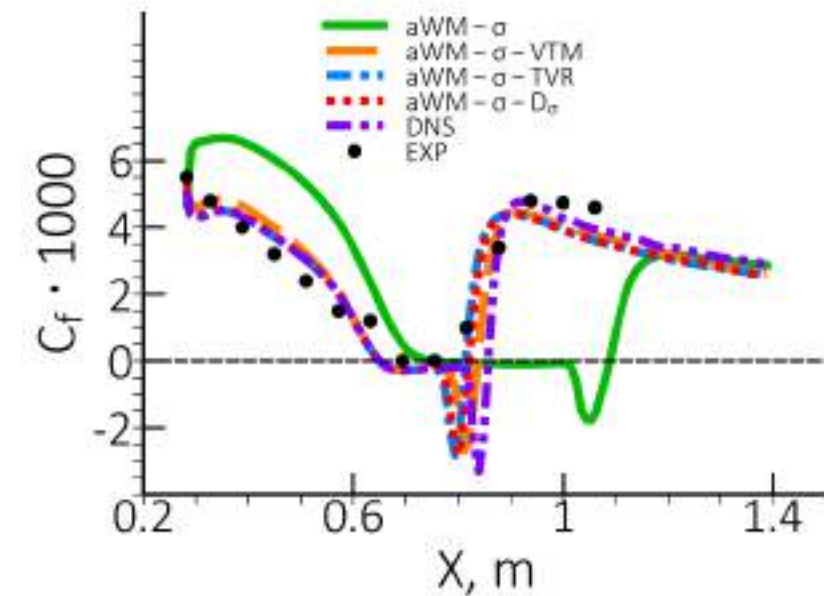
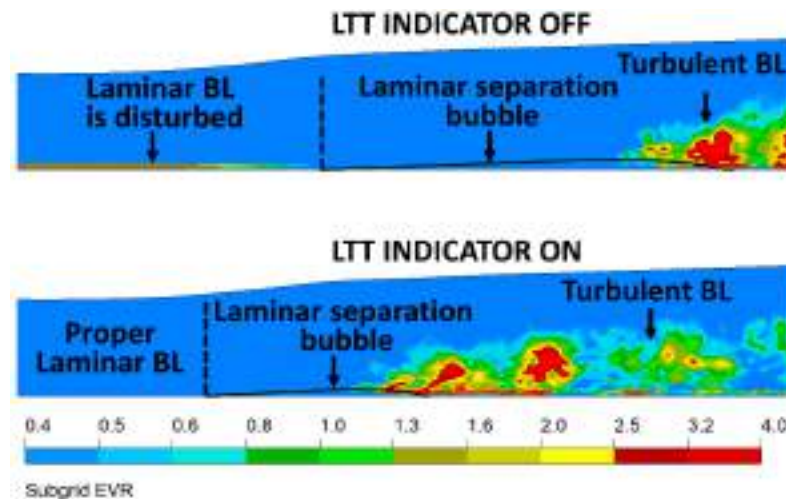
- A major difficulty arises when laminar-turbulent transition needs to be predicted
- DDES/SBES
 - Transition needs to be treated by a RANS transition model
- WFLES
 - In most cases the grid for WFLES is too coarse to resolve the thin laminar boundary layer as y^+ is too large
 - Solution – local wall-normal resolution based on indicator for laminar flow
- WMLES
 - RANS layer in near wall region will not de-activate automatically in laminar boundary layers
 - Need indicators to turn it off in laminar regions

Volino: Comparison of Different Indicators (The Coarsest Mesh)

aWM- σ -VTM

aWM- σ -TVR

aWM- σ - D_σ

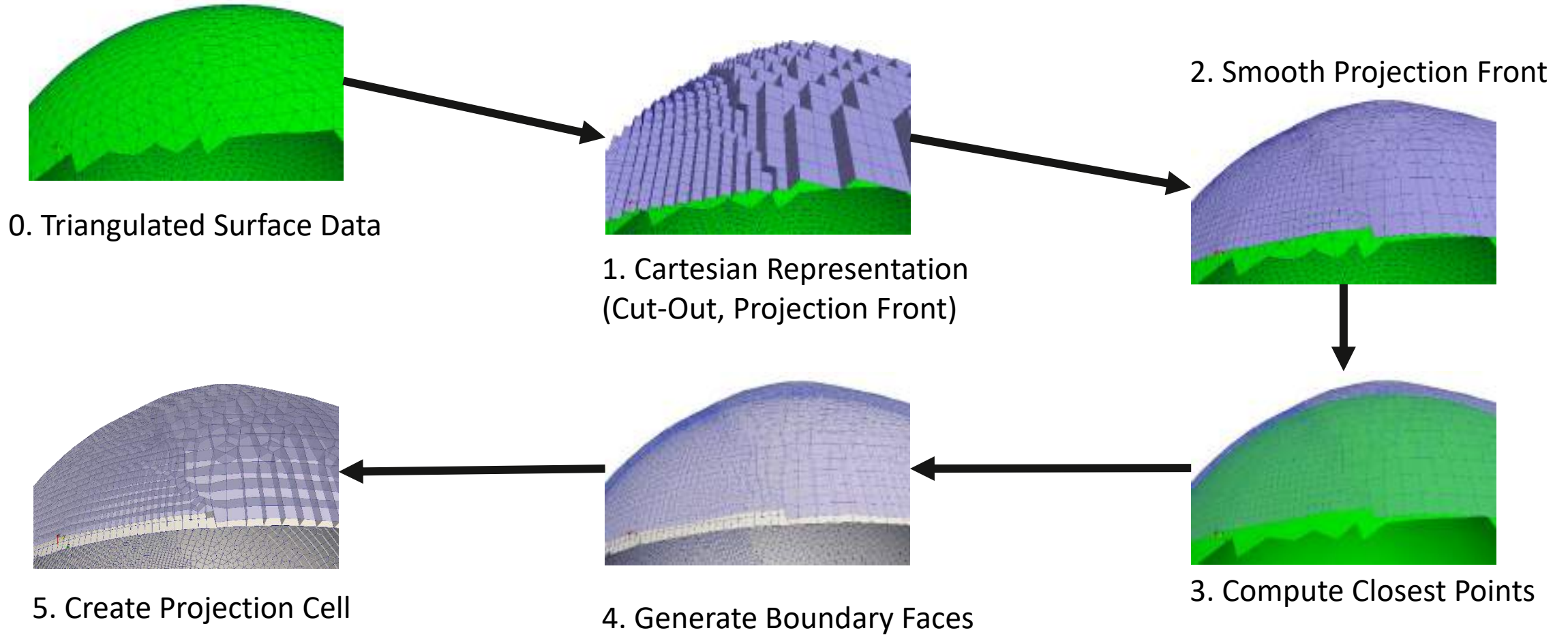


- For all the methods activated Laminar-Turbulent activator I_{LT} improves prediction of the flow
 - Proper resolution of the laminar boundary layer
 - The size of laminar separation bubble is in good agreement with DNS results and experimental data

Top-Down Meshing Strategies

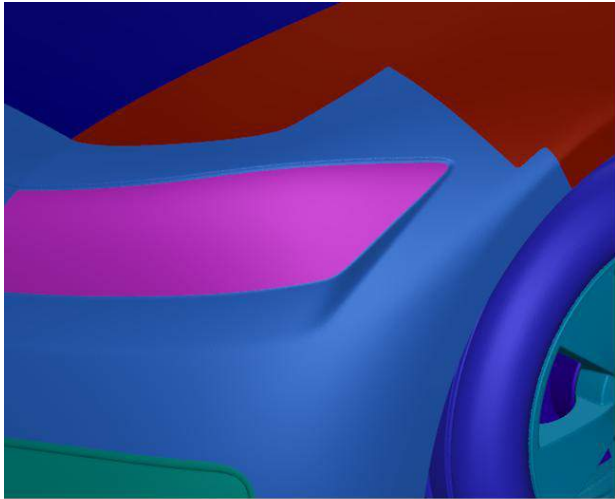
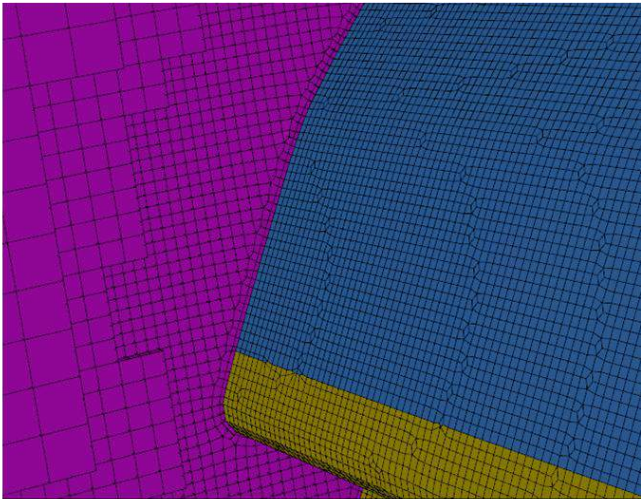
- For a body-fitted approach, meshing complex geometries can be bottle neck
- Classical bottom-up meshing do not scale well in highly parallel settings
 - From surface mesh to volume mesh
- Alternative approach is Top-Down meshing
 - Form volume to surface

Basic Steps in Normal Projection

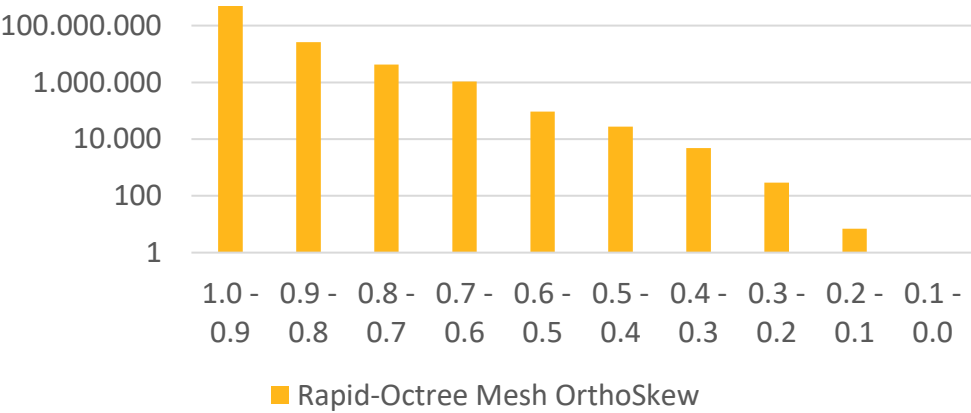
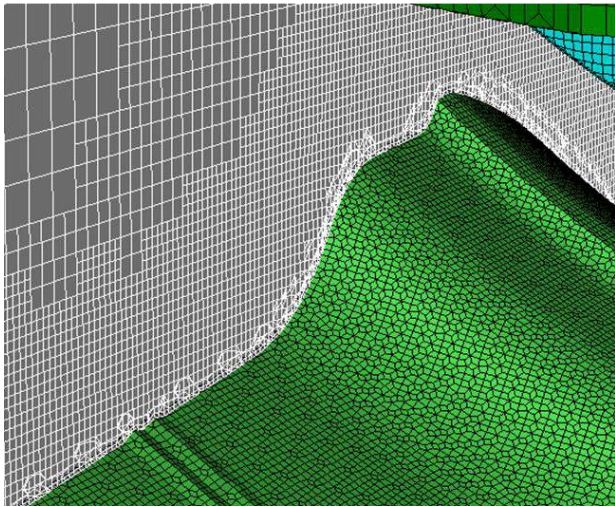
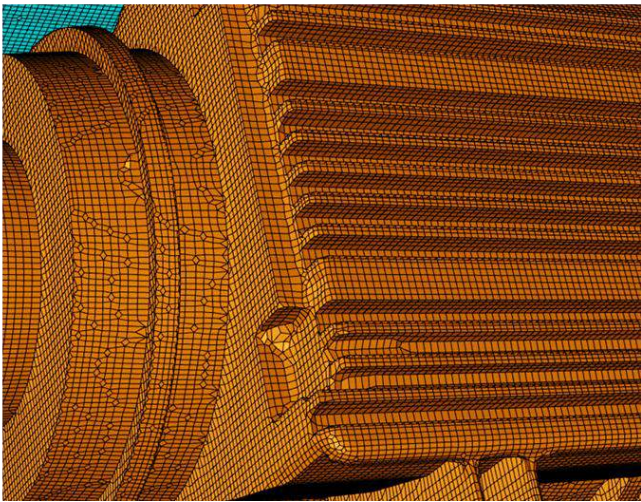


Base Mesh DrivAer Car

Number of CPU Cores for Meshing	480
Cell Count	525 252 268
Boundary Face Count	35 258 803
Meshing Time	1338.7s (~22min)
Worst Cell Orthogonal Skew	0.134
Maximum Aspect Ratio	24

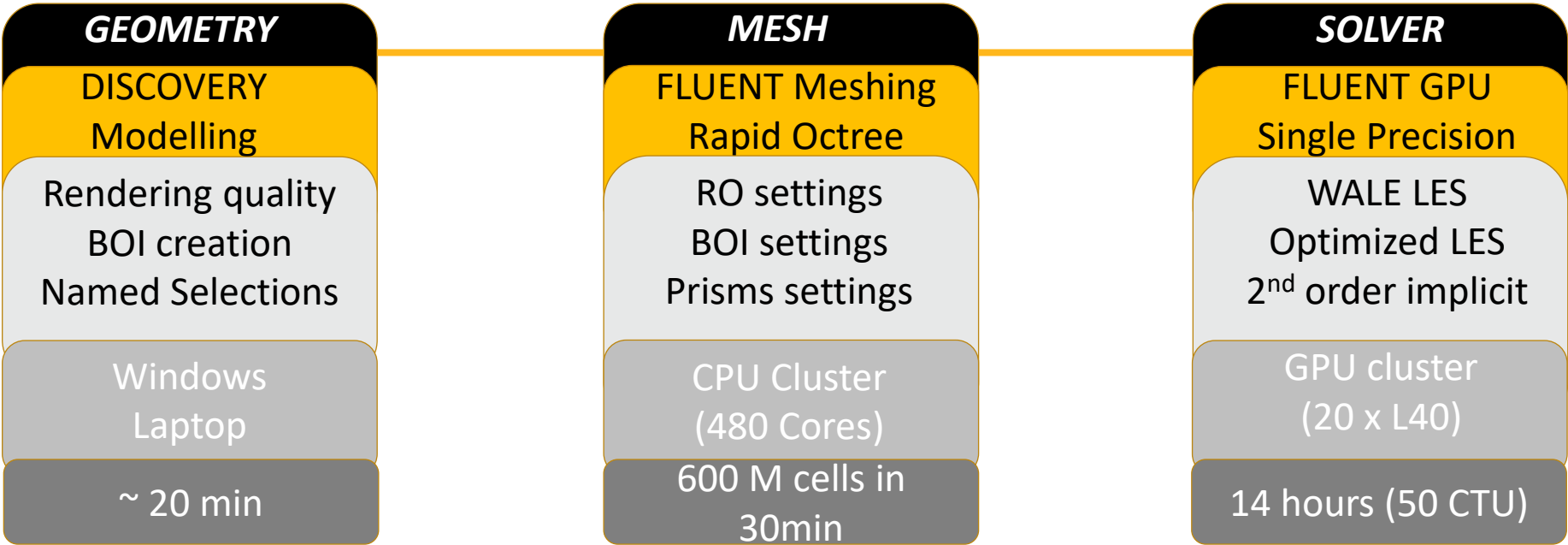
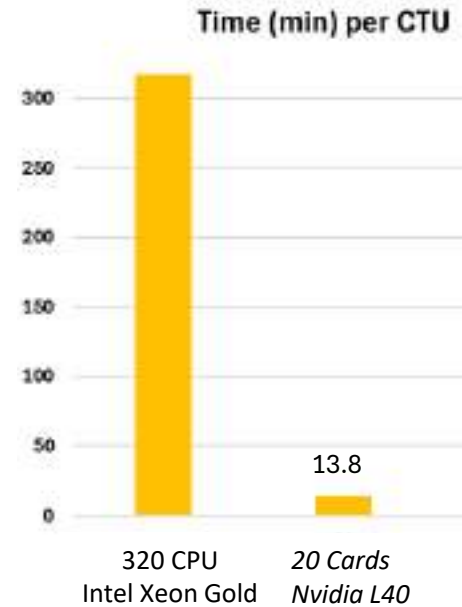
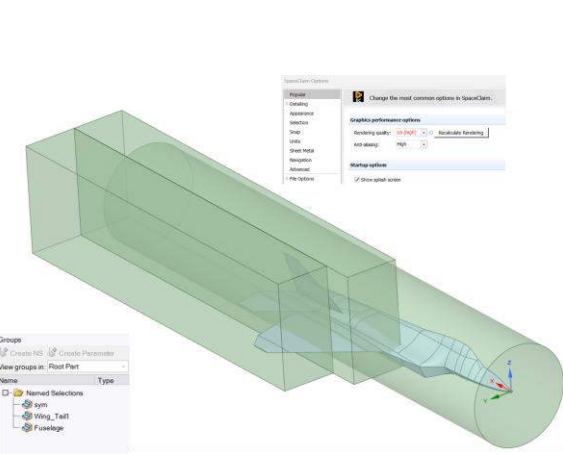
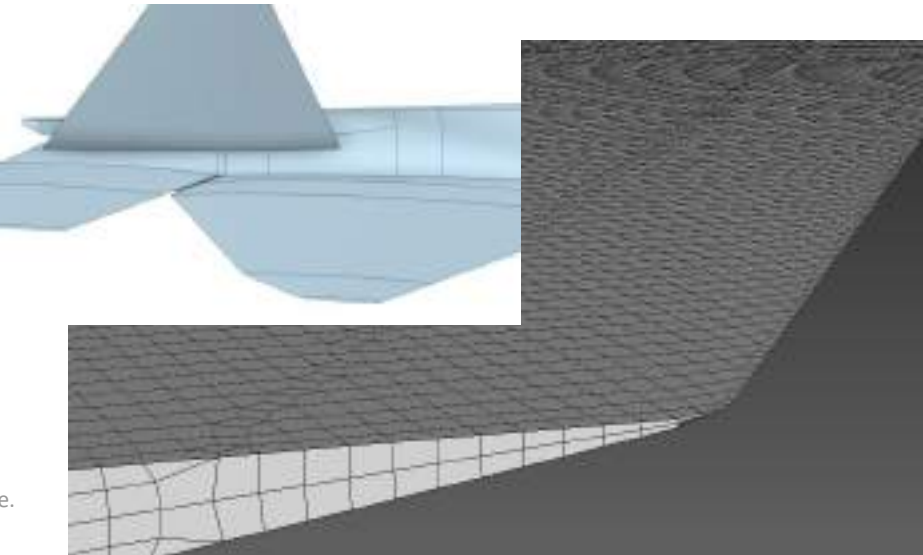
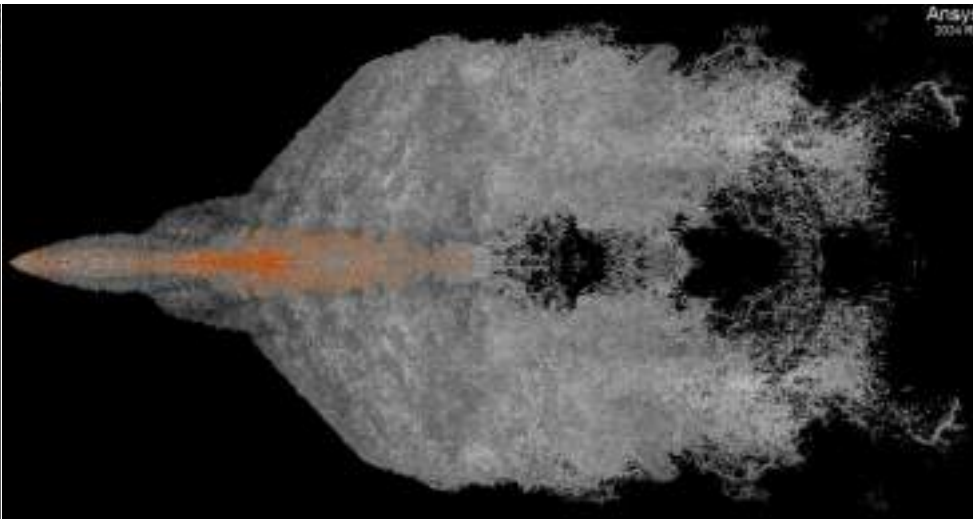


Orthogonal Quality Histogram



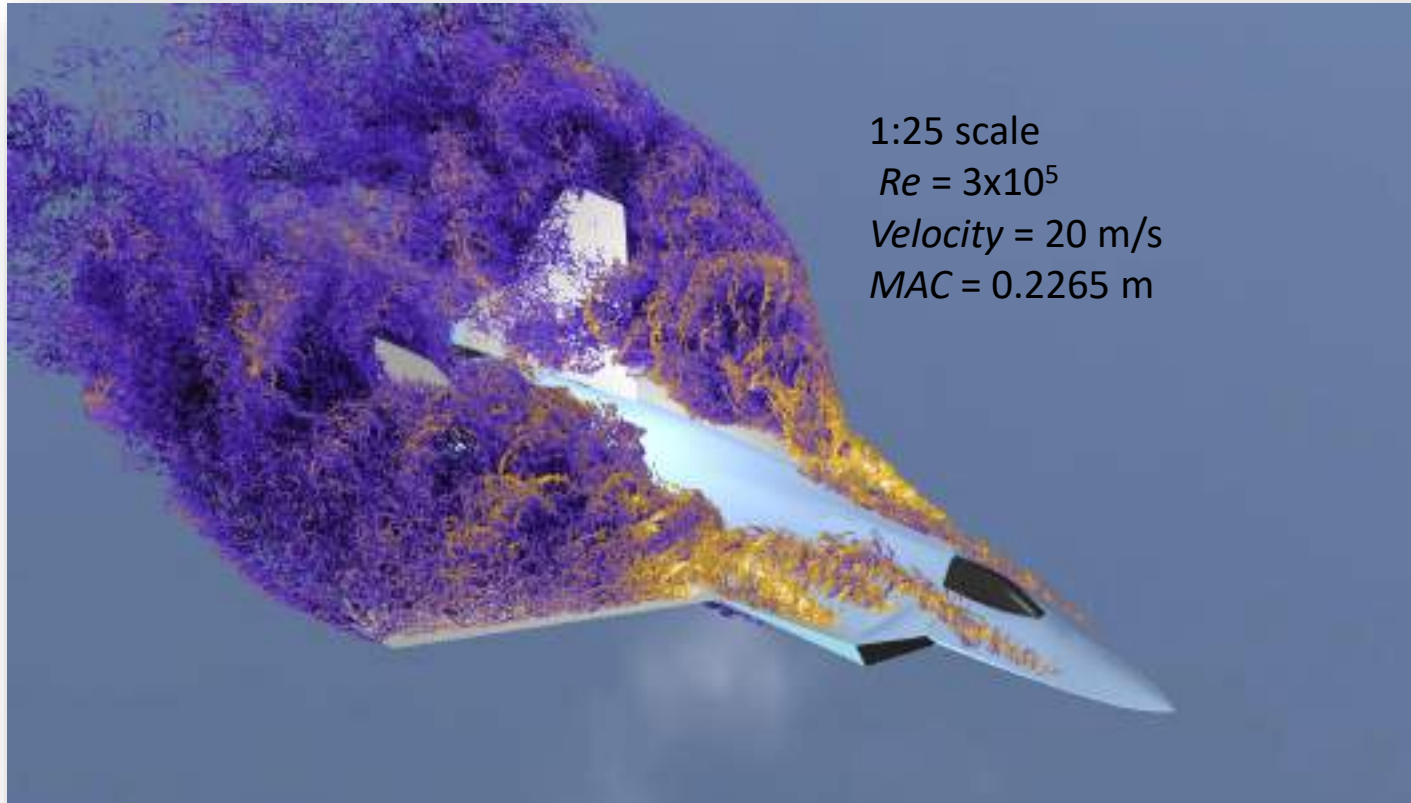
5th Gen Aircraft Workflow

Giannelis, N.F.; Bykerk, T.; Vio, G.A. A Generic Model for Benchmark Aerodynamic Analysis of Fifth-Generation High-Performance Aircraft. Aerospace 2023, 10, 746.

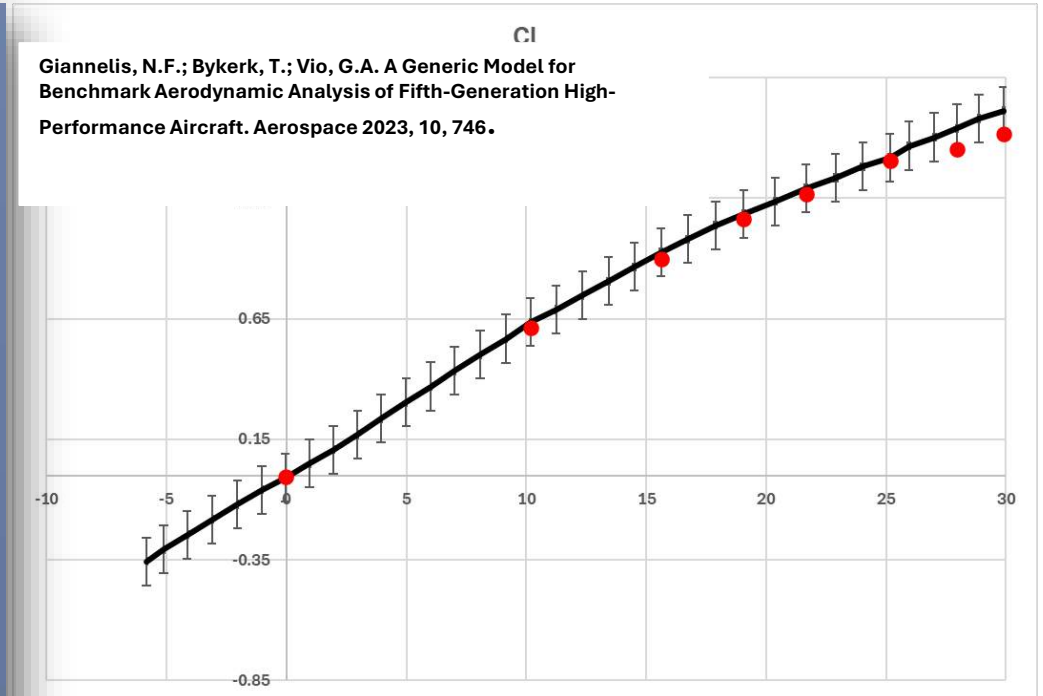


The Ansys Fluent GPU Solver in Practice

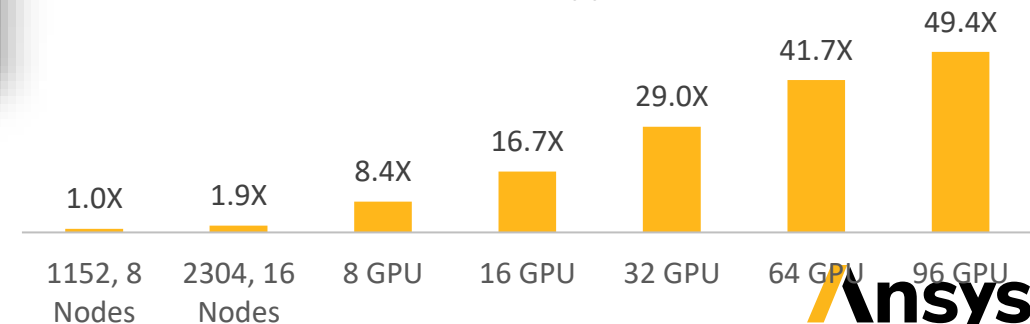
5th Gen High-Performance Aircraft



Lift coefficients
at different attack angles

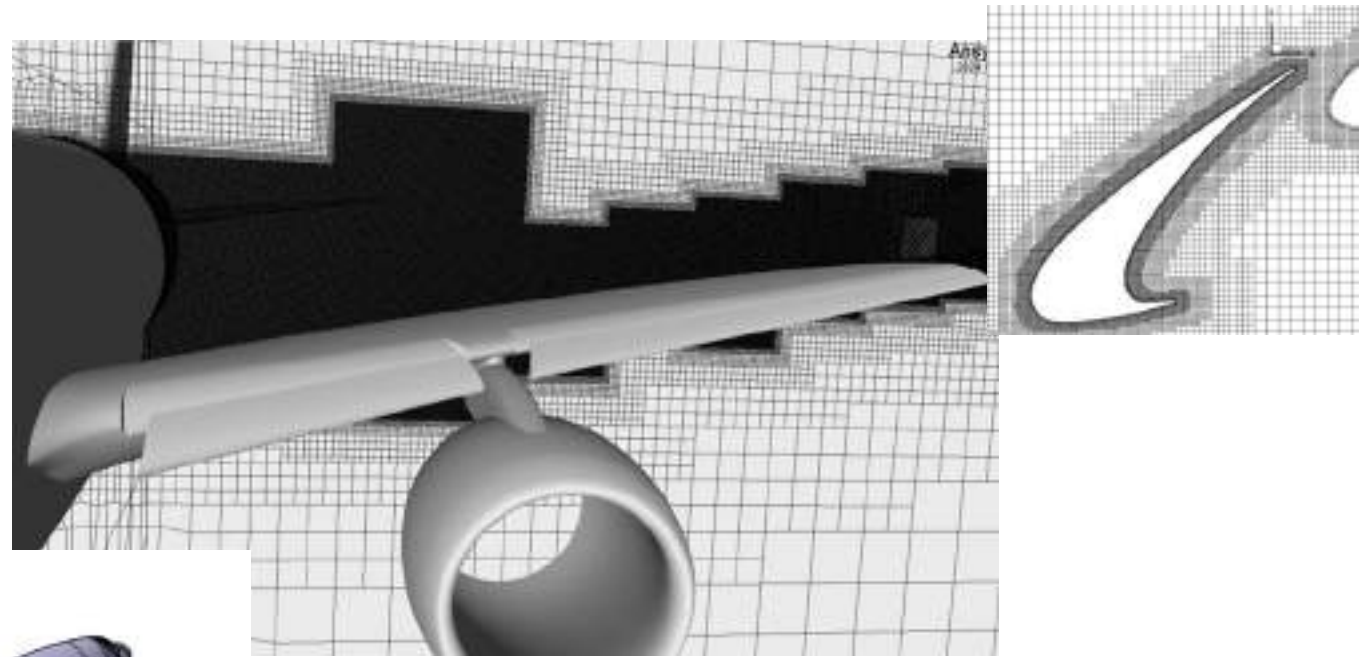


NVIDIA Grace CPUs and Grace Hopper GPUs



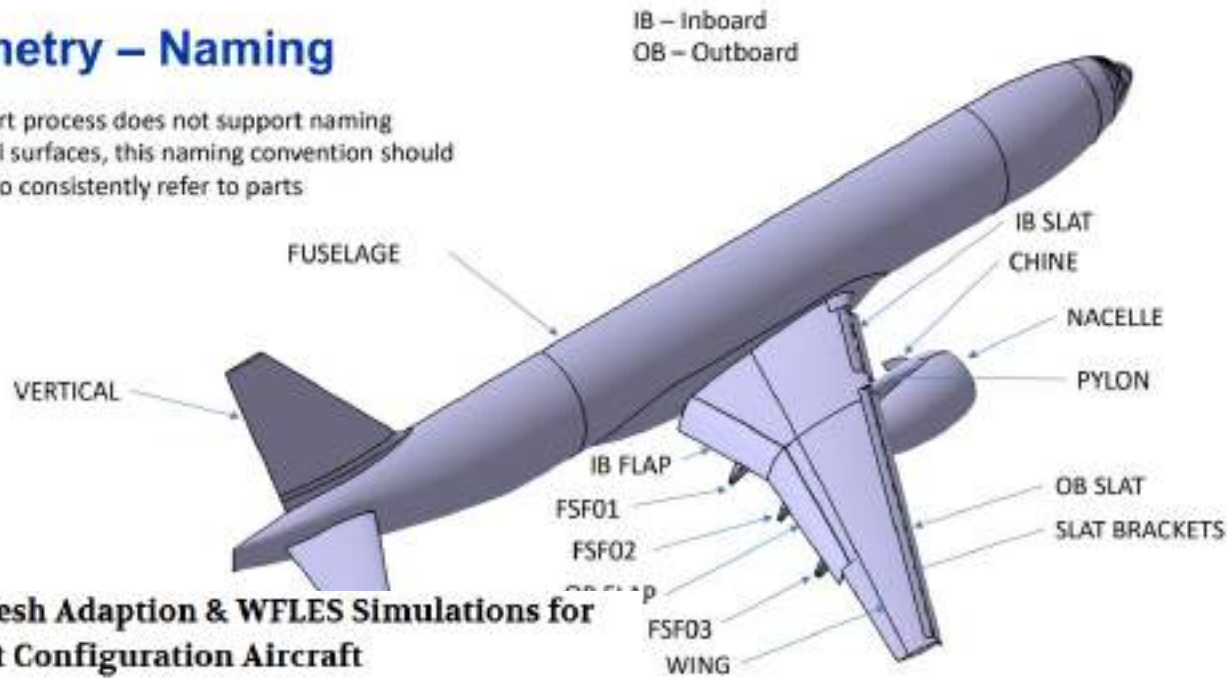
High-Lift AIAA Workshop Case

- High Lift Workshop 5 config 2.4
- Octree grid 20 min on 500 CPUs
- WFLES on GPU solver is tested on Rapid Octree mesh



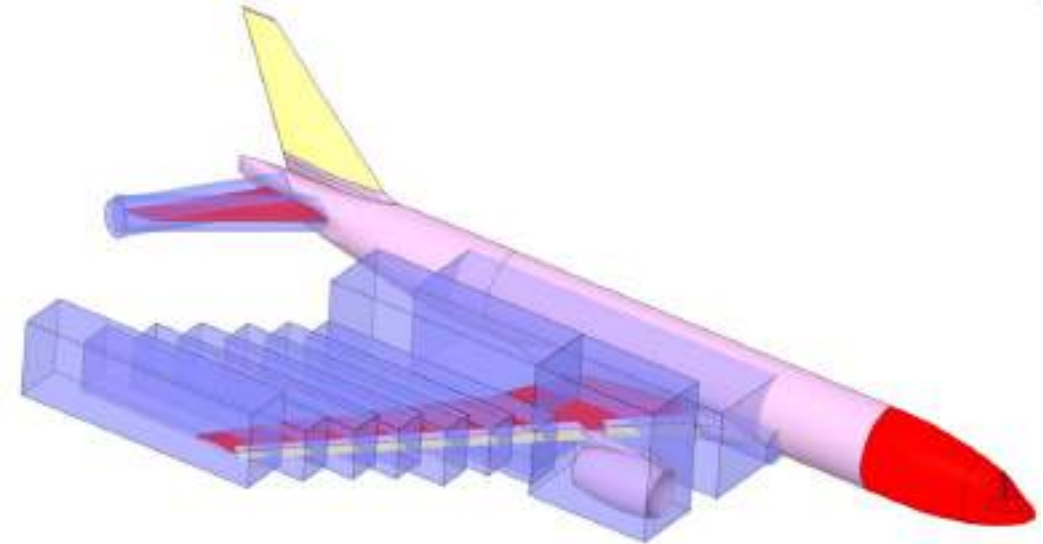
Geometry – Naming

The export process does not support naming individual surfaces, this naming convention should be used to consistently refer to parts



RANS Mesh Adaption & WFLES Simulations for High Lift Configuration Aircraft

Corrections for this article
Cridhian/Ajiga, Krishna Zora, Rachid Madi, Tobias Gierg, Achilles Nikitas, Jeya Selva, Latif Zori and Boris Makarov
AIAA 2025-0496
Session: Special Session: High-Lift Prediction Workshop (HLPW-5): RANS and Adaptive Simulations (Invited)
Published Online: 3 Jan 2025 • <https://doi.org/10.2514/6.2025-0496>

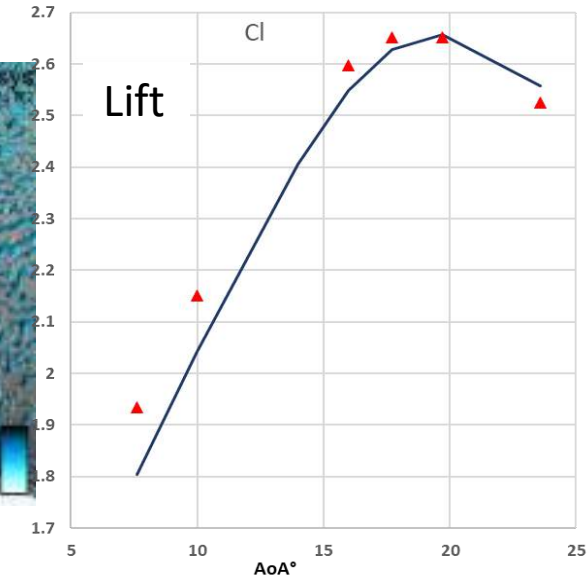
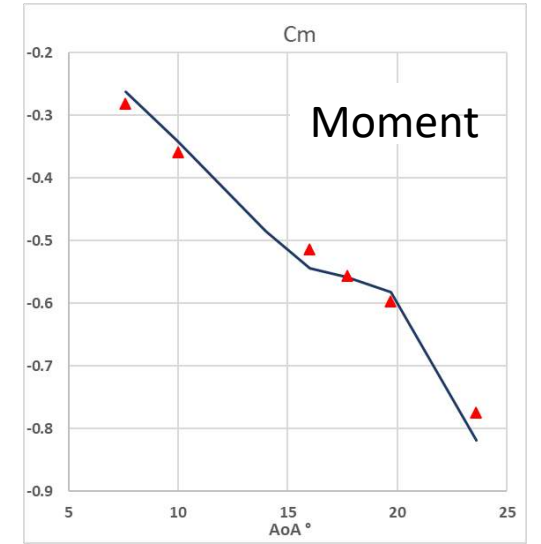
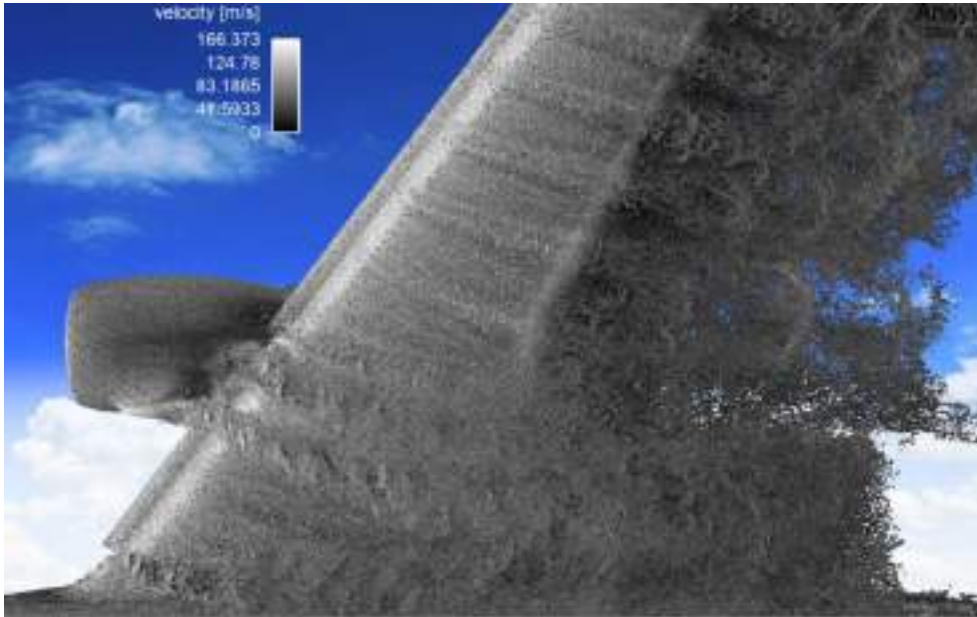
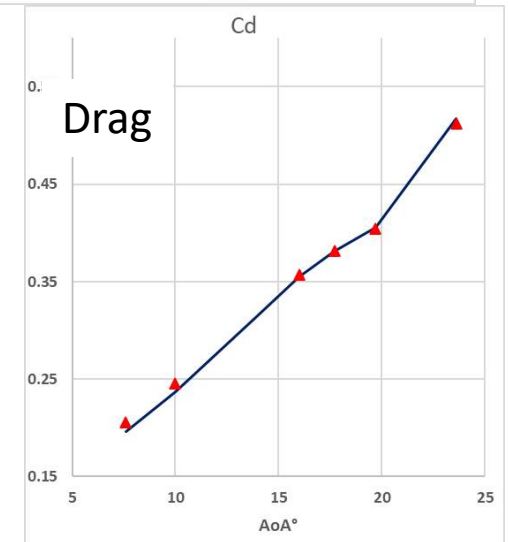


1 billion cells in 19 min on 512 Cores

New Era for Fast Transient

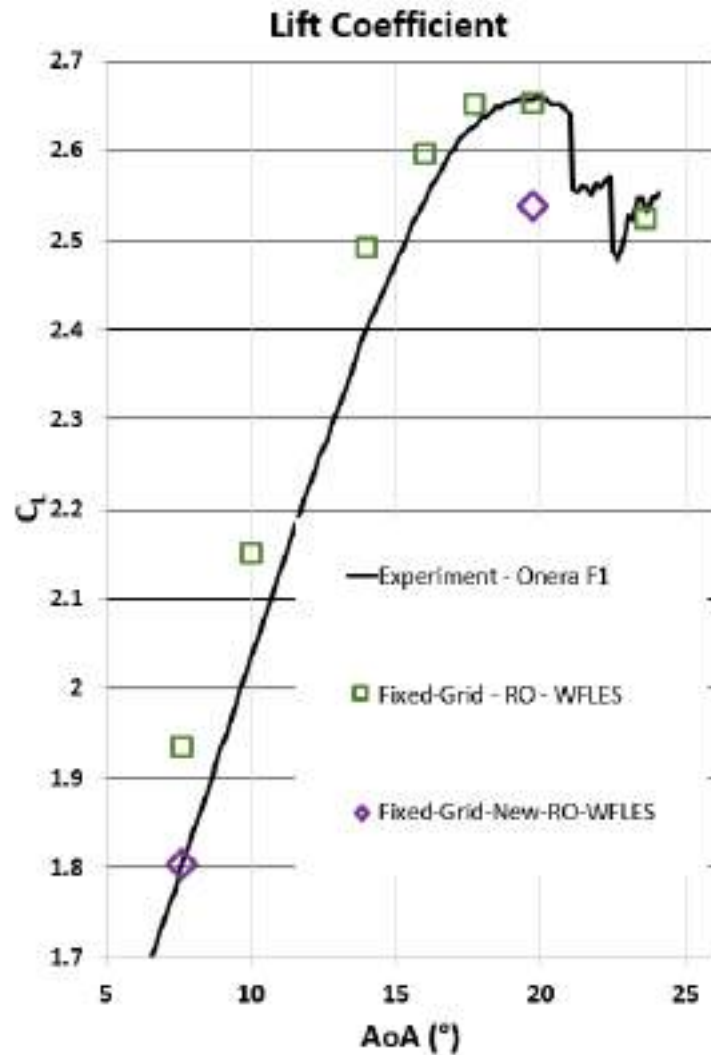
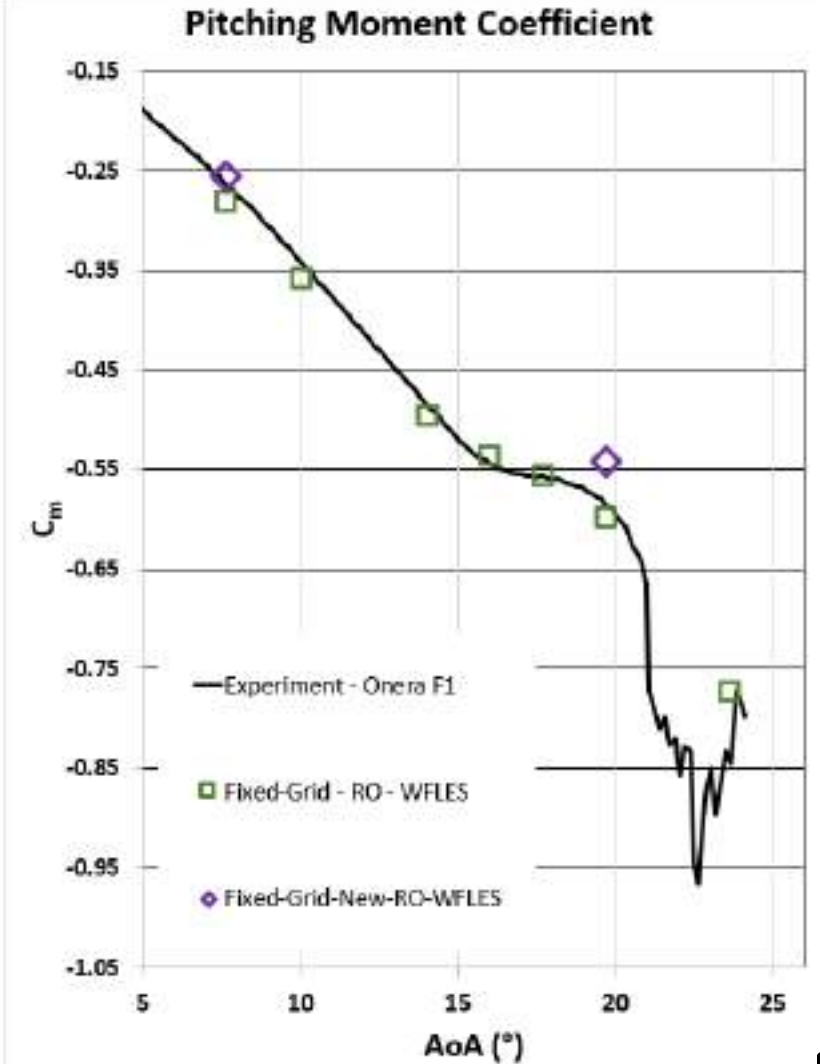


Solutions



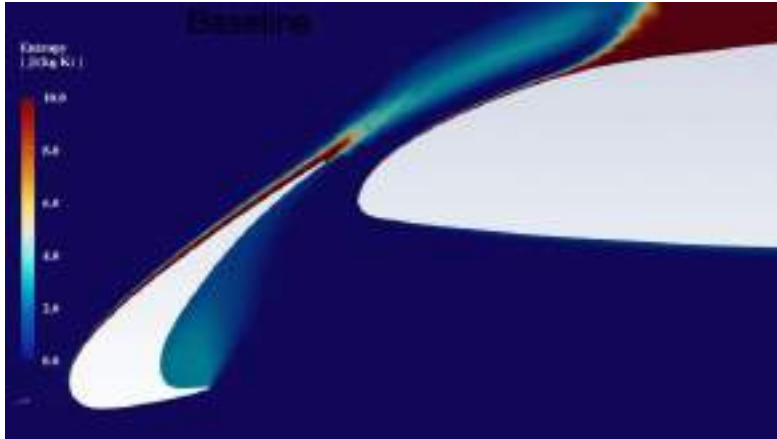
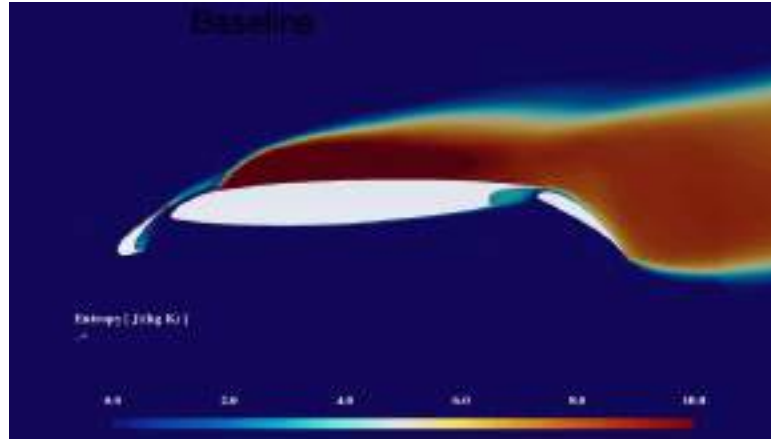
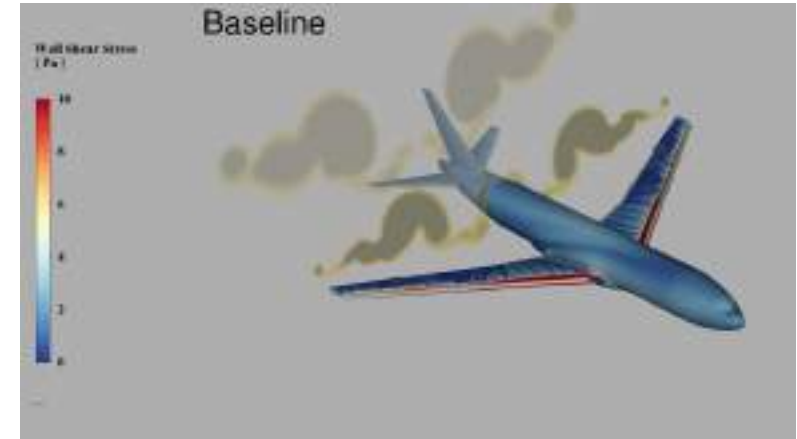
- Optimized LES numerics
- 2nd order Implicit time discretization
- Time step $4e-5$ s / 2it per timestep
- 32 QuadL40 Nvidia cards (44 Gb)
- 1 hour / CTU (0.1s)

Aerodynamic Coefficient

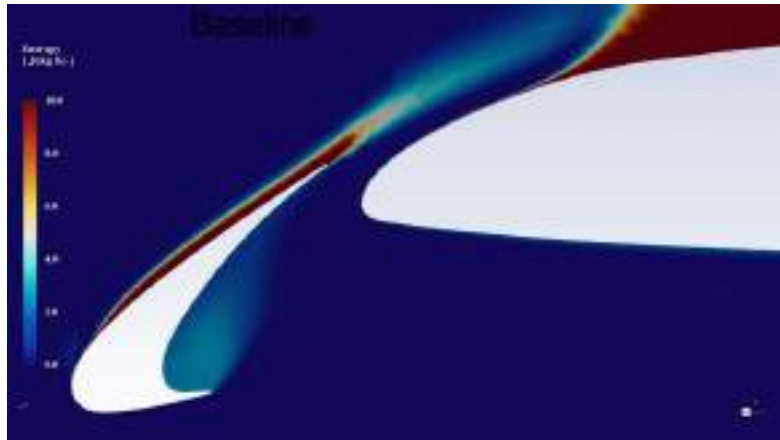
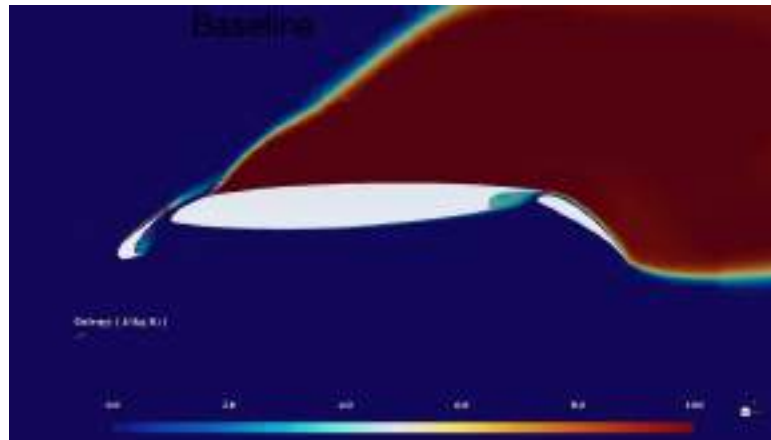


Case 2.3: RANS Mesh Adaption Solution - $AoA = 20.7^\circ$ - Stall AoA

SA



K- ω SST + γ -Re θ



RANS Mesh Adaption & WFLES Simulations for High Lift Configuration Aircraft

1 Corrections for this article: [▼](#)

Christian Allaga, Krishna Zora, Rachid Malik, Tobias Berg, Achilleas Kyriak, Jaya Selva, Laith Zori and Boris Maysan

AIAA 2025-0498

Session: Special Session: High-Lift-Prediction-Workshop (HLPW-5): RANS and Adaptive Simulations (Invited)

Published Online: 3 Jan 2025 • <https://doi.org/10.2514/6.2025-0498>



Case 2.3: RANS Fixed Grid vs RANS Mesh Adaption

- Experimental results in solid lines
- Fluent Fixed Grid RANS SA results in dash lines
 - ANSA Level C (Fine) mesh
- Fluent PUMA Combined Hessian Indicator (RANS Primitive + Entropy) with RANS results in no fill markers
 - ANSA Level A (71 million cells) is used to start the adaption
 - No surface mesh adaption
 - Adapt ANSA Level A with SA in yellow squares
 - Adapt ANSA Level A with K- ω SST + γ -Re $_{\theta}$ in red triangles

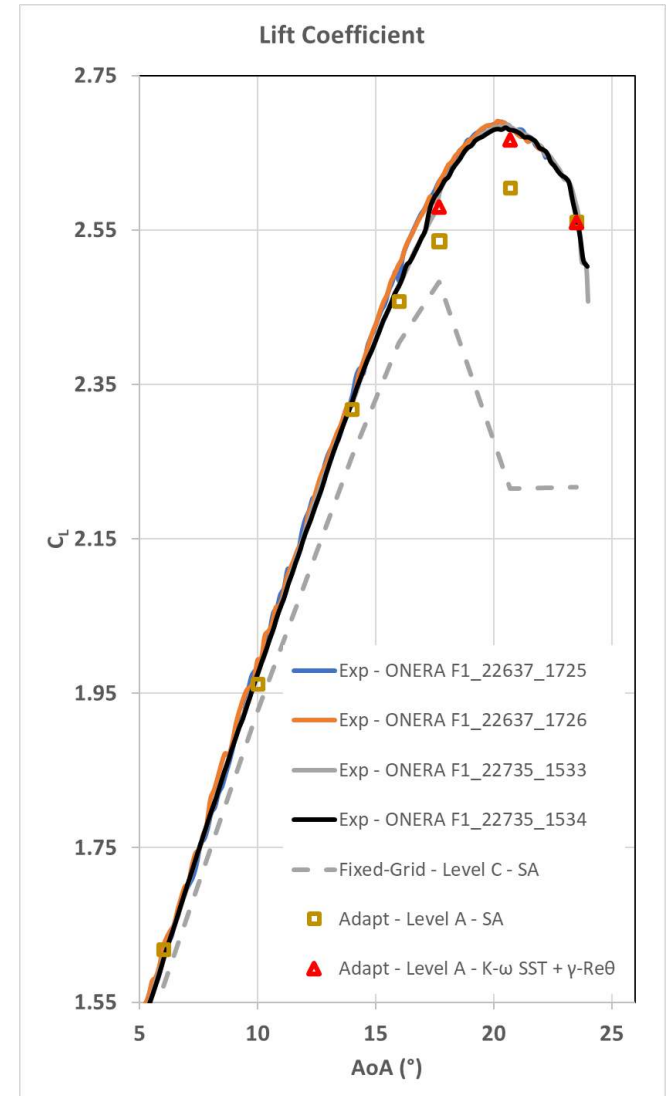


Case 2.3: ONERA_LRM-WBSFHV

Slats & Flaps Deployed

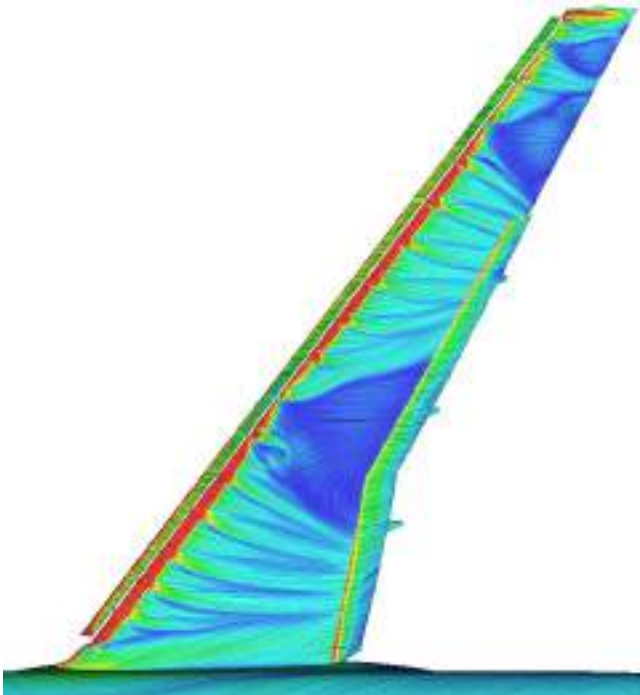
Mesh Size in Million Cells

	Coarse	Medium	Fine
Fluent SA	71	107	226
Fluent PUMA SA	~300		
Fluent PUMA K- ω SST + γ -Re $_{\theta}$	~400		

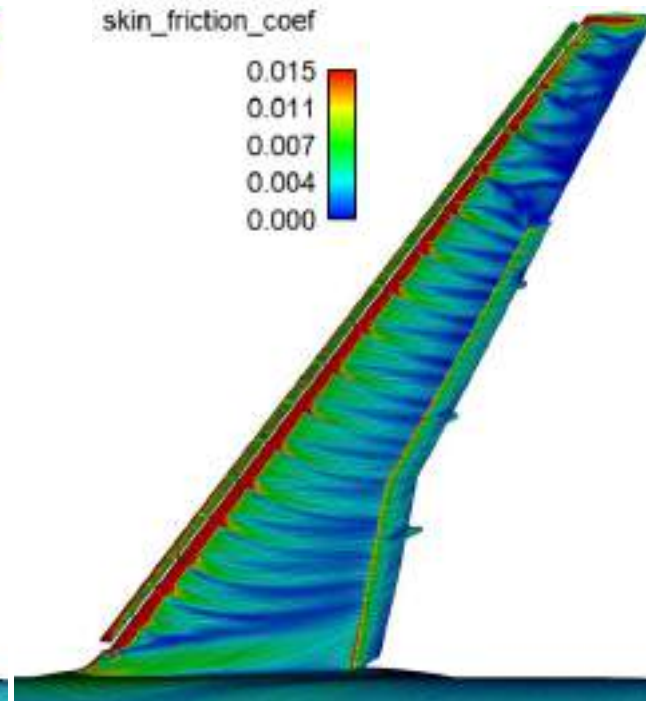


Case 2.3: RANS OilFlow Solutions - $AoA = 20.7^\circ$

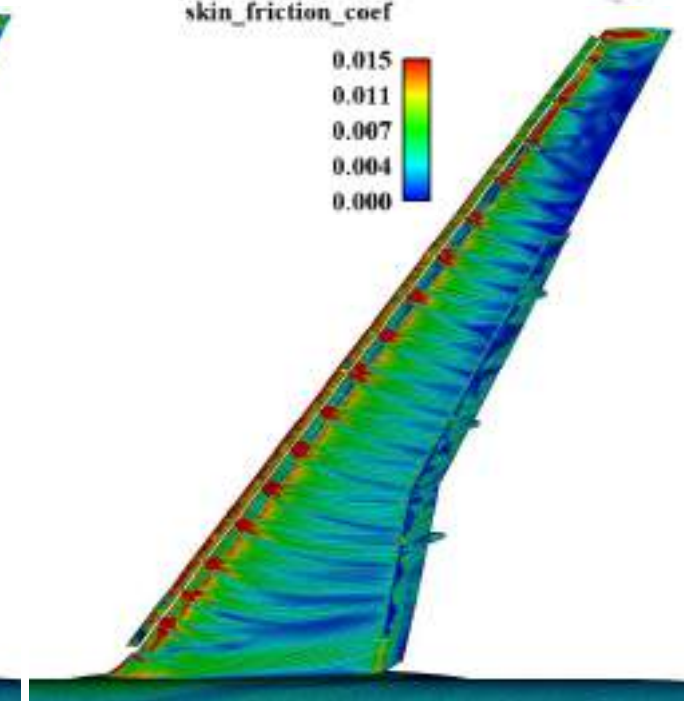
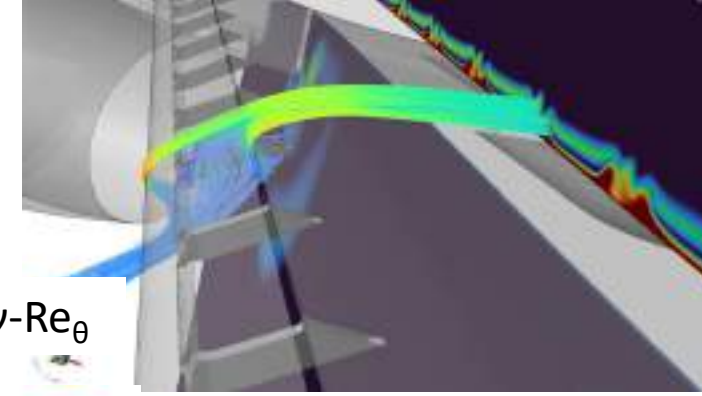
Fixed Grid - Level C - SA



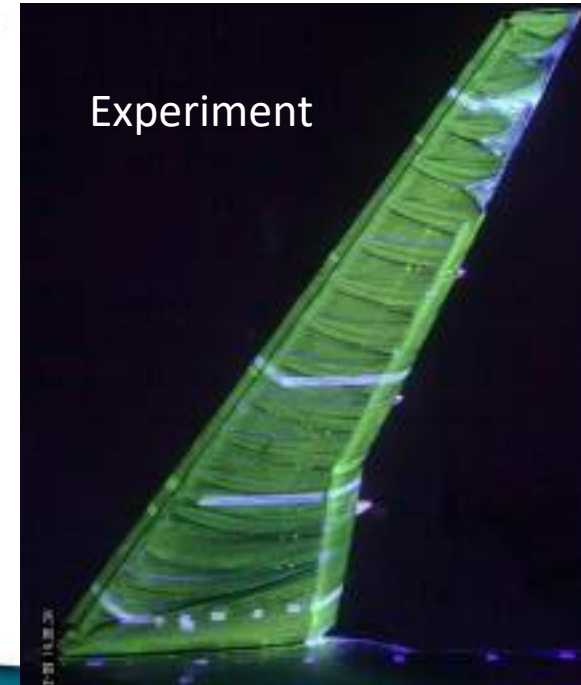
Adapt - Level A - SA



Adapt - Level A - K- ω SST + γ - Re_θ



Experiment



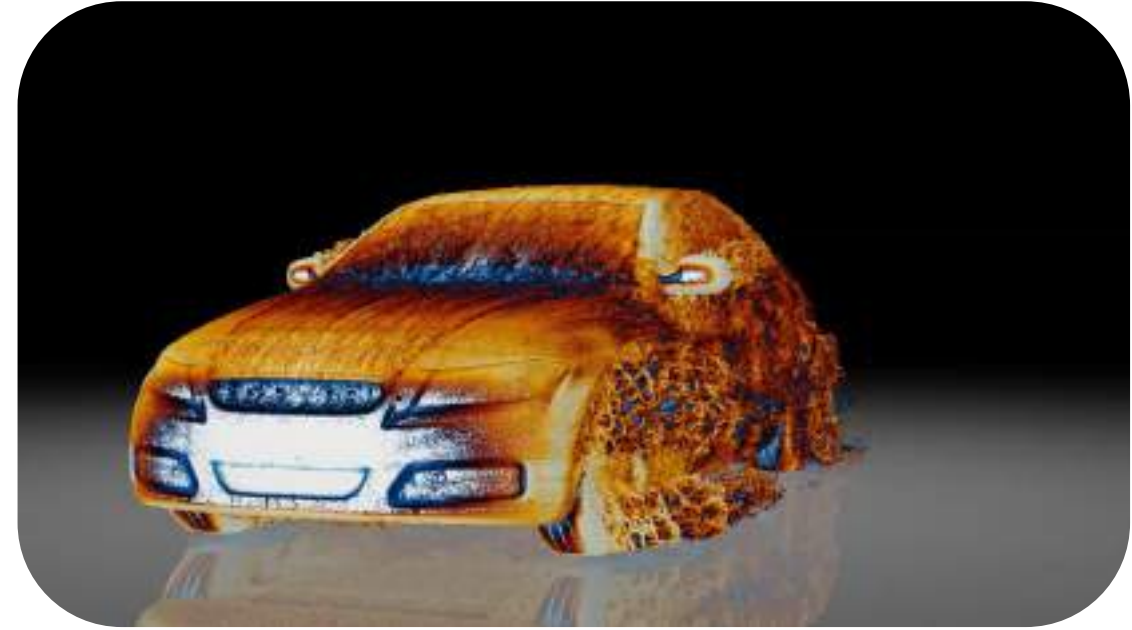
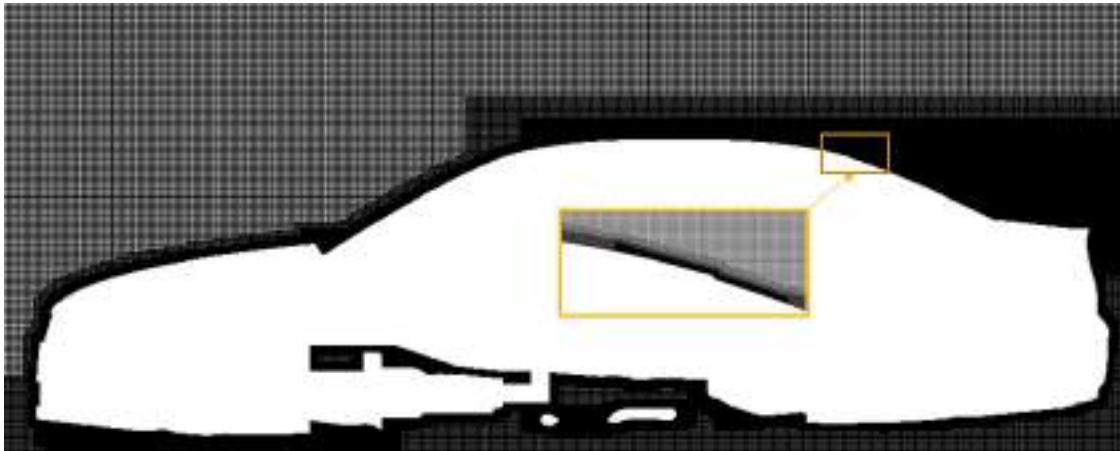
RANS is still relevant even after 100 Years

Auto CFD Workshop

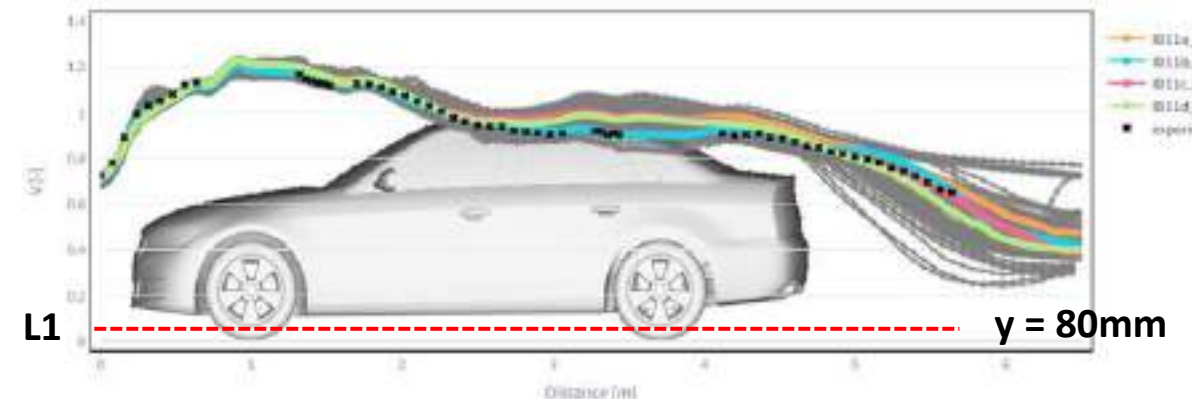
5th Automotive CFD Prediction Workshop

October 8th-9th 2026, Dama Technopolo, Bologna, Italy

- DrivAer Hatchback
- Showcasing our solution for auto external aero
 - Application of new, automated meshing
 - Emphasis on speed



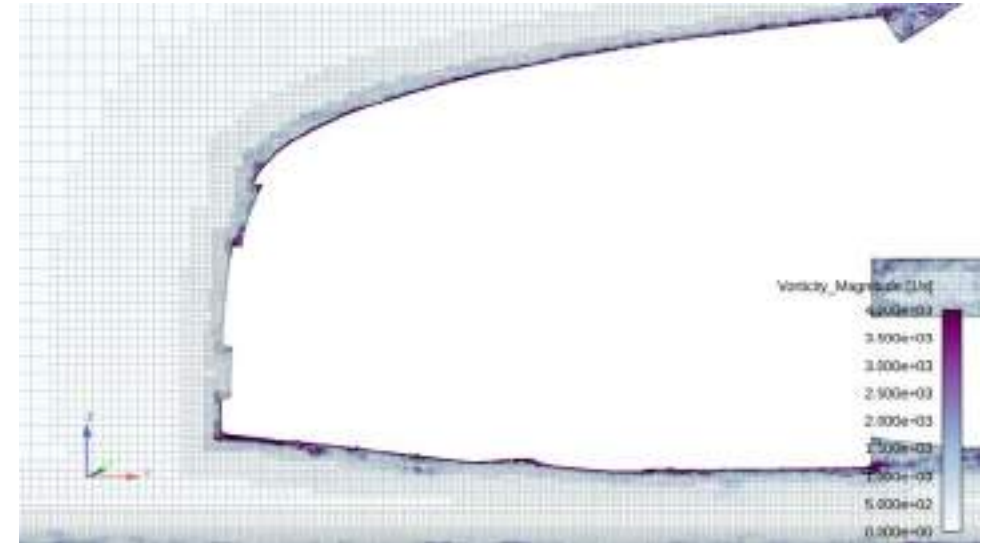
Ansys solutions (colour) for WFLES and SBES
against all workshop results and data



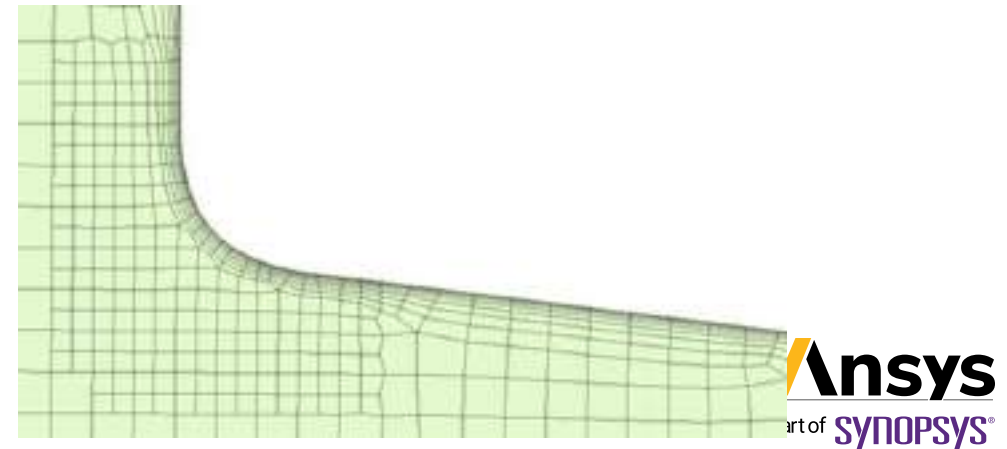
Work in progress for AutoCFD5 Workshop

- Run WFLES and SBES on 'minimum meshes'
 - WFLES has 128 million cells
 - SBES has 135 million cells (same mesh + 8 prisms)
- Explore relative accuracy of both methods
- Use best method and explore potential for improved predictions
 - Overall forces
 - Delta between case 2a/2b variants

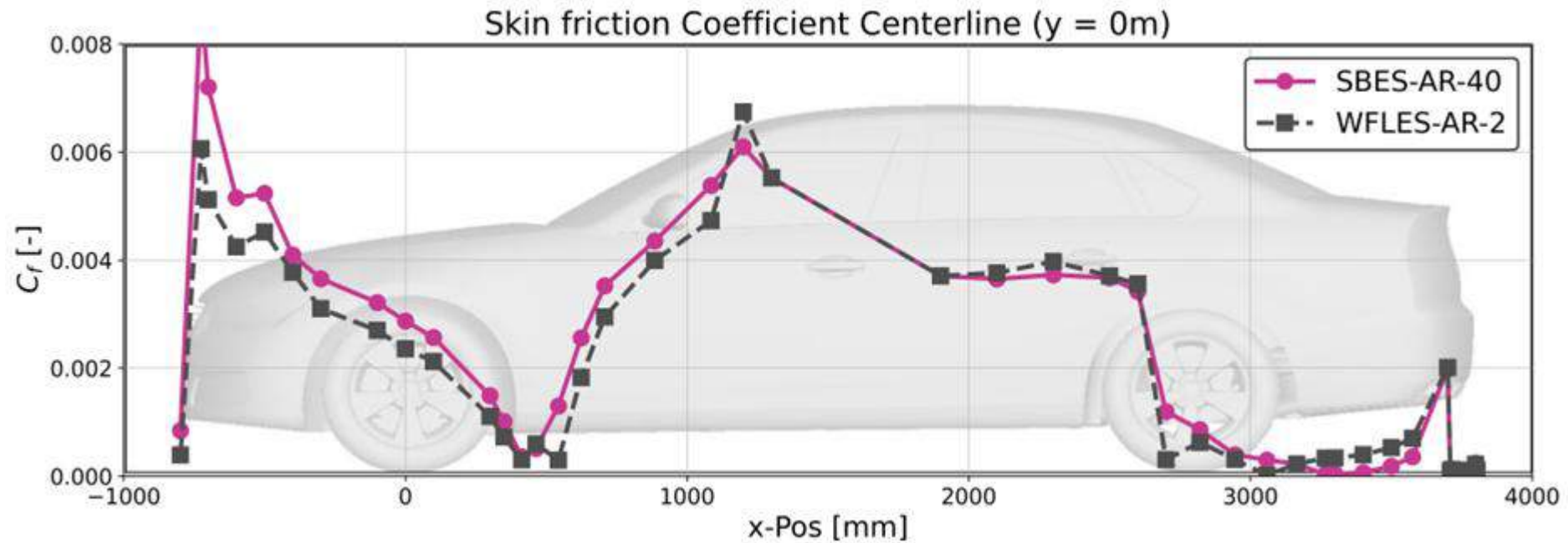
Base mesh for WFLES



Base mesh + prisms for SBES



Skin Friction Coefficient

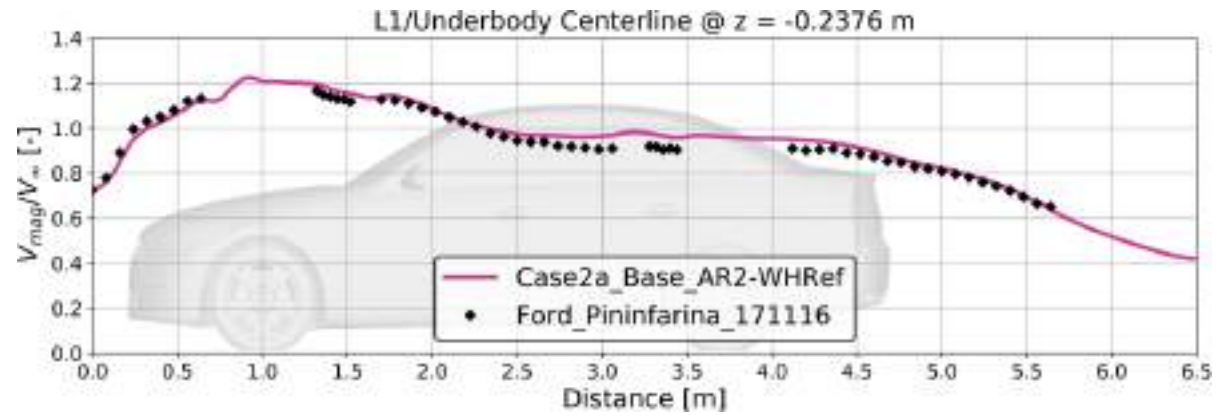


Consistency check for WFLES

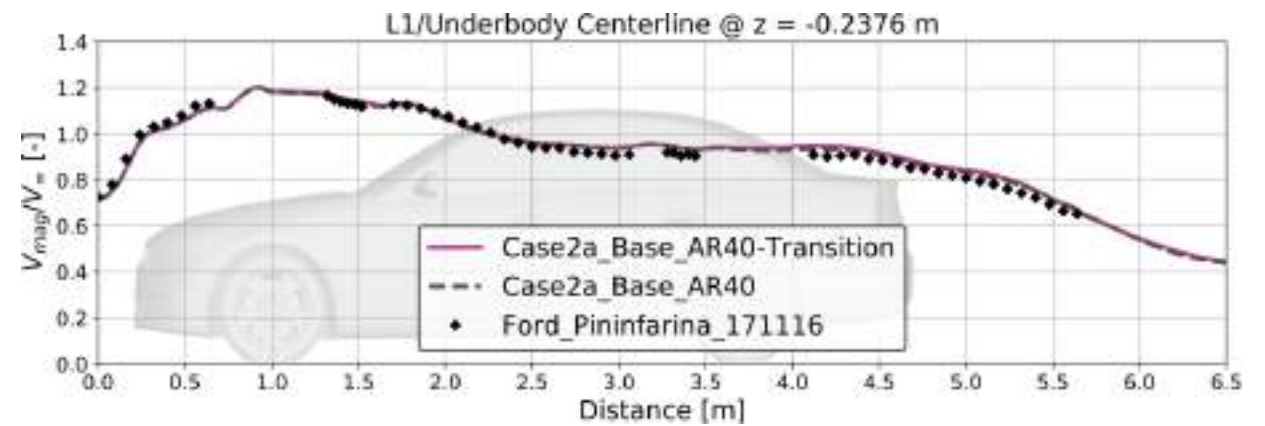
- WFLES gives C_f very similar to SBES/RANS

L1-Underbody Centreline case 2a

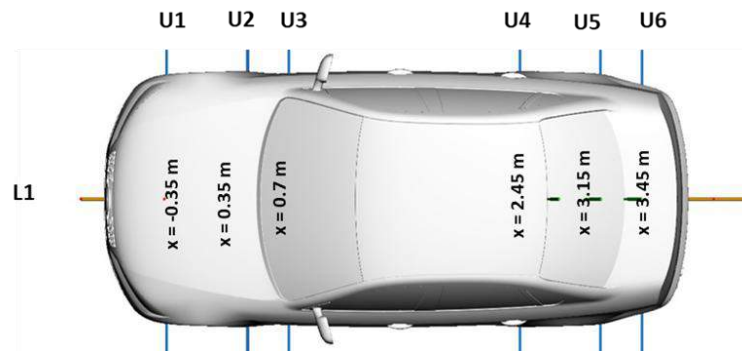
WFLES



SBES



U-Velocity

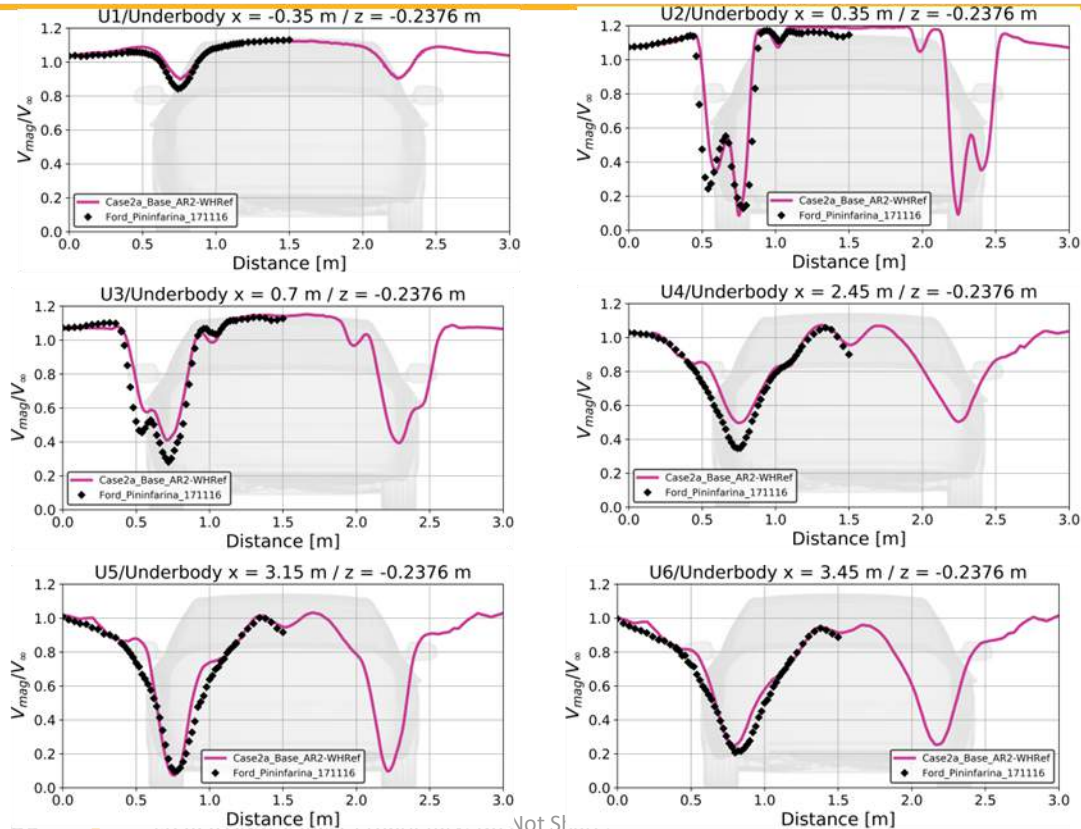
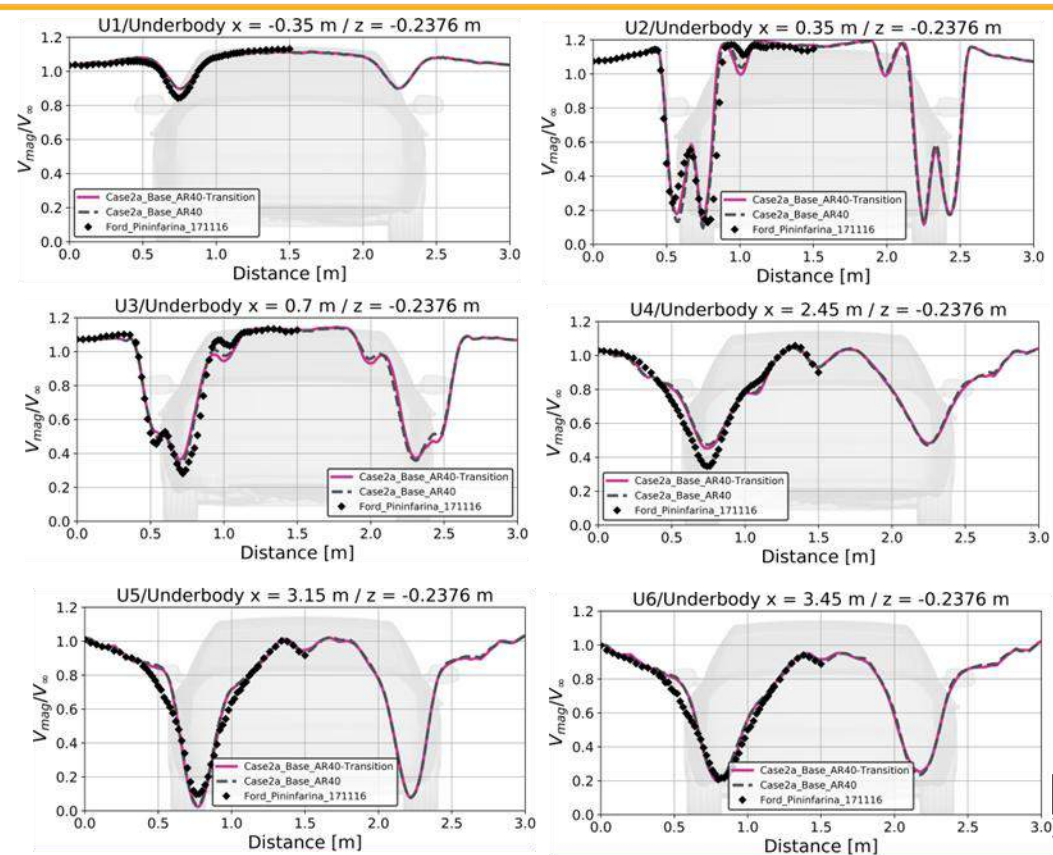


WFLES

Case2a

SBES

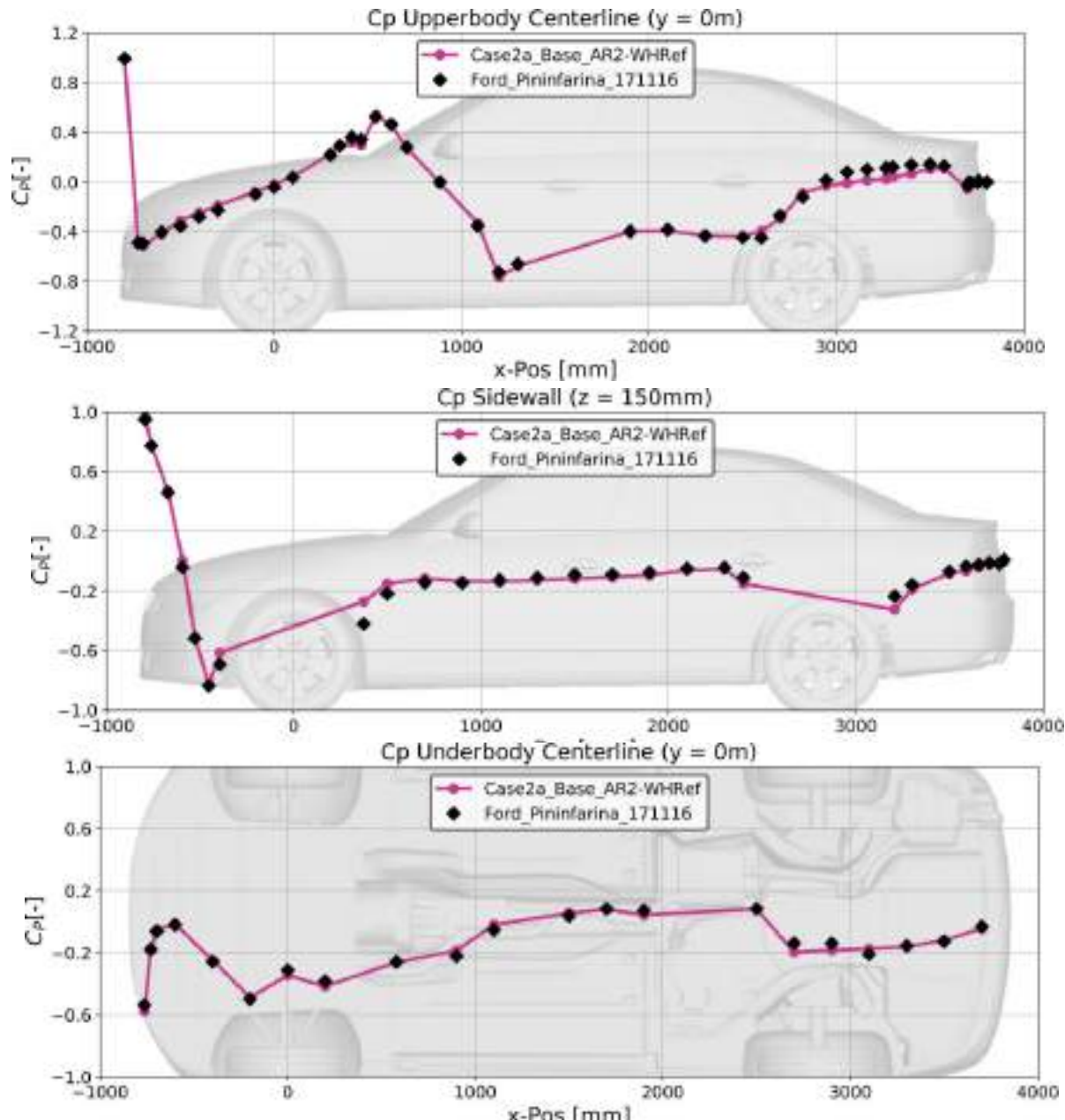
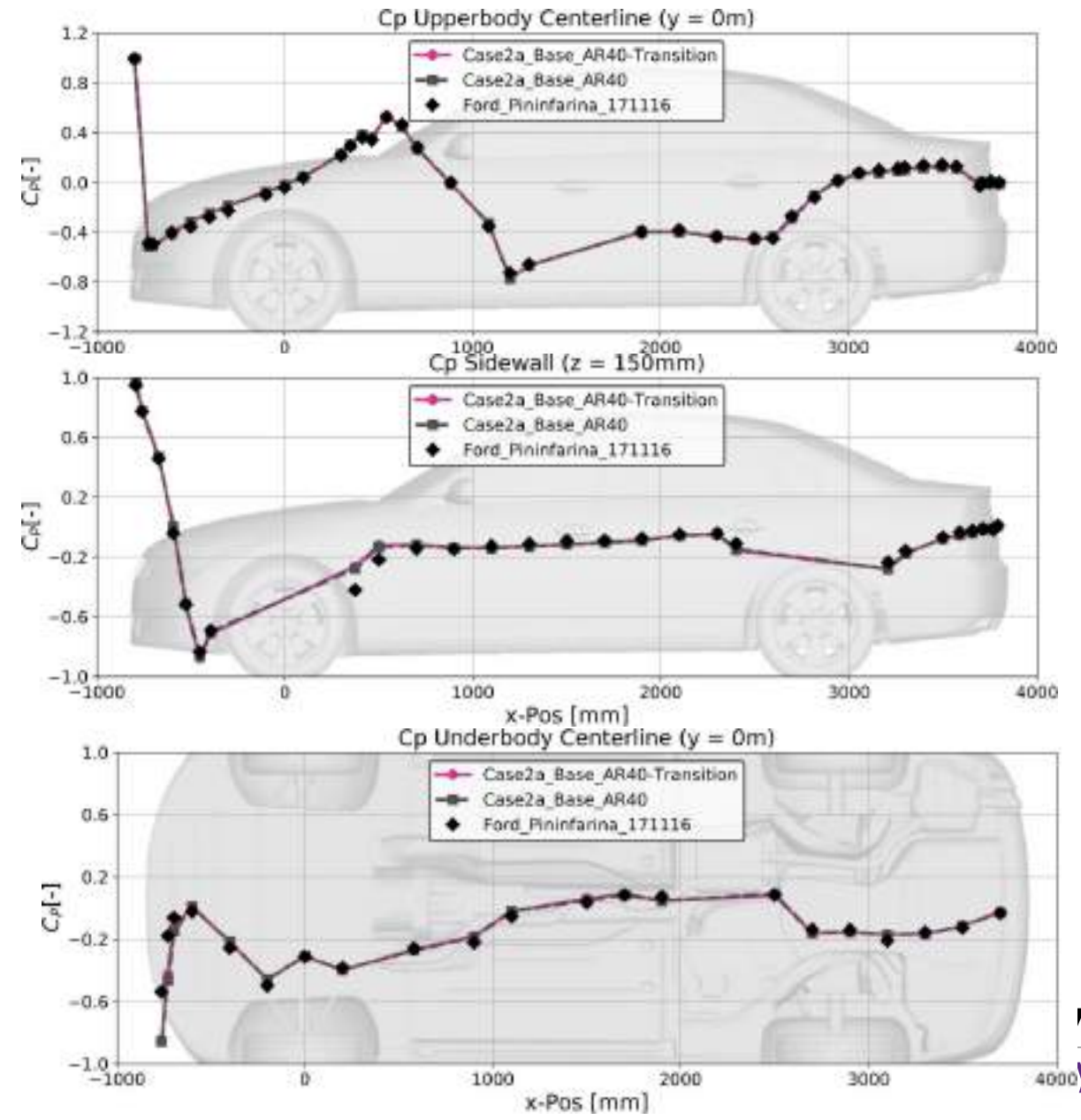
Case2a



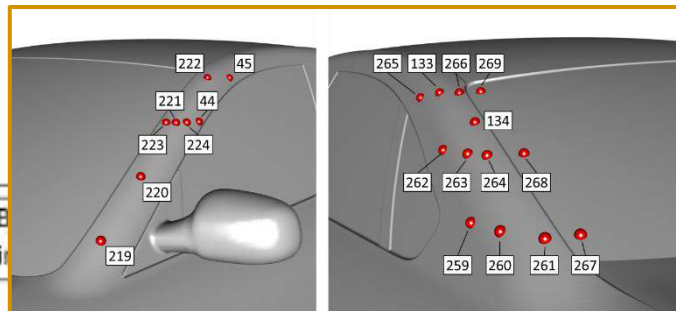
WFLES

Surface Pressure

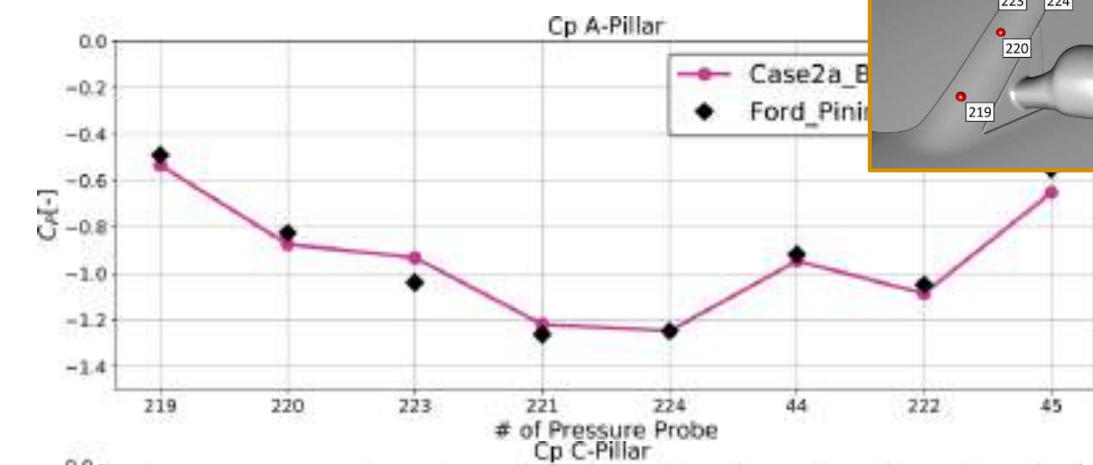
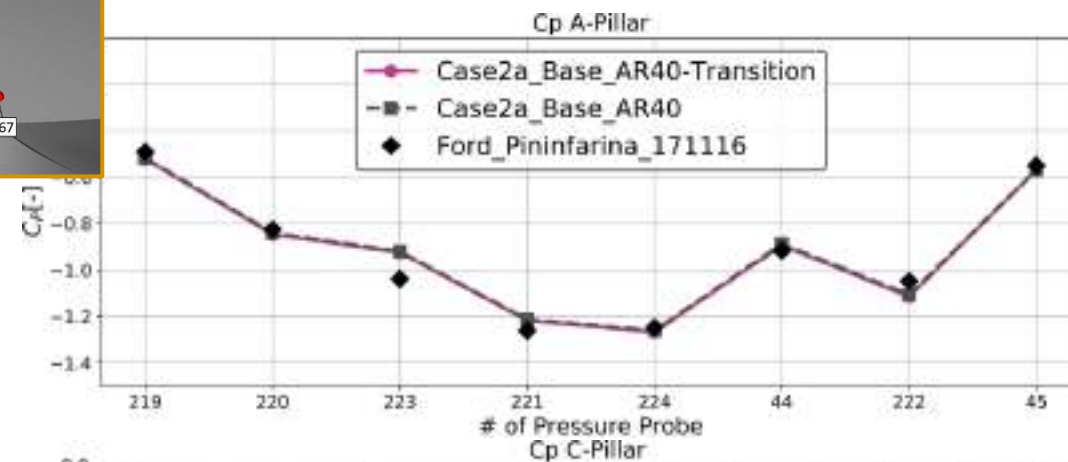
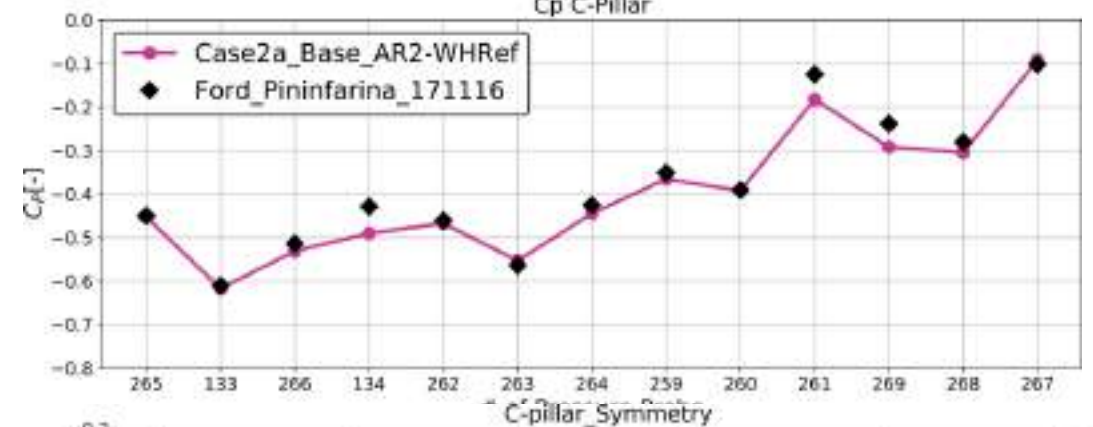
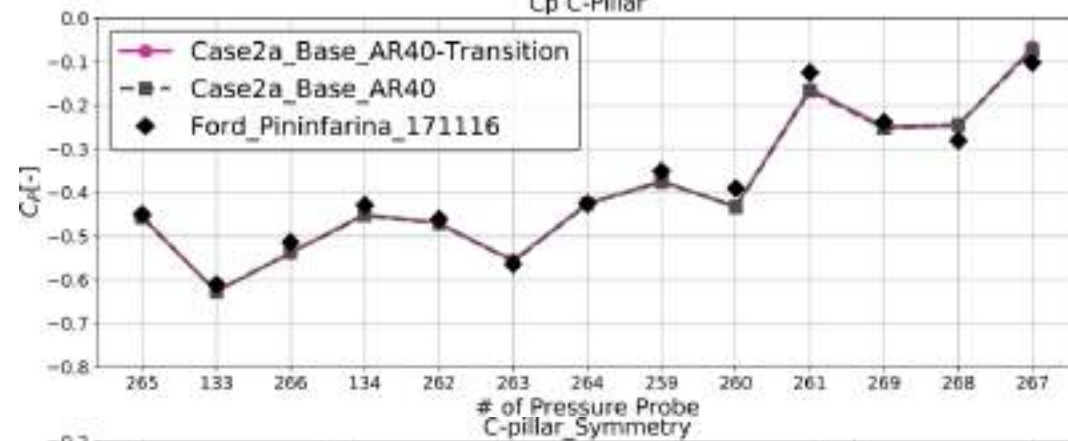
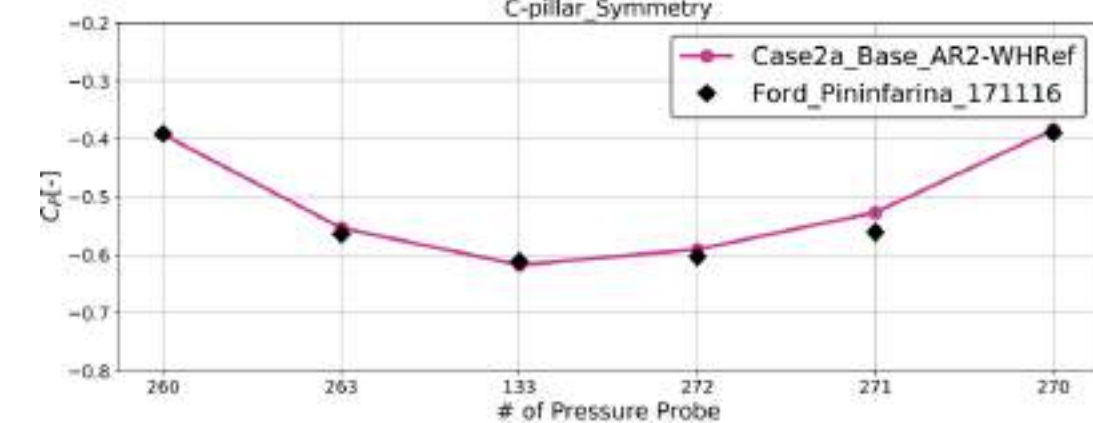
SBES



WFLES



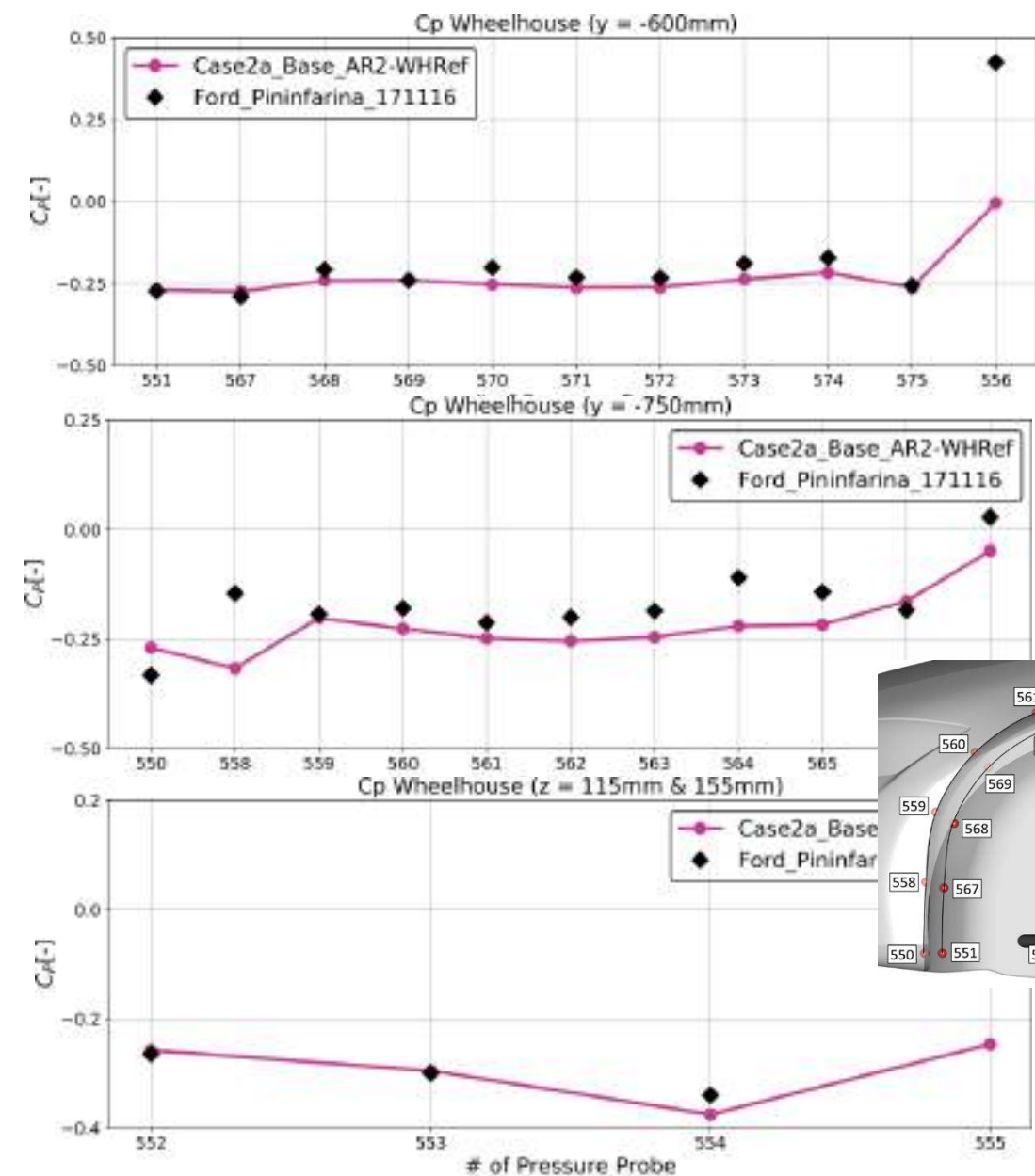
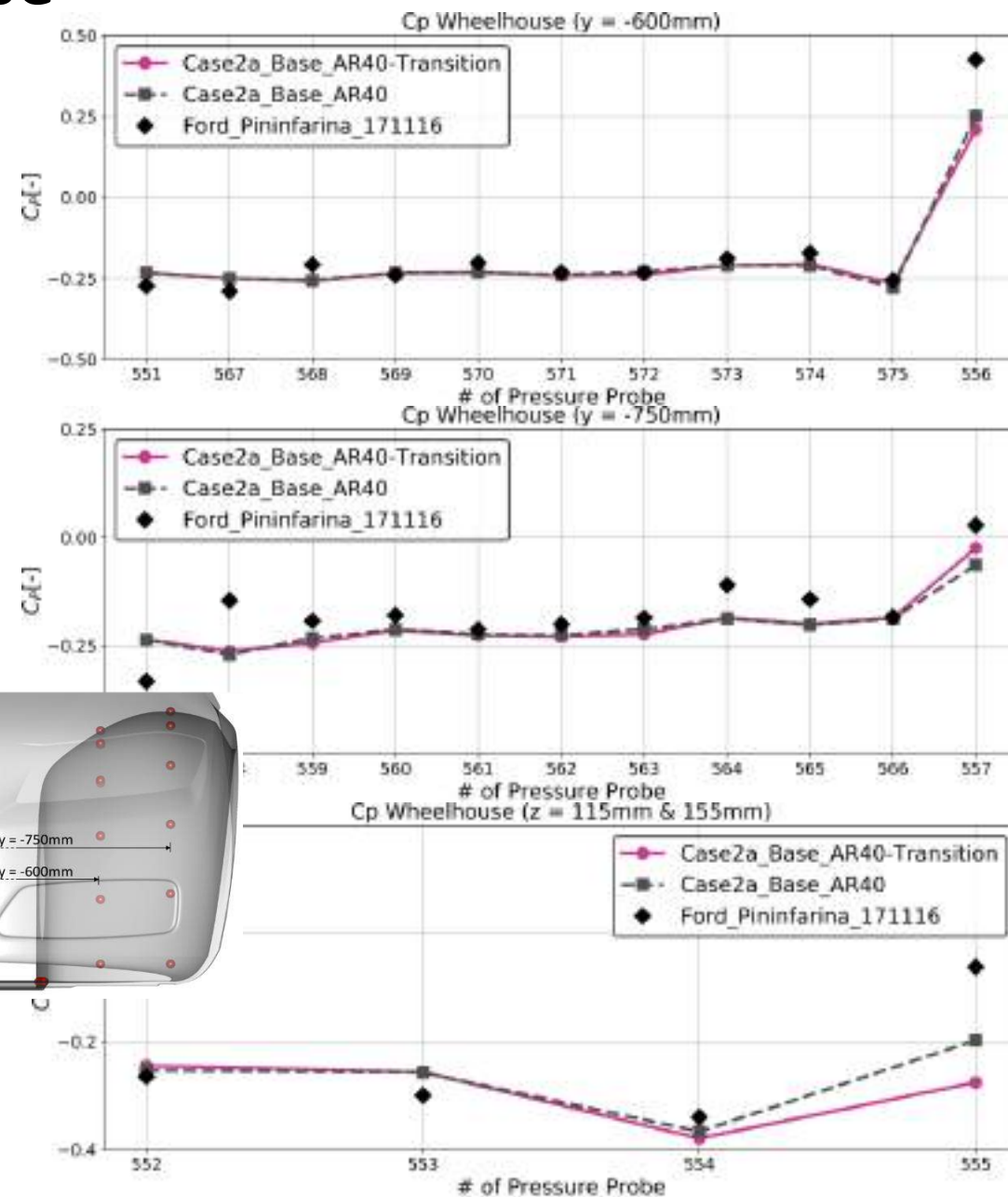
SBES



WFLES

Wheelhouse

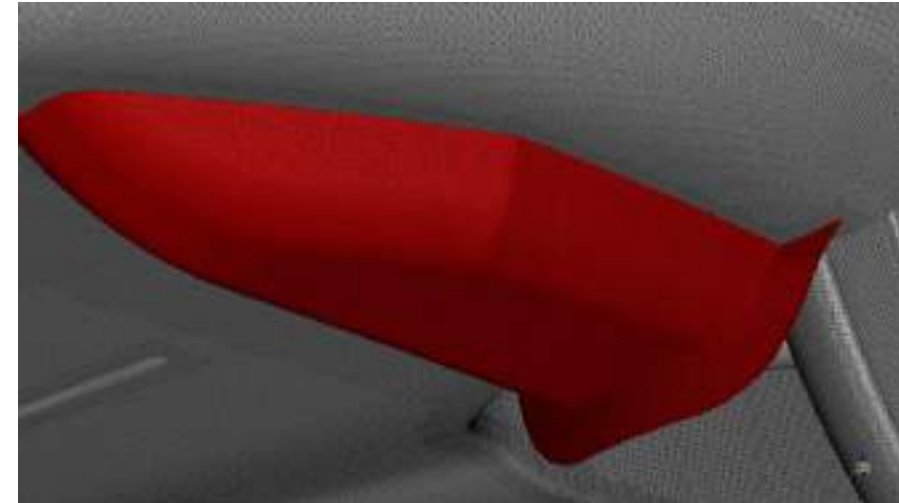
SBES



Aerodynamic Coefficient

- There was a tendency in the workshop to over-predict drag
- Differences in C_D by bulk of participants currently not understood
- Current results on minimum meshes
- Goals:
 - Get closer on overall drag
 - Get closer of ΔC_D

Front Wheel Deflector in 2b

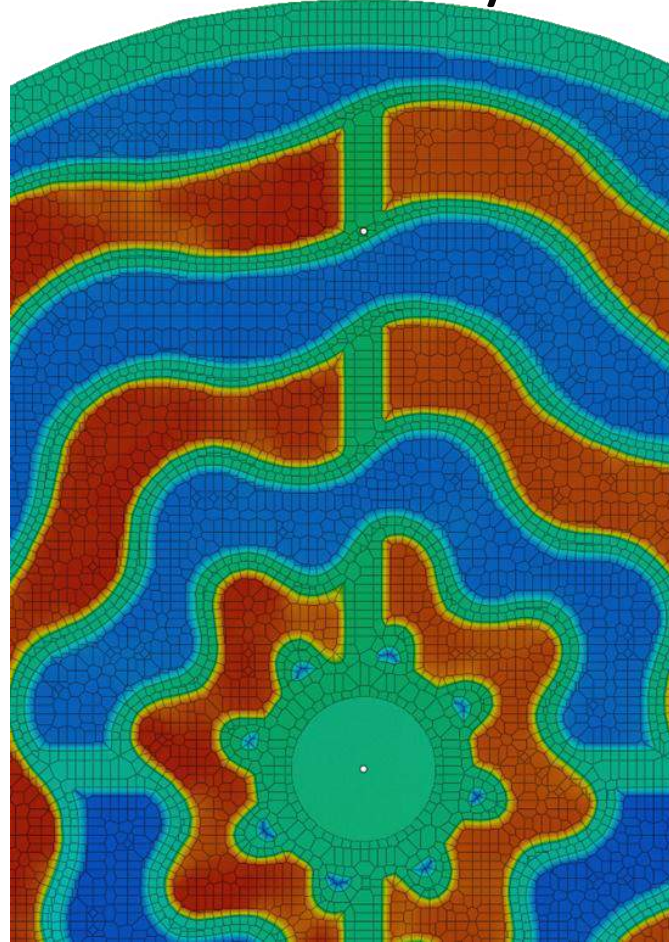
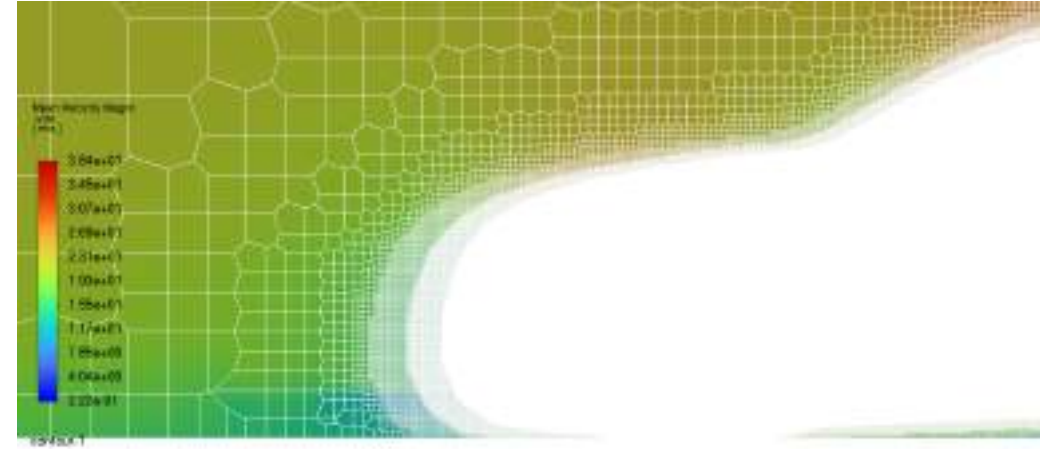
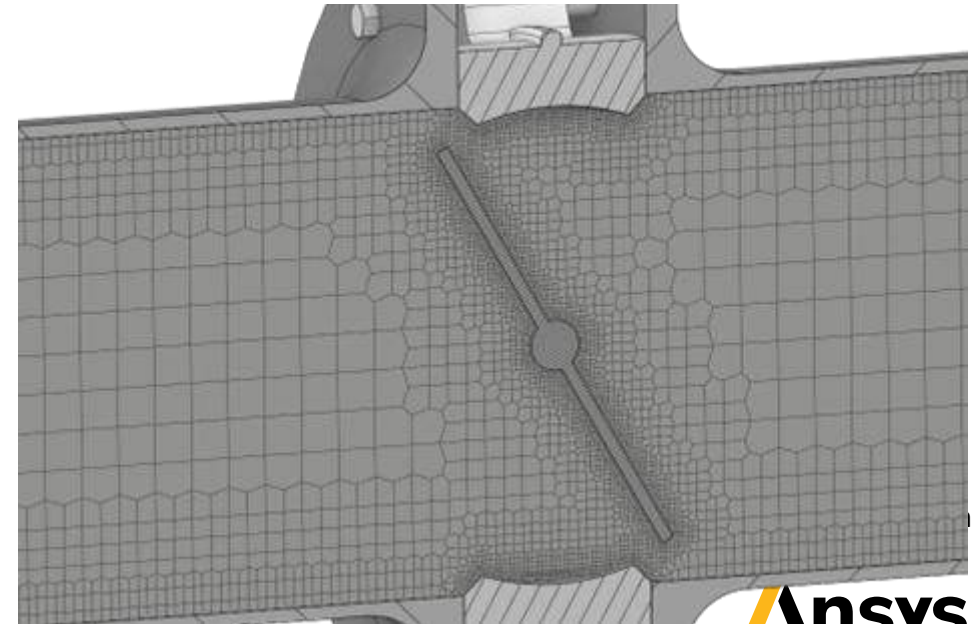


C_D	2a	2b	ΔC_D
Exp	0.2546	0.2420	0.0126
WFLES	0.2670	0.2600	0.0070
SBES	0.2700	0.2627	0.0073

Groundwork for MultiGPU – Octree/Voronoi Meshing

- Top-down meshing

- Both Octree and Voronoi technologies under development
- Highly scalable (1000s of CPUs, GPUs)
- Can handle ‘dirty’ geometries



	1 GPU	2 GPUs	3 GPUs	4 GPUs
13M Cells	1.19 s	1.06 s	0.92 s	0.77 s
69M Cells	-	4.5 s	3.65 s	2.28 s

Higher Order Discontinuous Galerkin Prototype - NACA4412

- Prototype development of higher order DG method
- First results for NACA 4412 airfoil
- 3 mm mesh 4th order DGSEM results comparable to 0.4 mm mesh 2nd order results
- Compared to: 2nd order FV, central scheme, mesh convergence study
 - 1.6 mm ; 0.8 mm ; 0.4 mm meshes

A GPU-native high-order DG solver for Large-Eddy-Simulation

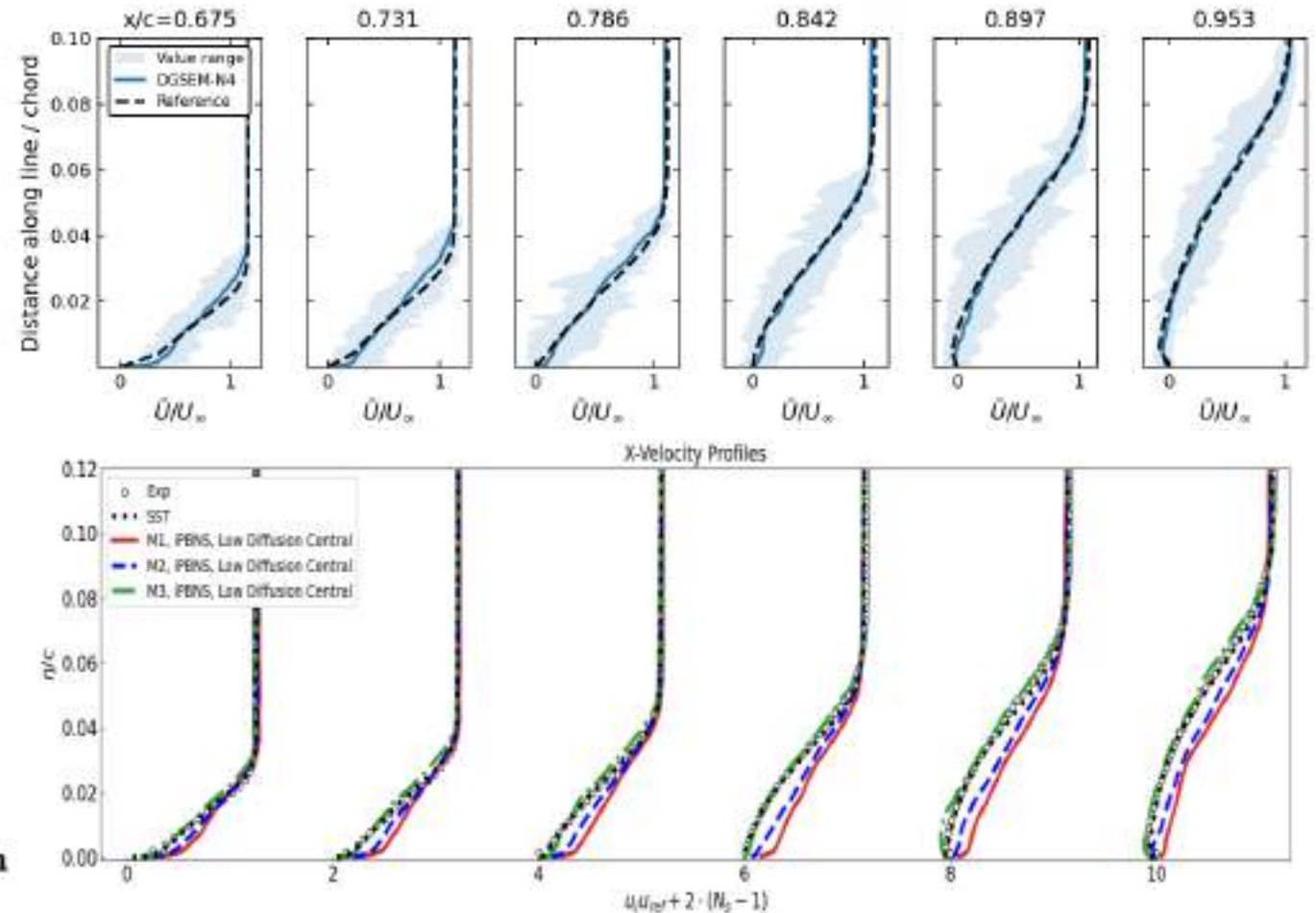
David G. Flad, Andreas Hueppe, Alex Main and Yllo Pasquariello

AIAA 2026-2333

Session: High-Order Numerical Methods

Published Online: 8 Jan 2026 + <https://doi.org/10.2514/6.2026-2333>

©2026 ANSYS, Inc. / Proprietary. DO NOT SHARE.



Overview Synopsys GPU-based CFD Solvers

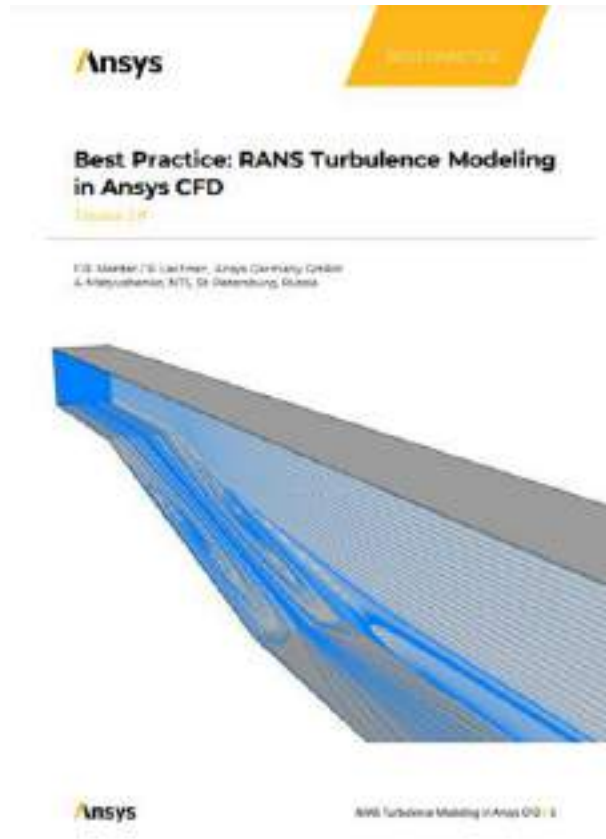
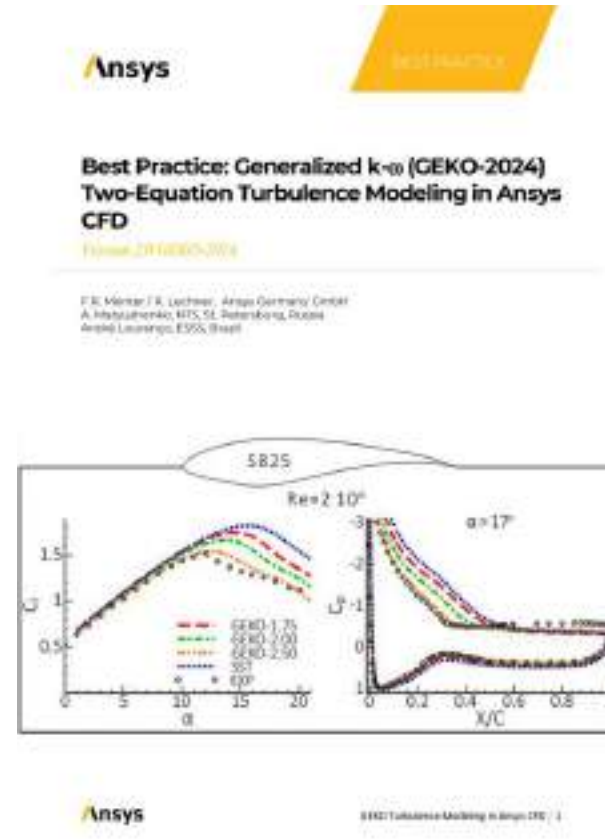
- Pressure-based Navier Stokes (PBNS) solver
 - Coupled
 - Segregated (SIMPLEC,...)
 - CPU and GPU
- Density-based Navier Stokes (DBNS)
 - Implicit coupled
 - Explicit Runge-Kutta (KEEP) scheme (prototype)
- Higher Order DG solver (R&D)
 - Density-based
 - Explicit
- SPH, DPM, ...

LES Summary

- GPU solver is 'enabling' technology
- GPU solver allows resolution of turbulence eddies in boundary layers for industrial flows
- Reynolds number scaling of LES can be reduced by WFLES/WMLES
- Highly optimized numerical methods essential
- Laminar-Turbulent transition is an open question both for WFLES and WMLES
- User adjustment is non-trivial
 - Meshing strategy different from RANS/SBES
 - Averaging requires flow-dependent process
 - Mesh independence hard to achieve

Ansys Best Practice Guidelines

- Can be found on public internet







Uncertainty quantification and Bayesian calibration for turbulence model parameters

WP5- Development of UQ approaches for FIV analyses

GO-VIKING Final Workshop: Flow-Induced Vibrations in Nuclear Systems,
GRS, Garching (Germany)- 20-22 May 2026- **Ali Eidi**- Richard P. Dwight, TU Delft



Funded by
the European Union



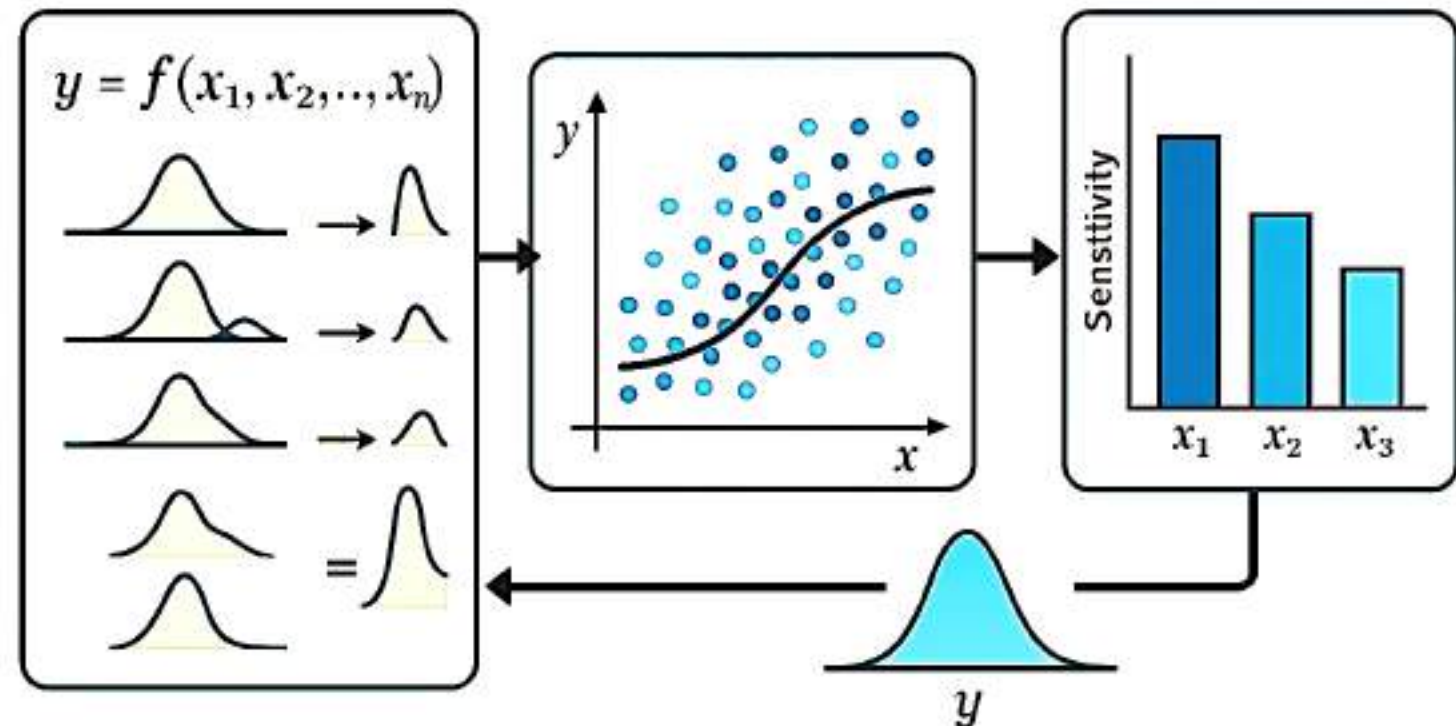
Contents

A. Task overview & Motivation

B. Methodology

C. Results & Discussion

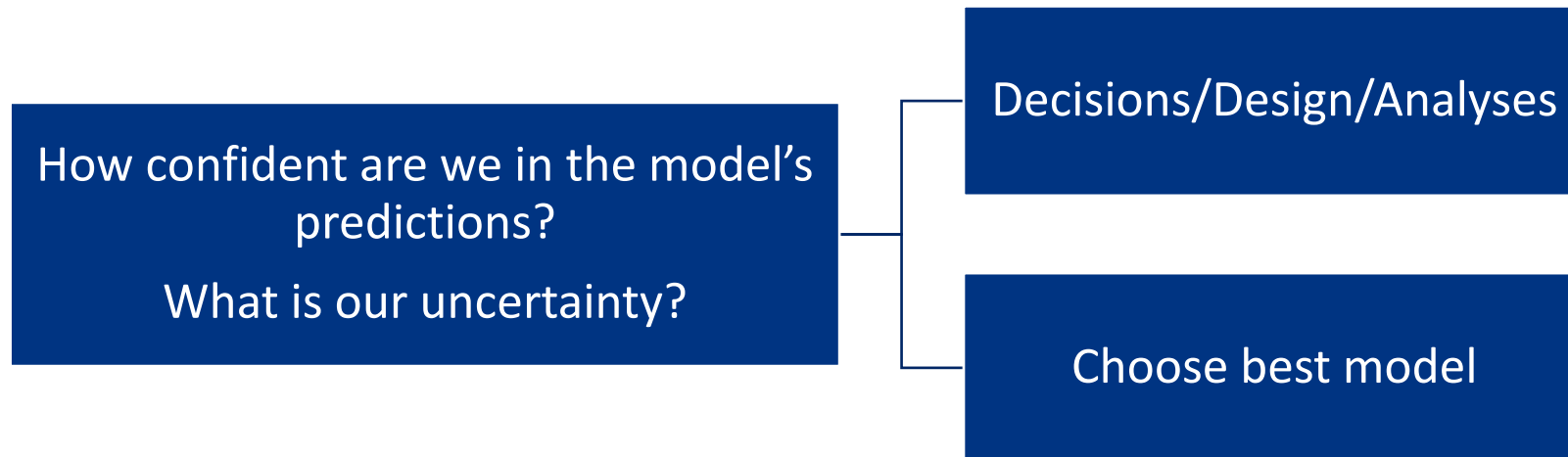
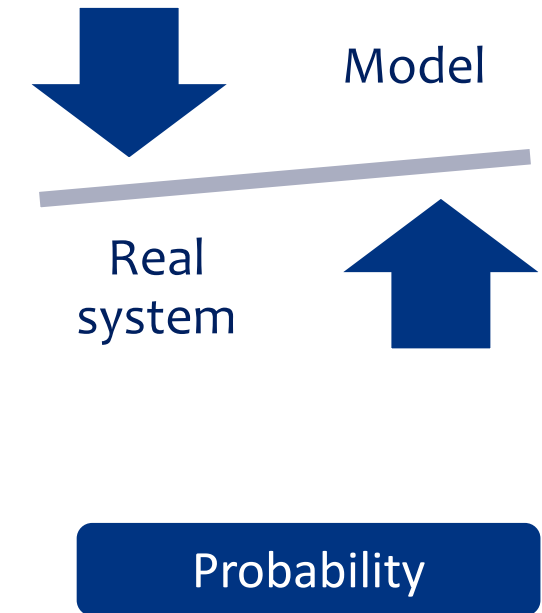
D. Toward FSI





Why UQ for FIV?

- “All models are wrong, some are useful” George Box (1976).
- Role of Uncertainty Quantification (UQ)





Why UQ for FIV?

- Why PROBABILITY?

Multi-variate

Bayesian

- What can we do?

Uncertainty
propagation

Calibration

Data assimilation

Sensitivity
analyses



Task 5.2: Uncertainty quantification for FIV

- Combine FSI analyses with non-intrusive UQ techniques (Polynomial Chaos Expansion, Monte Carlo) [UGENT]
- Quantify the RANS turbulence closure error on the output of interest with Bayesian estimates of the model corrections, derived from high-fidelity data [TUD]
- Perform global sensitivity analysis for a FIV run [TUD]
- Demonstrate the UQ methods on experiments from WP2-4 and synthesize [UGENT, TUD]



Project workflow — from canonical cases to the GO-VIKING benchmark

Canonical cases:
Method development
and validation

- Sensitivity analysis,
surrogate modeling and
uncertainty quantification

Target case
(GO-VIKING
benchmark)

- Surrogate modeling and
uncertainty quantification



Most influential
parameters are
identified



$k - \omega$ SST model¹

- Combines $k-\omega$ (near-wall) + $k-\varepsilon$ (free-stream)
- Includes many empirical coefficients $\rightarrow$ uncertain in complex flows

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho k \mathbf{u}) = P_k - \beta^* \rho \omega k + \nabla \cdot [(\mu + \alpha_k \mu_t) \nabla k] + S_k$$

$$\frac{\partial(\rho \omega)}{\partial t} + \nabla \cdot (\rho \omega \mathbf{u}) = \gamma \frac{\omega}{k} P_k - \beta \rho \omega^2 + \nabla \cdot [(\mu + \alpha_\omega \mu_t) \nabla \omega] + 2\rho(1 - F_1)\alpha_{\omega 2} \frac{1}{\omega} \nabla k \cdot \nabla \omega + S_\omega$$

$$\nu_t = a_1 \frac{k}{\max(a_1 \omega, F_2 S)}$$



$k - \omega$ SST model

Parameter	Description	Normal value	Sampling interval ¹
$\alpha_{\omega 1}$	Specific dissipation rate coefficient	0.5	[0.3, 0.7]
$\alpha_{\omega 2}$	Specific dissipation rate coefficient	0.856	[0.7, 1.0]
α_{k1}	Turbulent kinetic energy coefficient	0.85	[0.7, 1.0]
α_{k2}	Turbulent kinetic energy coefficient	1.0	[0.8, 1.2]
β_1	Destruction coefficient for ω	0.075	[0.06, 0.09]
β_2	Destruction coefficient for ω	0.0828	[0.07, 0.1]
β^*	Dissipation coefficient for k	0.09	[0.784, 0.1024]
a_1	Cross-diffusion term coefficient	0.31	[0.31, 0.4]
κ	Von Karman constant	0.41	[0.38, 0.42]
γ_1	Derived coefficient	computed	computed
γ_2	Derived coefficient	computed	computed



anisotropic pressure fluctuation model (aniPFM)¹

- Reconstructs missing pressure fluctuations via synthetic anisotropic velocity fields

$$\frac{\partial^2 p'}{\partial x_i \partial x_i} = -\rho \left[\frac{\partial}{\partial x_i} \left(u'_j \frac{\partial \bar{u}_i}{\partial x_j} \right) + \frac{\partial^2}{\partial x_i \partial x_j} (u'_i u'_j - \overline{u'_i u'_j}) \right]$$

- Based on Fourier decomposition + modified von Kármán spectrum

$$\omega_i(\mathbf{x}) = \sqrt{6} \sum_{n=1}^N \sqrt{q_n} \sigma_{in} \cos(\mathbf{k}_n \cdot \mathbf{x} + \varphi_n)$$

$$E(k) = \frac{(k/k_t)^4}{[1+2.4(k/k_t)^2]^{17/6}} \exp\left(-\frac{12k}{k_\eta}\right)$$

- Enforces anisotropy with Cholesky decomposition of Reynolds stress

$$u'_i = a_{ij} \omega_j, \text{ with } a_{ik} a_{kj} = R_{ij} E(k)$$

$\bar{u}_i$	Mean velocity
p'	Pressure fluctuation
u'_i	Fluctuating velocity component
$\omega_i(\mathbf{x}, t)$	Dimensionless fluctuations
q_n	Mode amplitude
$\mathbf{k}_n$	Wavenumber vector
σ_{in}	Directional cosine
φ_n	Random phase
$E(k)$	Modified von Karman energy spectrum
k_t	Energy-containing peak wavenumber
k_η	Dissipation-range cut off
R_{ij}	Reynolds stress tensor



Sensitivity analysis — finding the parameters that drive the QoI

Reconstructs Sobol indices — fraction of QoI variance attributable to each parameter ($S_i \in [0,1]$)

Saltelli sampling¹ — quasi-random sequences; cost scales as $N(d+2)$ CFD runs for d parameters and N base samples

Latin Hypercube Sampling² (LHS) designs a space-filling training ensemble

For expensive case: train a cheap surrogate; do SA and Bayesian inference on it.



Two surrogate models — and why we use both

Kriging (Gaussian Process Regression)^{1,2}

- Probabilistic interpolator — gives **mean prediction + uncertainty** at every parameter point
- Flexible: shape comes from a chosen kernel, not a fixed functional form
- Augmented with a **discrepancy term $\delta(\mathbf{x})$** (Kennedy & O'Hagan) → absorbs the systematic URANS-vs-LES gap
- Doesn't pretend the URANS is perfect: parameter uncertainty *and* model inadequacy enter the calibration jointly

PCE (Polynomial Chaos Expansion)²

- Spectral surrogate: QoI expanded in orthogonal polynomials of the parameters
- **Legendre basis** for uniform priors
- Coefficients fit non-intrusively from the **same** LHS ensemble — no extra CFD runs
- Sobol indices and variance fall out **analytically** from the coefficients — sensitivity for free

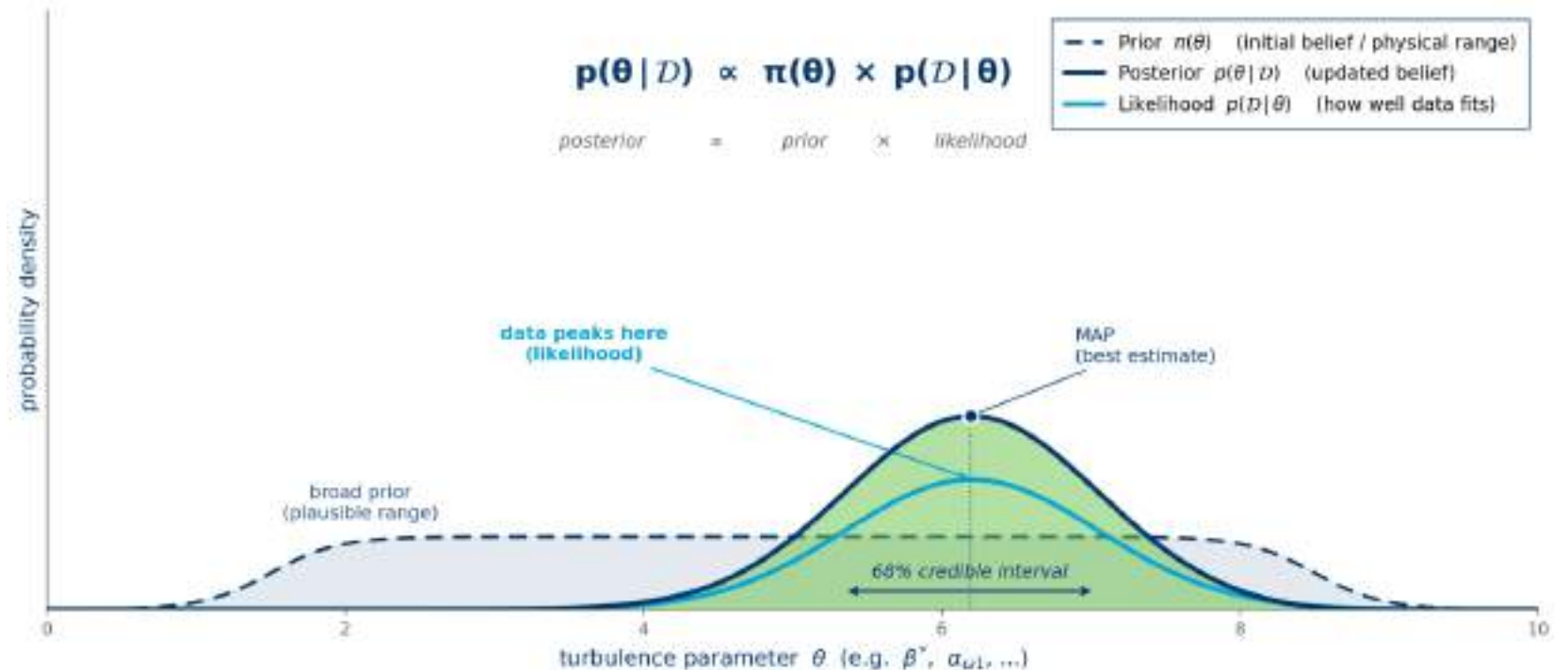
Why two surrogates? Same posterior from independent methods = confidence the result is not a surrogate artefact.



Bayes' theorem — turning data into a probability over parameters

Prior $\pi(\theta)$	Likelihood $p(\mathcal{D} \theta)$	Posterior $p(\theta \mathcal{D})$
what we'd accept <i>a priori</i> — uniform over physical rangest.	how well each θ reproduces the LES / DNS data	calibrated belief — narrower than either alone

- The output is a distribution of plausible parameter values, not a single best fit — that is the input uncertainty we then propagate through AniPFM into the FIV prediction.

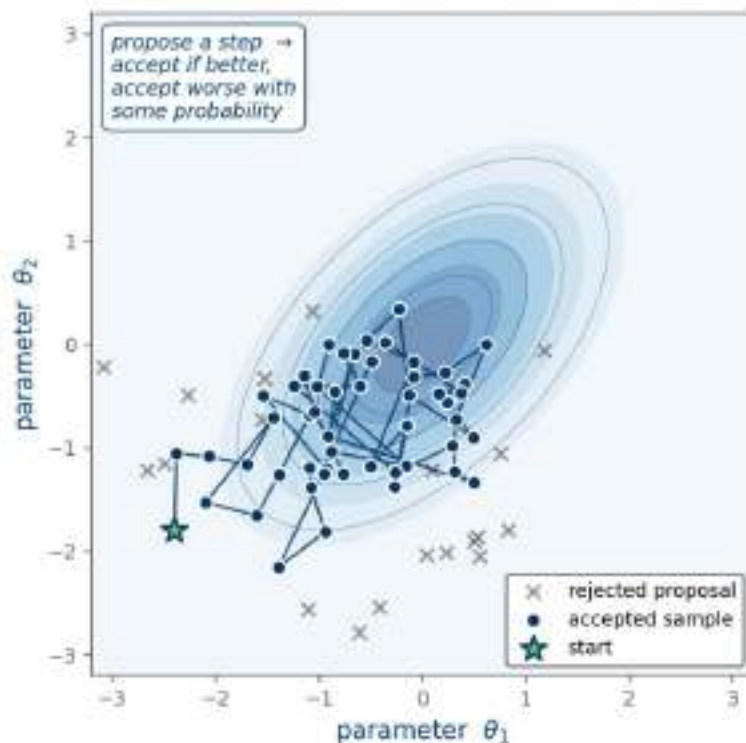




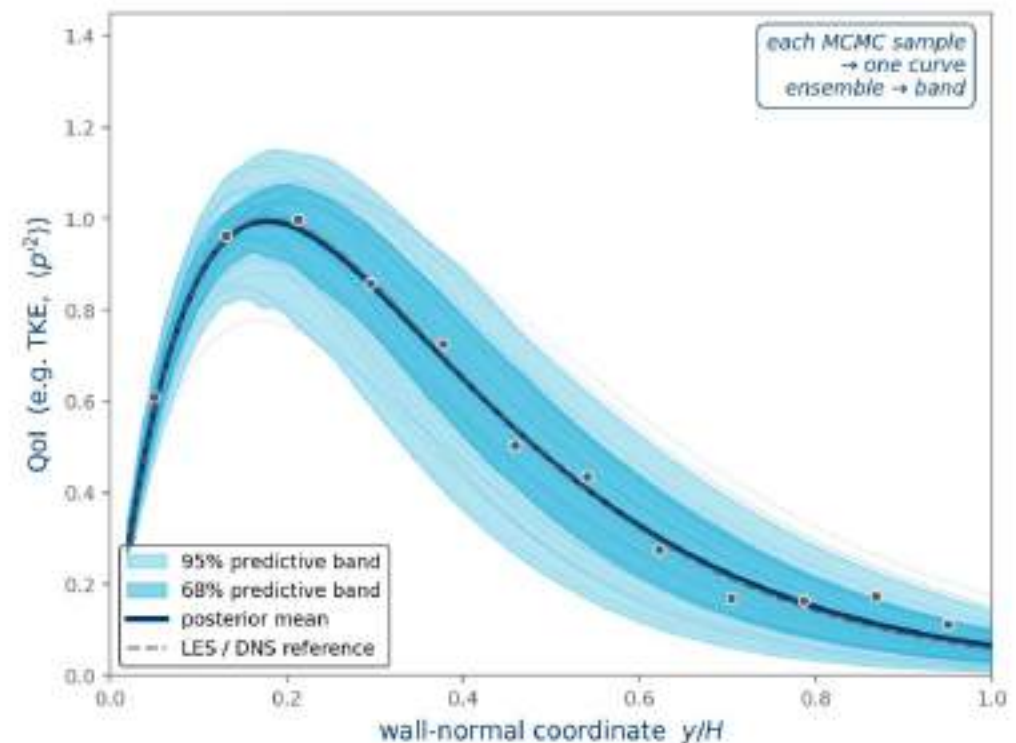
From posterior to predictive — MCMC in practice

- Posterior $\rightarrow$ MCMC samples $\rightarrow$ predictive band on QoIs
calibrated prediction with quantified uncertainty, not a single best-fit curve.

Metropolis-Hastings random walk on the posterior

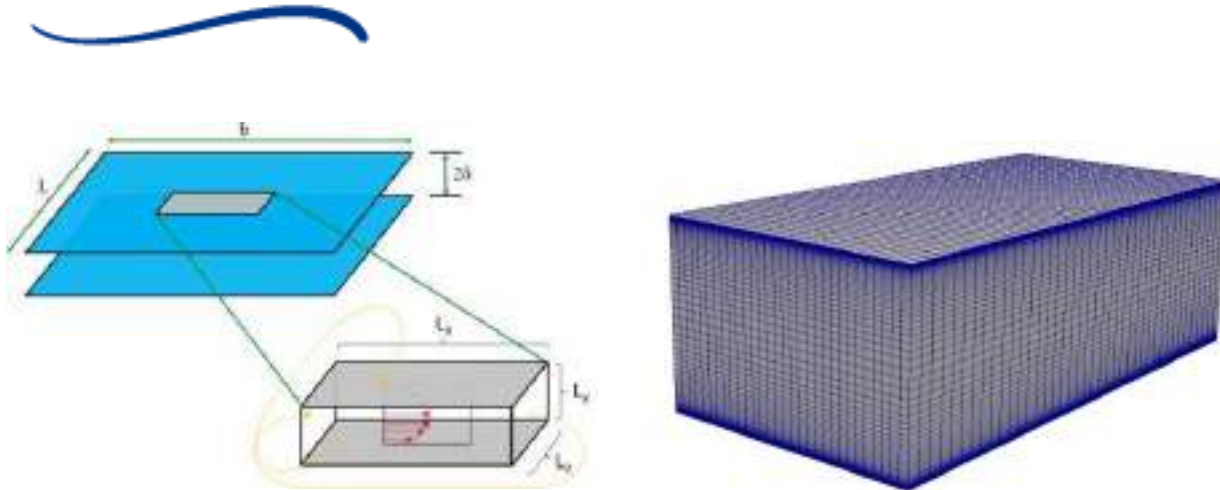


Predictive posterior on the QoI





Simulation Cases: Turbulent Channel Flow (TCF)



- $Re_\tau = 180, 395, 640, 1020$
- Dimensions: $6\delta \times 2\delta \times 3\delta$
- $y^+ \leq 1$
- Turbulence model : SST $k - \omega$
- Validation: DNS data¹

Gridding ($N_x \times N_y \times N_z$)

Coarse mesh	$40 \times 80 \times 30$
Medium mesh	$50 \times 96 \times 40$
Fine mesh	$80 \times 112 \times 60$

- Solver details (OpenFOAM):
 - Steady-state RANS simulation to initialize domain: SIMPLE (simpleFoam) + aniPFM¹
 - URANS (main simulation): PISO (pimpleFoam) + aniPFM
 - URANS simulation for 300s (equally 12 flow-through)
- Computational cost: ~10 CPU hours for each case

Simulation Cases: Turbulent Annular Flow (TAF)

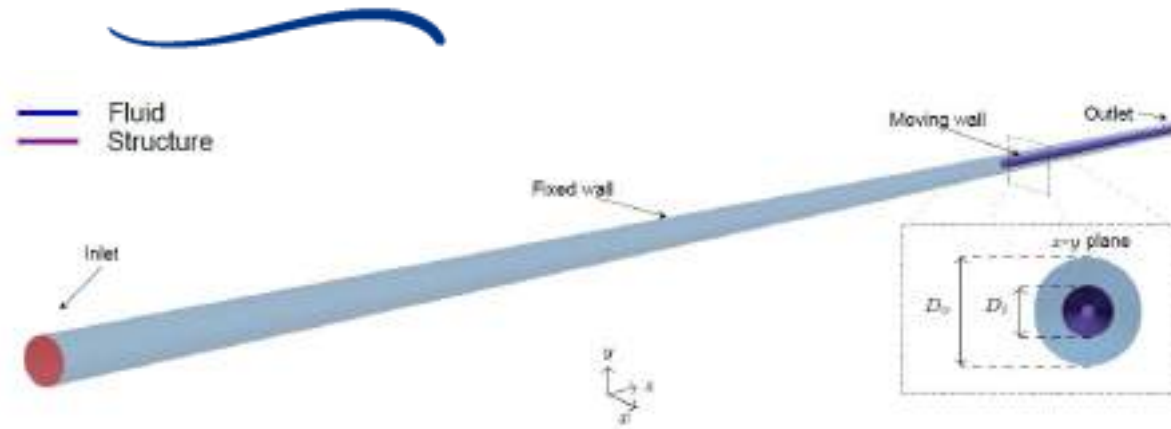


Gridding ($N_r \times N_\theta \times N_x$)	
Coarse mesh	$24 \times 20 \times 30$
Medium mesh	$36 \times 30 \times 54$
Fine mesh	$50 \times 40 \times 60$

- $Re_H = 45000$ ($U_{bulk} = 3.45 \text{ m/s}$)
- Dimensions: $L = 6D_h$
- Coarse mesh: $r_{avg}^+ = 2$, Medium mesh: $r_{avg}^+ = 0.85$
- Turbulence model : SST $k - \omega$
- Validation: LES data¹
- Solver details (OpenFOAM):
 - Steady-state RANS simulation to initialize domain: SIMPLE (simpleFoam) + aniPFM

- URANS (main simulation): PISO (pimpleFoam) + aniPFM
 - URANS simulation for 2s (equally 100 flow-through)
- Computational cost:
 - Coarse mesh: ~ 11 CPU hours for each case
 - Medium mesh: ~ 62 CPU hours for each case

Simulation Cases: Cantilevered-rod (blunt-end)

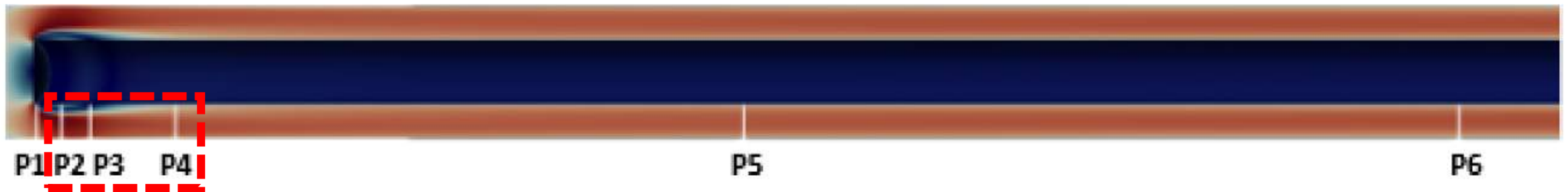


Mesh ¹	# cells	$r_{inner_{avg}}^+$	$r_{inner_{max}}^+$	$z_{tip_{avg}}^+$	$z_{tip_{max}}^+$
Coarse	1.9 M	0.85	1.89	24.33	42.69
Medium	4.2 M	0.63	1.28	18.09	35.67
Fine	8.2 M	0.58	0.58	7.84	22.92





Simulation Cases: Cantilevered-rod (blunt-end)



- $Re_H = 5.82 \times 10^4$ ($Re_{bulk} \approx 60000$)
- $U_{bulk} = 3.45 \text{ m/s}$
- Channel diameter = 0.021 m
- Channel length = 2.5 m
- Hydraulics diameter = 0.011 m
- Turbulence model : SST $k - \omega$
- Validation: LES data¹
- Solver details (OpenFOAM):
 - URANS: PISO (pimpleFoam)
 - $CFL \leq 1$
- Computational cost: ~ 2000 CPU hours for each case



Summery of workflow for UQ

Define Prior

- Specify parameter bounds using literature or expert knowledge.



SA analysis

- Determine most influential model parameters to reduce problem dimension



Build Surrogate Model

- Train Kriging + GP inadequacy model on CFD simulation data.
- Run secondary SA to validate surrogate model



Formulate Likelihood

- Quantify mismatch between surrogate predictions and high-fidelity data using a Gaussian error model.



Apply Bayes' Theorem

- Combine prior and likelihood to define the posterior distribution.



Run MCMC Sampling

- Use Adaptive Metropolis to generate samples from the posterior.



Overview of Simulations- TCF

4 × SA analysis

- Coarse mesh
- 9D sampling space
- Baseline + 1408 simulations: ~14k CPU hours
- QoIs: Pressure fluctuations, Velocity fluctuations, Reynolds stress components (aniPFM), TKE

Surrogate modeling

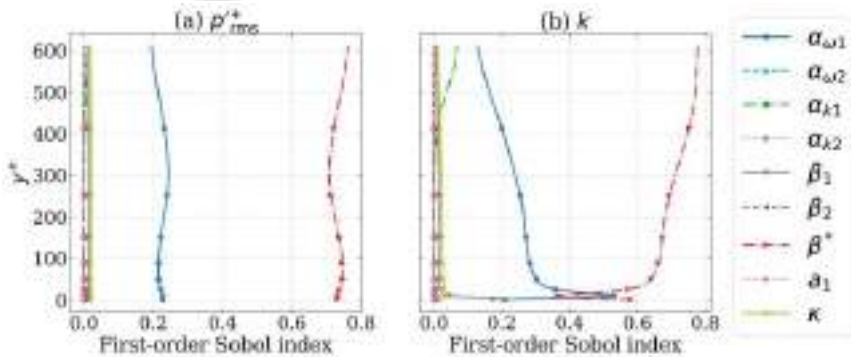
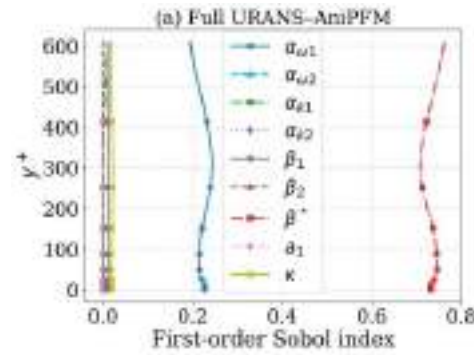
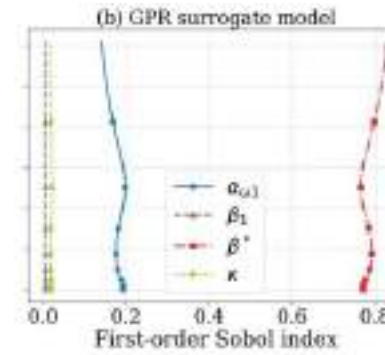
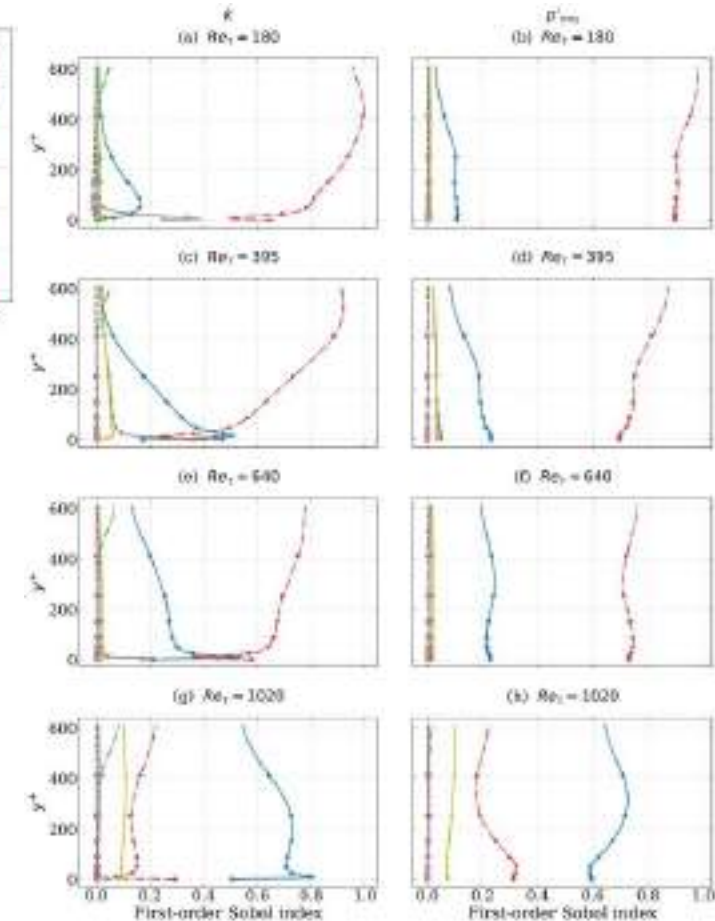
- Coarse mesh
- 4D sampling space
- 222 simulations: ~2.2k CPU hours
- QoIs: Pressure fluctuations, Reynolds stress components (aniPFM)

UQ

- Adaptive Metropolis MCMC algorithm based on Gaussian proposal and covariance is updated after burn-in
- 50000 MCMC steps with first 10000 as burn-in
- Noise level is estimated based on residuals over LHS samples



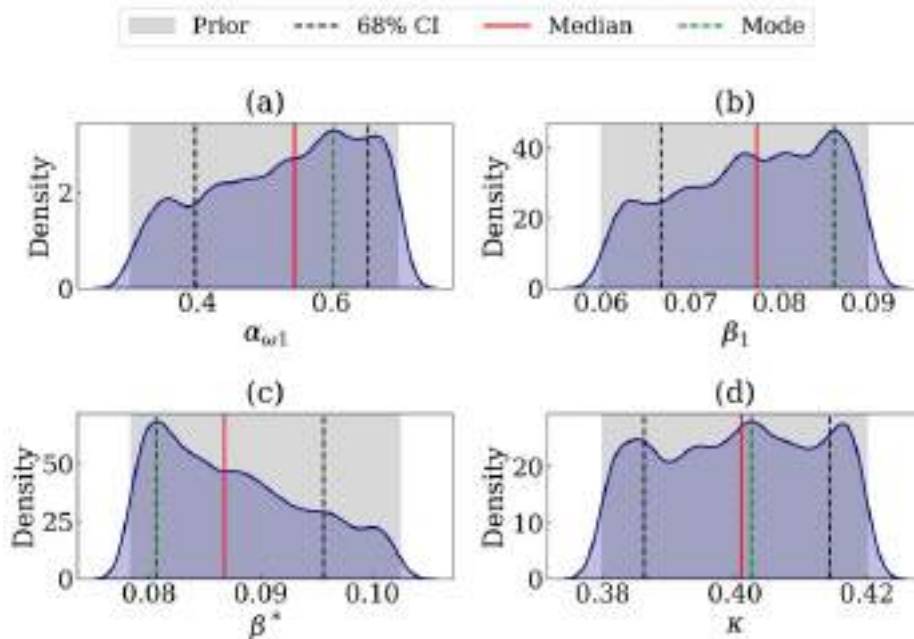
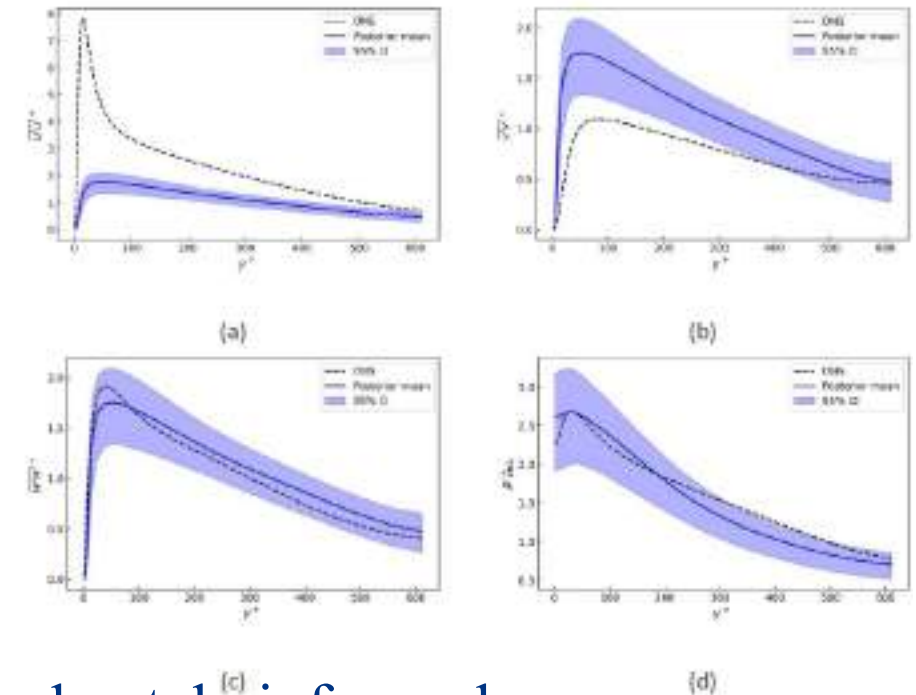
TCF: Sensitivity Analysis



- β^* , $\alpha_{\omega 1}$, κ and β_1 dominate across QoIs and Re_τ values.
- Their influence is strongest near the wall.
- The GP surrogate reproduces consistent sensitivity trends.
- Dominant parameters are consistent within the Re ranges.



TCF: Posteriors & Predictive Posteriors



- β^* is strongly informed; α_{w1} and β_1 are moderately informed.
- Posterior trends agree with Sobol sensitivity results.
- Predictive posteriors capture $\overline{w'w'}^+$ and near-wall pressure fluctuations, but not other Reynolds stress components
- Remaining QoI errors indicate URANS model-form limitations.

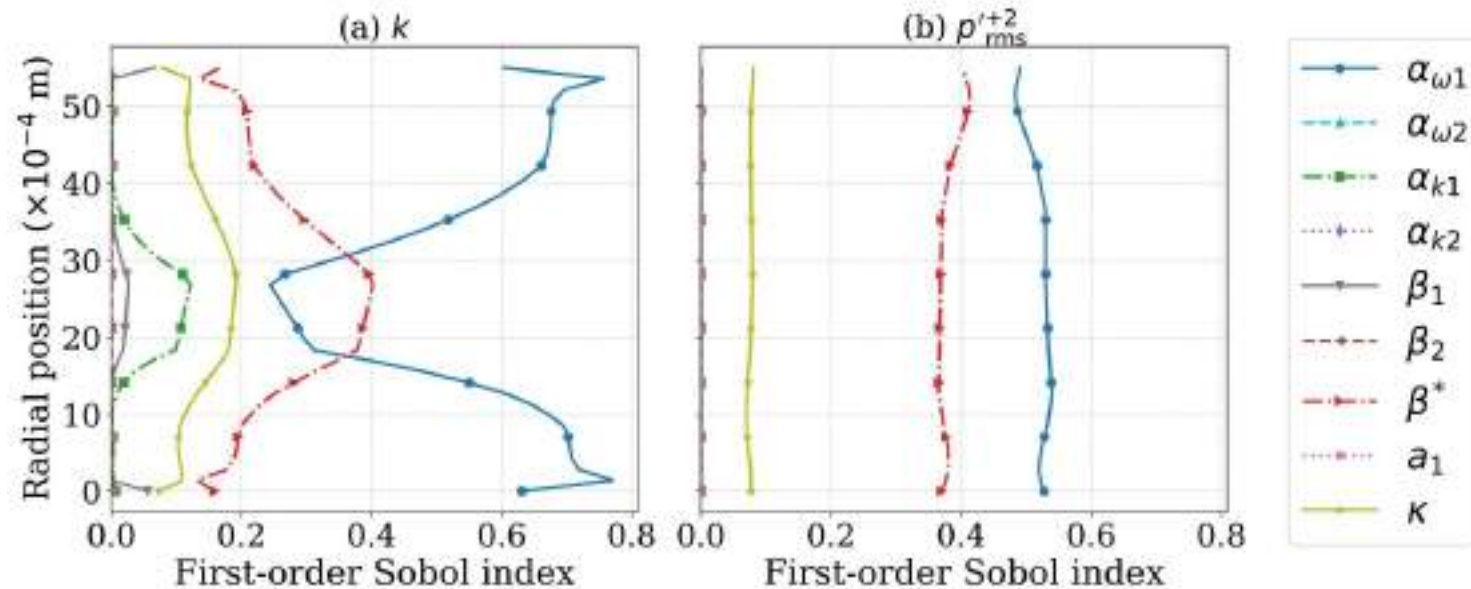


Overview of Simulations- TAF

SA analysis	<ul style="list-style-type: none">• Coarse mesh• 9D sampling space• Baseline + 1408 simulations: ~15k CPU hours• QoIs: Streamwise velocity, Streamwise and wall-normal Reynolds stress components (aniPFM), TKE, Pressure fluctuations
Surrogate modeling	<ul style="list-style-type: none">• Medium mesh• 4D sampling space• 222 simulations: ~13k CPU hours• QoIs: Streamwise velocity, Streamwise and wall-normal Reynolds stress components (aniPFM), TKE
UQ	<ul style="list-style-type: none">• Adaptive Metropolis MCMC algorithm based on Gaussian proposal and covariance is updated after burn-in• 50000 MCMC steps with first 10000 as burn-in• Noise level is estimated based on residuals over LHS samples



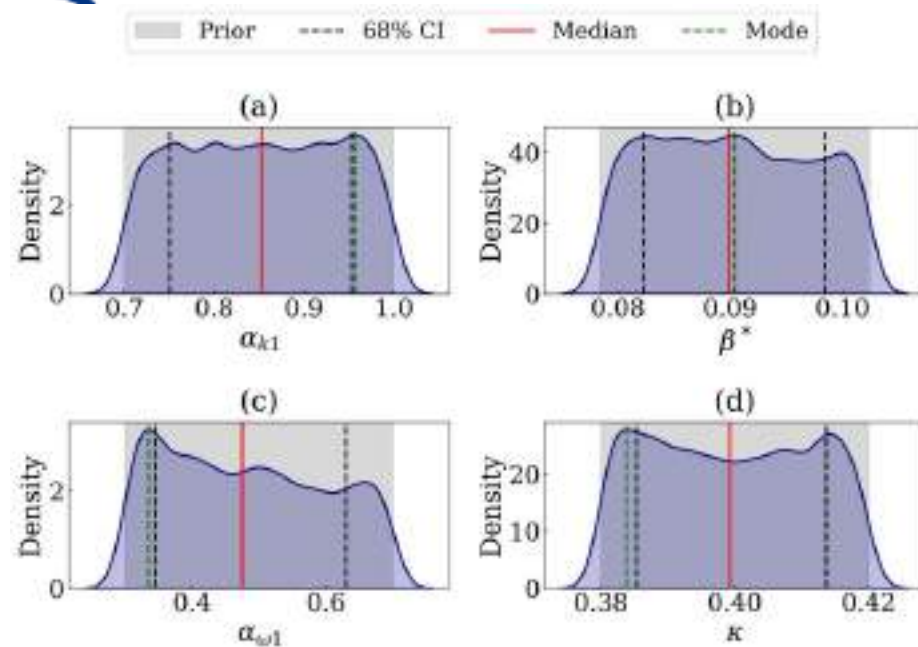
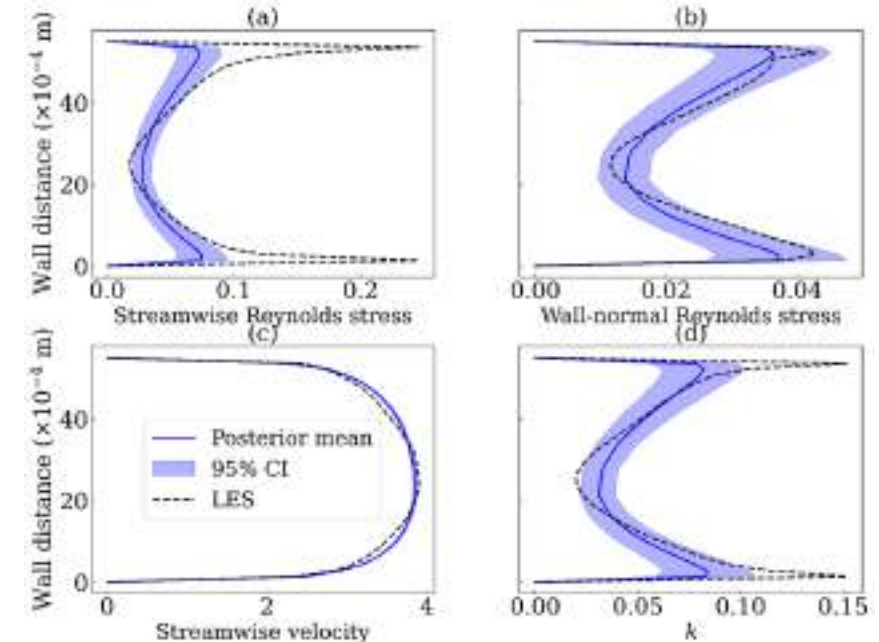
TAF: Sensitivity Analysis



- β^* , $\alpha_{\omega 1}$, κ and α_{k1} dominate across QoIs.
- Their influence is strongest near the wall.
- The GP surrogate reproduces consistent sensitivity trends.



TAF: Posteriors & Predictive Posteriors



- All parameters except $\alpha_{\omega1}$ remain weakly informed in annular flow.
- Limited posterior contraction agrees with weaker Sobol sensitivities.
- Predictive posteriors capture mean streamwise velocity and wall-normal Reynolds stress but not hardly capture near-wall peaks.
- Remaining prediction errors suggest URANS–AniPFM model-form limitations.



Overview of Simulations- Cantilevered-rod case

Surrogate modeling
Both Kriging and PCE

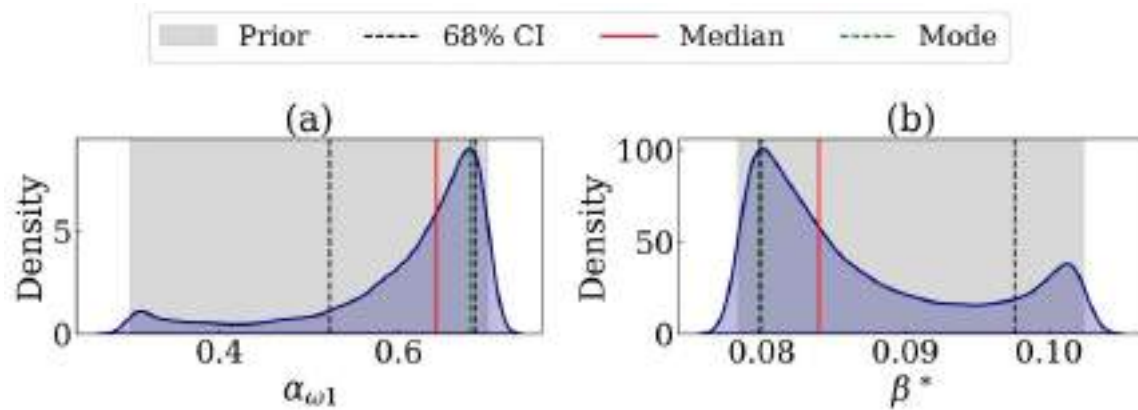
- Coarse mesh
- 2D sampling space (β^* and $\alpha_{\omega 1}$)
- 65 simulations ~130k CPU hours
- QoIs: Streamwise velocity, TKE

UQ

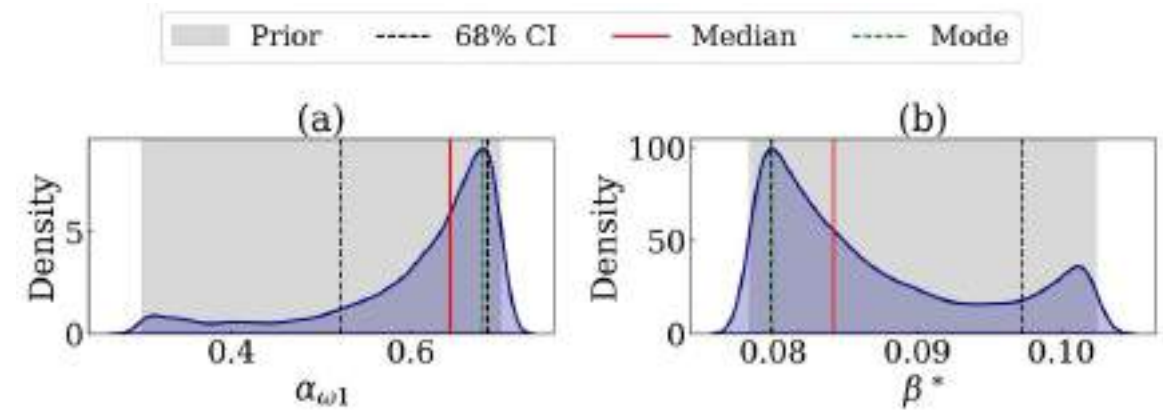
- Adaptive Metropolis MCMC algorithm based on Gaussian proposal and covariance is updated after burn-in
- 50000 MCMC steps with first 10000 as burn-in
- Noise level is estimated based on residuals over LHS samples



Cantilevered-rod case: Posteriors



GPR posteriors

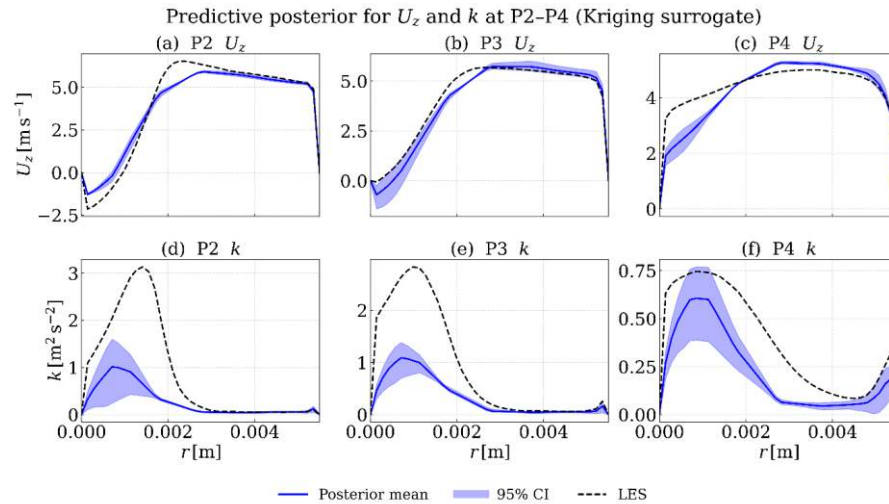


PCE posteriors

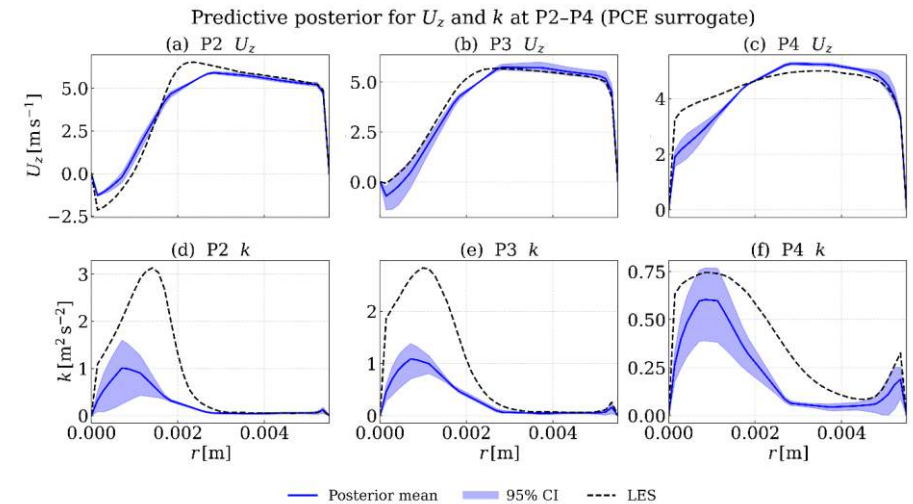
- Bayesian calibration uses LES data at stations P2–P4.
- Both $\alpha_{\omega 1}$ and β^* show clear posterior contraction.
- GPR and PCE posteriors agree, confirming surrogate-robust trends.



Cantilevered-rod case: Predictive posteriors



GPR predictive posteriors



PCE predictive posteriors

- Predictive posteriors capture the velocity-profile shape only qualitatively.
- TKE is consistently underpredicted at P2–P4.
- GPR and PCE give nearly identical predictive trends.
- Narrow bands indicate dominant model-form error.



Cross-case Discussion

Channel and annular flows provide transferable parameter rankings.

$\alpha_{\omega 1}$ and β^* consistently dominate across all cases.

Reduced two-parameter calibration makes rod-case UQ feasible.

Predictive errors mainly reflect structural limits of URANS models.

Calibration improves consistency but cannot remove model-form uncertainty.



Conclusions

Sobol analysis identified a Reynolds-number-invariant SST parameter hierarchy in channel flow.

$\alpha_{\omega 1}$ and β^* were the most transferable dominant parameters across channel, annular, and rod cases.

This enabled reduction of the rod-case calibration problem from 9 parameters to 2.

GPR and PCE surrogates produced consistent posterior and predictive results.

Bayesian calibration constrained $\alpha_{\omega 1}$ toward higher values and β^* toward lower values.

Predictive discrepancies remained after calibration, showing dominant model-form uncertainty.

The framework makes UQ feasible for expensive FSI-relevant URANS configurations.



Dissemination & outputs

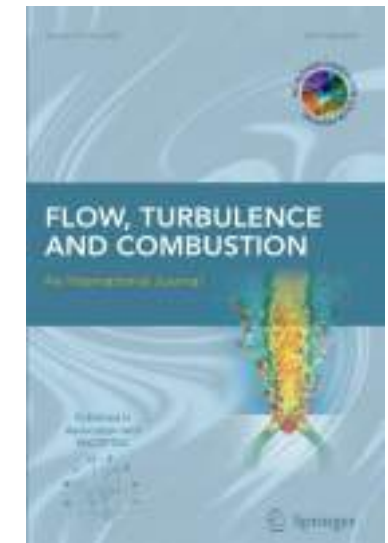
- Contribution to D5.3 & D5.4 within GO-VIKING project.
- Presented at ETMM 15, Dubrovnik, Croatia, 2025.
- Journal paper accepted in *Flow, Turbulence and Combustion* (Springer)



Special Issue

Title: “A surrogate-based Bayesian framework for parametric calibration of URANS turbulence models informed by anisotropic pressure fluctuations”

Authors: Ali Eidi and Richard P. Dwight



Partners



Associated Partners



*EDF R&D UK Center



Thank you!



Ali Eidi



TU Delft



a.eidi@tudelft.nl



Website:
www.go-viking.eu



Mail:
contact@go-viking.eu



Twitter:
[@GOVIKING_EU](https://twitter.com/GOVIKING_EU)



LinkedIn:
[GO-VIKING](https://www.linkedin.com/company/go-viking)



**Funded by the
European Union**

Funded by the European Union. Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union or European Atomic Energy Community ('EC-Euratom'). Neither the European Union nor the granting authority can be held responsible for them.



Uncertainty Quantification for Flow-Induced Vibration using Polynomial Chaos Methods

WP5 – UQ procedures suited for FSI

GO-VIKING Final Workshop - Garching, 20-22 May 2026 - Henri Dolfen, Joris Degroote - Ghent University



Funded by
the European Union



FACULTY OF ENGINEERING
AND ARCHITECTURE

VLAAMS
SUPERCOMPUTER
CENTRUM



Vlaanderen
is supercomputing

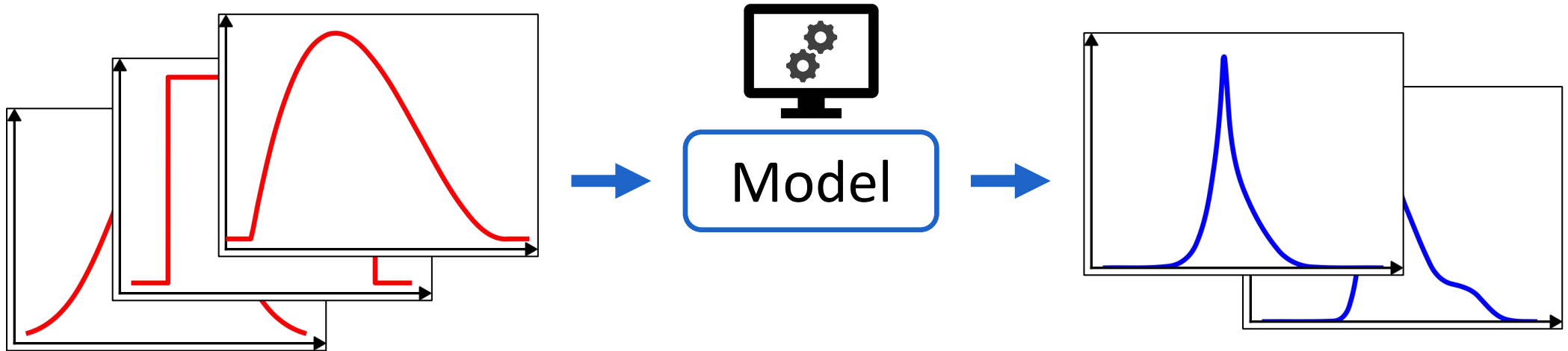
Outline

- Uncertainty quantification
- Steady CFD
- Unsteady FSI single-phase
- Unsteady FSI two-phase
- Conclusions



Uncertainty Quantification

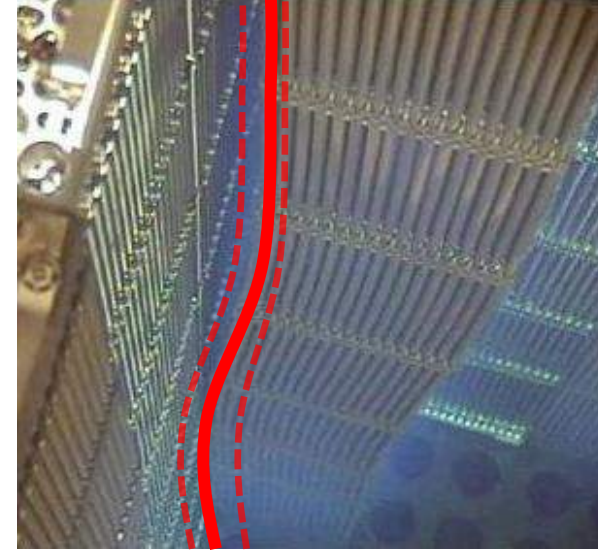
- Any **input uncertainty** propagated through model into **output uncertainty**



- Possible stochastic inputs to numerical model
 - Boundary conditions: free stream state, void fraction, wall temperatures, support stiffness, ...
 - Material properties: elasticity, viscosity, ...
 - Geometry: tolerances, misalignments, deformations, ...

C-shaped Deformed Tube

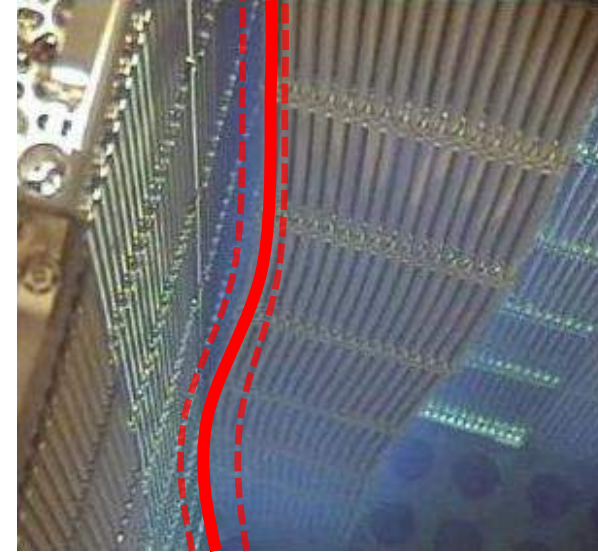
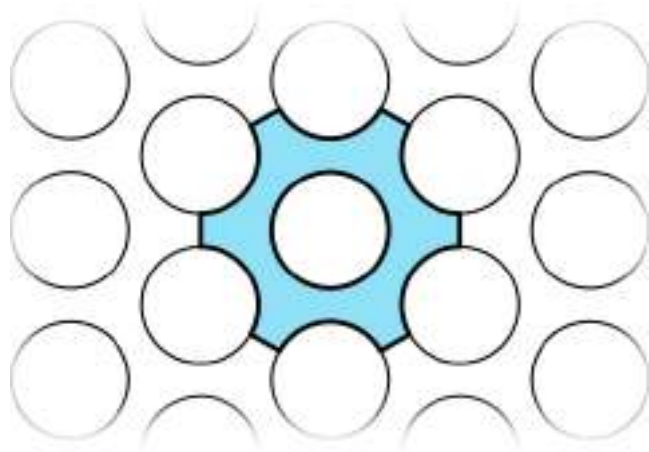
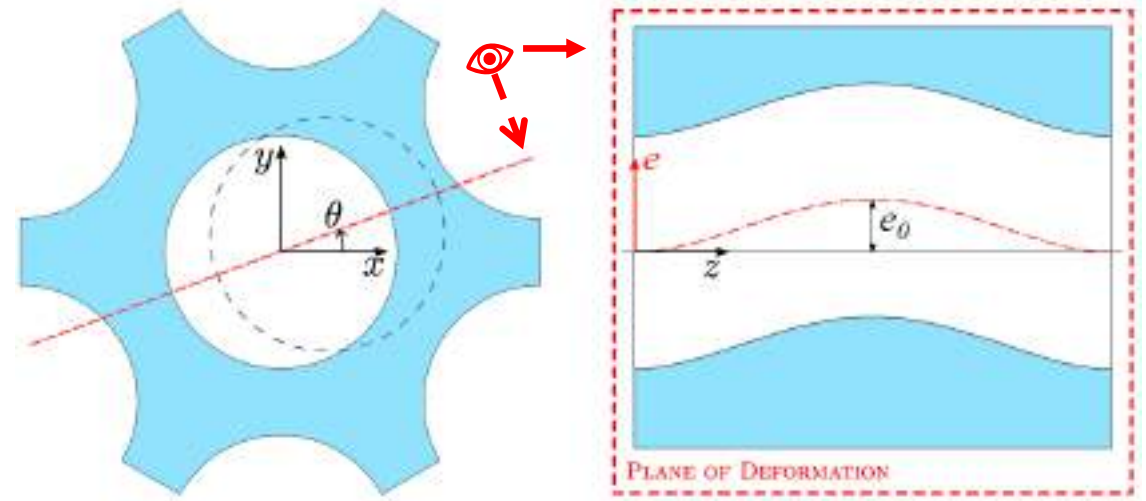
- Fuel assembly deformation due to thermal, hydraulic and irradiation
- Compare Monte Carlo with Polynomial Chaos Expansion and validate workflow



Input Uncertainty

Two inputs with uniform distribution for stressless geometry

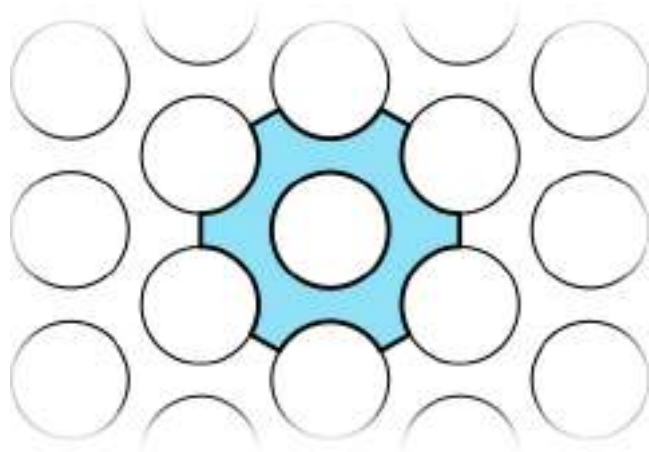
- Deformation extent $e_0 \sim U(0; 0.001 \text{ m})$
- Deformation direction $\theta \sim U(0; 30^\circ)$



Models and Output Uncertainty

Model 1: Steady CFD

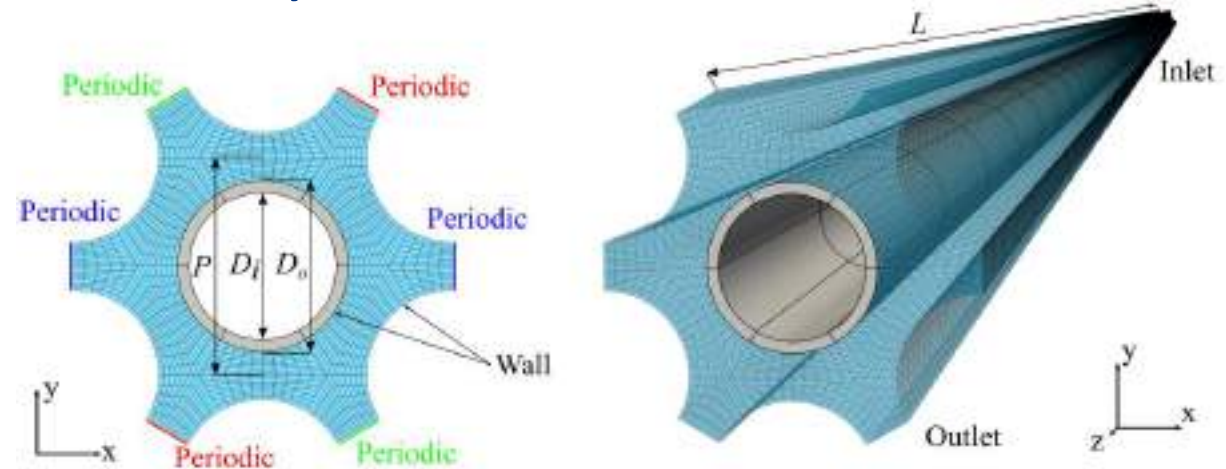
- Goal: Test polynomial chaos method and verify with Monte Carlo method
- Output quantities of interest
 - Pressure drop Δp
 - Maximal velocity V_{max}



Model Steady CFD

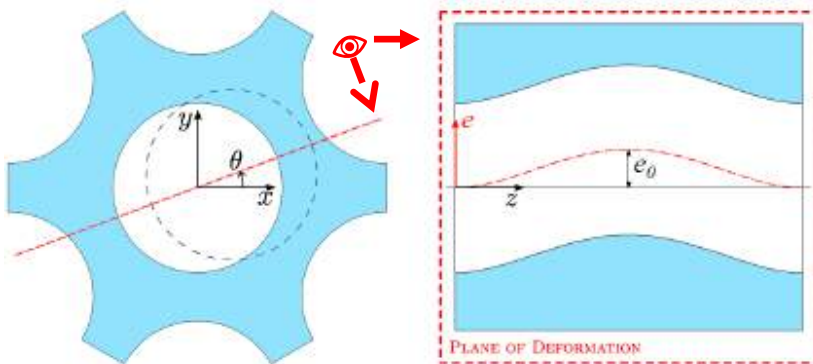
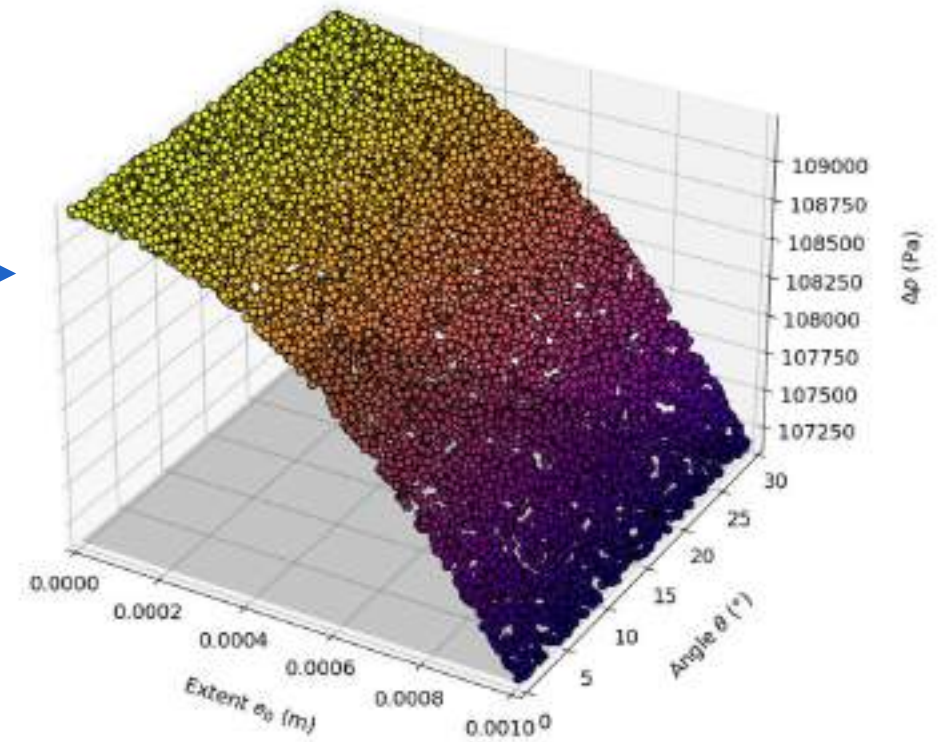
- Grid
 - 700 axial
 - 60 circumferential
 - 14 from one rod to another
- Solver
 - Ansys Fluent
 - RANS $k-\omega$ SST
 - Wall functions for $30 < y^+ < 100$

- Boundary conditions



Monte Carlo

- Random sample to compute statistics
- $$\text{Average}(\Delta p) = \frac{1}{10000} \sum_{i=1}^{10000} \Delta p_i$$



Non-intrusive Polynomial Chaos

- Project on space spanned by orthogonal polynomials up to finite degree equipped with inner product
- Typically, fewer samples needed than MC in case of low-dimensional stochastic input, but prone to curse of dimensionality

$$s(\vec{\xi}) = \sum_{k=0}^P s_k \Psi_k(\vec{\xi})$$

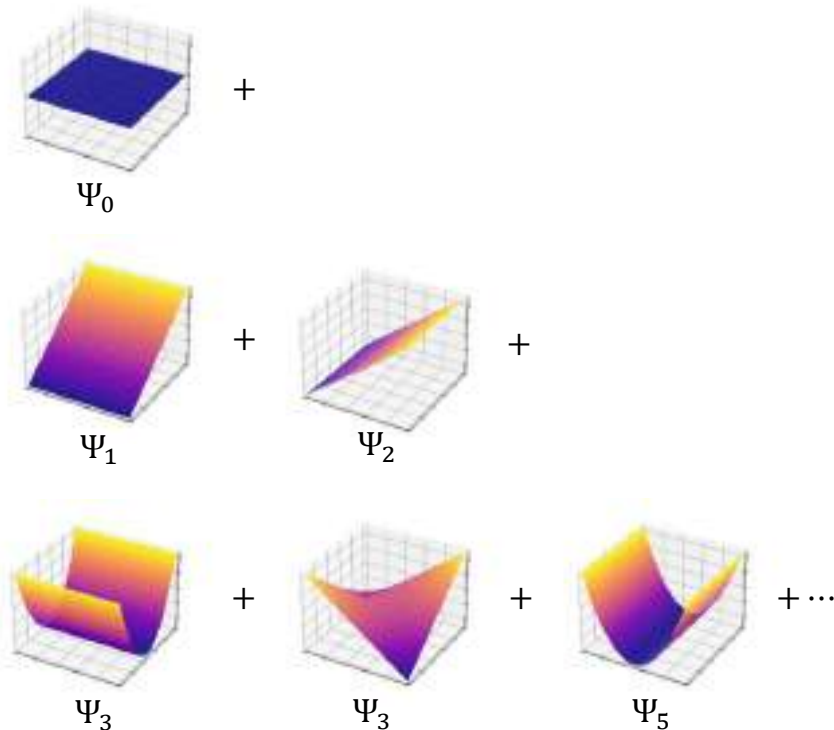
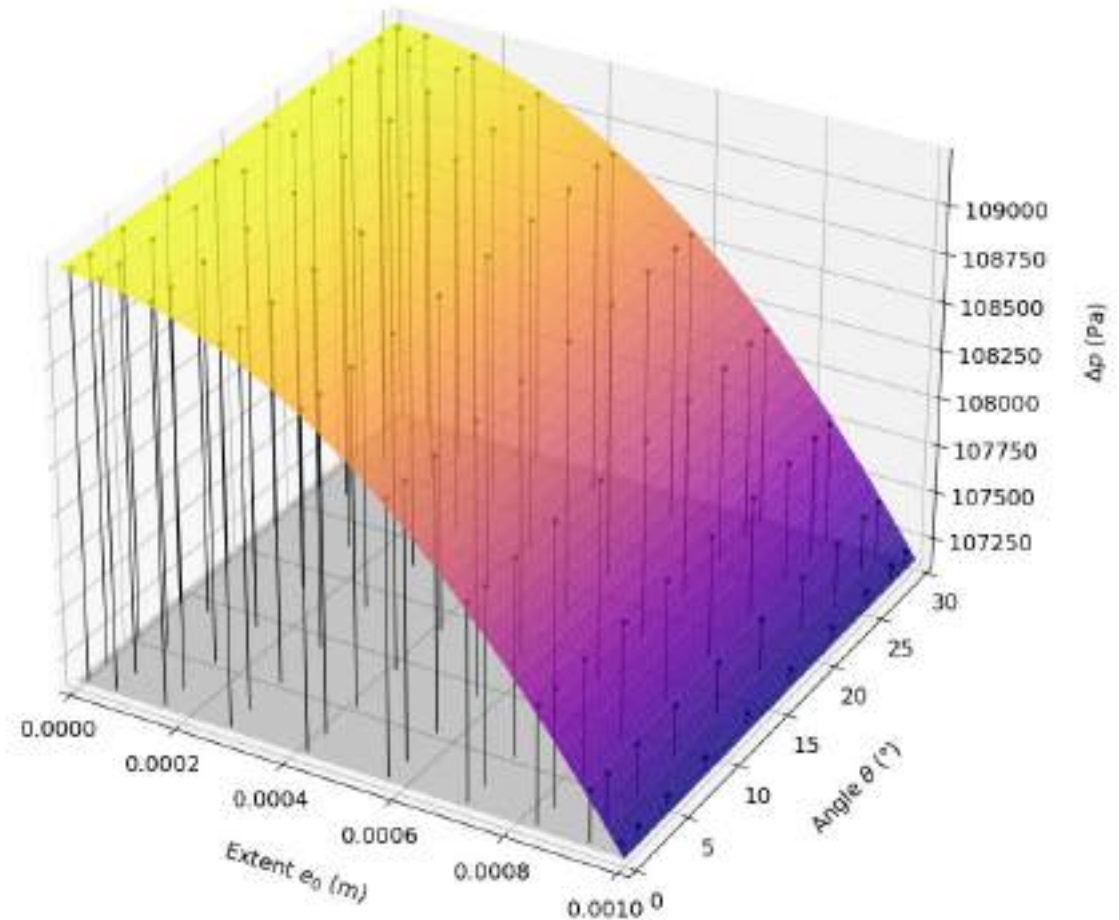
$$\langle \Psi_i, \Psi_j \rangle = \int_{\Xi} \Psi_i(\vec{\xi}) \Psi_j(\vec{\xi}) p_{\xi}(\vec{\xi}) d\vec{\xi} = \langle \Psi_i, \Psi_i \rangle \delta_{ij}$$

$$\rightarrow s_k = \frac{\langle s(\vec{\xi}), \Psi_k(\vec{\xi}) \rangle}{\langle \Psi_k, \Psi_k \rangle} = \int_{\Xi} s(\vec{\xi}) \Psi_k(\vec{\xi}) p_{\xi}(\vec{\xi}) d\vec{\xi}$$

Non-intrusive Polynomial Chaos

- $\Delta p(e_0, \theta) \approx s(\vec{\xi}) = \sum_{k=0}^P s_k \Psi_k(\vec{\xi})$
- $\text{Average}(\Delta p) = s_0$

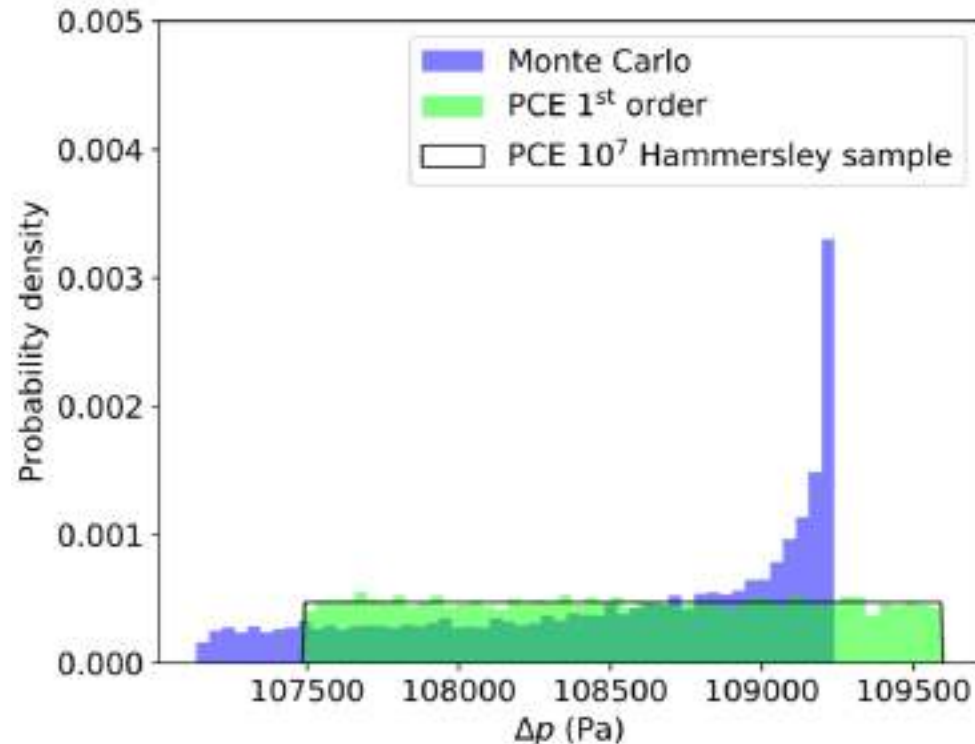
Polynomial order: 9
Number of evaluations: 100
Average $\Delta p = 108544.55$



Results Steady CFD: Pressure Drop

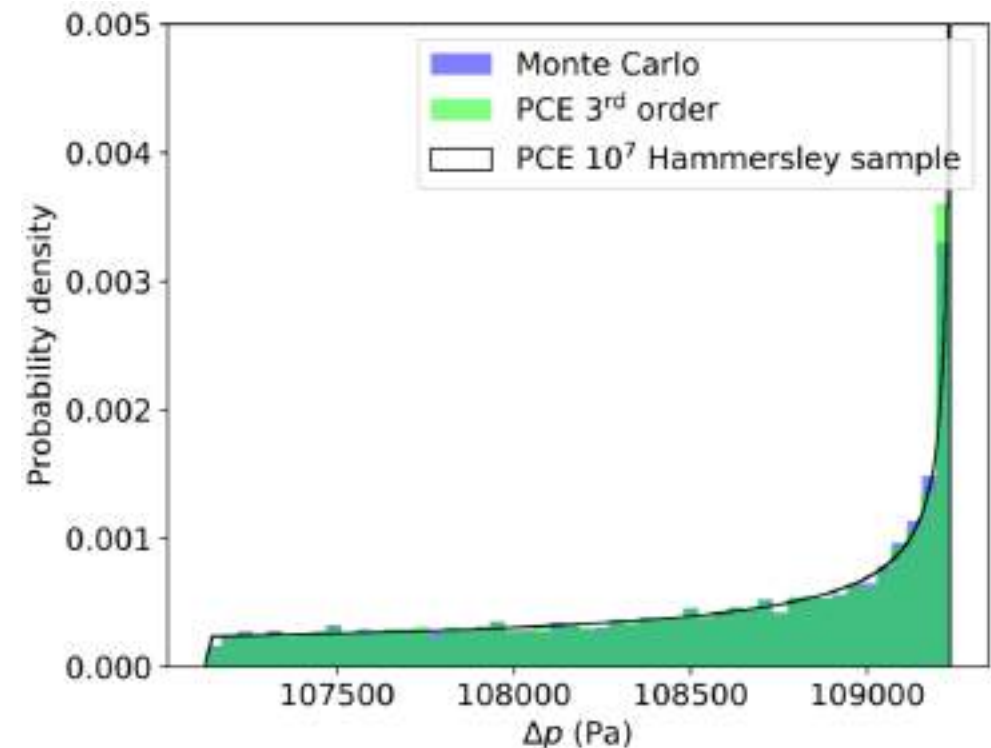


1st order PC expansion



4 model evaluations for polynomial chaos
10000 evaluations in Monte Carlo

3rd order PC expansion

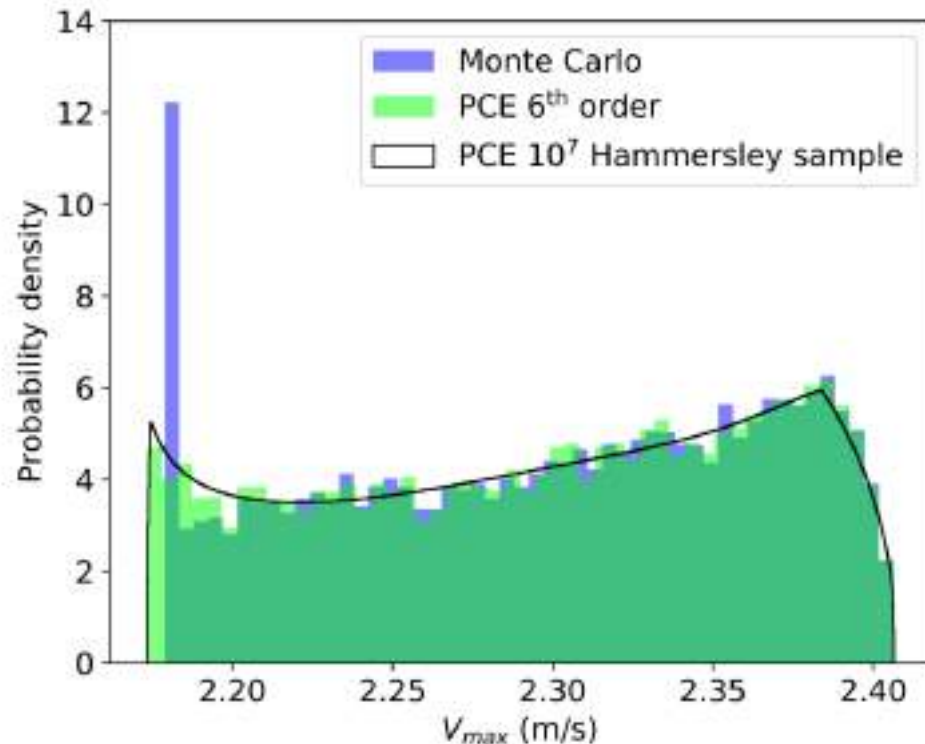


16 model evaluations for polynomial chaos
10000 evaluations in Monte Carlo

Results Steady CFD: Maximal Velocity

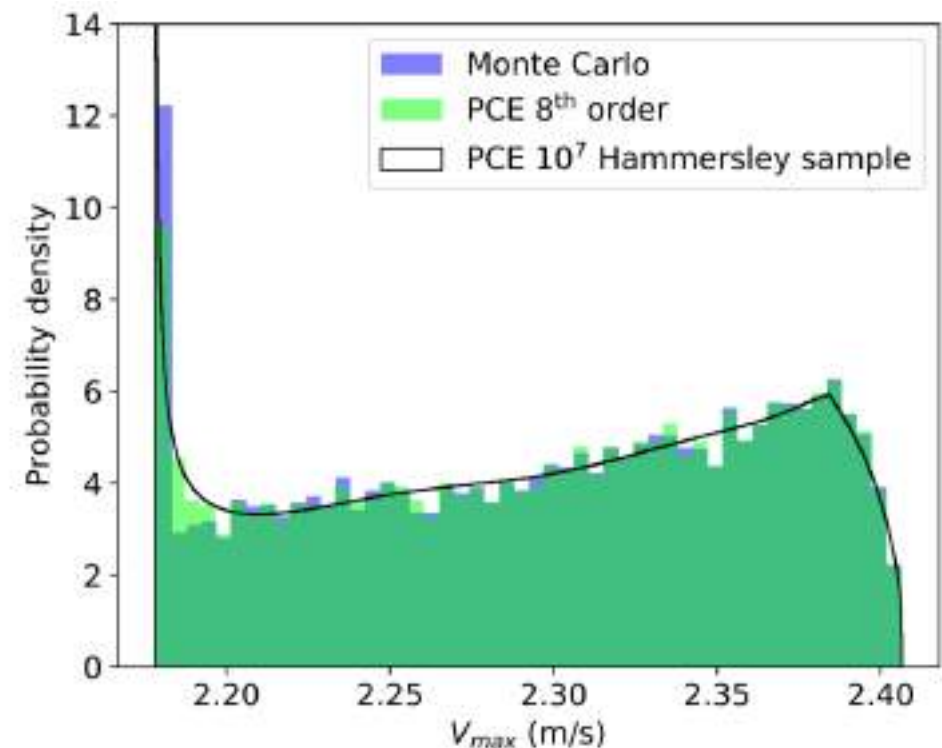


6th order PC expansion



49 model evaluations for polynomial chaos
10000 evaluations in Monte Carlo

8th order PC expansion

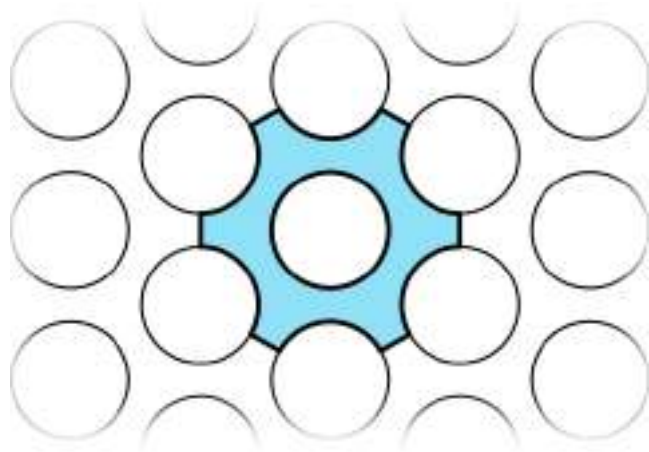
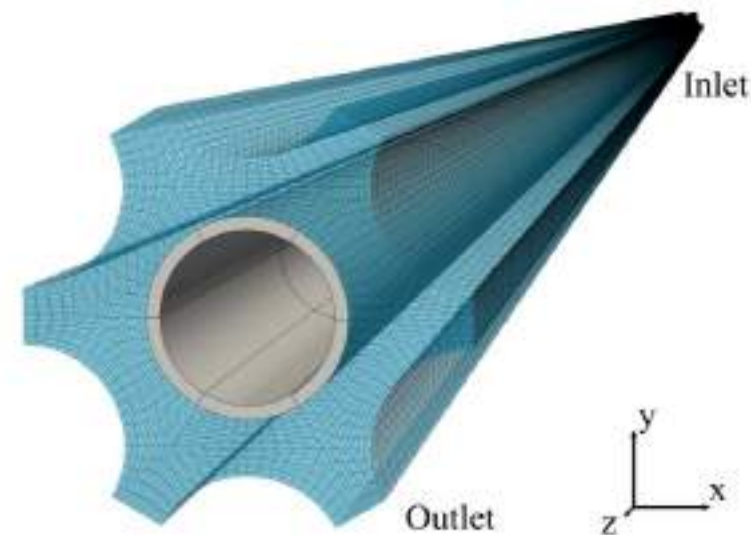


81 model evaluations for polynomial chaos
10000 evaluations in Monte Carlo

Models and Output Uncertainty

Model 2: Unsteady FSI

- Goal: Only use polynomial chaos method
- Output quantities of interest
 - Natural frequency f
 - Damping ratio ζ



Model Unsteady FSI



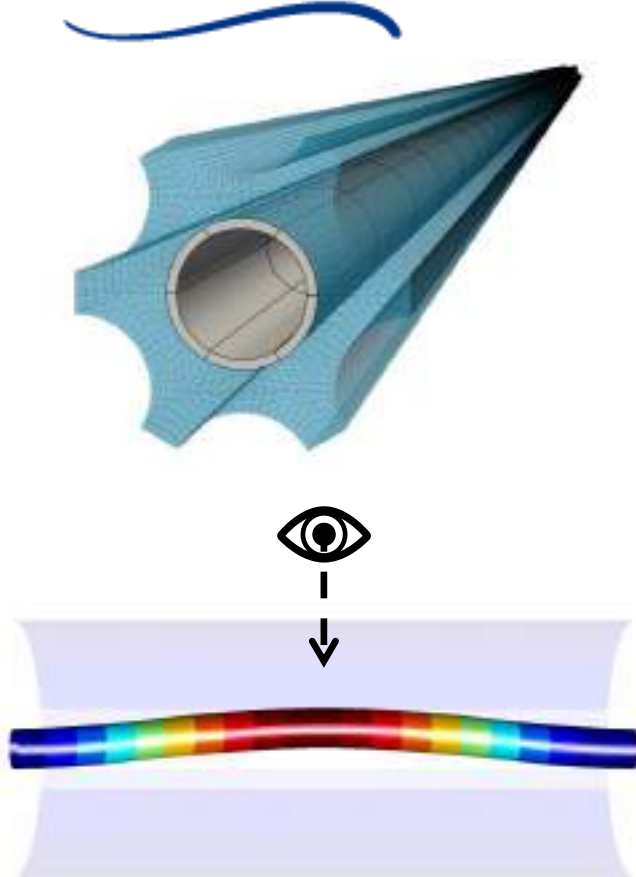
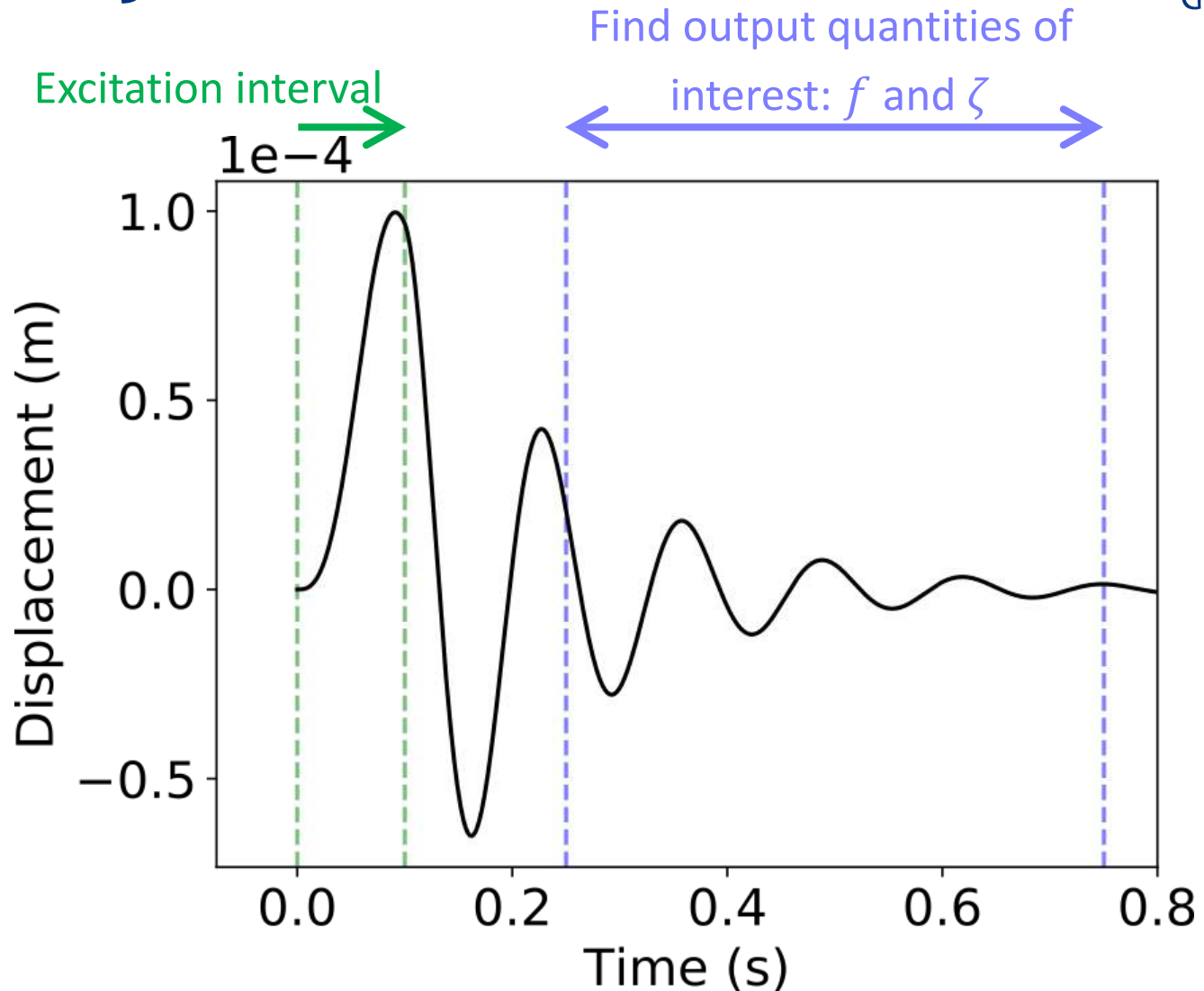
Solid

- Grid
 - 50 axial
 - 6 circumferential
- Solver
 - Simulia ABAQUS
 - Quadratic shell elements
- Boundary conditions
 - Clamped at both ends

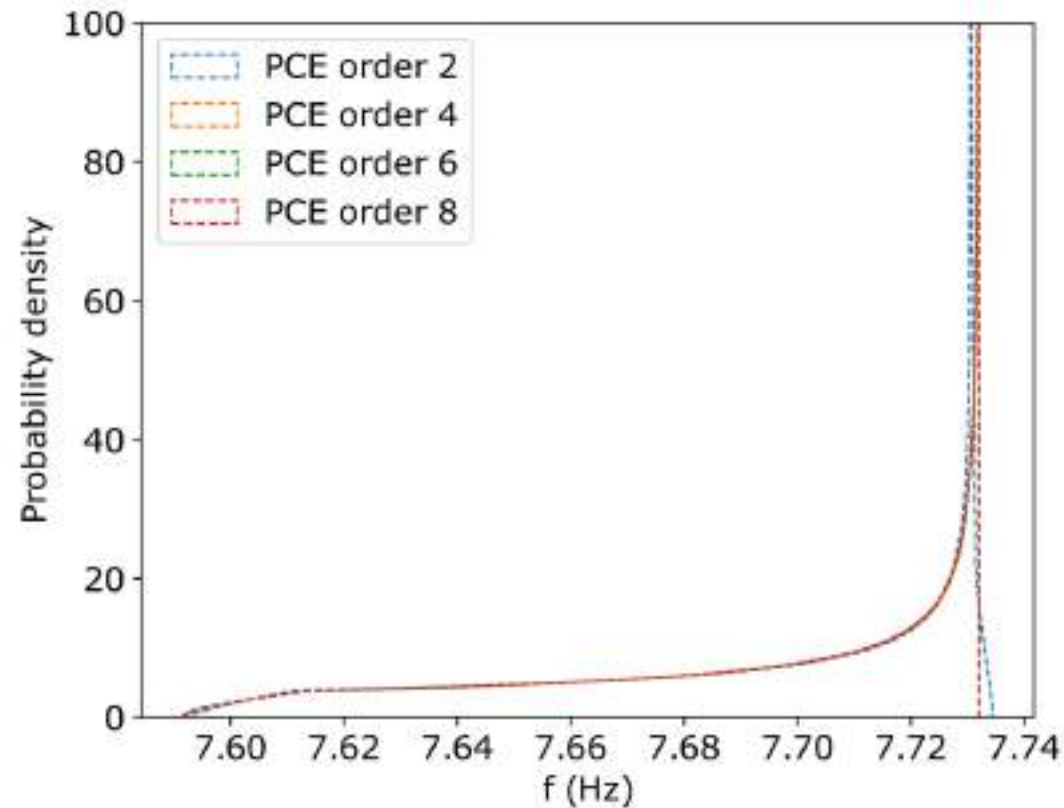
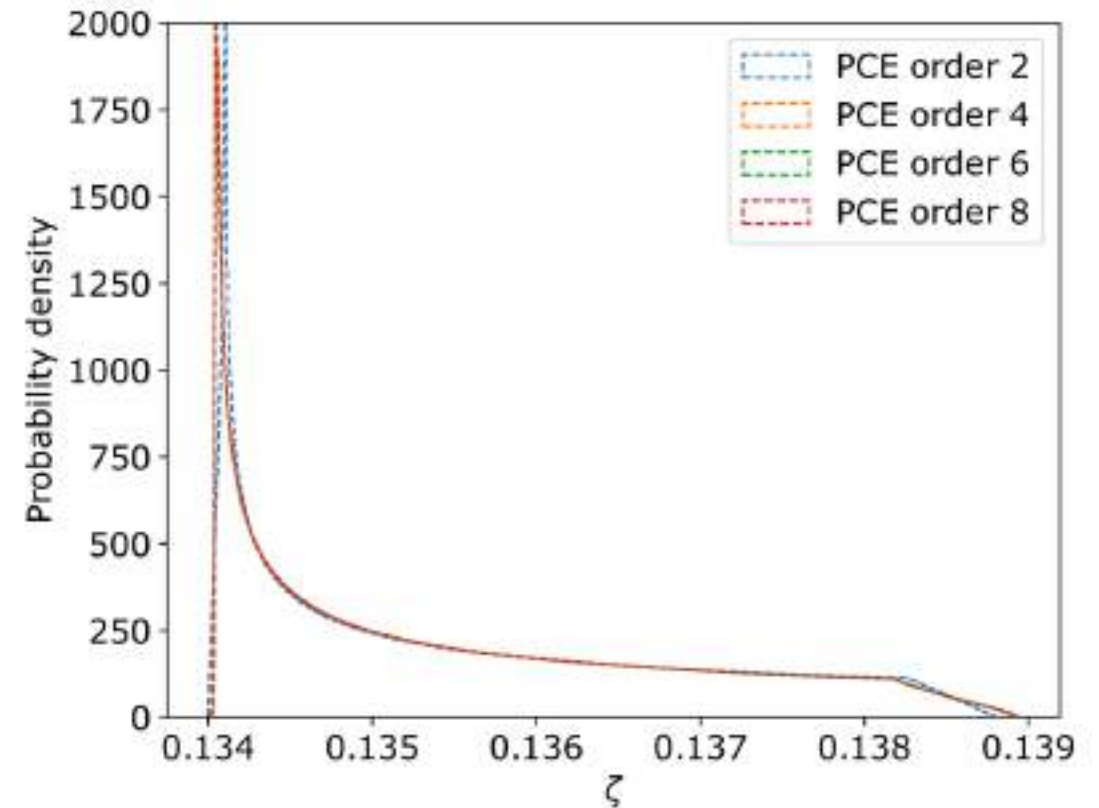
Coupling

- CoCoNuT
- ALE with grid motion like linear-elastic solid in fluid domain
- IQN-ILS

Model Unsteady FSI



Results Unsteady FSI: Histograms

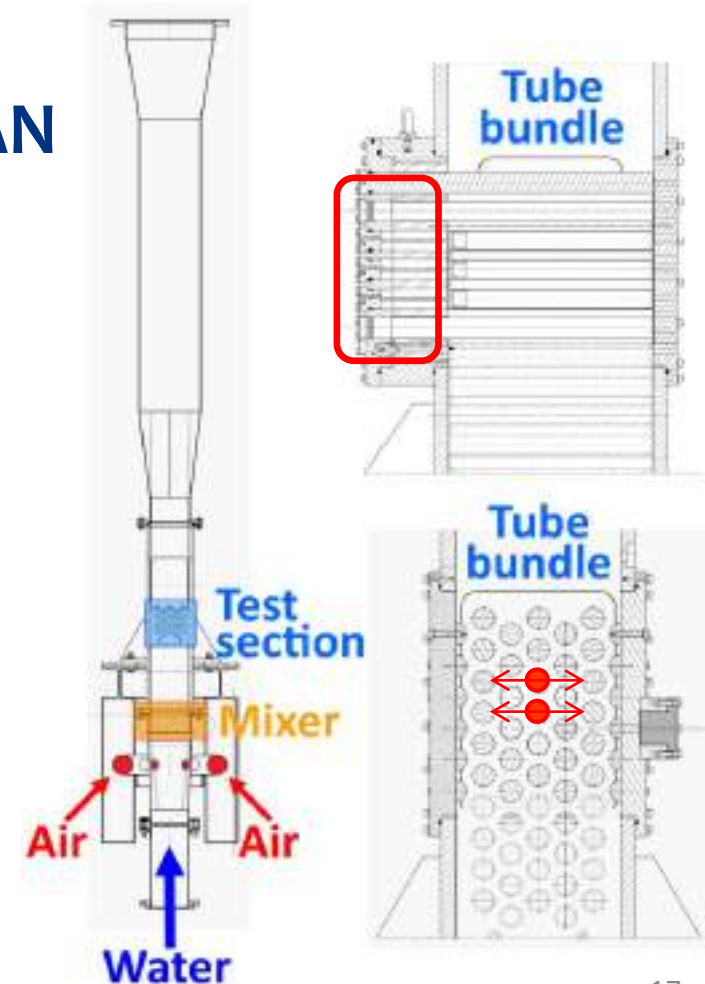


Models



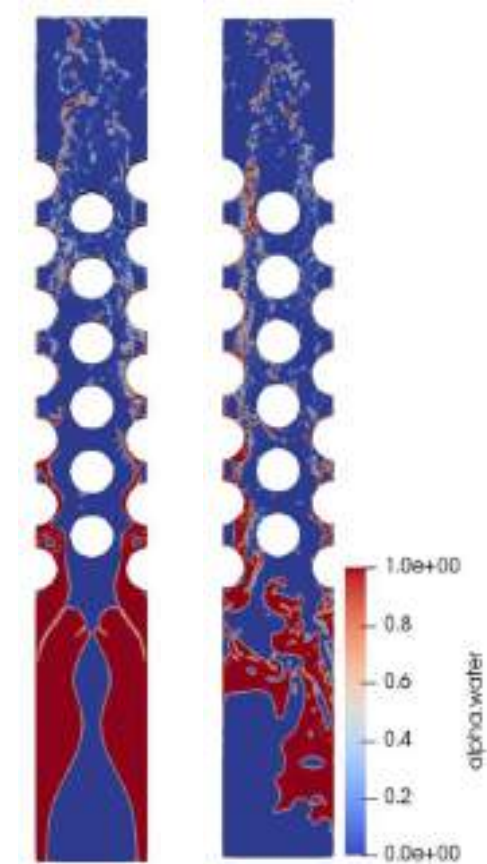
Model 3: Unsteady FSI with two-phase flow of TITAN

- Tube bundle in two-phase cross-flow (water/air)
- 1-15 tubes mounted on flexible blade
 - Approximately 1-DoF rigid body vibration in lift direction
- Simulation goals
 - FSI simulation of 2 tubes
 - Propagate effect uncertain velocity to vibration amplitude
- Requirements
 1. Rigid body motion solver
 2. Two-phase flow boundary conditions for volume of fluid (VOF)



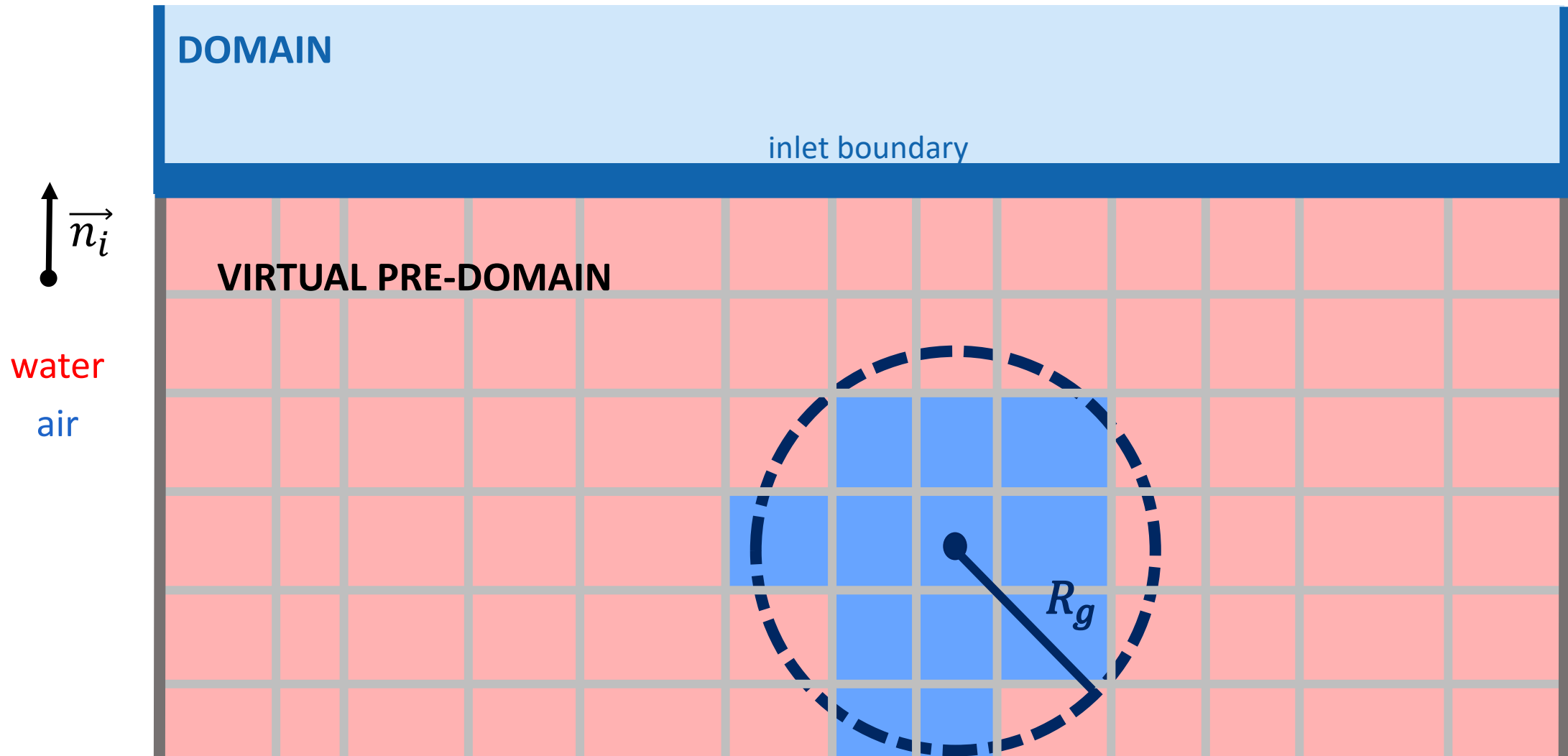
Synthetic Bubble Model (SBM)

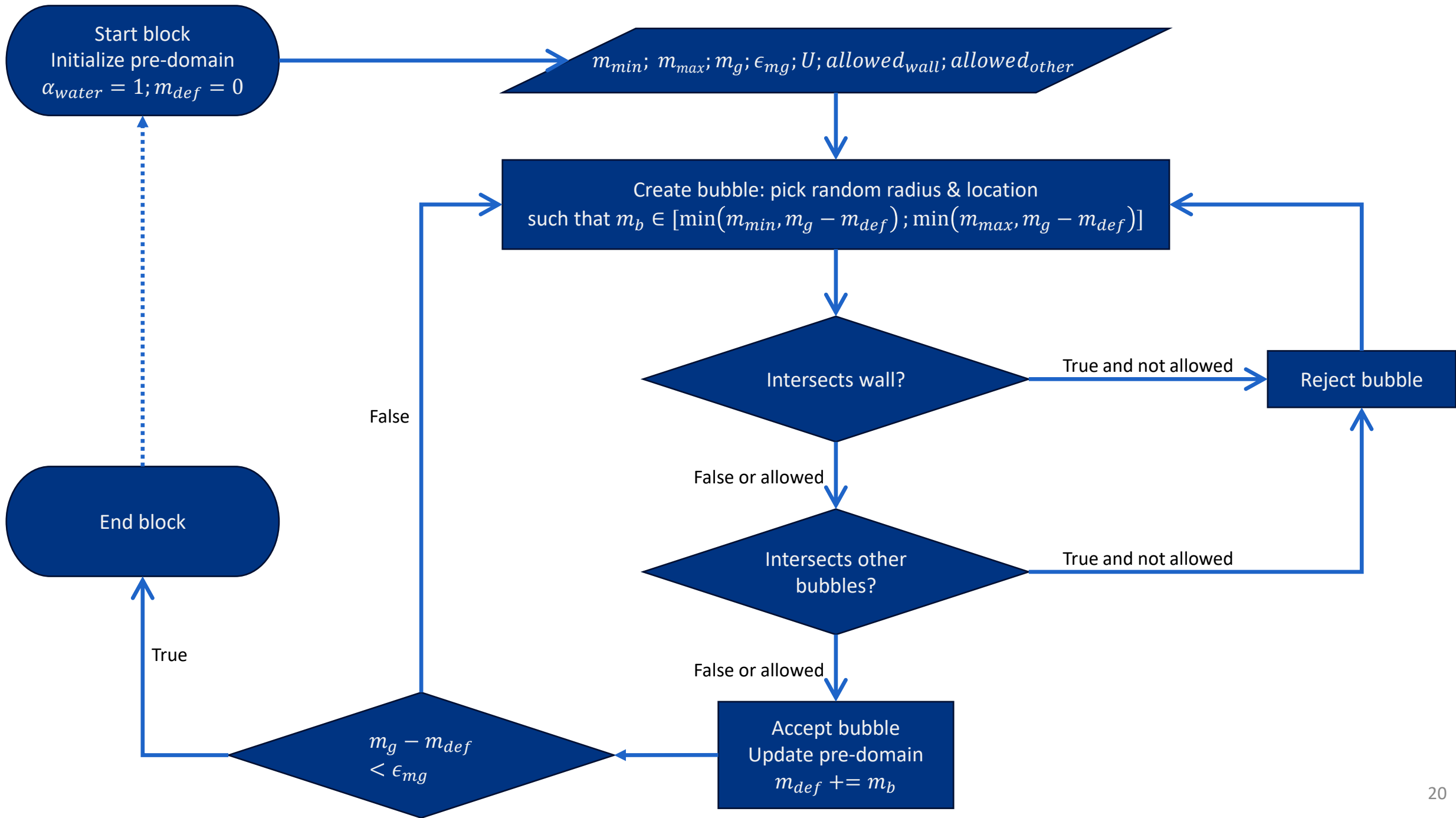
- Inlet boundary has large effect on results → search for better model
- Desired characteristics:
 - Transient (no steady jet at inlet)
 - Eulerian (mainly large bubble important for FSI)
 - Random
- Originally in OpenFOAM, now also implemented in Fluent
- Accelerated $\times 20$ for GO-VIKING
- Available on GitHub: <https://github.com/pyfsi/SBM>



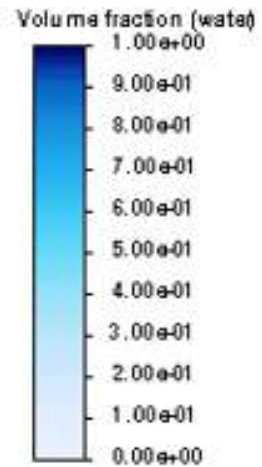
1 air jet SBM

Synthetic Bubble Model (SBM)

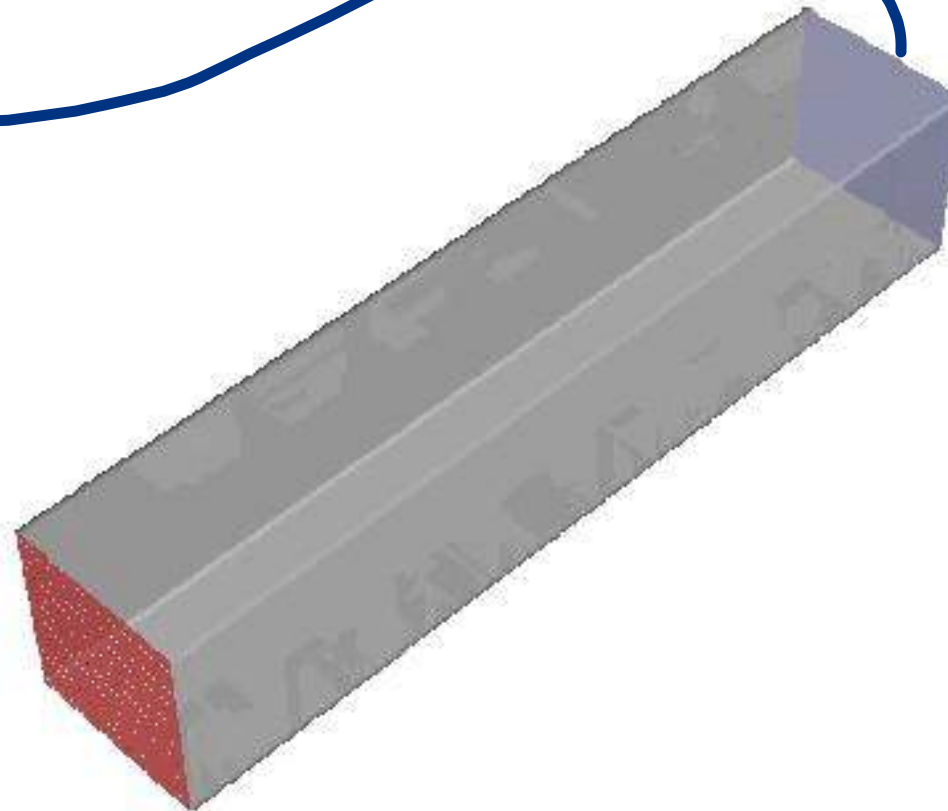




Synthetic Bubble Model (SBM)

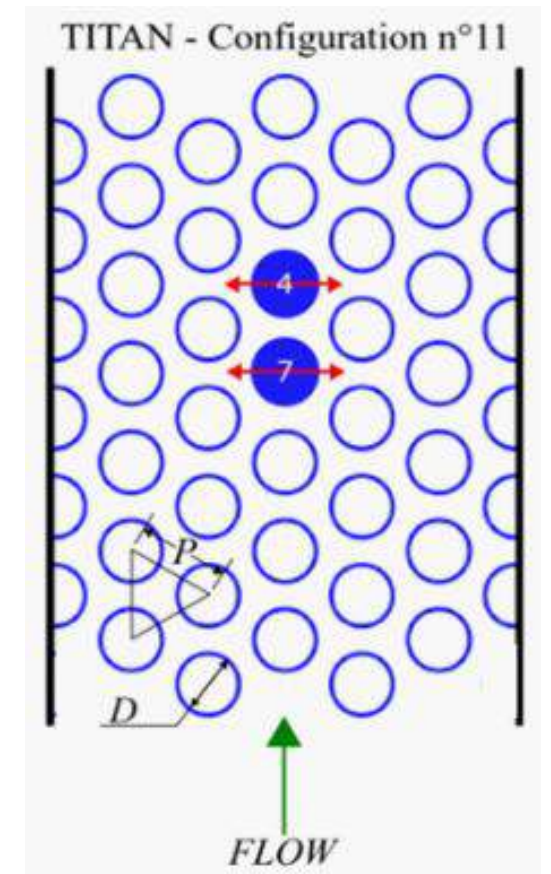


Transient inlet profile



Rigid Body Motion Solver

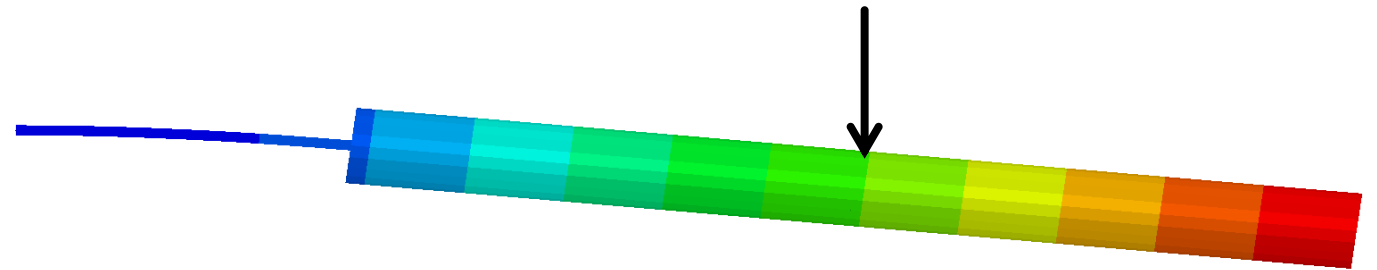
- Two coupled 1-DoF rigid body solver via Fluent UDF
 - Cylinder 4 and 7 vibrate in 1D as rigid body
 - Newmark integration scheme
 - Forces and displacements implicitly coupled, accelerated using Jacobian information on $\mathbf{F}(\ddot{\mathbf{x}})$ from finite differences
- Validation with experimental data using AMOVI Test 27



Rigid Body Motion Solver

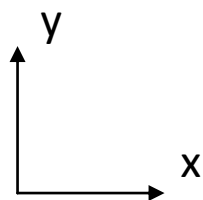
Material properties

- Stiffness determined from displacement at middle of tube
- Mass and damping constant derived from stiffness, natural frequency and damping ratio
- Displacement from FSI simulation scaled up ($\times 1.74$) to represent end of tube
 - $k = 6857.14 \text{ N/m}$
 - $m = 0.86 \text{ kg}$
 - $c = 0.069 \text{ Ns/m}$

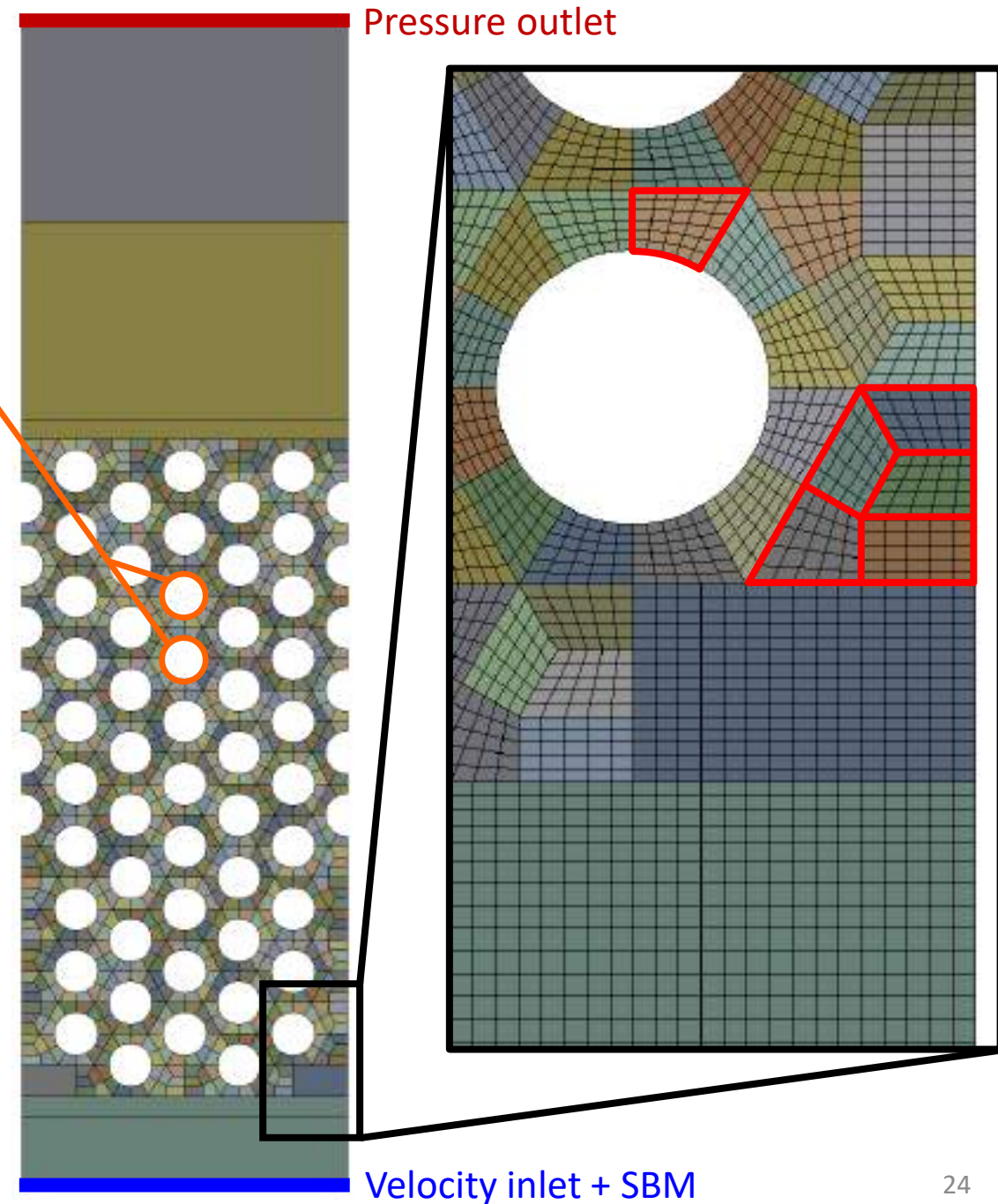


CFD Model

- 1/3rd of depth → 1.37M cells
- Ansys Fluent 2023R1
- URANS k- ω SST, 1st order implicit in time with $\Delta t = 0.0005 \text{ s}$
- Volume of fluid (VOF) method with implicit formulation
- Free slip at front and back
- Velocity inlet with SBM for TITAN conf. n°11:
 - $U_P \sim \mathcal{N}(1.2259 \text{ m/s}; 2.74 \times 10^{-3} \text{ m}^2/\text{s}^2)$
 - $\alpha_{air} = 34.6 \%$
 - $U_P = U \cdot \frac{P}{P-D}$



Moving wall



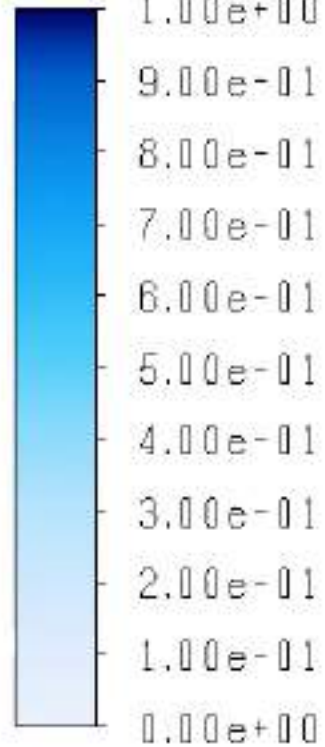
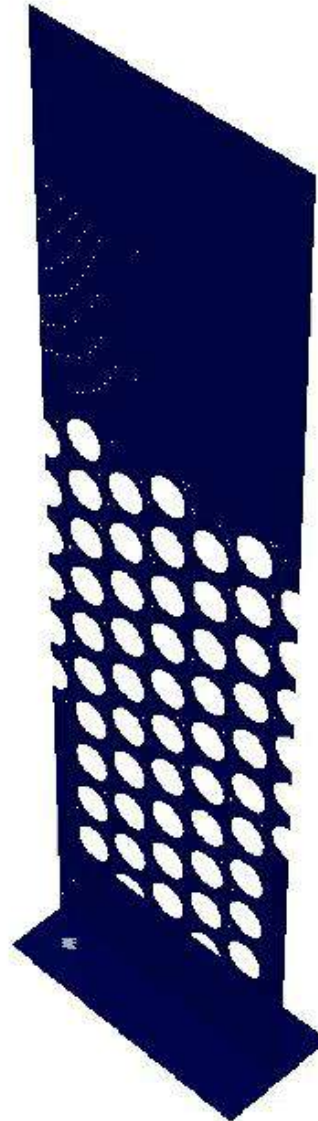
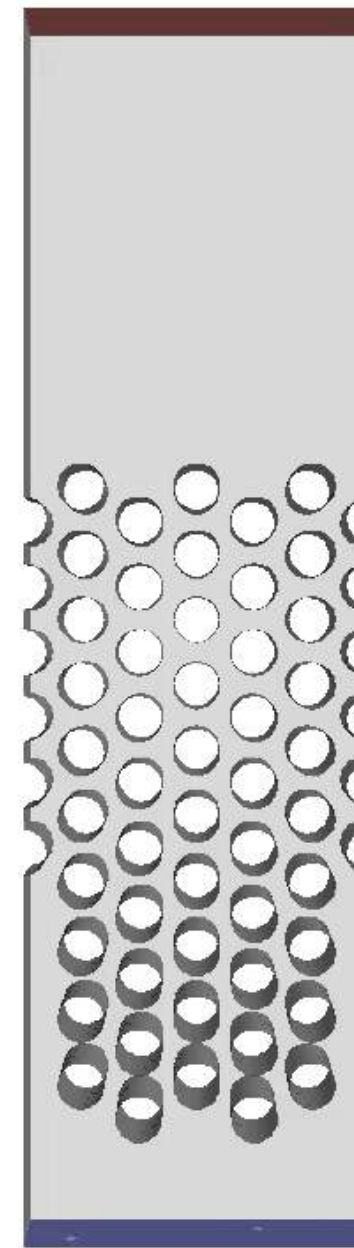
Simulations done for 5 different values of U_P and 5 different bubble diameters $d \rightarrow 25$ FSI simulations in total

$U_P \left(\frac{m}{s}\right)$	1.0764	1.1549	1.2259	1.2968	1.3753
×					
$d \text{ (mm)}$	20	22.628	25	27.372	30

CFD Results

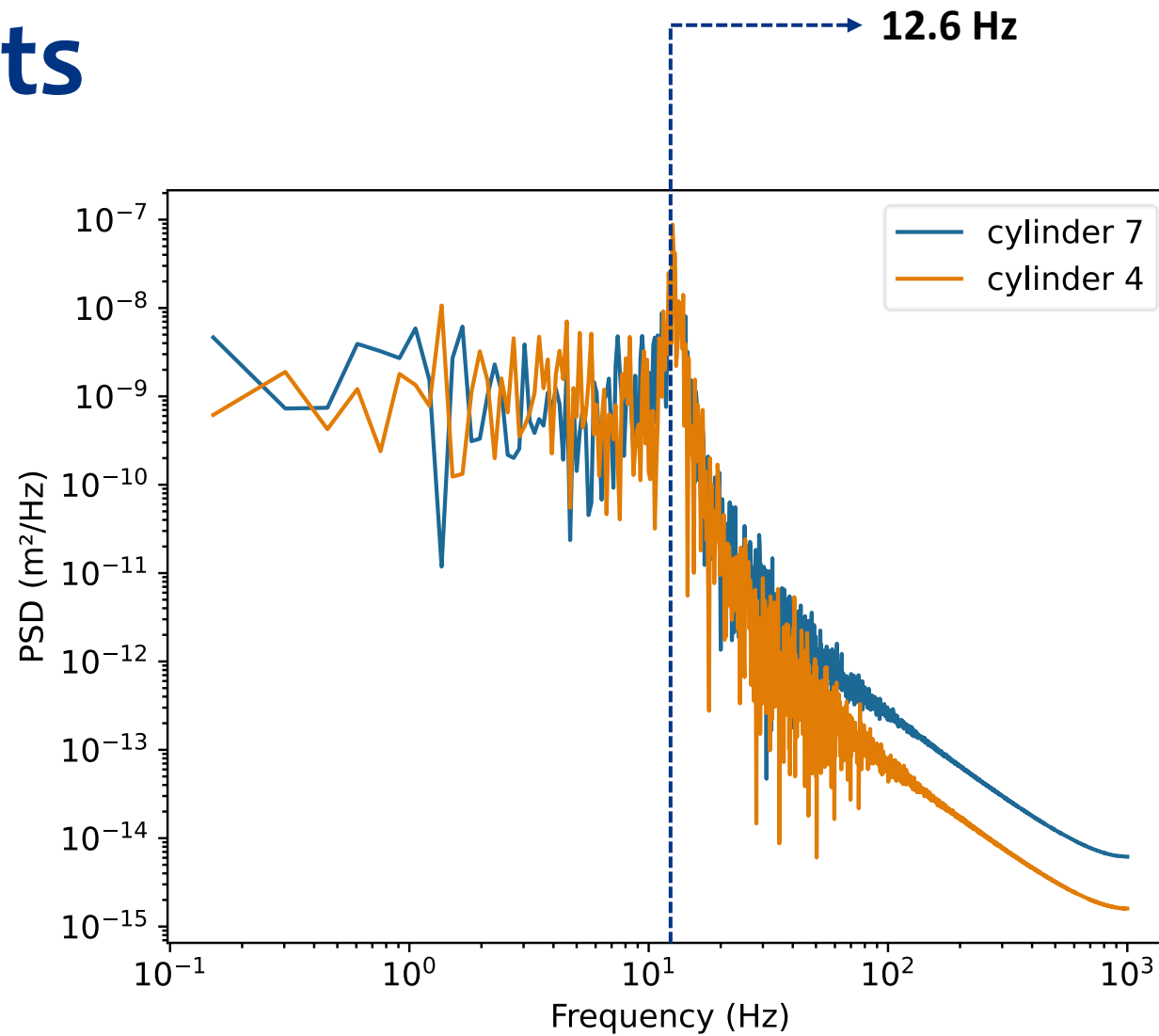


Volume fraction (water)



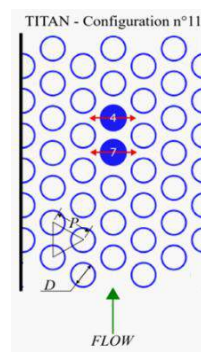
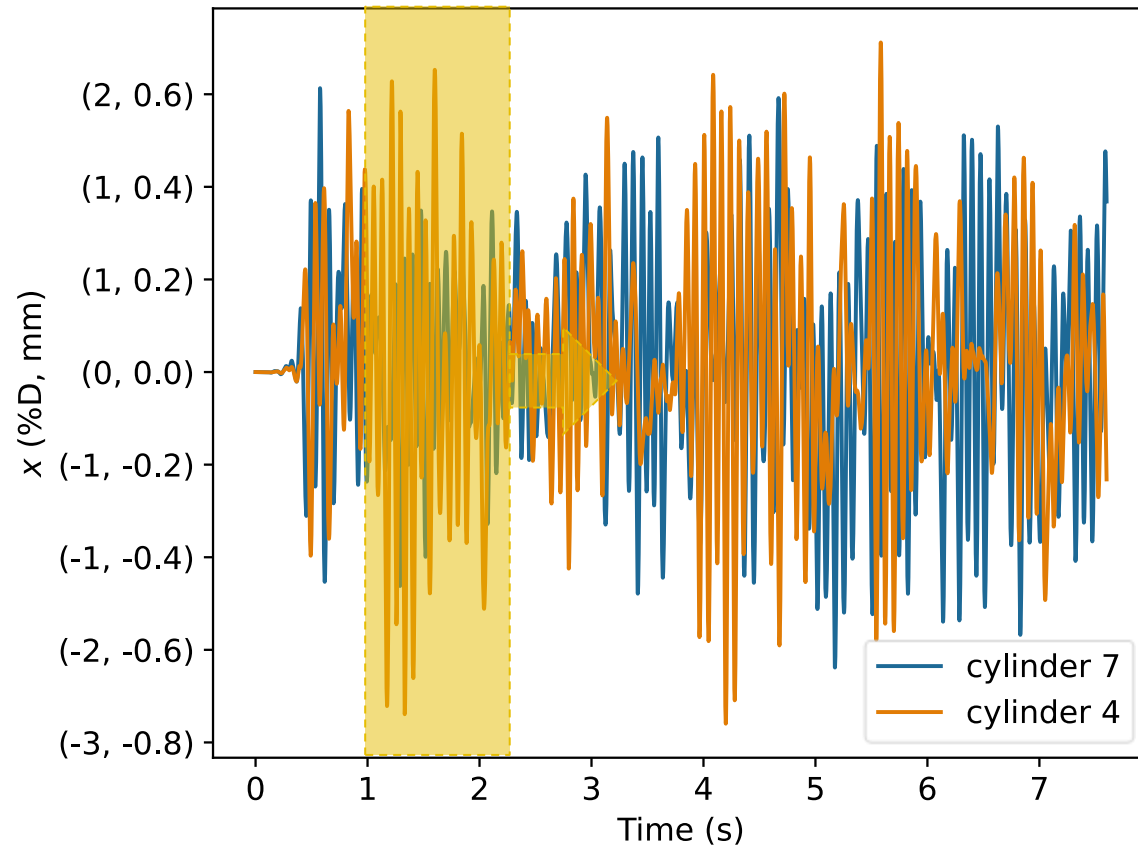
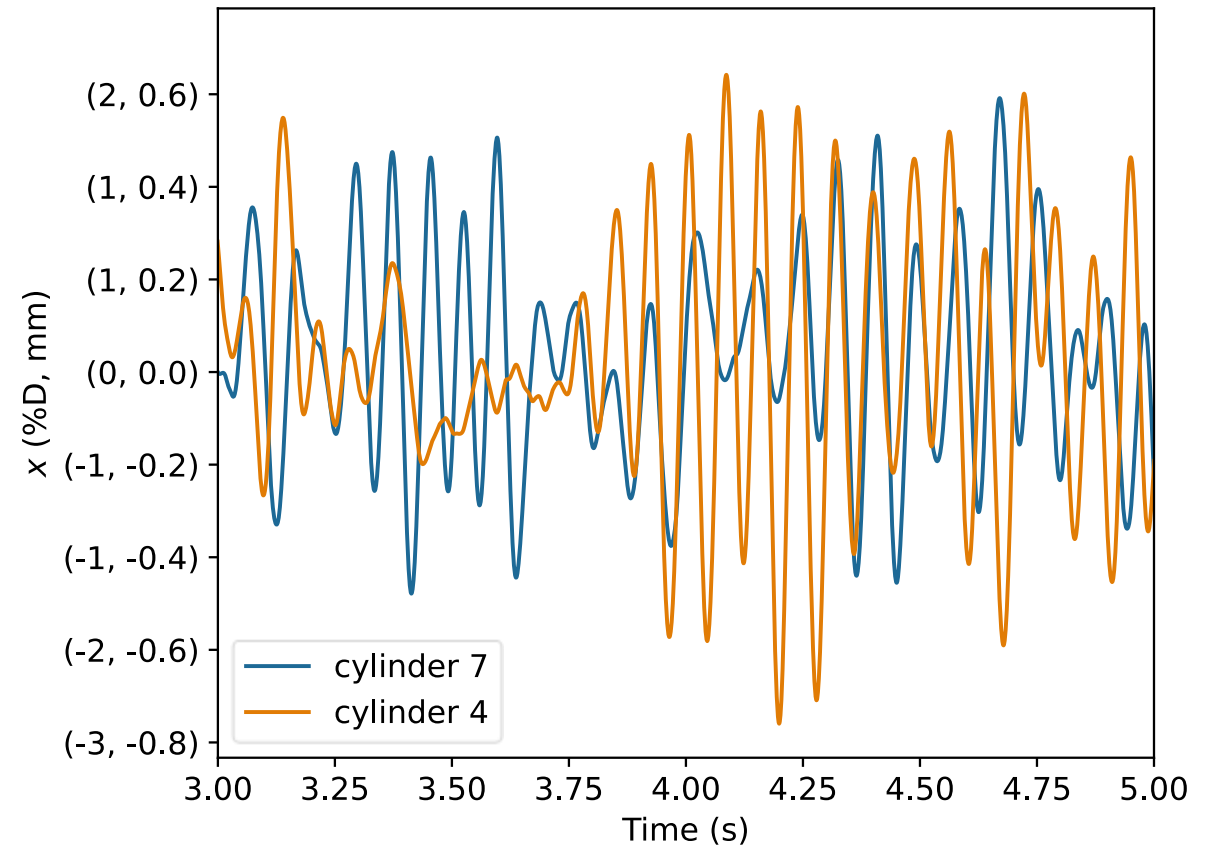
Simulation with $U_P = 1.2259 \text{ m/s}$ and $d = 25 \text{ mm}$. Left: contour of α , Right: iso-surface of $\alpha = 0.5$.

FSI Results

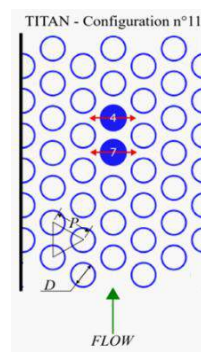
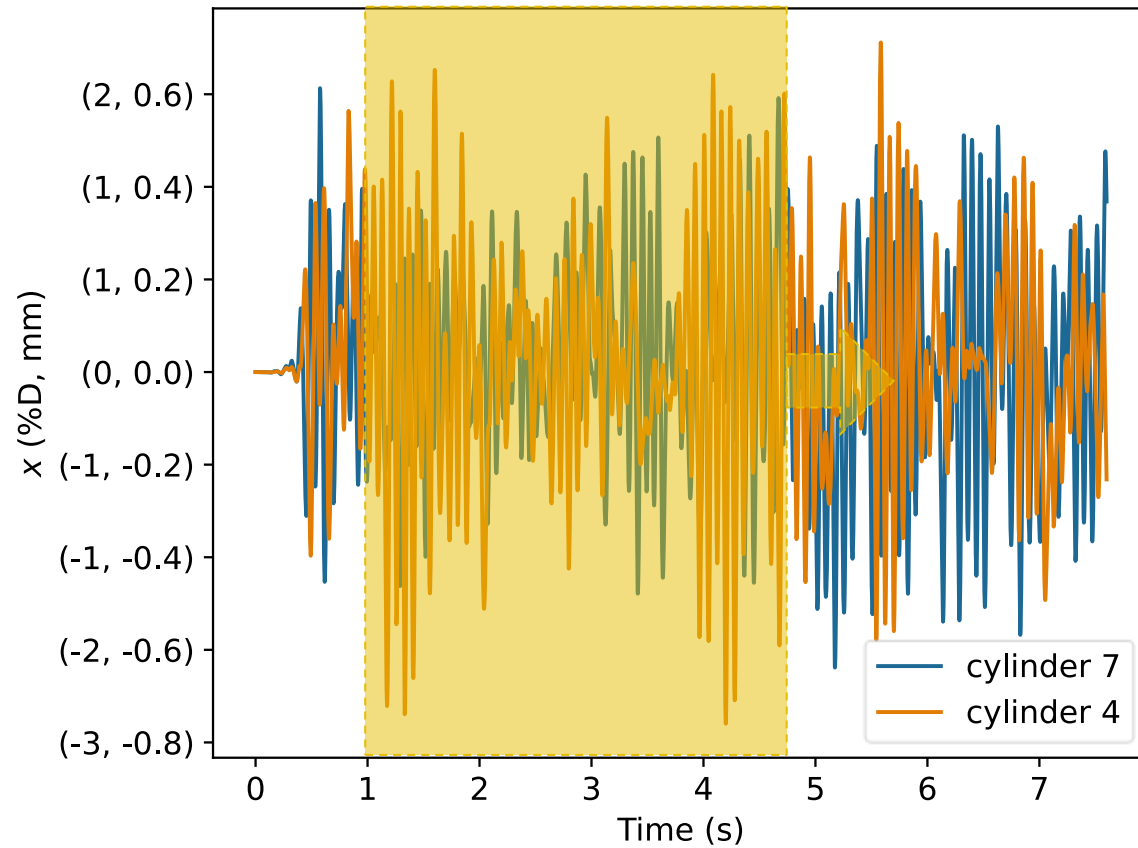
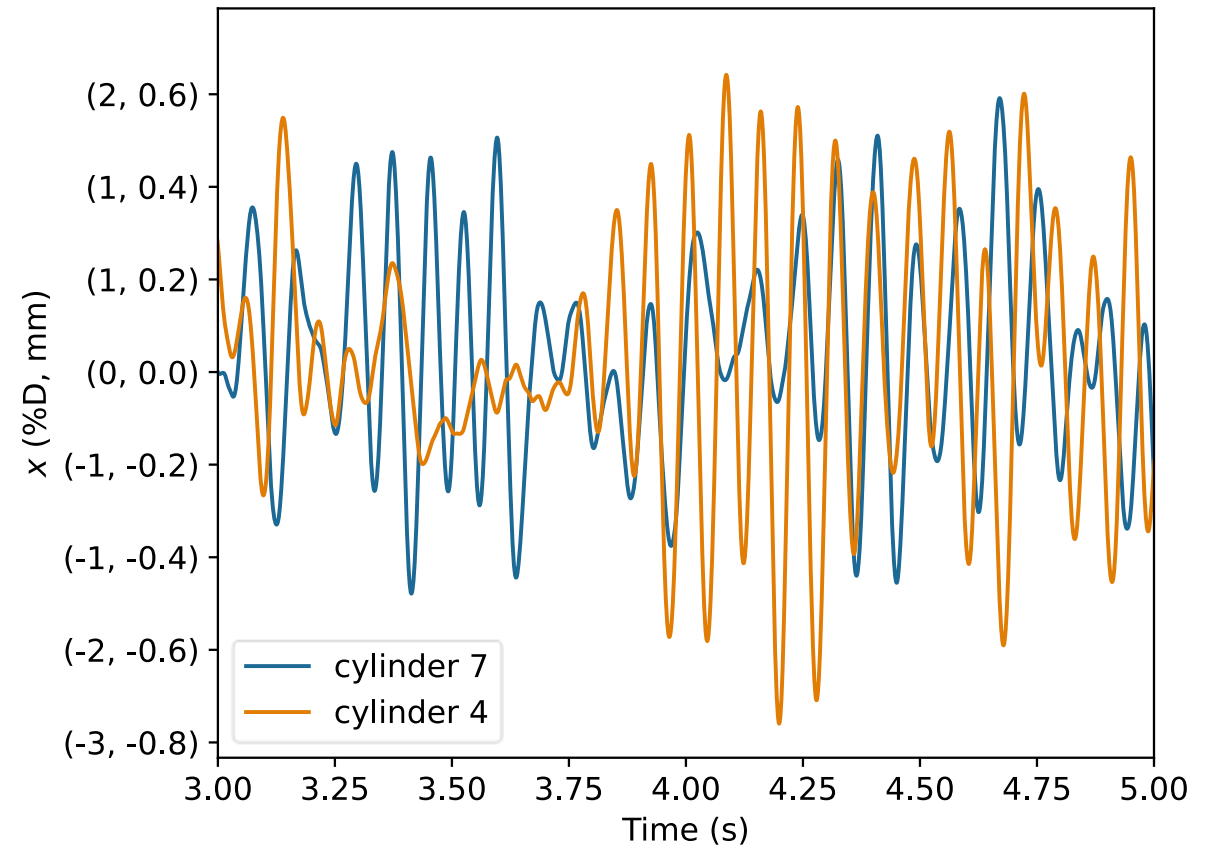


Simulation with $U_P = 1.2259 \text{ m/s}$ and $d = 25 \text{ mm}$.

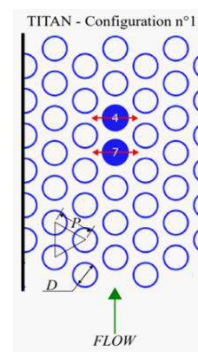
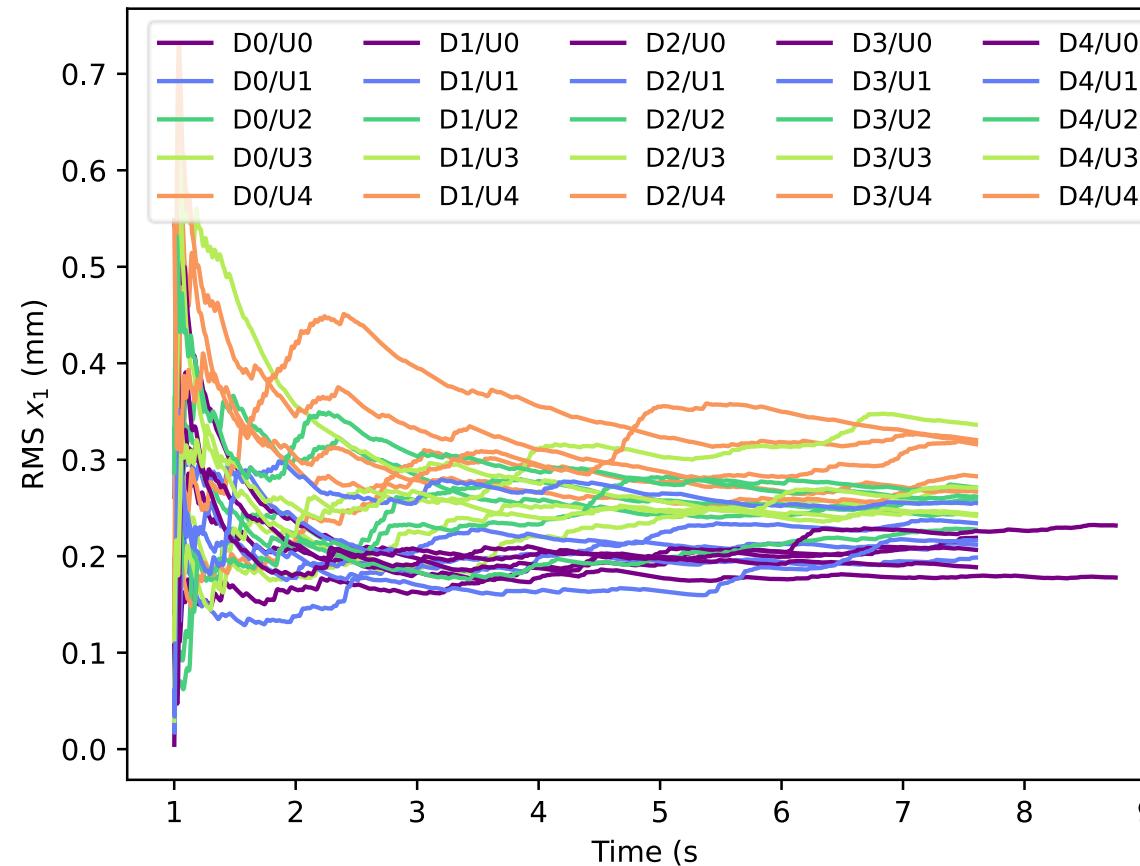
FSI Results



FSI Results



RMS Convergence

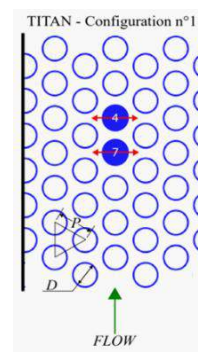


Convergence of the value of the RMS vibration amplitude for cylinder 7

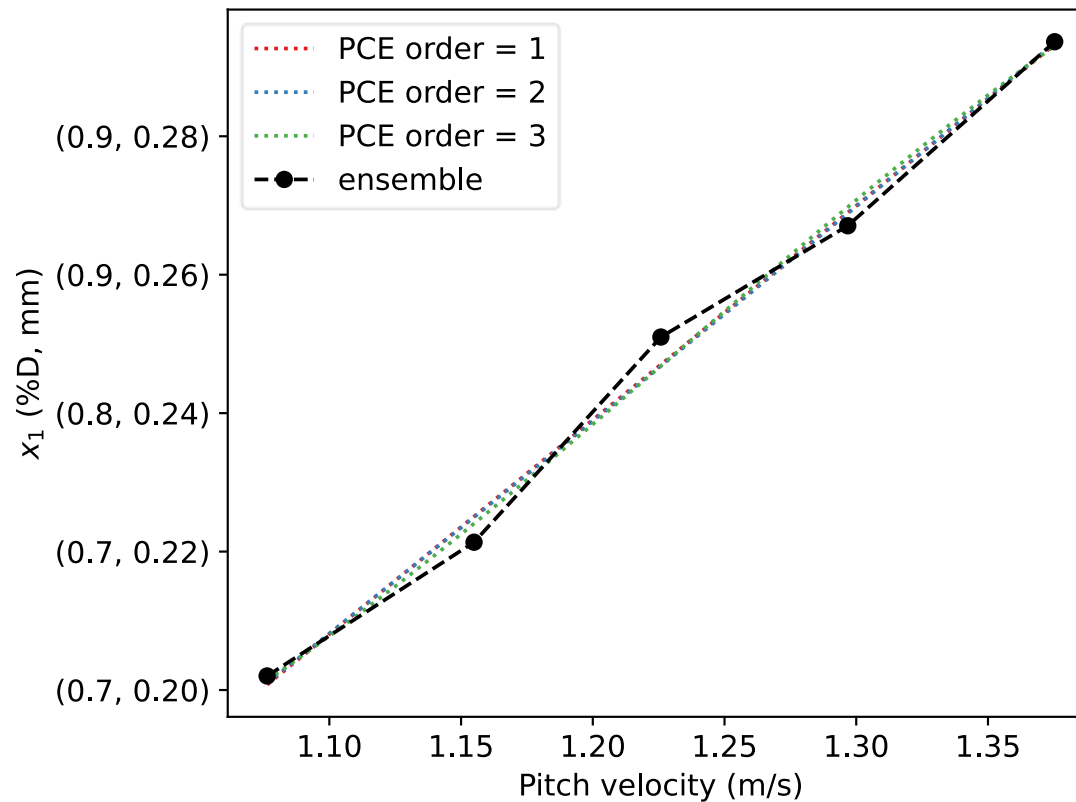
RMS Convergence

- Slow convergence RMS → difficulties for UQ
 - Residual error deteriorates quality of polynomial chaos expansion
 - Much longer simulations not feasible
- Ensemble average over diameter taken
 - 25 simulations → 5 results
 - As if simulation becomes 5 times longer
- Least squares method used to be able to test multiple orders of PCE
 - Up to 3rd order worked well
 - 4th order overfitting phenomenon due to remaining error on RMS

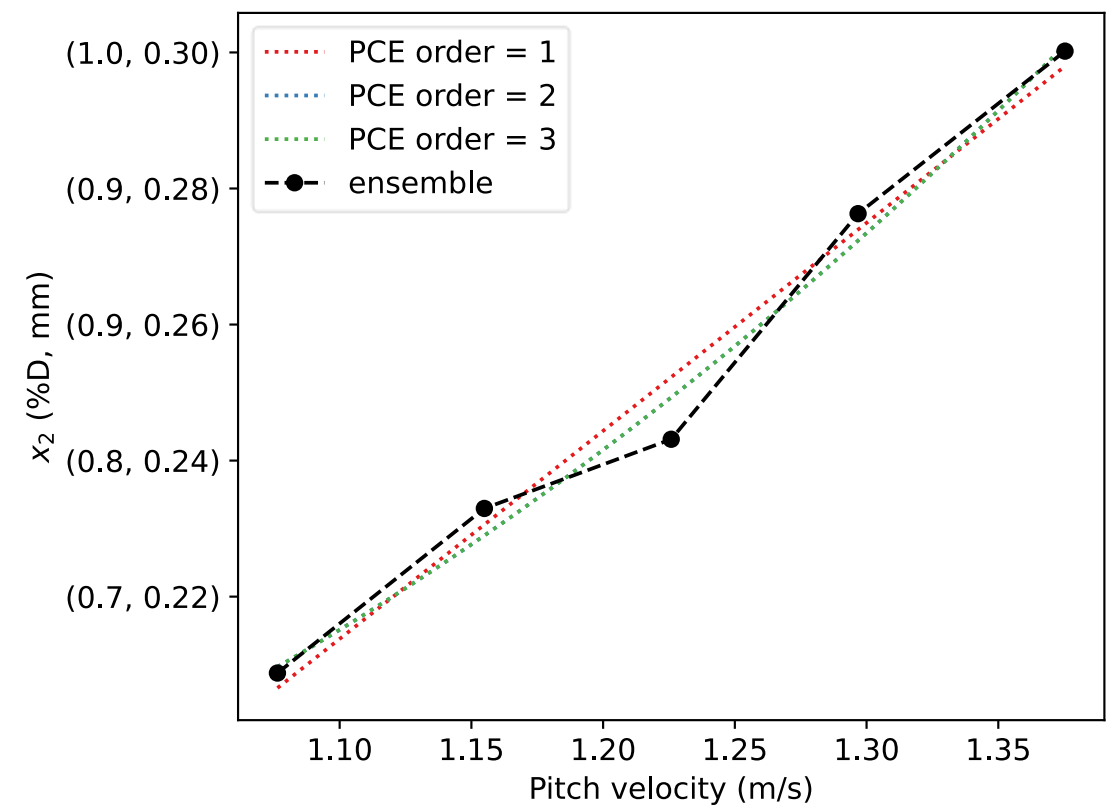
UQ Results



PCE model cylinder 7



PCE model cylinder 4



Conclusions

- Compared Monte Carlo with Polynomial Chaos for pressure drop and maximal velocity in tube bundle to validate workflow
- Developed approach to transform uncertainty in initial shape of tube bundle into distribution of modal characteristics using unsteady FSI
- Demonstrated UQ approach on two-phase TITAN experiment using unsteady FSI with Synthetic Bubble Model and two coupled 1-DoF rigid body solvers
- Biggest challenge: slow convergence of RMS value and high computational cost of time-dependent FSI simulation

References

1. H. Dolfen, S. Vandewalle, and J. Degroote. Effect of stochastic deformation on the vibration characteristics of a tube bundle in axial flow. *Nuclear Engineering and Design*, 411(112412):1–14, 2023. doi: [10.1016/j.nucengdes.2023.112412](https://doi.org/10.1016/j.nucengdes.2023.112412).
2. N. Delaissé, T. Demeester, R. Haelterman, and J. Degroote. Quasi-Newton methods for partitioned simulation of fluid-structure interaction reviewed in the generalized Broyden frame work. *Archives of Computational Methods in Engineering*, 30:3271–3300, 2023. doi: [10.1007/s11831-023-09907-y](https://doi.org/10.1007/s11831-023-09907-y).
3. L. De Moerloose, A. Bral, T. Demeester, M. De Paepe, and J. Degroote. Effect of a new synthetic bubble model on forces in simulations of two-phase flows in tube bundles. *European Journal of Mechanics B-Fluids*, 90:49–62, 2021. doi: [10.1016/j.euromechflu.2021.08.003](https://doi.org/10.1016/j.euromechflu.2021.08.003).



Thank you!



Joris Degroote



Ghent University



joris.degroote@ugent.be



Website:
www.go-viking.eu



Mail:
contact@go-viking.eu



Twitter:
[@GOVIKING_EU](https://twitter.com/GOVIKING_EU)



LinkedIn:
[GO-VIKING](https://www.linkedin.com/company/go-viking)



**Funded by the
European Union**

Funded by the European Union. Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union or European Atomic Energy Community ('EC-Euratom'). Neither the European Union nor the granting authority can be held responsible for them.



WP4: FSI in two-phase Cantilever Case

Ewan Harris, Constantinos Katsamis, Ehimen Iyamabo

21/05/2026

EDF UK



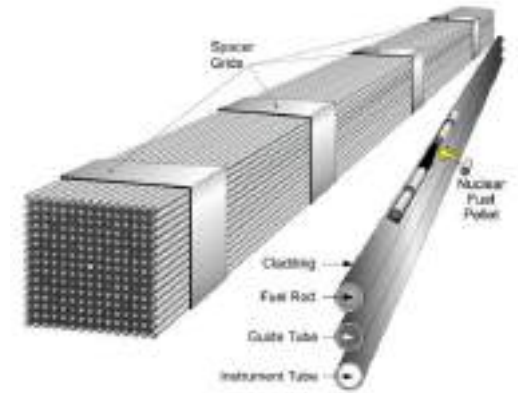
Funded by
the European Union

Contents



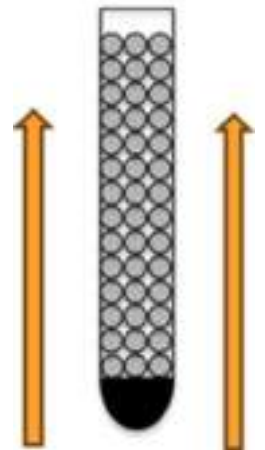
- Background and motivation
- Experimental benchmark
- Synthetic Bubble Method / inlet challenge
- EDF UK modelling approach
- Flow-induced vibration results in StarCCM+
- Lessons learned and future improvements

Background and Motivation



Why this benchmark matters

- Slender structures exposed to axial multiphase flow can experience flow-induced vibration.
- This can affect component integrity, wear and long-term reliability in critical reactor core components
- The GO-VIKING WP4 benchmark investigates a cantilever rod exposed to upward air-water flow.
- The objective is to assess whether numerical CFD-FSI methods can reproduce measured vibration behaviour at reasonable cost

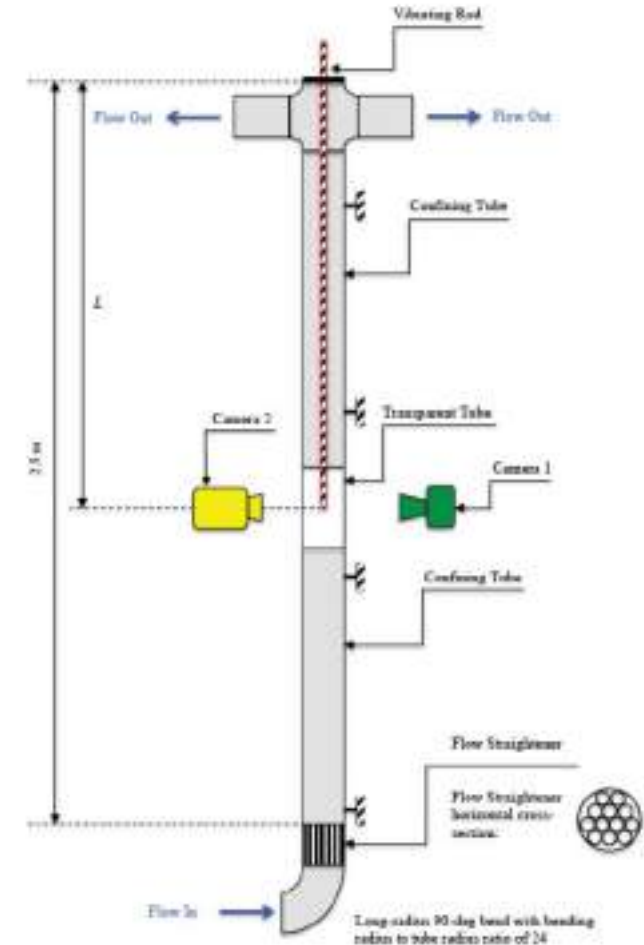


Experimental Benchmark



UoM benchmark cantilever rod in axial air-water flow

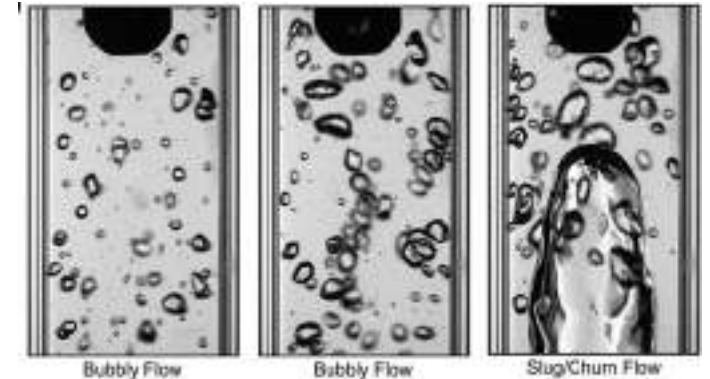
- Rod clamped at the top and free at the bottom.
- Flow passes through the annulus between the rod and confining tube.
- Rod-tip motion measured using a non-intrusive Hall-effect sensor method.
- EDF UK focused on the UoM P0T3 cantilever benchmark case:
 - Liquid superficial velocity, $U_{ls} = 1.7 \text{ m/s}$
 - Gas superficial velocity, $U_{gs} = 0.35 \text{ m/s}$



Modelling Objective



- Build a two-phase CFD-FSI model of the cantilever rod experiment
- Use STAR-CCM+ v2024.6 to couple the air-water flow field with the structural
- Perform comparisons against experimental data and predict:
 - rod-tip displacement in two transverse directions
 - frequency-domain response
- Compare numerical results with The University of Manchester's experiments.

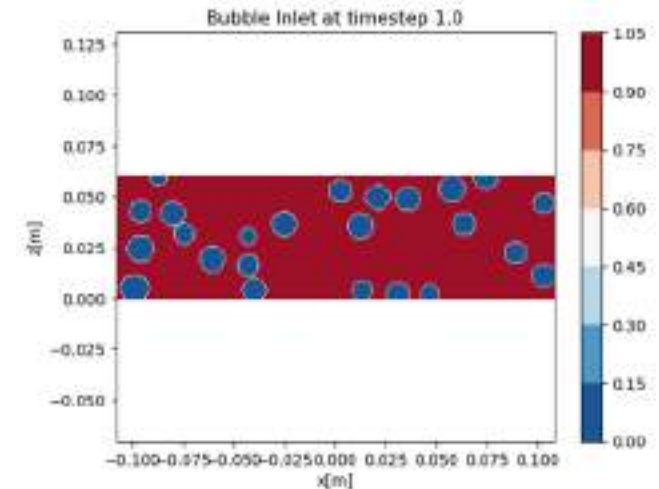
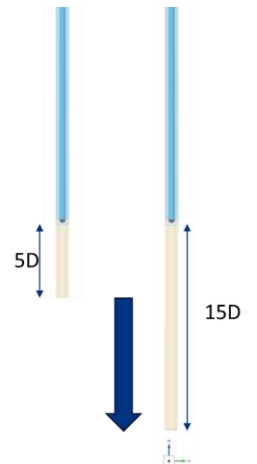


Synthetic Bubble Method / Inlet Challenge



Key modelling challenge: two-phase inlet definition

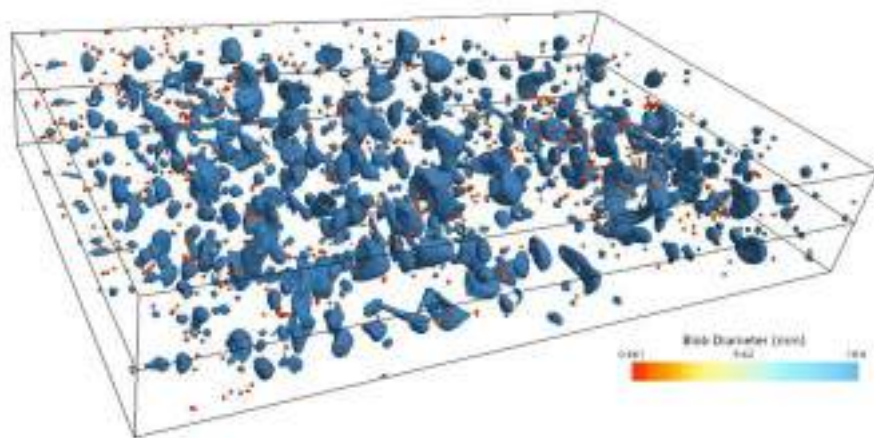
- Adopted transient inlet from SBM of Degroote (2021):
 - Modified the code to impose the experimental bubble size and gaussian distribution aiming at bubbly/intermittent regime.
 - Still the method raises concern of gas injected and void fraction.
- High uncertainty about inlet conditions: Experimental bubble size distribution and detailed void-fraction profile were not available.
- This made it difficult to prescribe a physically representative bubble inlet.
- EDF UK did not rely on a final synthetic bubble inlet approach in the reported StarCCM+ calculation.
- Instead, the final EDF UK approach used a mixture velocity inlet:
 - Inlet extended to allow sufficient development of the flow and volume fraction profiles through the interaction of inertia, gravity, shear, and surface tension effects
 - Successfully deployed for TITAN and TREFLE cases
- The two-phase flow structures were then allowed to develop within an extended upstream inlet section.



Synthetic Bubble Method / Inlet Challenge



- Contours of VOF and particle representations of detected blobs (non-contiguous regions of air)
- Much less diffusive interface and large-scale bubble break-up coalescence captured
- Air bubble size distribution by bubble count (not to same timescale as animation)
- Extracted using the blob detection algorithm



Solution Time 2.03099 (s)

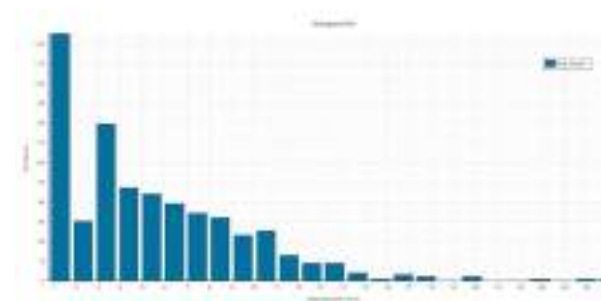
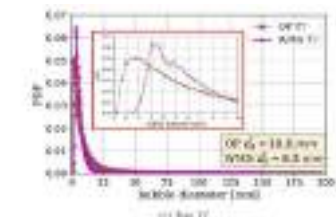
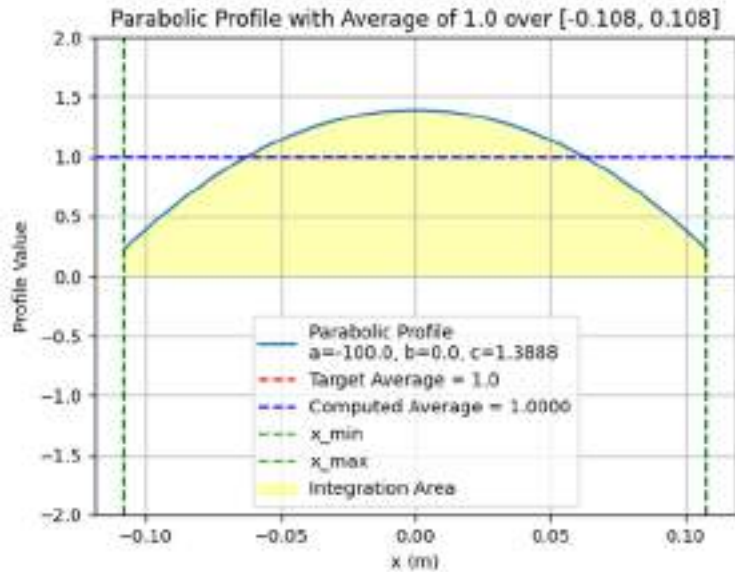
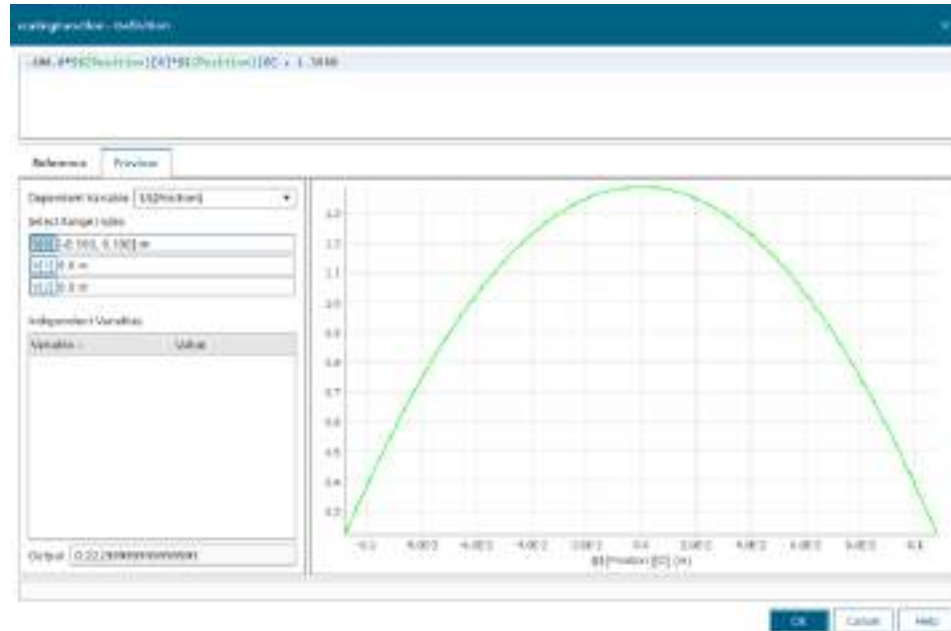


Figure 4.12: Comparison of PDF of bubble size distribution obtained with optical analysis and VOF. The average diameter d_{50} is obtained according to Equation 4.6: $d_{50} = \sum d_i \cdot P(d_i) / \sum P(d_i)$.



Scaling factor introduced and used to transform flat inlet velocity profile into a parabolic profile



EDF UK Numerical Method



StarCCM+ VOF-URANS FSI approach

- CFD code: StarCCM+ v2024.6
- Fluid discretisation: finite volume
- Solid discretisation: finite element
- Two-phase model: Volume of Fluid, VoF
- Turbulence model: $k-\omega$ SST URANS
- Fluid-structure coupling: two-way FSI with Solid Stress model
- Spatial discretisation: Second Order Upwind
- SIMPLEC for pressure velocity coupling

Numerical Modelling Details

Multiphase model

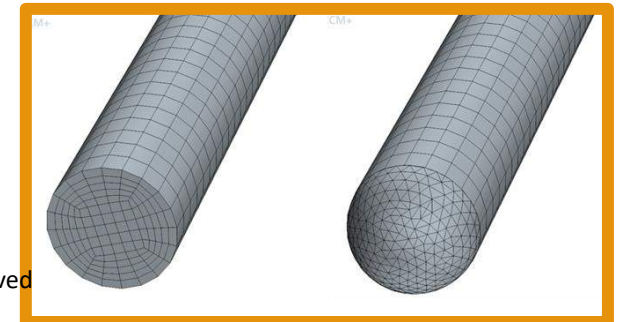
- Volume of Fluid (VoF) method
- HRIC convection scheme selected for the VoF solver.
- HRIC sharpening factor set to 0.9 to reduce smearing at phase boundaries.

Two-phase flow sub-models enabled:

- Surface tension force .
- Interface turbulence damping .
- Interaction area density between the two fluids .
- These settings supported development of bubble structures and gas-liquid segregation.

Structural and FSI Setup

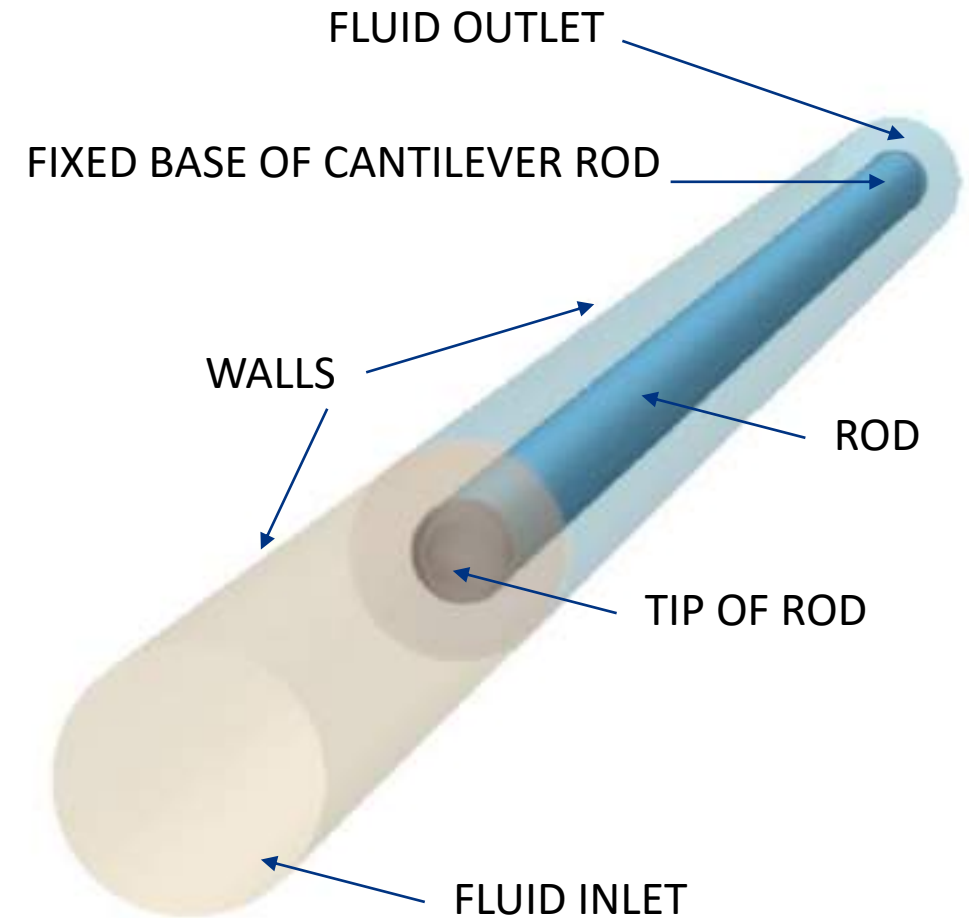
- Rod was idealised as a homogeneous solid.
- Density = 7490 kg/m^3 , Young's modulus = 76.4 GPa
- Rod base fixed using a solid stress constraint.
- Rod surface coupled to the fluid through an FSI interface.
- Rod motion specified as solid displacement.
- FSI activated after 2 seconds to allow the flow field to develop.
- FSI coupling ramped over 0.01 seconds to reduce non-physical start-up instabilities.
- Boundary added mass treatment used to improve coupling stability.



Boundary Conditions



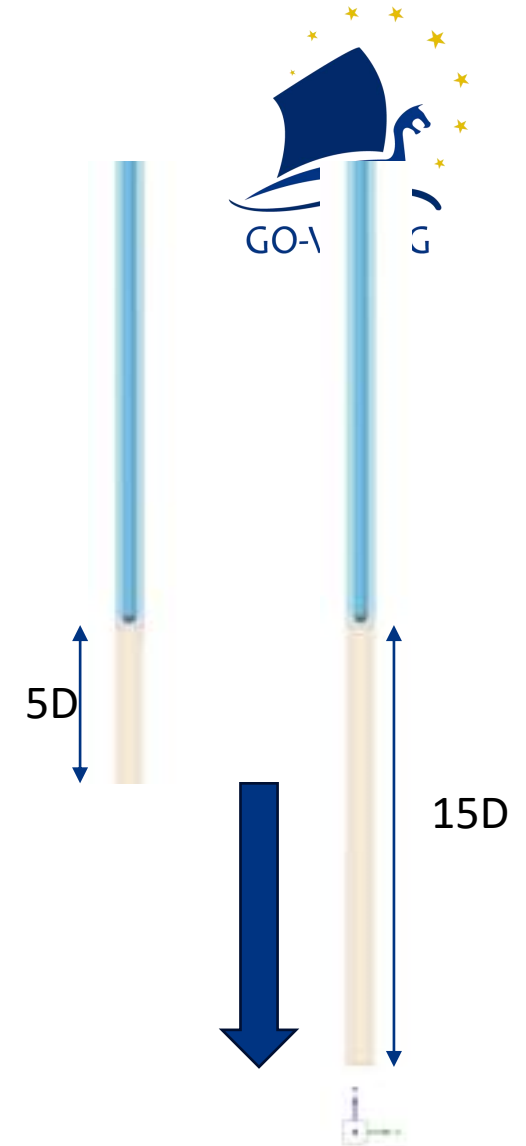
- INLET – uniform – gas 0.35 m/s , water 1.7 m/s
- ROD-FLUID interface – Mapped FSI interface
- ROD BASE – the base of the solid rod is a built-in symmetry and fixed boundary condition
- Annular OUTLET adjacent to rod base



Computational Domain and Mesh

Extended Inlet to allow two-phase flow development

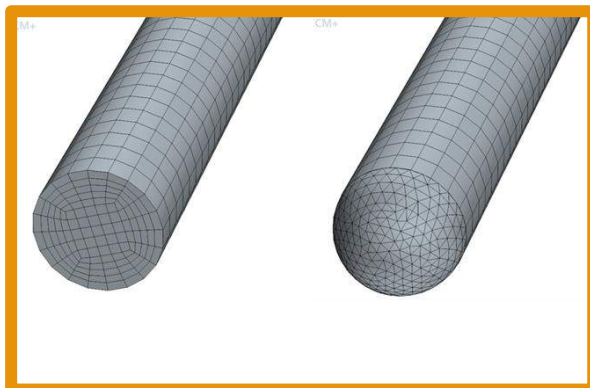
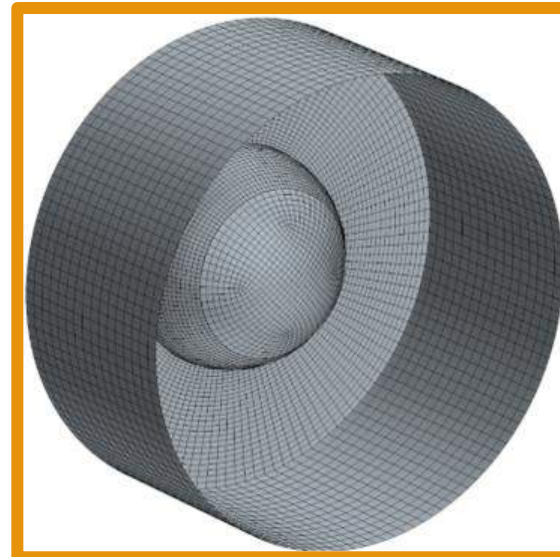
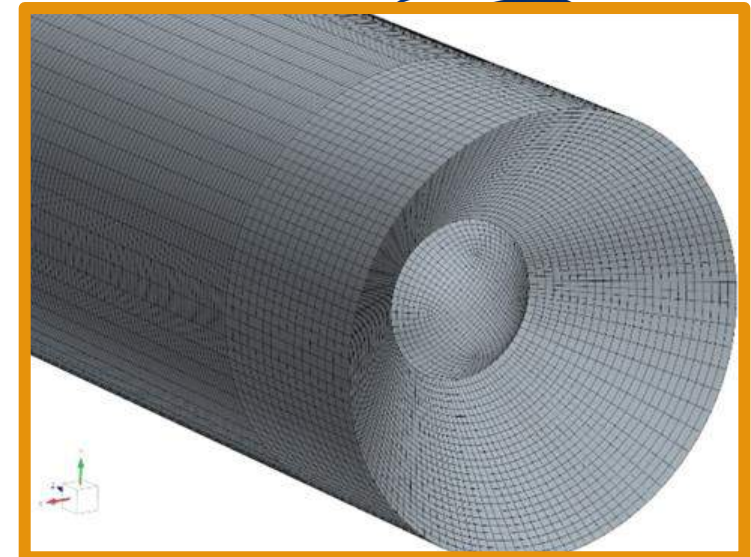
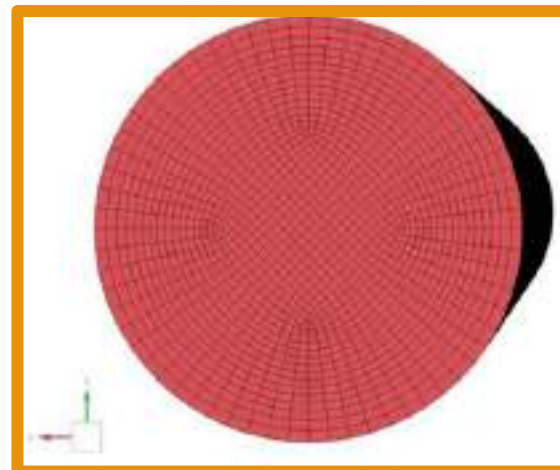
- Simulated configuration: POT3 cantilever case
- Inlet length extended from 5D to 15D
- Reason: allow development of bubbles and two-phase flow development upstream of the rod surface.
- Non-conformal interface used between fluid and solid domains.
- Hexahedral elements used for most of the solid mesh.
- Local refinement applied around the rod tip where flow acceleration occurs.





Mesh Details

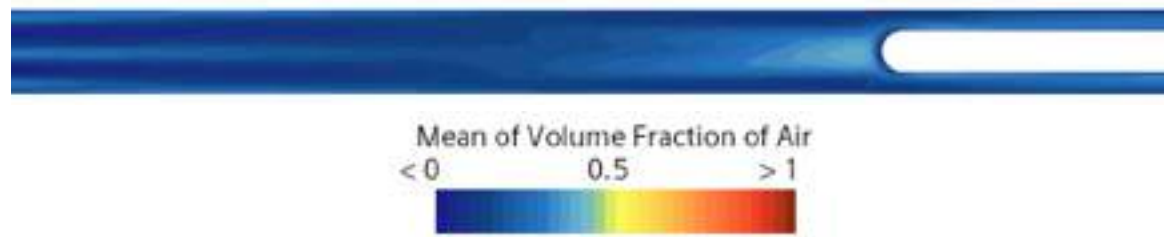
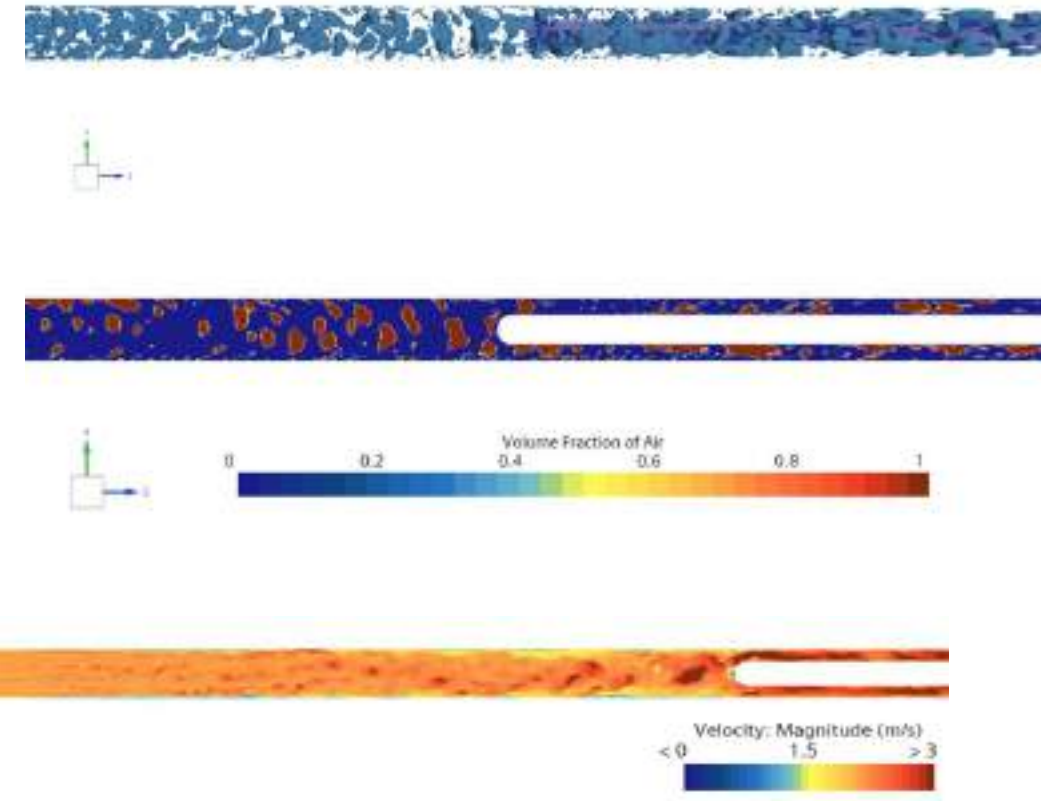
	Value
Fluid Mesh size	2M
Solid Mesh size	62500
Maximum Wall y+	39.4
Surface Average Wall y+	31.6
Near-wall y+ model / Wall Treatment Model	All y+ wall treatment
Maximum Courant Number	0.5
Volume Averaged Courant Number	0.1
Number of inner iterations per time step	6
Time Step Size (s)	0.00015
Total Simulation Time (s)	8



Flow-field Results



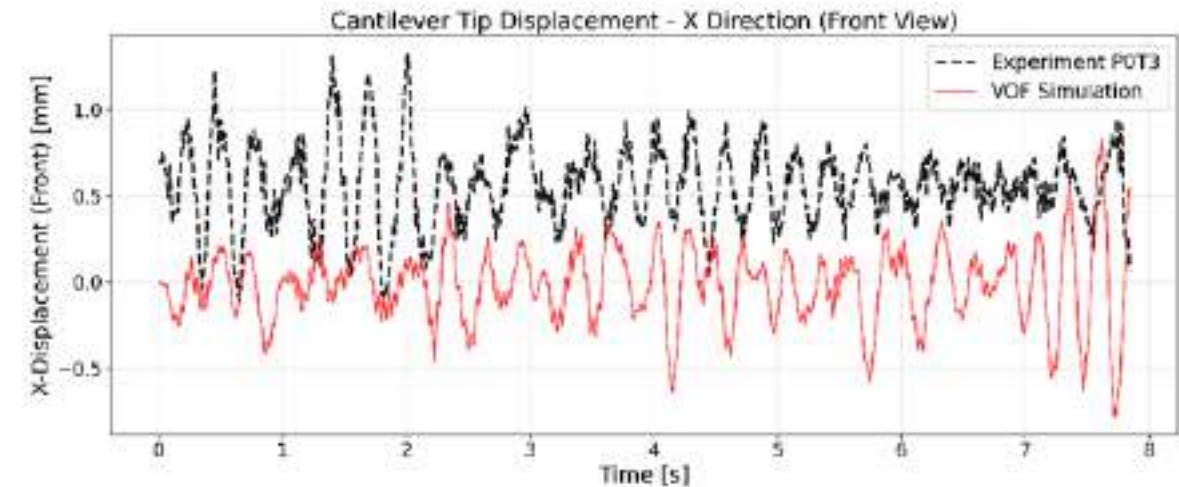
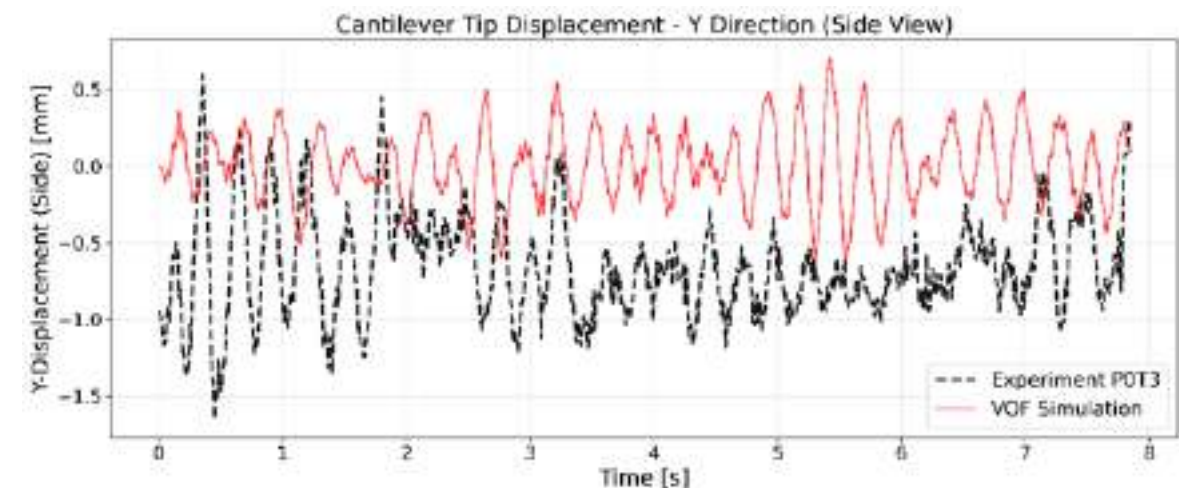
- EDF UK predicted bubble structures and void-fraction distribution near the rod tip.
- Time-averaged velocity and void fraction were extracted at the centre plane.
- The flow-field results provided the forcing environment for the rod vibration response.



Time-domain Displacement Results

- Front and side rod-tip displacement were compared with experimental data.
- The predicted vibration amplitude was visually similar to the experiment.
- RMS displacement values after mean subtraction:

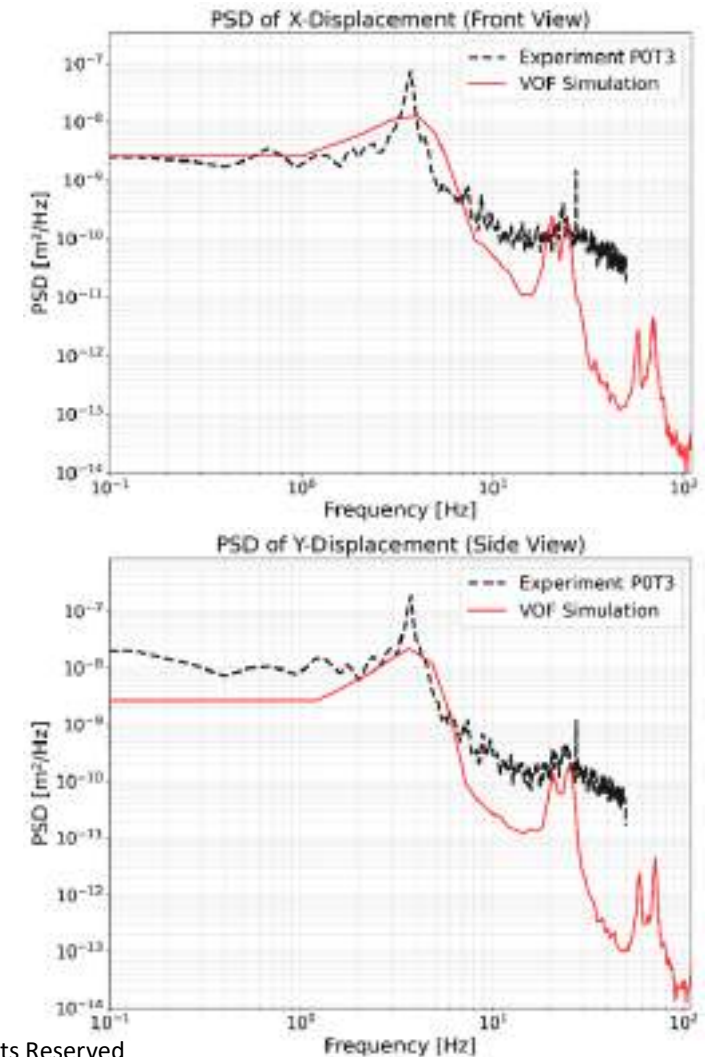
Direction	Simulation RMS	Experiment RMS	Error
Front / X	0.1968 mm	0.2278 mm	-13.6%
Side / Y	0.2724 mm	0.3628 mm	-24.9%



Frequency Domain Results

Comparison of PSD and natural frequency

- PSD results captured the main structural vibration frequencies.
- First mode predicted at approximately 3.64 Hz.
- Second mode observed around 23 Hz.
- PSD peaks were broader and flatter than the experiment.
- Simulation duration was 8 s, compared with 60 s for the experiment.
- Agreement was stronger at lower frequencies; higher-frequency discrepancies increased.



Key Findings

- A StarCCM+ VOF-URANS two-way FSI model was successfully applied to the two-phase cantilever rod benchmark.
- The model captured the first and second vibration modes.
- RMS displacement was predicted within approximately 25% of experiment.
- The dominant vibration frequencies were reproduced.
- The model slightly under-predicted vibration amplitude.
- The inlet two-phase condition remained the largest uncertainty.

$t = 2.000 \text{ s}$



$t = 2.000 \text{ s}$



Conclusions



- EDF UK developed a StarCCM+ VoF-URANS two-way FSI model for the GO-VIKING two-phase cantilever rod benchmark.
- The final approach used a mixture inlet velocity and an extended inlet region to allow two-phase structures to develop.
- The model captured the main vibration frequencies:
 - first mode around 3.64 Hz
 - second mode around 23 Hz
- Predicted RMS displacement was within approximately 25% of the experimental values.
- The main modelling uncertainty was the lack of detailed inlet bubble/void-fraction data.
- Overall, the method provides a credible engineering approach for simulating multiphase flow-induced vibration of slender structures.

Lessons Learned and Future Improvements



- Better representation of the experimental inlet condition is needed for the CFD model to capture:
 - bubble size distribution
 - inlet void-fraction profile
 - phase distribution at inlet
 - time-resolved inlet gas-liquid structure
- Longer simulation time would improve PSD resolution.
- Sensitivity studies could assess:
 - inlet development length
 - bubble generation assumptions
 - mesh refinement around the rod tip
 - turbulence modelling approach (such as use of Reynolds Stress Models)



Thank you!



Ewan Harris



EDF ENERGY R&D (UK)



ewan.harris@edfenergy.com



Website:
www.go-viking.eu



Mail:
contact@go-viking.eu



Twitter:
[@GOVIKING_EU](https://twitter.com/GOVIKING_EU)



LinkedIn:
[GO-VIKING](https://www.linkedin.com/company/GO-VIKING)



**Funded by the
European Union**

Funded by the European Union. Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union or European Atomic Energy Community ('EC-Euratom'). Neither the European Union nor the granting authority can be held responsible for them.



Benchmark – two-phase axial flow around cantilever rod simulation

VTT, EDF UK and EDF R&D

WP4 final meeting

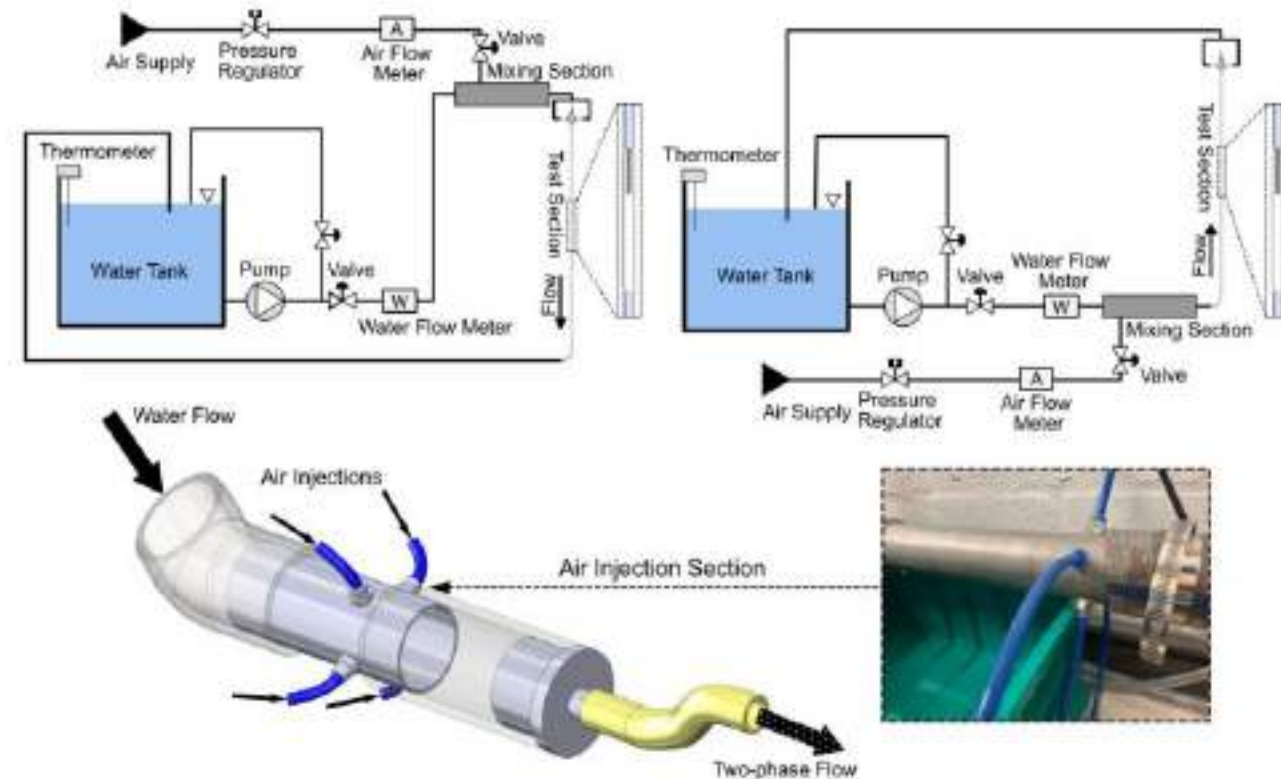


Funded by
the European Union

Experimental setup



- Experiment at the university of Manchester (Li 2026)
- Initially: single phase experiment
- Two-phase axial flow around a cantilever rod (**towards** or away from the rod)
- Different rod tips (focus on round tip)
- Study the **vibration** of rod in **axial two phase flow**



Unveiling the fundamentals of two-phase axial-flow-induced vibrations of cantilever rods

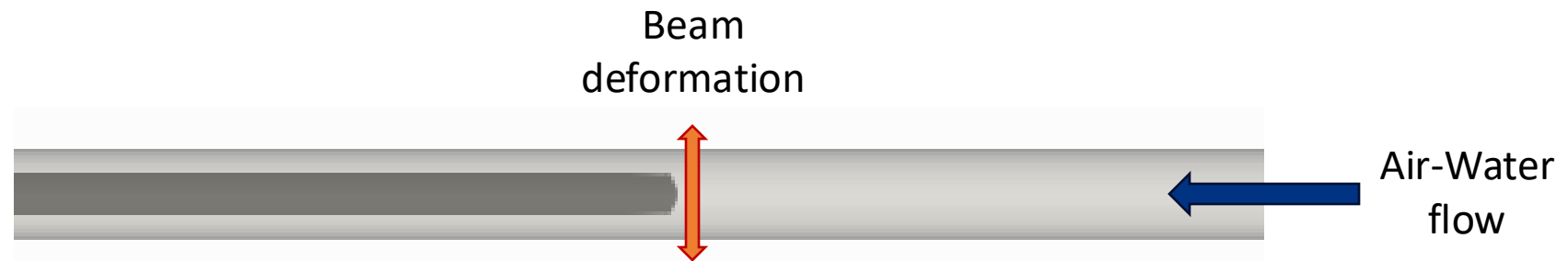
Hao Li, Andrea Concalini, Hector Iacovides, William Benquigui & Mostafa R. A. Nabawy

Scientific Reports 16, Article number: 5102 (2026) | Cite this article

Experimental setup



- Channel length 2m
- Beam length 1m
- Beam diameter 0.01m (round tip radius 5mm)



- Results
 - o Tip displacement (temporal data)

See Li 2026 for more details

Experimental setup

- Inlet condition:
 - Liquid inlet flux 1.7 m/s
 - Air inlet flux 0.35 m/s
- No further details on the inlet conditions (bubbles are millimetric)
- Standard properties for air and water

Fluid	Mass per unit volume (kg/m ³)	Viscosity (kg/(ms))	Surface tension (N/m)
Water	998	0.001	0.072
Air	1.225	1.79e-5	0.072

- Solid properties (homogeneous hypothesis)

L (mm)	D (mm)	E (GPa)	Density	Damping coefficient	Tip radius (mm)
1000	10	193	7.49	0.013	5

Numerical modelling

Geometry

- Modelling of the vertical test section only
- 3D simulations for every participant
- Explicit solid representation for VTT and EDF UK
- IBM representation for EDF R&D

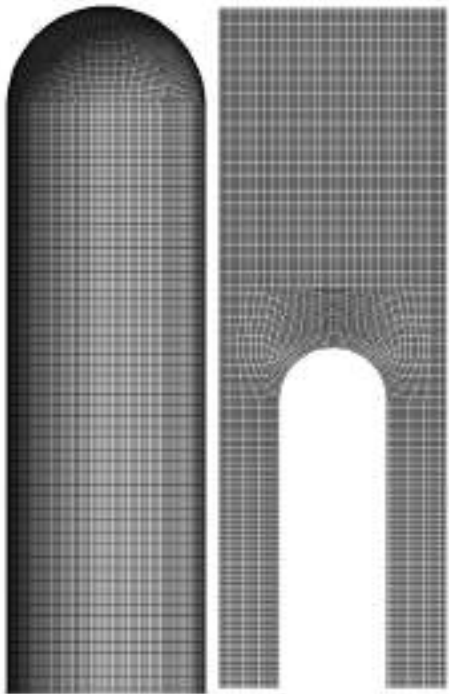
	Tube length	Type of mesh	Number of fluid cells	Number of solid cells	Dr and Dx
EDF R&D	2m	Hexahedral	300k - 3M	10 (quadratic)	0.6mm and 1mm
EDF UK	2m	Hexahedral + Tetrahedral	2.5M - 6.2M	62.5k (linear)	0.28mm and 0.45mm
VTT	1.16m	Hexahedral	2M	212 (linear)	1.5mm and 3mm

Numerical modelling

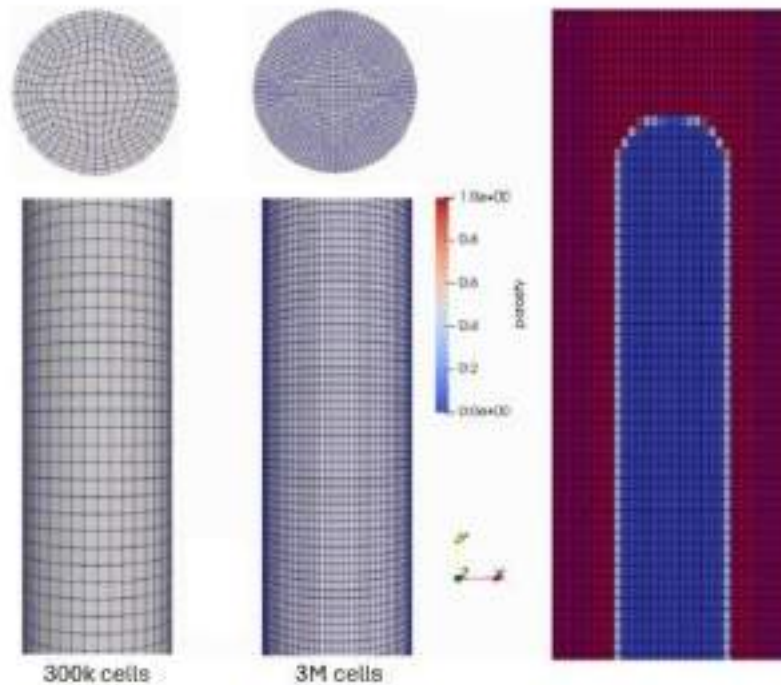
Mesh and boundary conditions



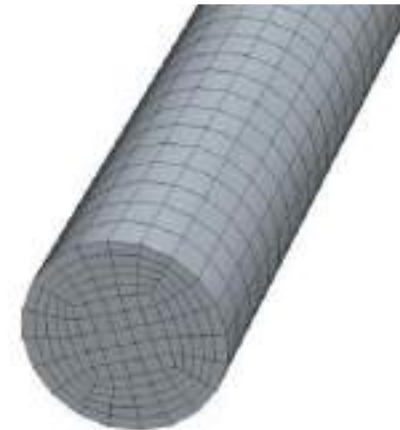
VTT



EDF R&D



EDF UK



Numerical modelling



EDF R&D

- Multi-Eulerian+URANS in `neptune_cfd`
- Rij-EBRSM turbulence model (water phase and air turbulence modulation)
- Generalized large interface model (for bubble and air pocket modelling)
- Strong two-way coupling for FSI with immersed boundary method
- Euler-Bernoulli FEM model for the rod

Numerical modelling



EDF UK

- VOF-URANS in StarCCM+
- kw-SST turbulence model
- High resolution interface capturing convection scheme (sharpness factor = 0.9)
- Activation of multiphase sub-models
- Partitioned FSI approach with remeshing for solid coupling

Numerical modelling



VTT

- VOF+LES in Ansys-Fluent
- Wale turbulence model
- Geometric reconstruction scheme and explicit VOF + continuum surface force model
- FSI with explicit coupling + Euler-Bernoulli FEM model
- SBM for the inlet

Numerical modelling



Summary of modelling approaches

Partner	Software	Two-phase modelling	Turbulence model	Type of inlet	Type of outlet	Space discretization	FSI
EDF R&D	Neptune_cfd	Multi-Eulerian	RIJ EBRSM	Velocity with explicit air inlet	Pressure	Second order linear upwind	Explicit two-way + EB FEM (IBM)
EDF UK	StarCCM+	VOF+HRIC	kw SST	Velocity with explicit air inlet	Pressure	Second order upwind	Explicit two-way + 3D FEM (body fitted)
VTT	Ansys-Fluent	VOF+PLIC	LES WALE	Velocity + SBM	Pressure	Bounded central differencing with factor 0.5	Explicit two-way + EB FEM (body fitted)

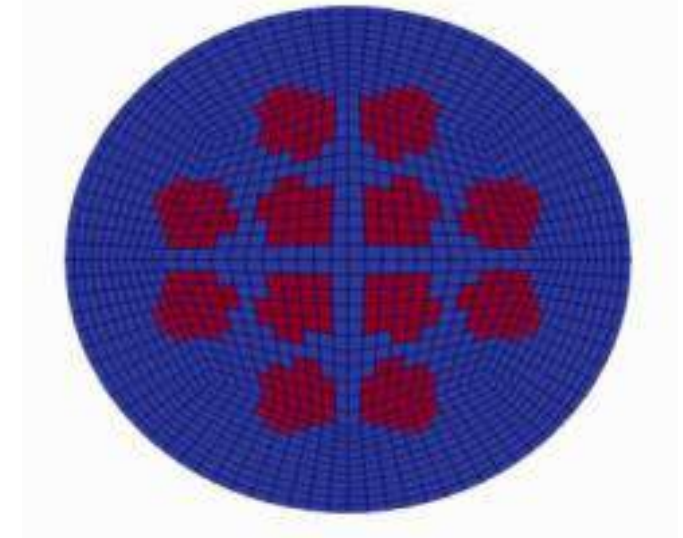
Numerical modelling



Example of explicit air-water inlet definition of the inlet in the EDF R&D simulations

Mesh and boundary conditions

- All participants worked on the same set of boundary conditions with some differences
 - Air-water inlet opposed to the rod tip
 - Outlet at the rod base
 - Walls around the pipe and at the rod surface
- VTT used SBM (synthetic bubble model) to create the inlet
- EDF UK and EDF R&D used an explicit air inlet to create air pockets at the inlet



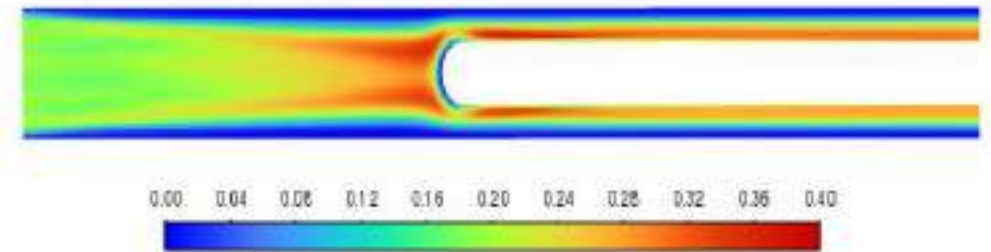
Numerical results: Qualitative analysis of the field variables



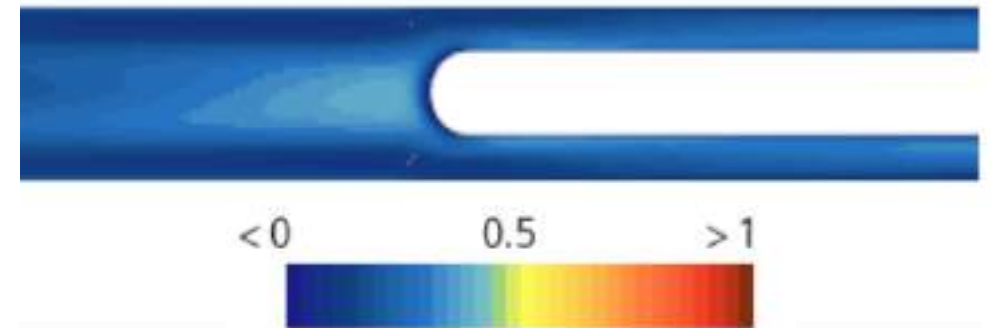
Average air volume fraction

- Bubbles more concentrated towards the centre in the VOF simulations
- Similar global air concentration
- More air close to the beam in the VOF simulations

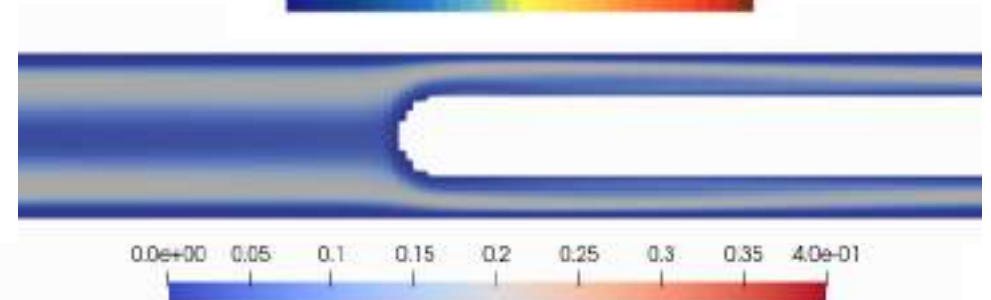
VTT



EDF
UK



EDF
R&D

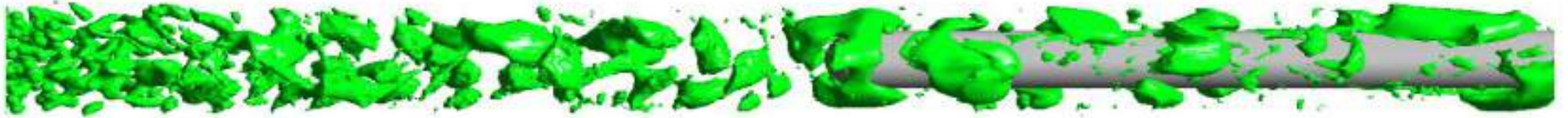


Numerical results: Qualitative analysis of the field variables



Instantaneous air volume fraction

VTT



EDF
UK



EDF
R&D



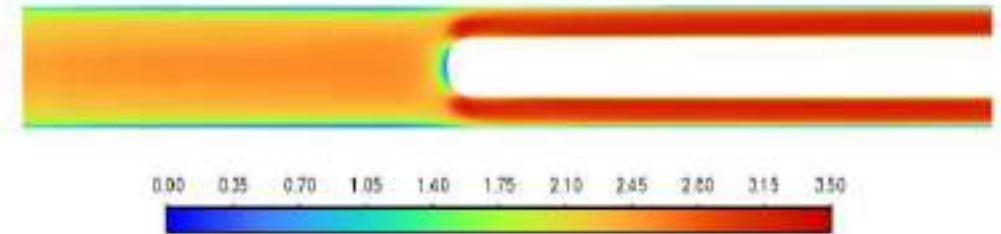
Numerical results: Qualitative analysis of the field variables



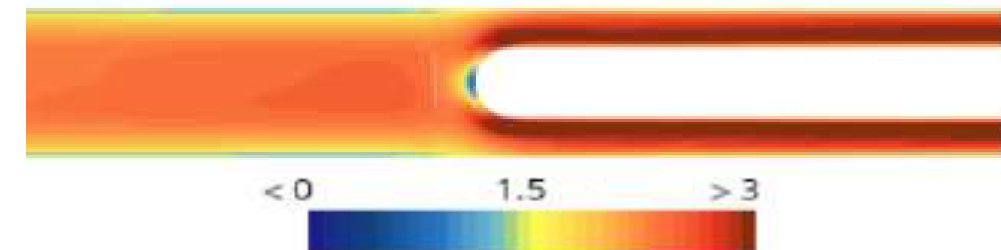
Average mixture velocity

- Similar patterns in all the simulations
- Differences would certainly appear in velocity fluctuation maps

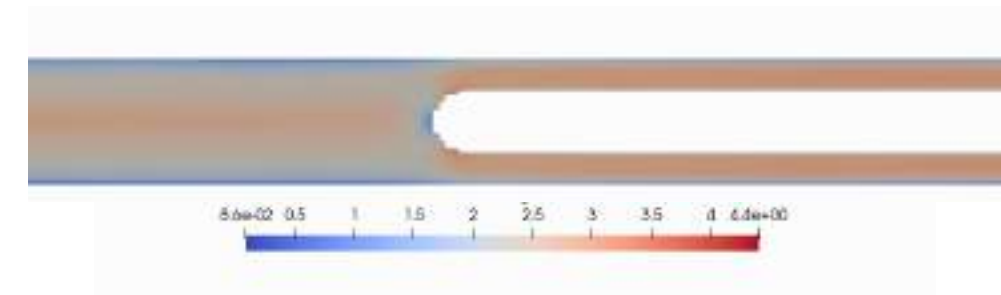
VTT



EDF
UK



EDF
R&D

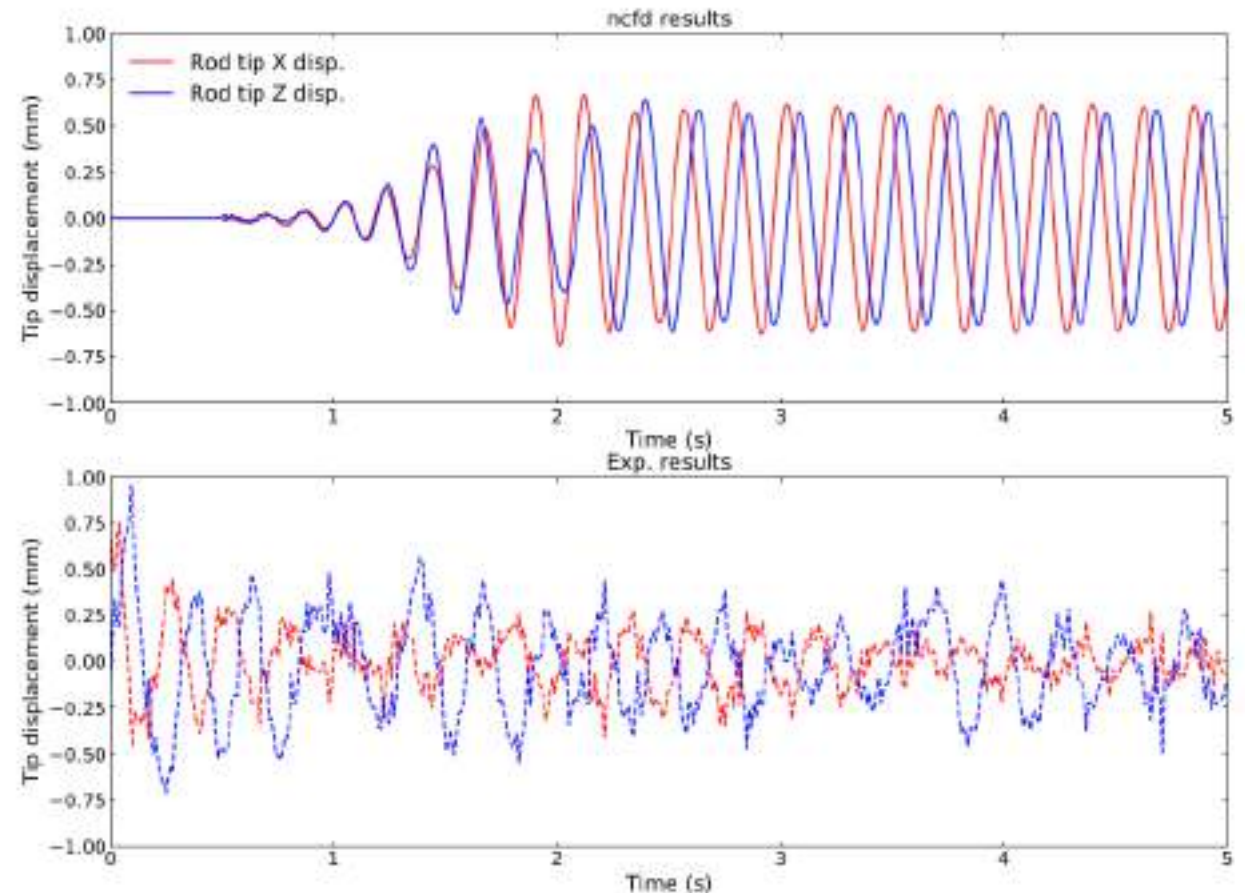


Numerical results



Rod tip displacement EDF R&D

- Very smooth results
- Overestimate slightly the amplitude
- Correct first frequency
- No differences in the two directions

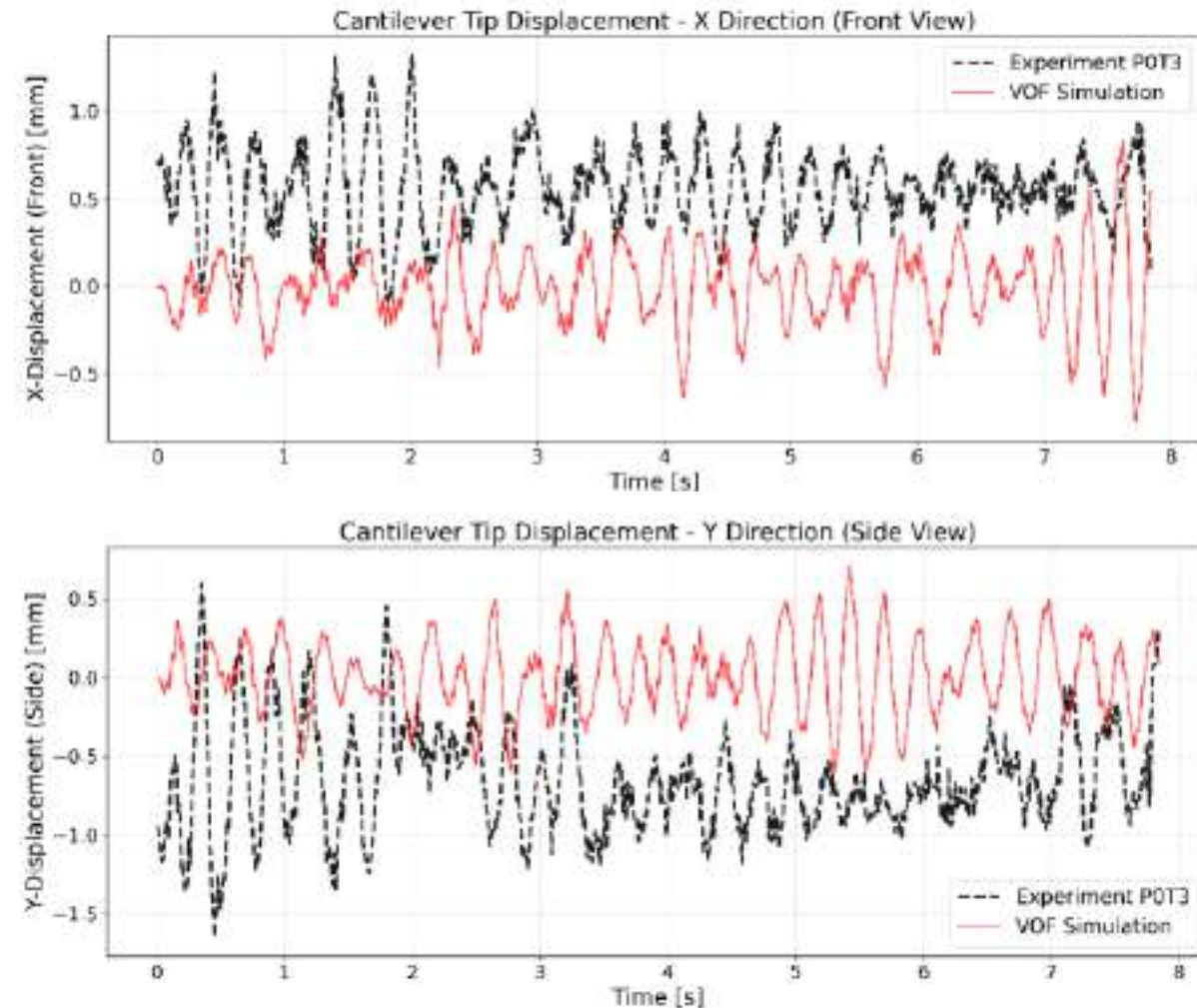


Numerical results



Rod tip displacement EDF UK

- Satisfactory results
- Amplitude, frequency and fluctuation seem consistent in both directions

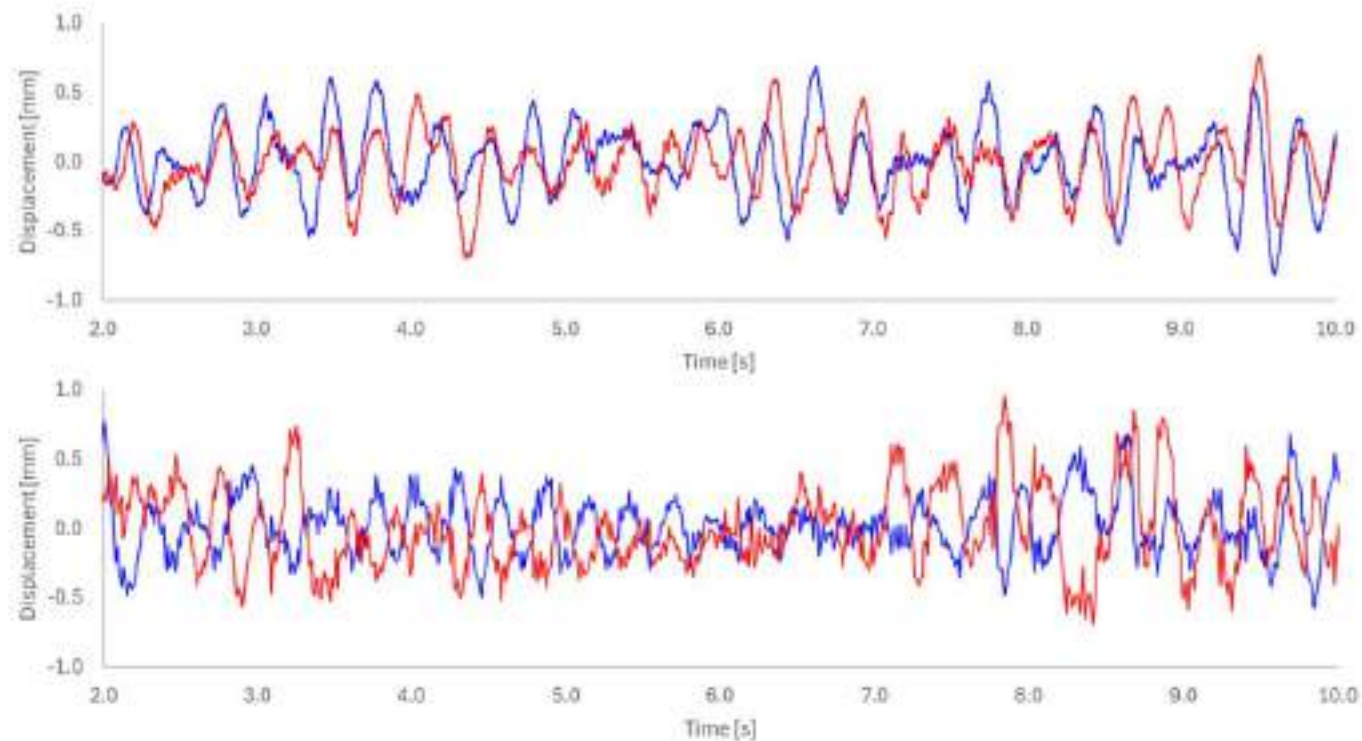


Numerical results



Rod tip displacement VTT

- Satisfactory results
- Amplitude, frequency and fluctuation seem consistent in both directions



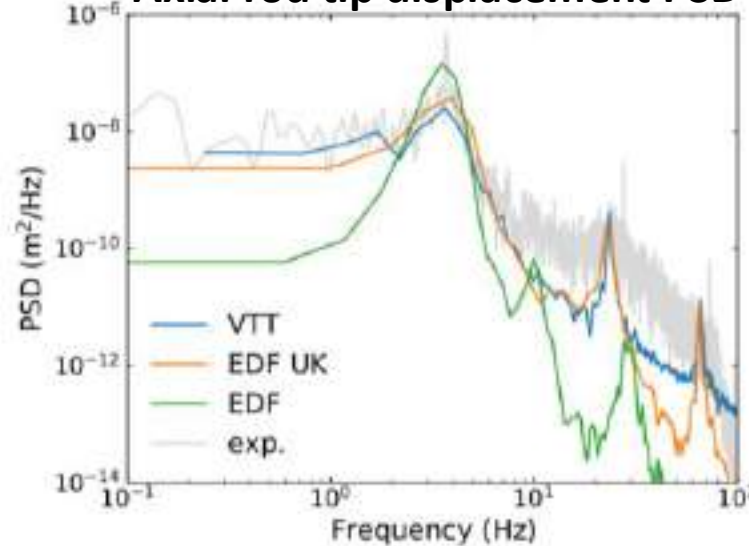
Numerical results



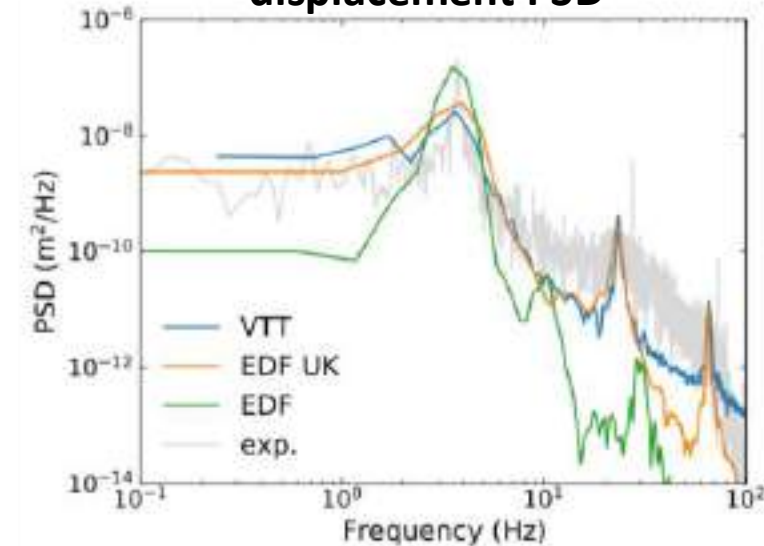
Rod tip displacement power spectral density

- First excited frequency captured by all partners.
- Better results in the VOF simulations for the second frequency
- Not enough fluctuations in the Euler+URANS simulations
- VTT (VOF+LES) is the only one with consistent frequency amplitude

Axial rod tip displacement PSD



Transverse rod tip displacement PSD



Conclusion, perspectives, and best practices



Comparison

- Satisfactory results for the three participants
- GLIM allows to simulate the effects of bubbles compared to VOF simulations
 - VOF > better results at this gas superficial velocity
 - Not necessary to capture the correct beam deformation
- Explicit solid representation clearly provide better results (at least at low Reynolds)
- In larger gas inlet fluxes, the results are better with the implicit solid Euler-Euler modelling approach compared to those obtained at lower gas inlet flux

Conclusion, perspectives, and best practices



Best practices

- Boundary conditions are key to correct simulations
 - It would be beneficial to get more detailed data in the future
- Meshes are key to consistent simulations

Partners



Associated Partners



*EDF R&D UK Center



Thank you!



Federico BARAGLIA



EDF R&D



federico.baraglia@edf.fr



Website:
www.go-viking.eu



Mail:
contact@go-viking.eu



Twitter:
[@GOVIKING_EU](https://twitter.com/GOVIKING_EU)



LinkedIn:
[GO-VIKING](https://www.linkedin.com/company/GO-VIKING)



**Funded by the
European Union**

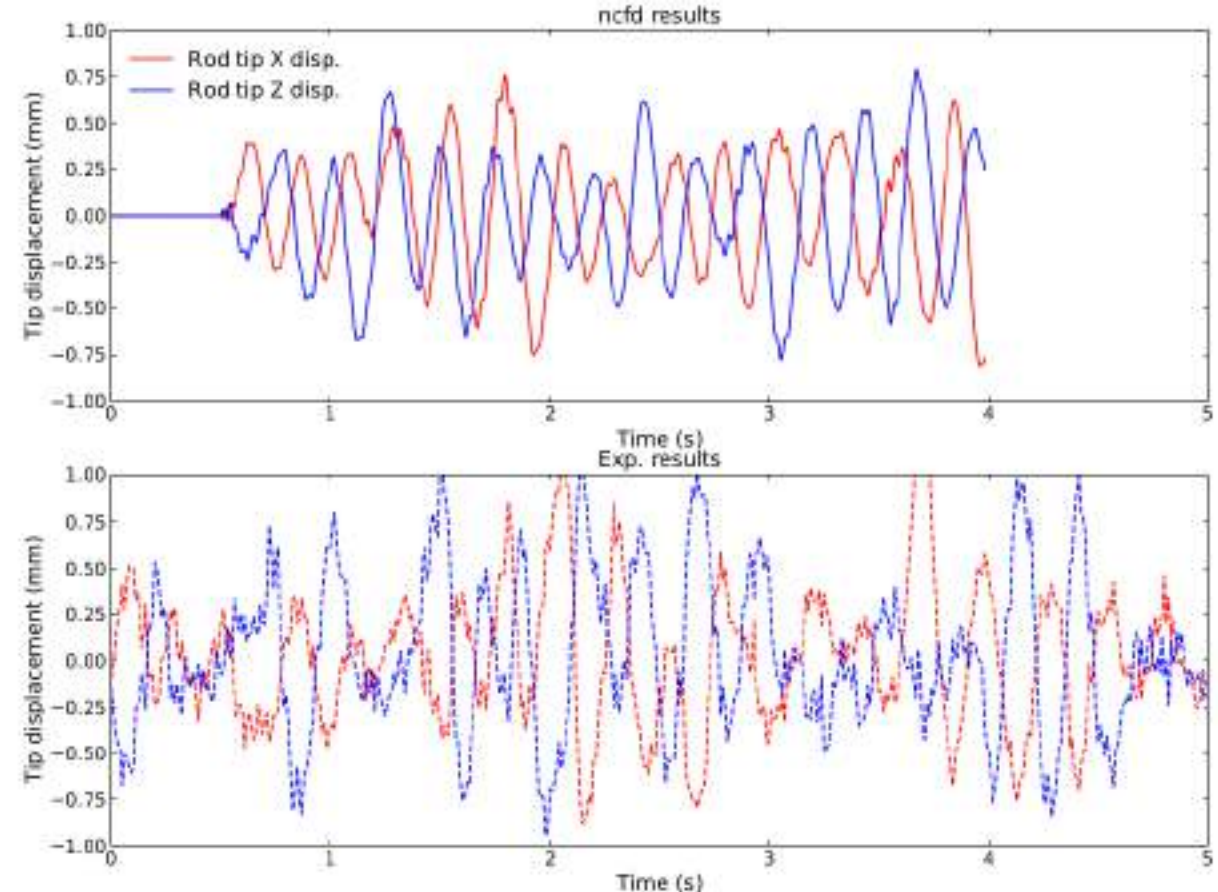
Funded by the European Union. Views and opinions expressed are however those of the author(s) only and do not necessarily reflect those of the European Union or European Atomic Energy Community ('EC-Euratom'). Neither the European Union nor the granting authority can be held responsible for them.

Numerical results



Rod tip displacement RMS = $f(Ug)$ EDF R&D

- Better results and better fluctuations for greater inlet gas fluxes

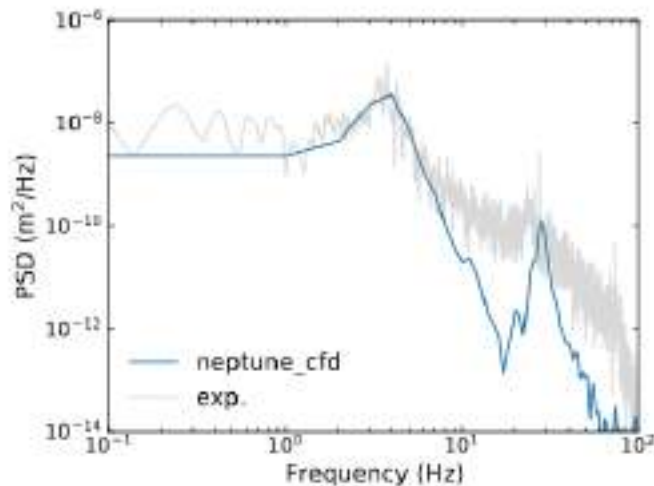
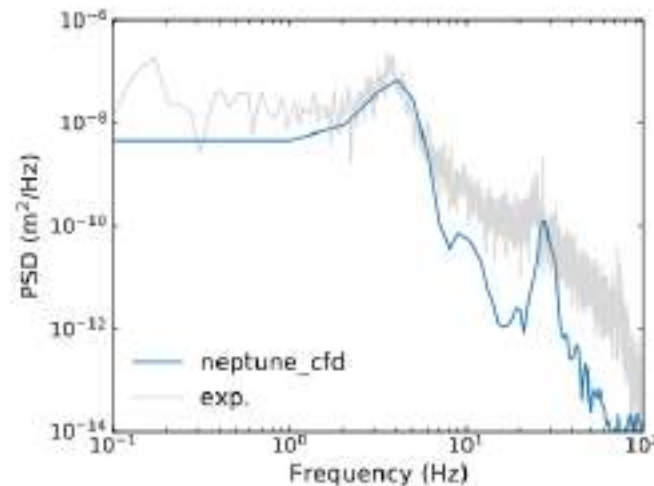
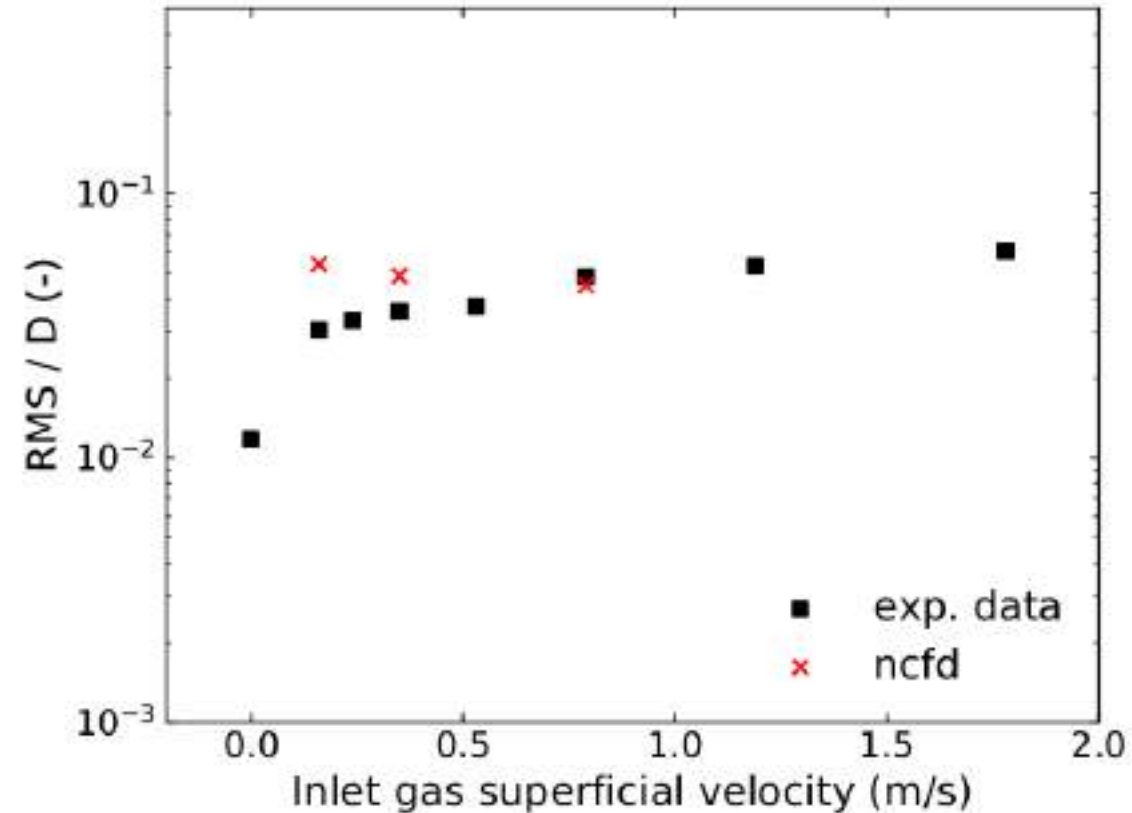


Numerical results



Rod tip displacement RMS = f(Ug) EDF R&D

- Better results and better fluctuations for greater inlet gas fluxes





WP4 – Flow-Induced Vibrations in Multiphase Flow

The TITAN experiment: vibrations in a tube bundle exposed to a two-phase cross-flow

Domenico Panunzio (CEA)



Funded by
the European Union

GO-VIKING Final workshop (GRS, Garching, Germany), 20-22/05/2026

Summary



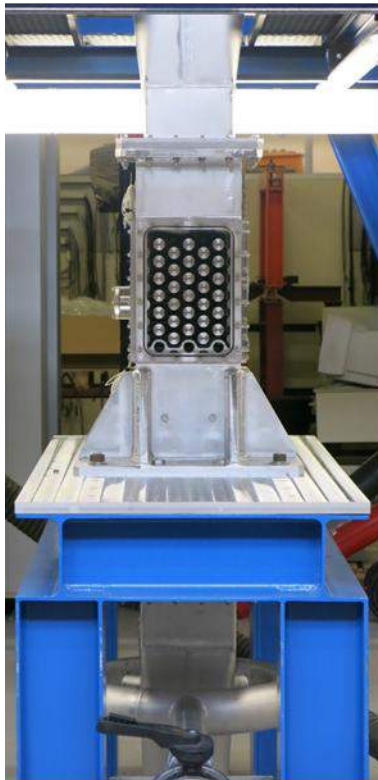
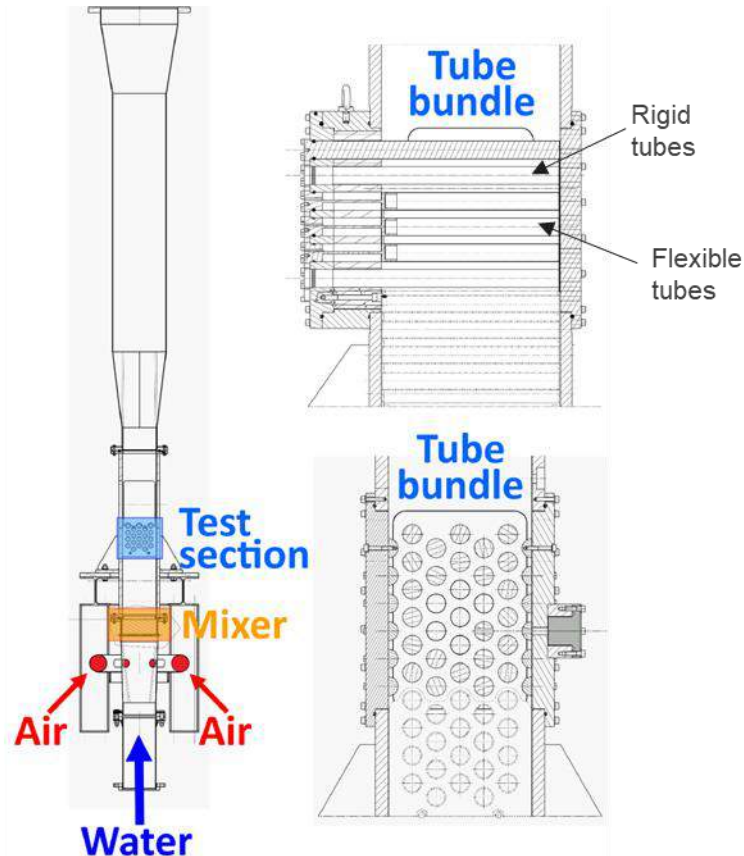
- **Introduction**
- **Experimental setup**
- **Results**
 1. Displacement measurements
 2. Force and void fraction measurements
- **Conclusions**

Introduction



- In the nuclear field, a frequently experienced phenomena is the flow-induced vibrations (FIV) in tube bundles.
- The WP4 of GO-VIKING aims to improve understanding of the FIV phenomena in tube bundles exposed to two-phase cross-flows, mainly relevant to SG applications.
- CEA produces experimental results to support the numerical activities within GO-VIKING.
- A rotated triangular tube bundle, named TITAN, was tested in single-phase (water and air) and in two-phase (air-water) cross-flow.

Experimental setup

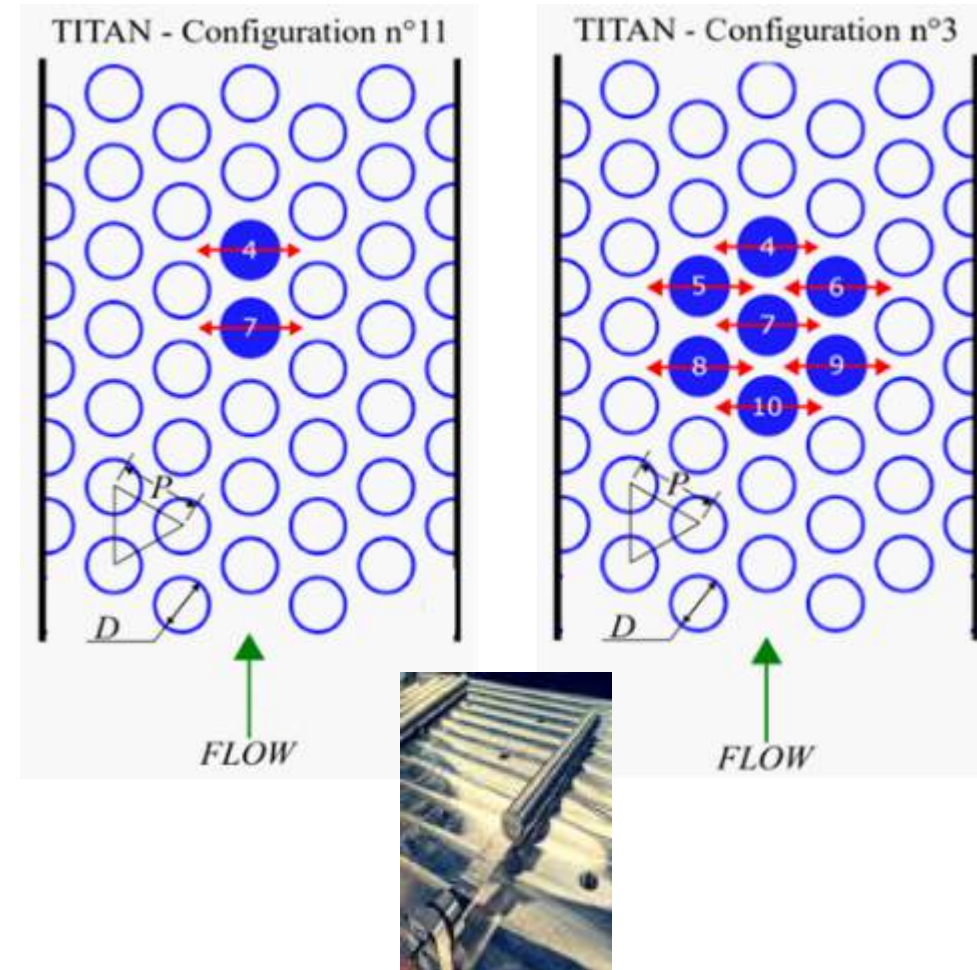


- Air-water loop supplied with a centrifugal pump ($Q_{\max} = 1080 \text{ m}^3/\text{h}$) and a compressor ($Q_{\max} = 3000 \text{ Nm}^3/\text{h}$)
- Rotated triangular (60°) tube bundle: $D = 30 \text{ mm}$, $L = 300 \text{ mm}$, $P/D = 1.44$,
- 30 tubes of which 15 tubes are potentially flexible + 6 half-tubes at each wall + 20 rigid tubes upstream

Results

Displacement measurements

- **Two flexible configurations tested:**
 - Configuration n°11: 2 flexible tubes at the bundle center
 - Configuration n°3: pattern of 7 flexible tubes at the bundle center
- **Experimental setup:**
 - Tubes mounted on a single flexible blade ($100 \times 25 \times 4$ mm)
 - Strain gauge installed at the blade root
- **Dynamic properties in air :**
 - Config. n°11: average natural frequency: 14.18 Hz; average damping ratio: 0.045%
 - Config. n°3: average natural frequency: 31.96 Hz; average damping ratio: 0.047%
- **Test conditions :**
 - Gradually increasing mass flow rate
 - Constant void fraction during each test
- **Data acquisition :**
 - Duration: 180 s; sampling rate: 2000 Hz

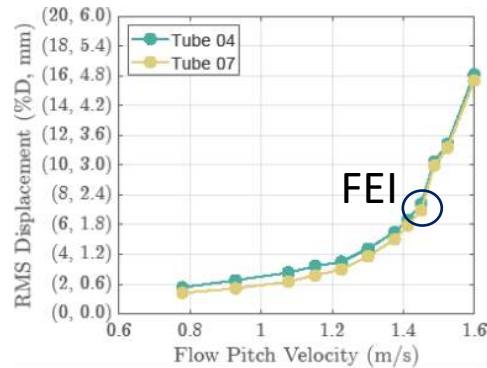
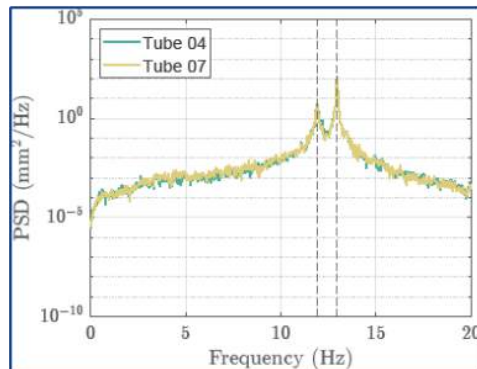


Results

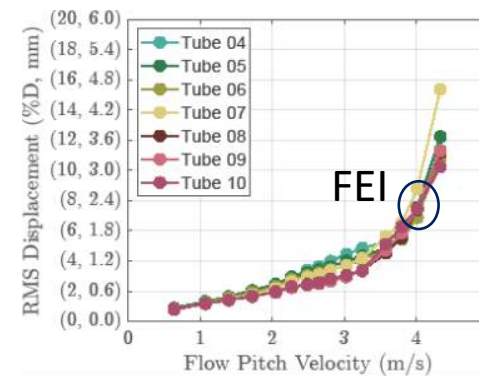
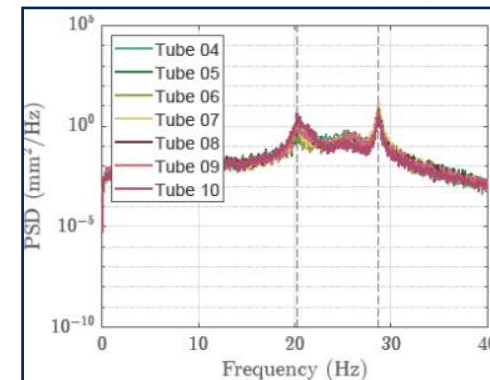
Displacement measurements



$$\alpha_H = 35\%, f_n = 14 \text{ Hz}$$



$$\alpha_H = 55\%, f_n = 32 \text{ Hz}$$

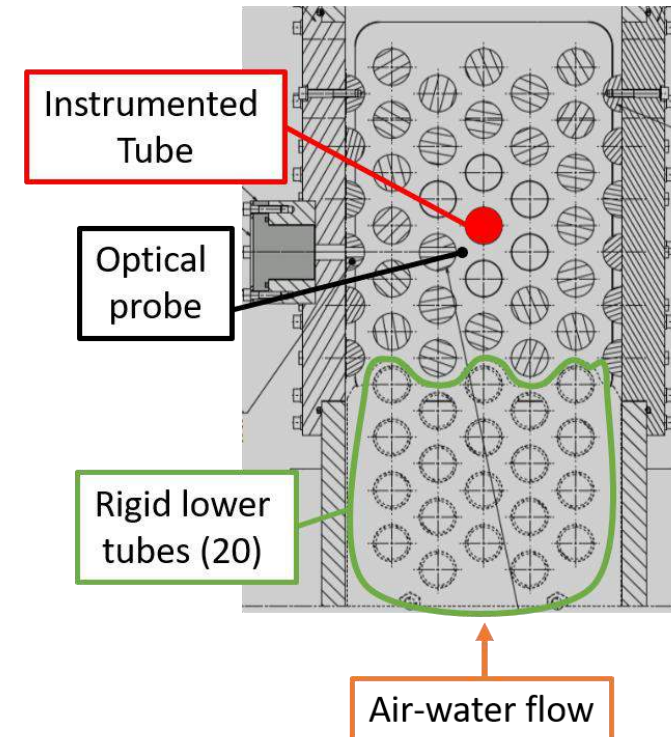


Results

Force and void fraction measurements

- **Experimental setup:**
 - Central tubes cantilevered
 - Force sensors installed at both ends
 - Optical probe installed inside the tube bundle
- **Test conditions :**
 - Constant void fraction during each test
- **Data acquisition :**
 - Duration: 600 s ; ; sampling rate: 512 Hz

Test	Q_w (l/s)	Q_a (Nm ³ /h)	J_w (m/s)	J_a (m/s)	Corrected void fraction (%)	Time length of force signals (s)	Sampling frequency (Hz)
1	10.1	24.3	0.49	0.33	40.2	600	512
2	10.1	54.6	0.49	0.74	60.2	600	512
3	10.1	145.7	0.49	1.97	80.1	600	512
4	9.9	321.9	0.48	4.35	90.1	600	512
5	8.1	262.8	0.39	3.55	90.1	600	512
6	6.6	214.5	0.32	2.90	90.1	600	512

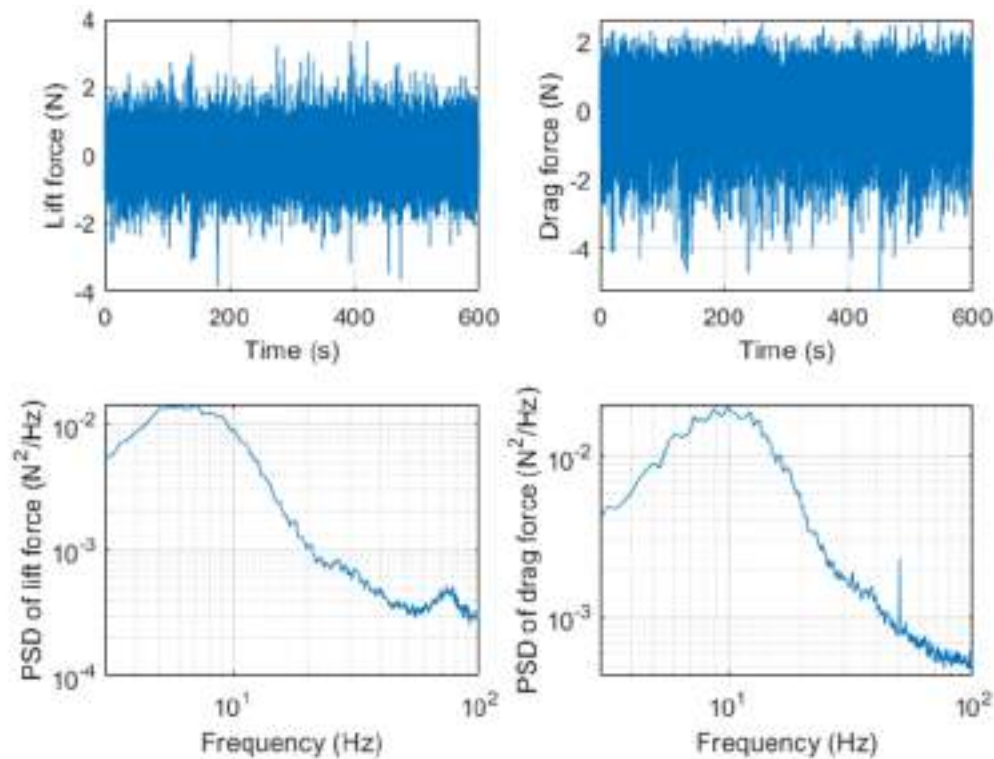


Results

Force and void fraction measurements

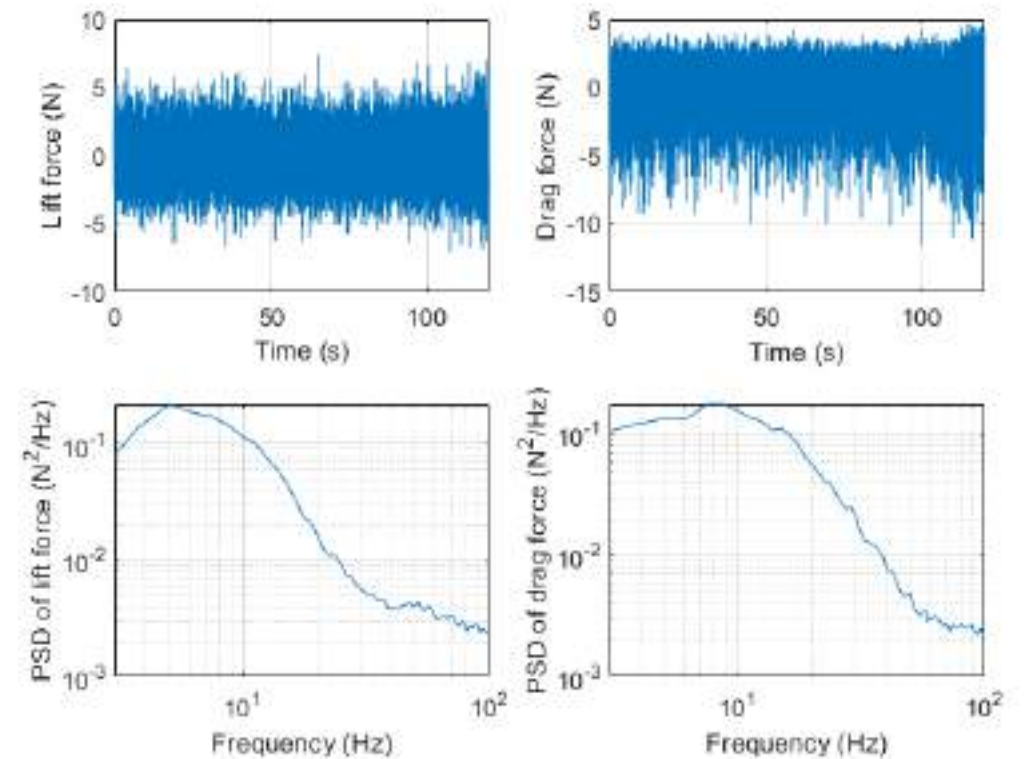


TEST 1, $\alpha_H = 40\%$



Void fraction	Interface frequency	Number of bubbles	Time of flight	Interfacial coefficient	Interface velocity	Interfacial area	Sauter diameter
0,254216	69,019753	41796	1,430204	0,844652	0,978881	285,362	5,382646

TEST 3, $\alpha_H = 80\%$



Void fraction	Interface frequency	Number of bubbles	Time of flight	Interfacial coefficient	Interface velocity	Interfacial area	Sauter diameter
0,584435	203,25291	123083	0,845714	0,855329	1,655405	495,94959	7,22164

Conclusions



- **Experimental Setup – TITAN**
 - Triangular tube bundle with pitch-to-diameter ratio $P/D=1,44$ in air–water cross-flow
 - Displacement test configuration (2 or 7 flexible tubes)
 - Rigid test configuration (centrally instrumented tube)
- **Force and Void Fraction Measurements**
 - Force spectrum exhibiting a distinct bump
 - Measured void fraction deviates from the homogeneous flow model
- **Displacement Measurements**
 - RMS displacement increases with flow velocity up to the onset of fluid-elastic instability (FEI)
 - Appearance of dominant peaks in the power spectral density (PSD)



GO-VIKING

Thank you!



Domenico Panunzio



CEA



domenico.panunzio@cea.fr

www.go-viking.eu



Funded by the
European Union



GO-VIKING

GO-VIKING FINAL WORKSHOP

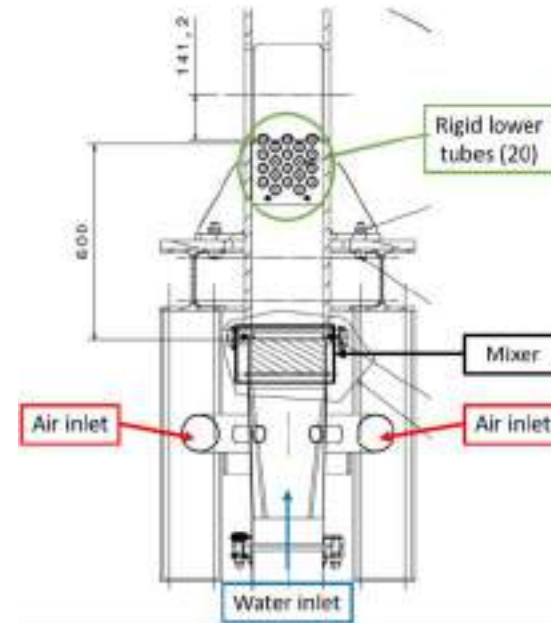
GRS Munich – 20-22 May 2026

THE *TITAN* BENCHMARK: PRESENTATION OF THE NUMERICAL RESULTS

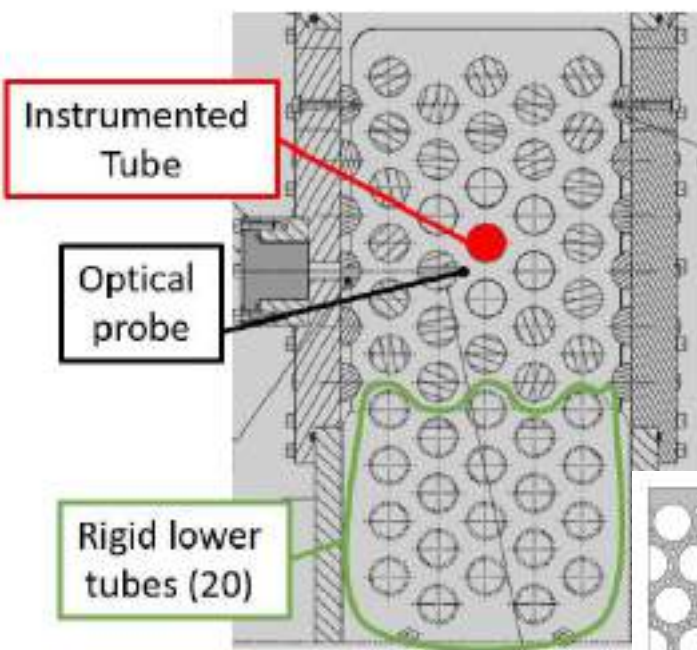
D. Vivaldi (ASNR Cadarache)

TITAN configuration

- Air-water
- Triangular array
- $D=30\text{mm}$, $P/D=1.44$



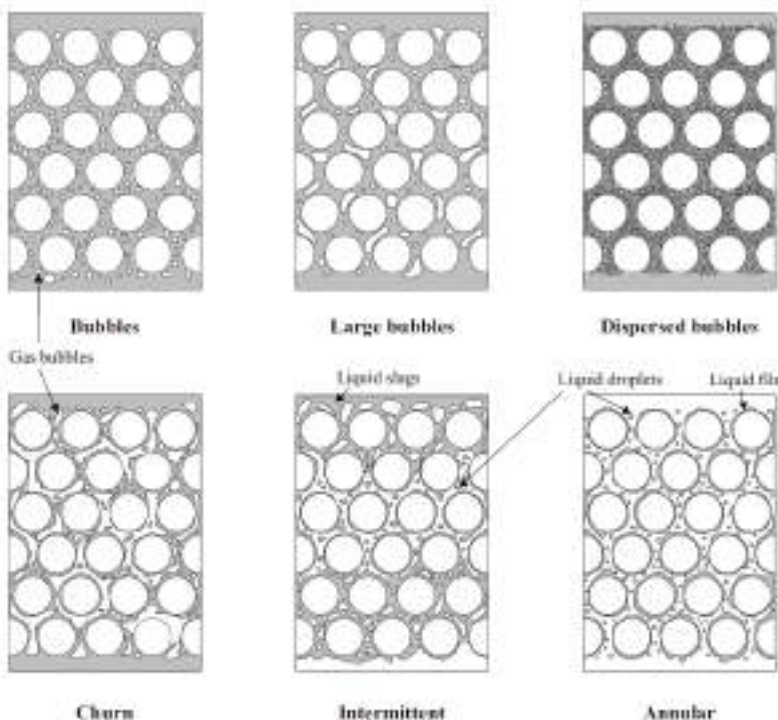
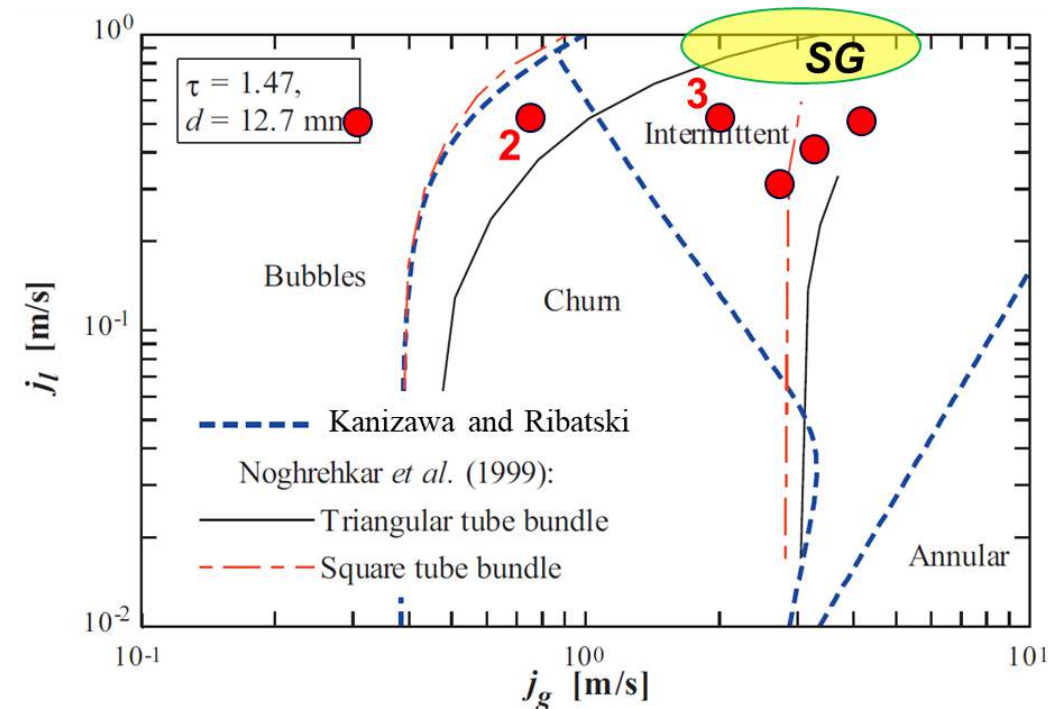
TITAN configuration



$\alpha_H = 0.6$

$\alpha_H = 0.8$

Test	Water flow rate (l/s)	Air flow rate (Nm ³ /h)	Superficial liquid velocity (m/s)	Superficial air velocity (m/s)	Homogeneous void fraction	Time length of force signals (s)	Sampling frequency (Hz)
1	10.1	24.3	0.5	0.25	33.3 %	600	512
2	10.1	54.6	0.49	0.74	60.2 %	600	512
3	10.1	145.7	0.49	1.97	80.1 %	600	512
4	9.9	321.9	0.48	4.35	90.1 %	600	512
5	8.1	262.8	0.39	3.55	90.1 %	600	512
6	6.6	214.5	0.32	2.90	90.1 %	600	512



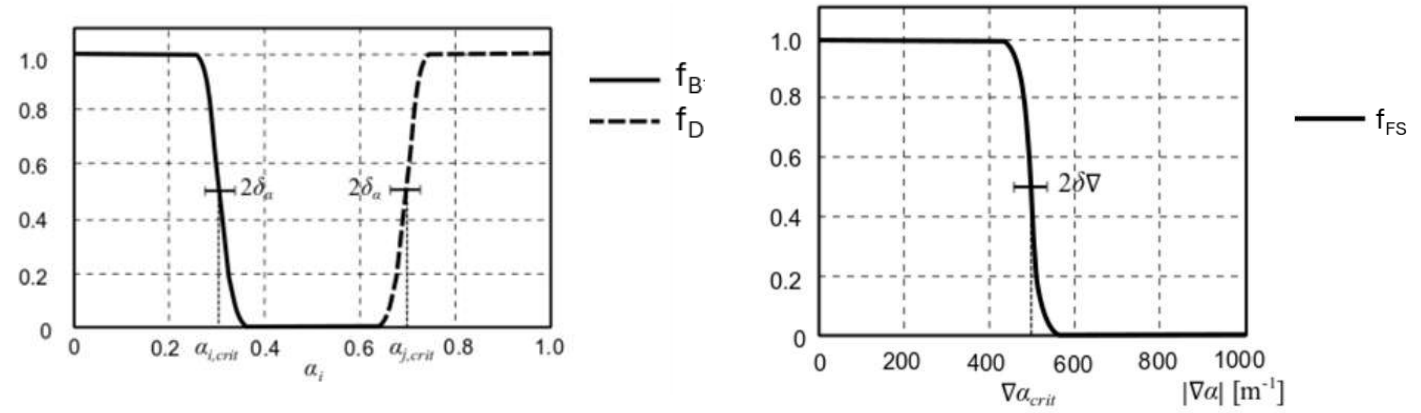
CFD approach

Algebraic Interfacial Area Density (AIAD)

- Proposed by Hohne and Valle (2012)
- Improved blending function by Hohne (2020)
- Implemented in Ansys Fluent

Physical approach

- 2-fluid model
- Morphology-adaptive: blending functions defining bubbly, free surface, droplet flow.



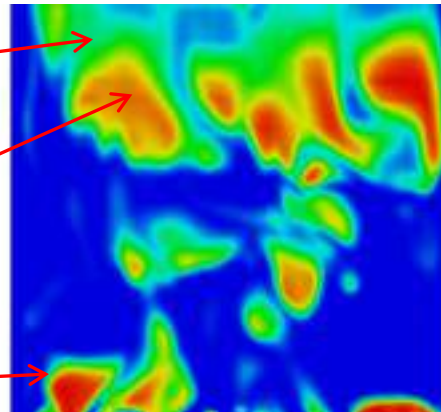
$$\psi_{fs} = \varphi_{fs}(f_B - f_D)$$

$$\psi_{fs} = \begin{cases} 1 & \text{bubbles} \\ 0 & \text{interface} \\ -1 & \text{droplets} \end{cases}$$

$$f_B = \frac{1}{2} \left[1 + \cos \left(\pi \frac{\widetilde{\alpha}^G - (\alpha_{b,crit} - \delta_\alpha)}{2\delta_\alpha} \right) \right]^{-1}$$

$$f_D = \frac{1}{2} \left[1 + \cos \left(\pi \frac{\widetilde{\alpha}^L - (\alpha_{d,crit} - \delta_\alpha)}{2\delta_\alpha} \right) \right]^{-1}$$

$$\varphi_{fs} = \frac{1}{2} \left[1 + \cos \left(\pi \frac{\widetilde{\nabla \alpha} - (\nabla \alpha_{crit} - \delta_\nabla)}{2\delta_{\alpha \nabla}} \right) \right]$$



$$F_D = \frac{1}{2} C_D a_i \rho |U_{gl}| U_{gl}$$

$$a_i = (1 - |\psi_{fs}|) a_{i,fs} + a_{sign} \psi_{fs} a_{i,b} + (1 - a_{sign}) |\psi_{fs}| a_{i,d}$$

$$C_D = (1 - |\psi_{fs}|) C_{D,fs} + a_{sign} \psi_{fs} C_{D,b} + (1 - a_{sign}) |\psi_{fs}| C_{D,d}$$

$$a_{sign} = \begin{cases} 1, & \text{if sign}(\psi_{fs}) = 1 \\ 0, & \text{else} \end{cases}$$

CFD approach

Algebraic Interfacial Area Density (AIAD)

- Proposed by Hohne and Valle (2012)
- Improved blending function by Hohne (2020)
- Implemented in Ansys Fluent

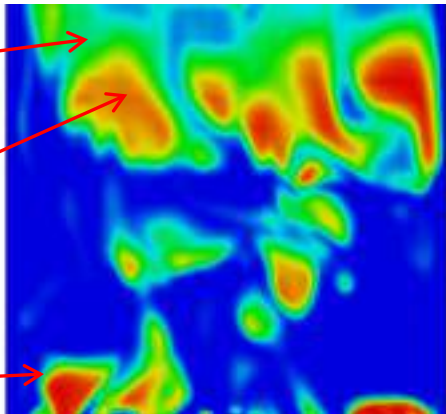
Physical approach

- 2-fluid model
- Morphology-adaptive: blending functions defining bubbly, free surface, droplet flow.

$$f_B = \frac{1}{2} \left[1 + \cos \left(\pi \frac{\alpha_G - (\alpha_{b,crit} - \delta_\alpha)}{2\delta_\alpha} \right) \right]^{-1}$$

$$f_D = \frac{1}{2} \left[1 + \cos \left(\pi \frac{\alpha_L - (\alpha_{d,crit} - \delta_\alpha)}{2\delta_\alpha} \right) \right]^{-1}$$

$$\varphi_{fs} = \frac{1}{2} \left[1 + \cos \left(\pi \frac{\widetilde{\nabla} \alpha - (\nabla \alpha_{crit} - \delta_{\nabla})}{2\delta_{\alpha \nabla}} \right) \right]$$



$$a_i = (1 - |\psi_{fs}|) a_{i,fs} + a_{sign} \psi_{fs} a_{i,b} + (1 - a_{sign}) |\psi_{fs}| a_{i,d}$$

$$C_D = (1 - |\psi_{fs}|) C_{D,fs} + a_{sign} \psi_{fs} C_{D,b} + (1 - a_{sign}) |\psi_{fs}| C_{D,d}$$

	C_D	a_i
Bubbly flow	« standard » (Ishii)	$\frac{6\alpha_G}{d_B}$
Droplet flow	« standard »	$\frac{6\alpha_L}{d_D}$
Free surface	shear stress	$ \nabla \alpha_G $

Free surface shear stress:

$$\tau_{w,x} = [n_x \mu (2 \frac{\partial u}{\partial x})] + [n_y \mu (\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})] + [n_z \mu (\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x})]$$

$$\tau_{w,y} = [n_x \mu (\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x})] + [n_y \mu (2 \frac{\partial v}{\partial y})] + [n_z \mu (\frac{\partial u}{\partial z} + \frac{\partial w}{\partial u})]$$

$$\tau_{w,z} = [n_x \mu (\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z})] + [n_y \mu (\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z})] + [n_z \mu (2 \frac{\partial w}{\partial z})]$$

$$\tau_w = \sqrt{\tau_{w,x}^2 + \tau_{w,y}^2 + \tau_{w,z}^2}$$

$$F_D = \frac{1}{2} C_{D,fs} a_{i,fs} \rho U_{gl}^2 = \tau_w a_{i,fs}$$

$$C_{D,fs} = \frac{2(\alpha_L \tau_{w,L} + \alpha_G \tau_{w,G})}{\rho_m U_{gl}^2}$$

CFD approach

Numerical setup:

➤ Fluent v2024 R2

Solver crashes if
switched-on!

Solver crashes with
2nd order

Details of implementation
not shared by Ansys

AIAD set-up	
Interfacial closure laws:	
• Bubbly flow <u>diameter</u>	
if constant	4·10 ⁻³ m
with population balance	7 classes; d_min=0.0015 – d_max=0.075; aggregation and breakage: Luo model
• Drag coefficient bubbly flow	Ishii and Zuber
• Droplet flow <u>diameter</u>	1·10 ⁻⁴ m
• Drag coefficient droplet flow	0.44
• Turbulent dispersion	Burns et al.
• Lift coefficient	none
• Virtual mass coefficient	none
• Wall lubrication	none
Turbulence:	
• model	k-w SST per phase
• turbulence interaction	none
• <u>Free surface damping</u>	Egorov
Surface tension:	
• coefficient	0.072 N/m
• model	Continuum Surface Stress
Space discretization:	
• Momentum, Turbulence	First order upwind
• Volume fraction	Compressive
• Pressure	Second order
• Gradient evaluation	Least Squares Cell-Based
Time discretization:	
	First order

$\alpha_f = \alpha_d + \beta \nabla \alpha_d \cdot d\vec{r}$

Slope limiter value β	Scheme
0	1 st order upwind
0< β <1 and 1< β <2	Blended: 1 st upwind - 2 nd upwind and 2 nd order - compressive
2	Compressive

Domain, BC and Mesh

- 2D simulations
- **3D simulation:**
 - ✓ tube length = $3 \cdot D$
 - ✓ 4.2 M cells

Inlet:

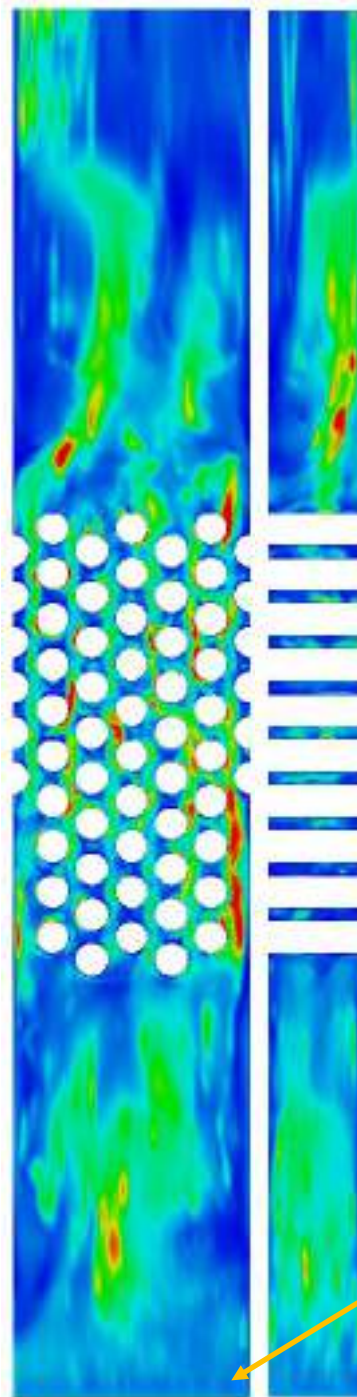
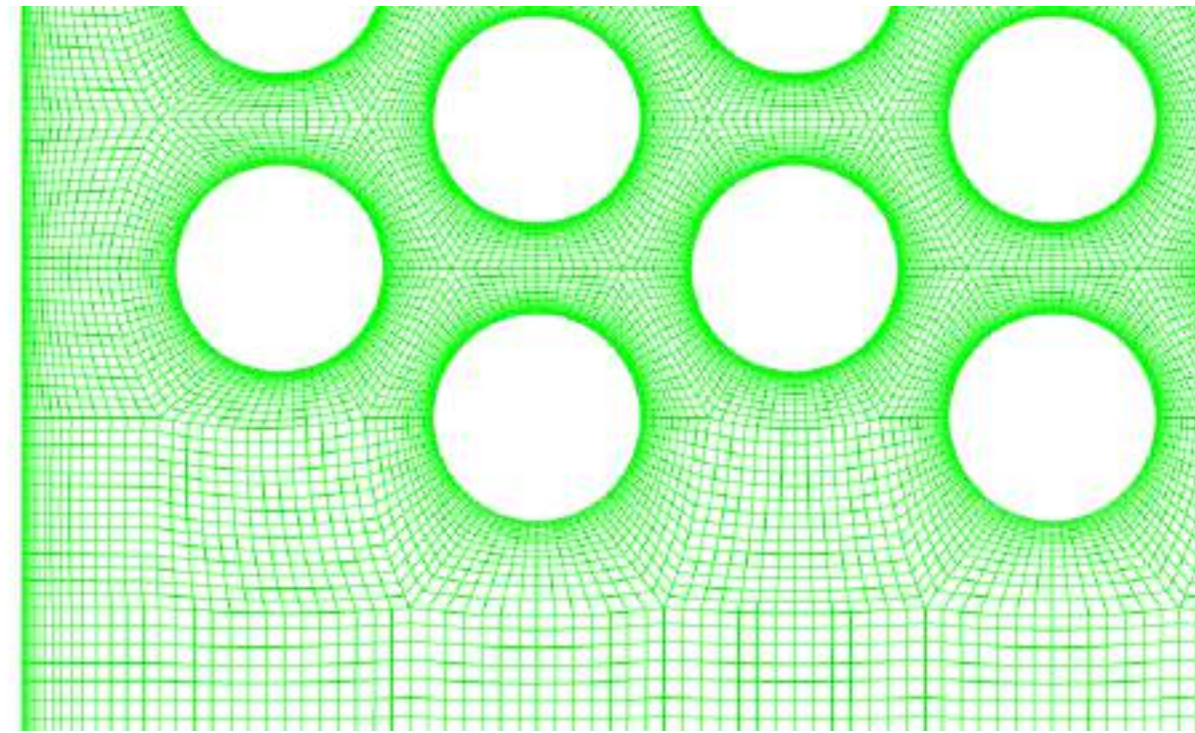
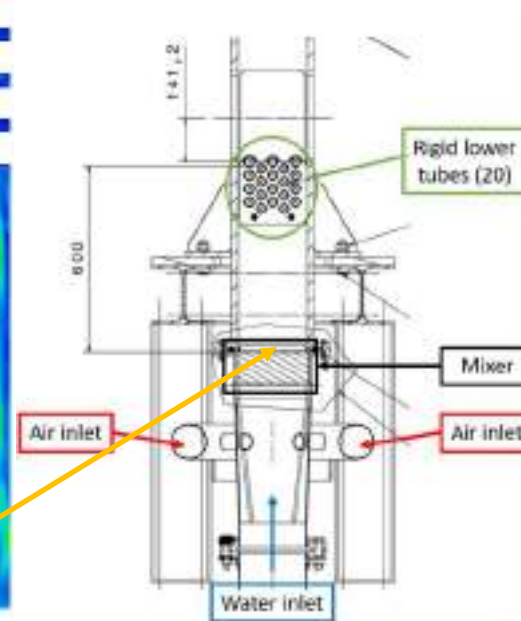
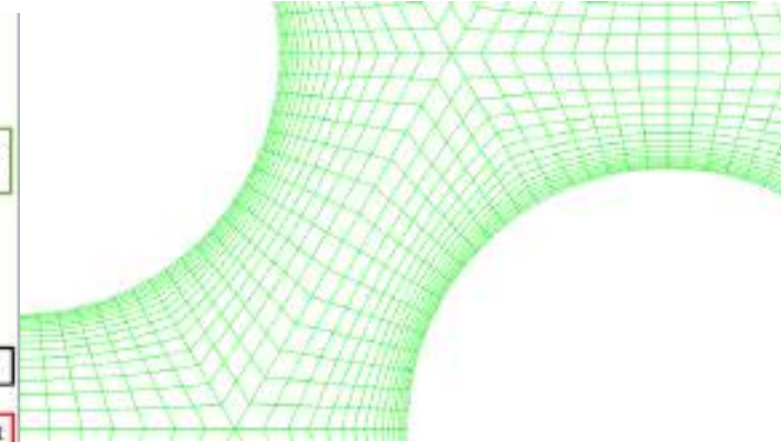
- Position corresponds to the mixing device;
- Flat and constant phase velocity and void fraction imposed.

Outlet:

- Constant P;
- Pressure drop to limit recirculation.

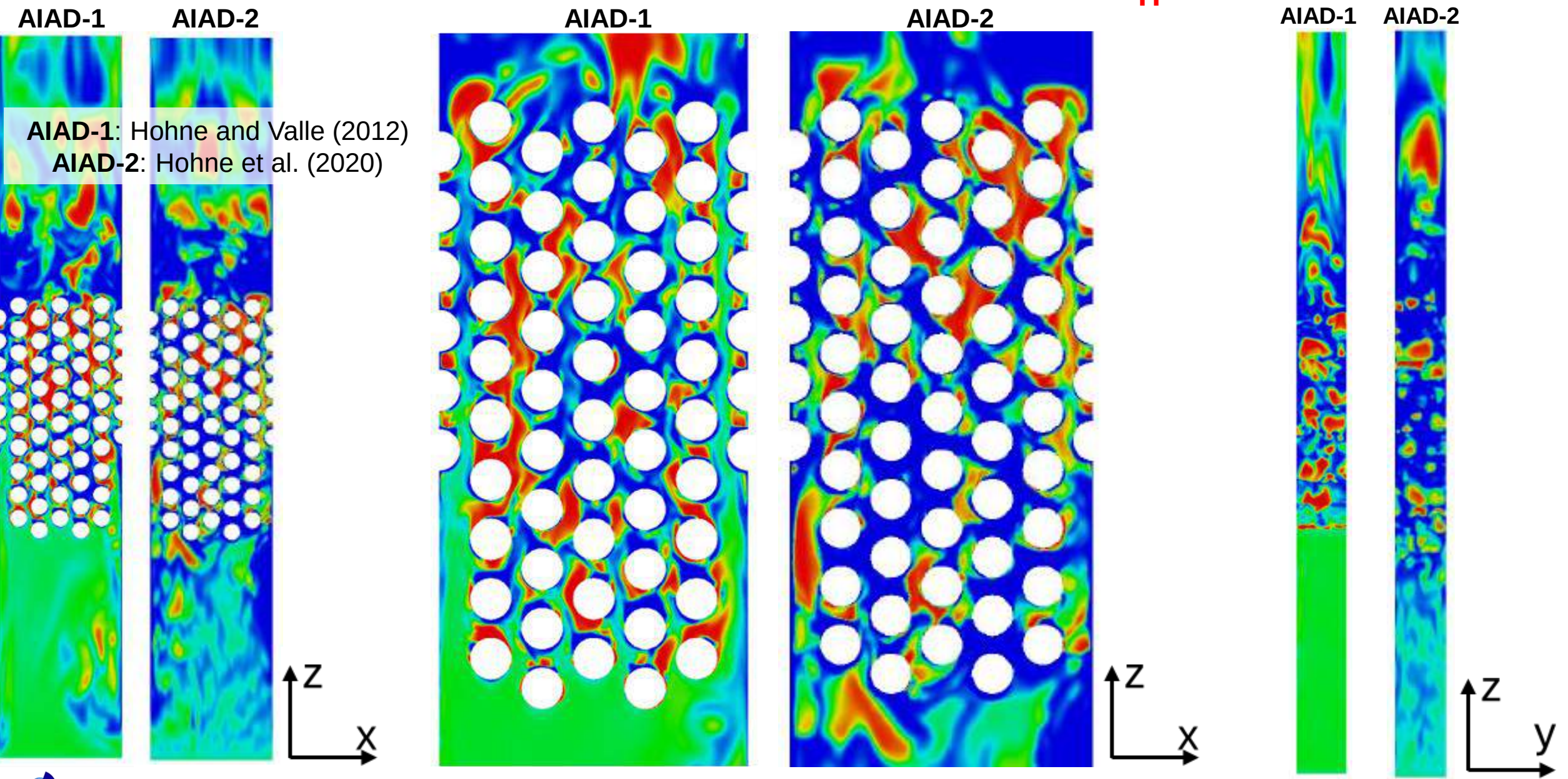
Simulated physical time:

- 10 seconds (4s « initialization » + 6s for post-processing)



Results – void fraction (3D sim.) - **case 2**

$\alpha_H=0.6$



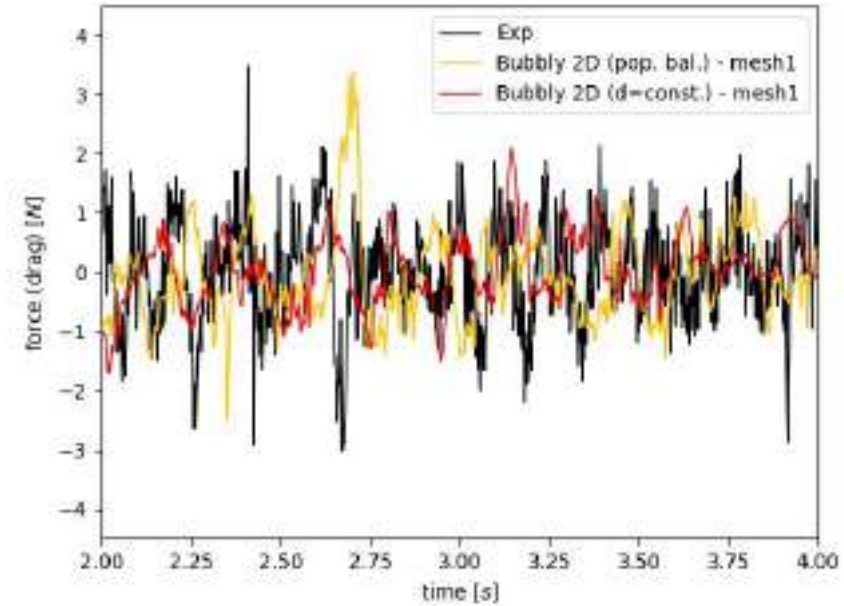
Results – force spectra (2D) - case 2

1) Bubble diameter (d_b) sensitivity

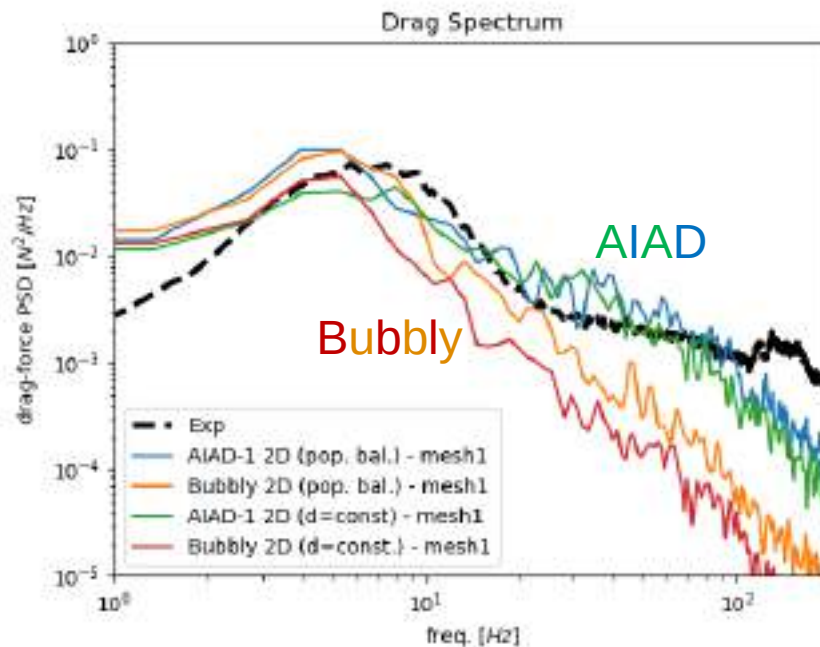
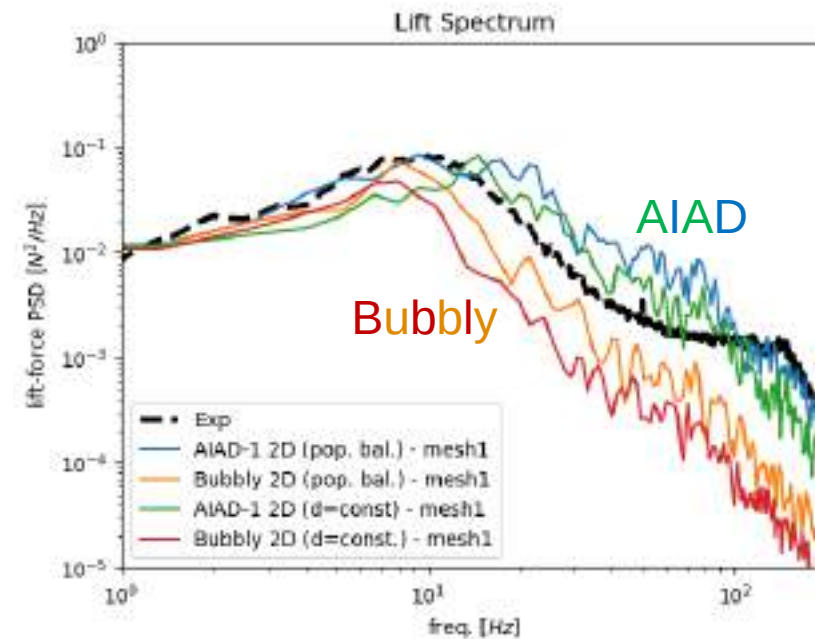
- Constant vs population balance
- Remember $F_D \propto \frac{\alpha}{d_b} \rho_c U_{gl}^2$

2) Dispersed bubbly vs AIAD sensitivity

Force signal:



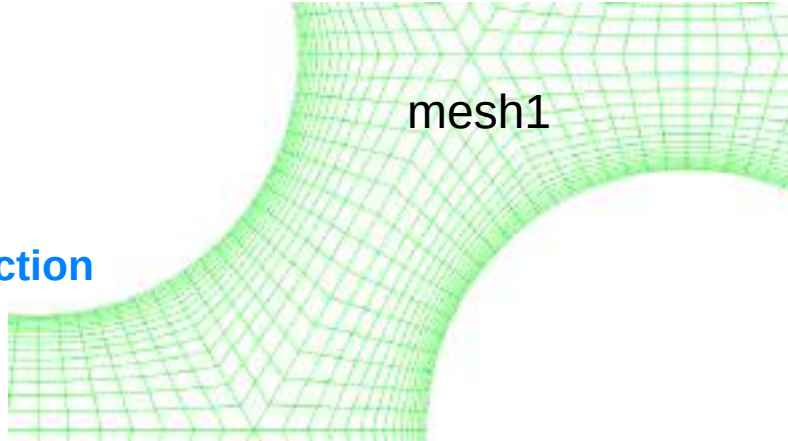
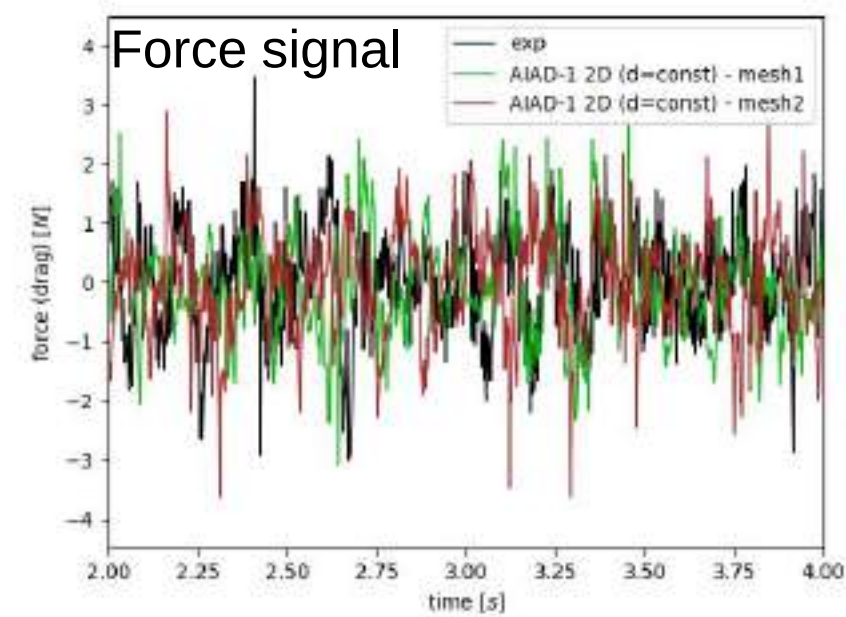
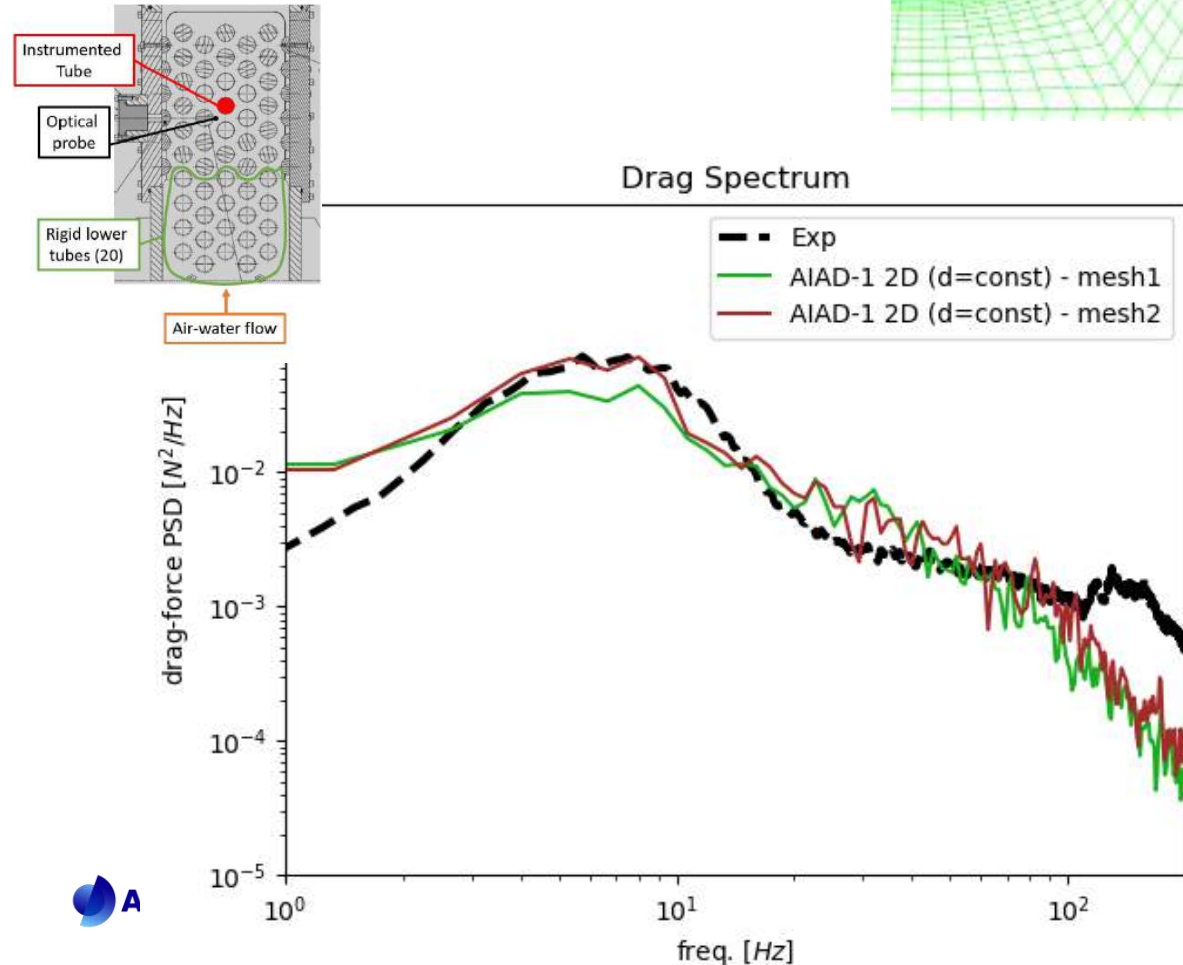
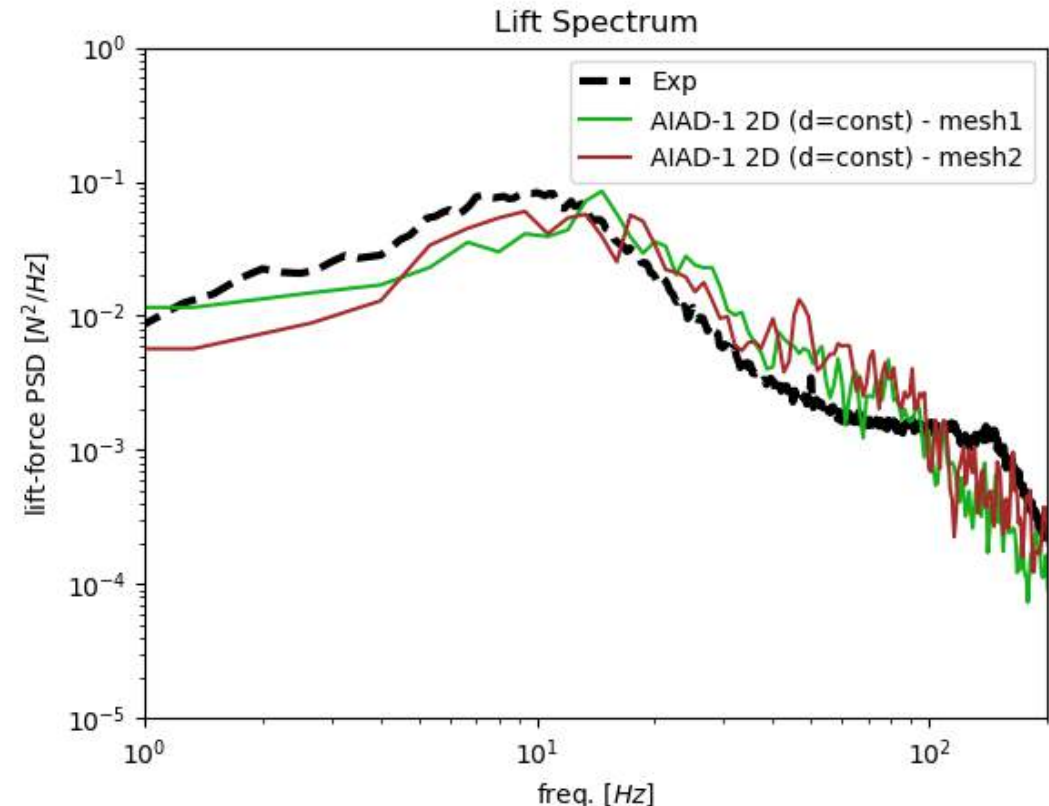
Force PSD:



Results – force spectra (2D) - case 2

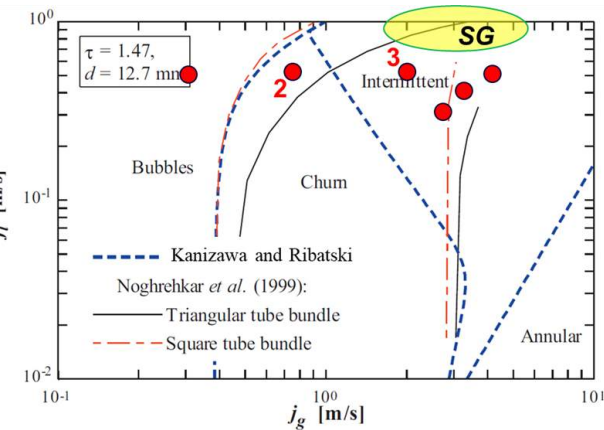
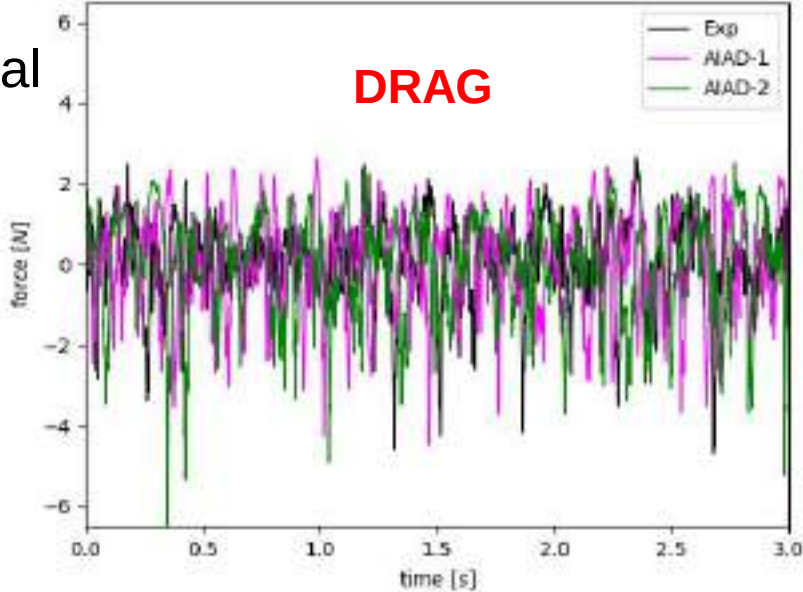
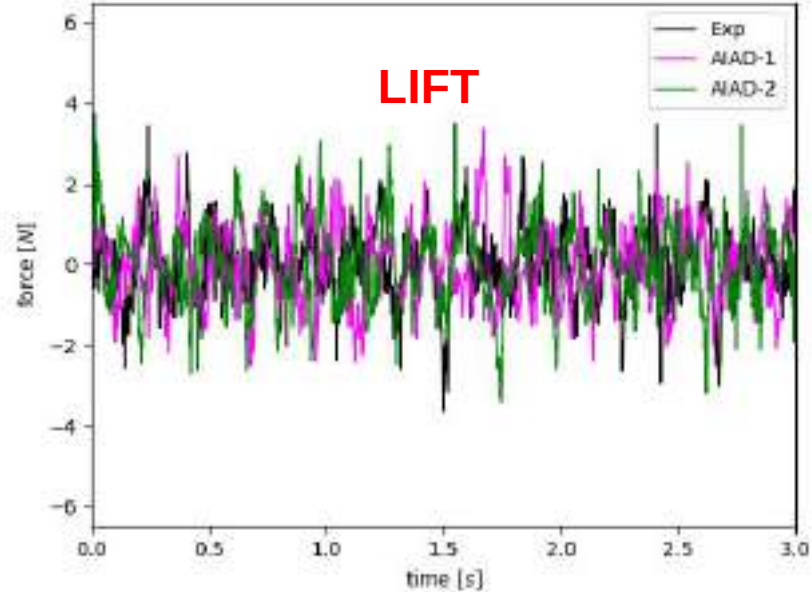
Mesh sensitivity (in 2D)

- *mesh1*: reference mesh
- *mesh2*: 20% refinement in each direction

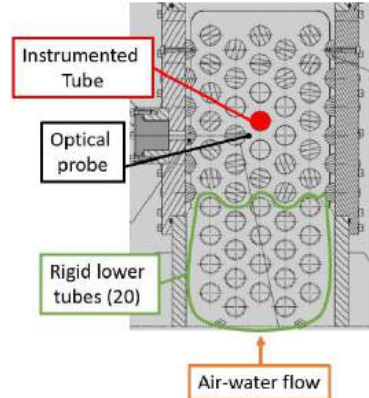
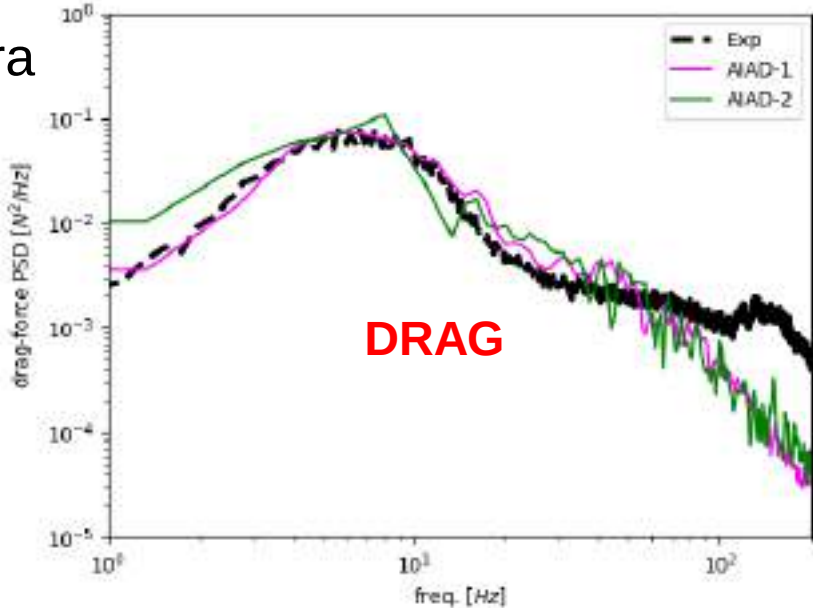
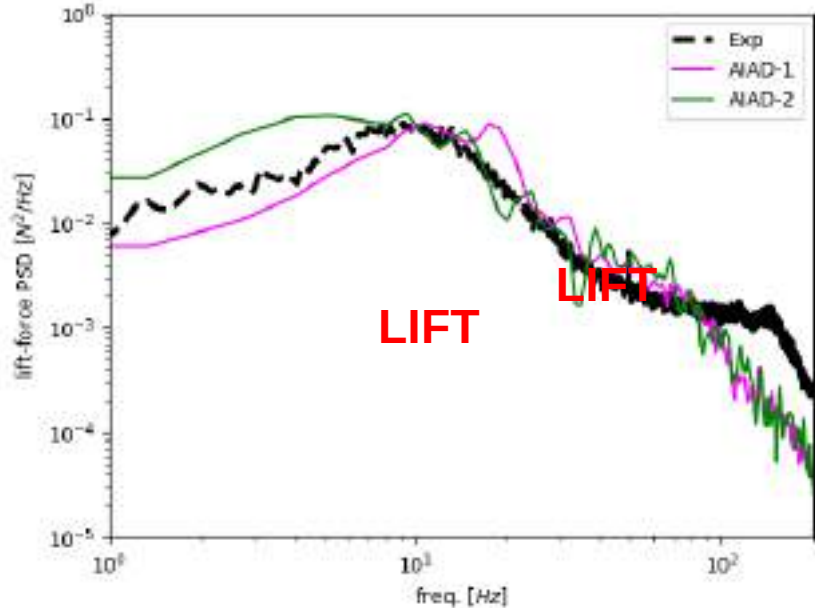


Results – force spectra (3D) - case 2

Force signal

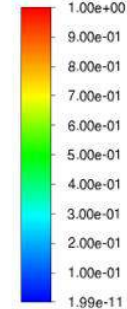
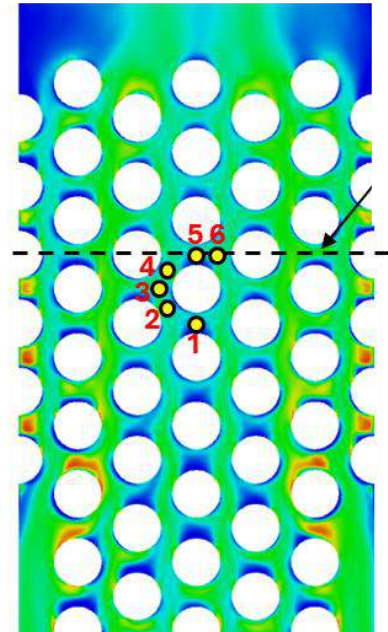
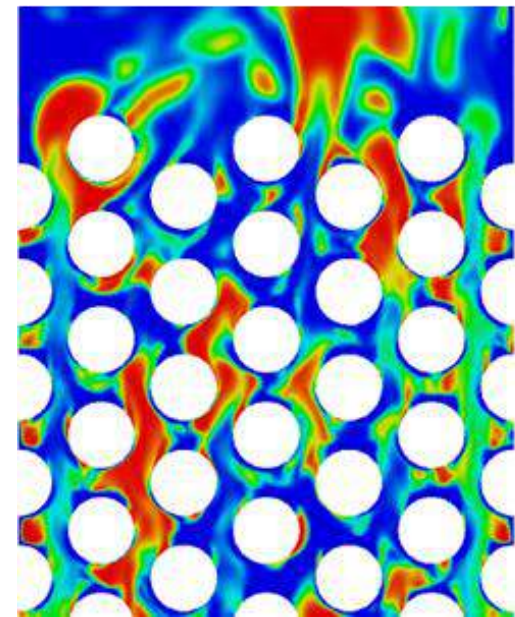


Force spectra

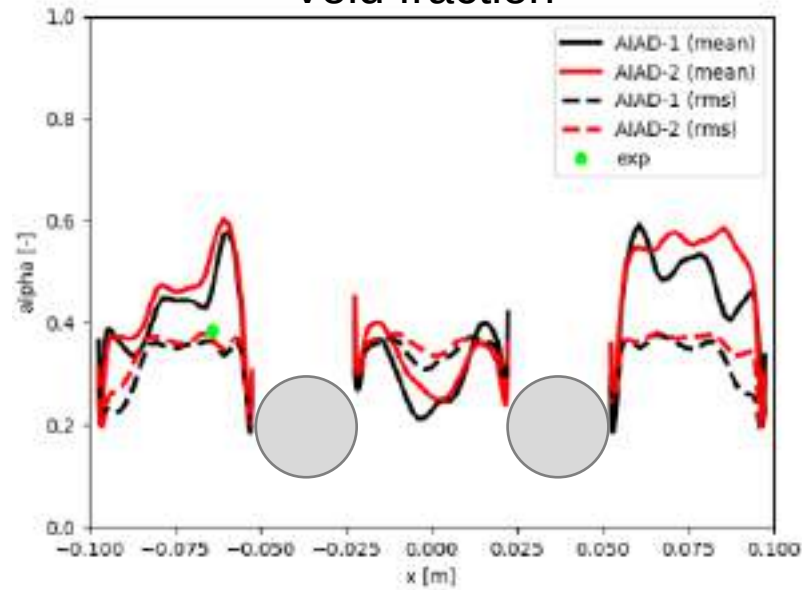


Results – void/velocities fraction (3D) - case 2

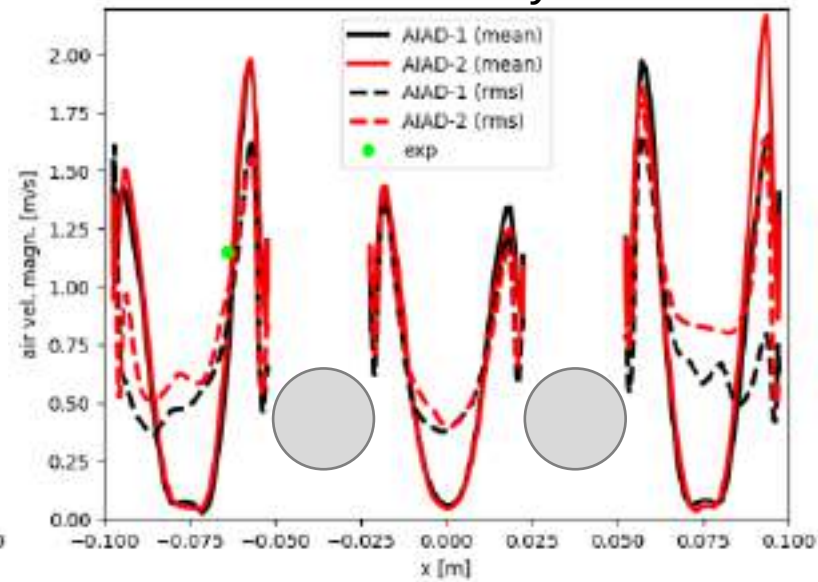
Mean
void
fraction
(-)



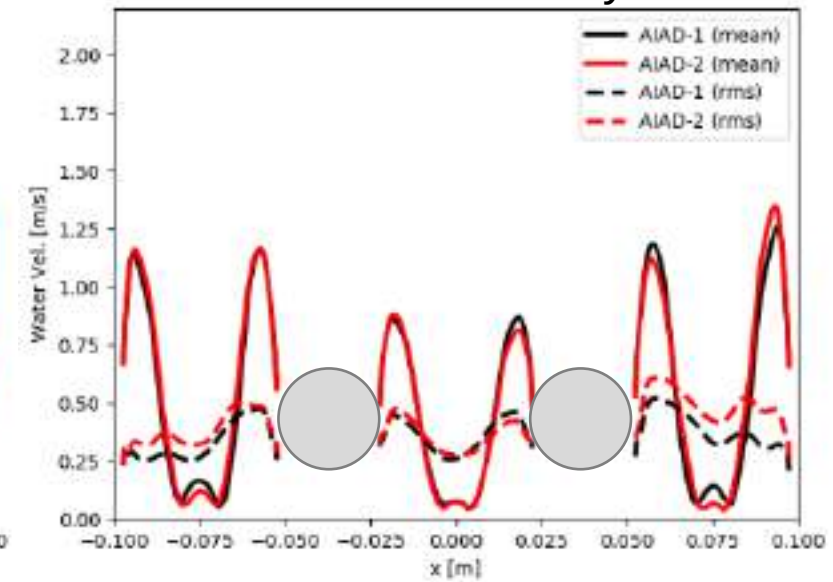
Void fraction



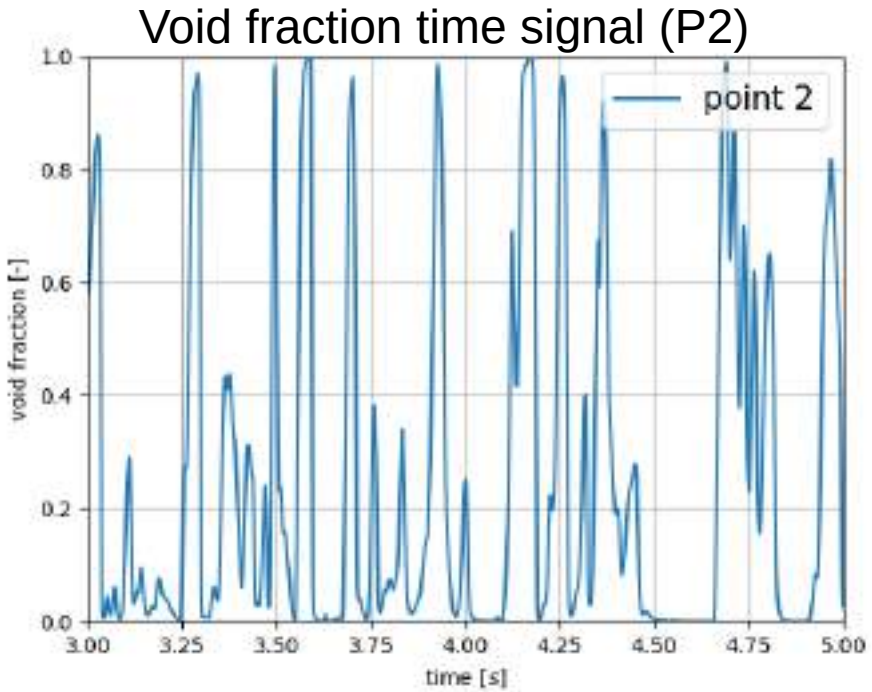
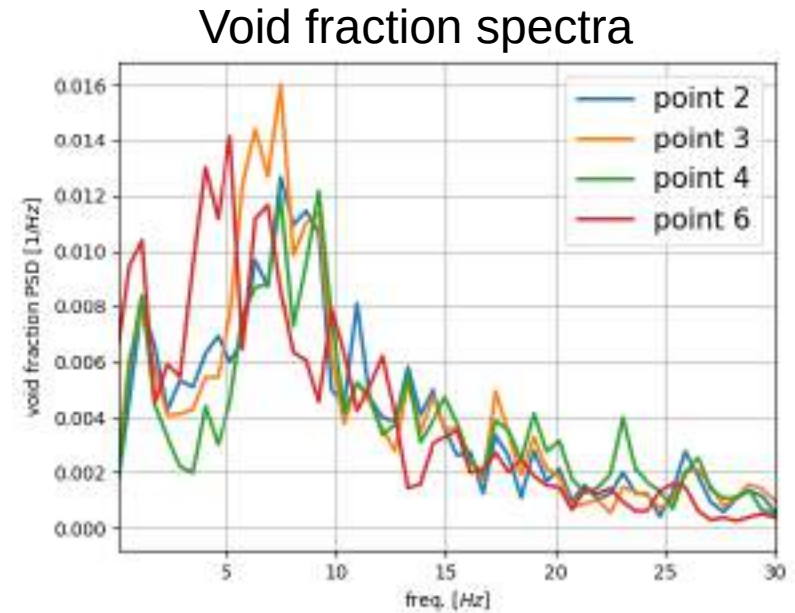
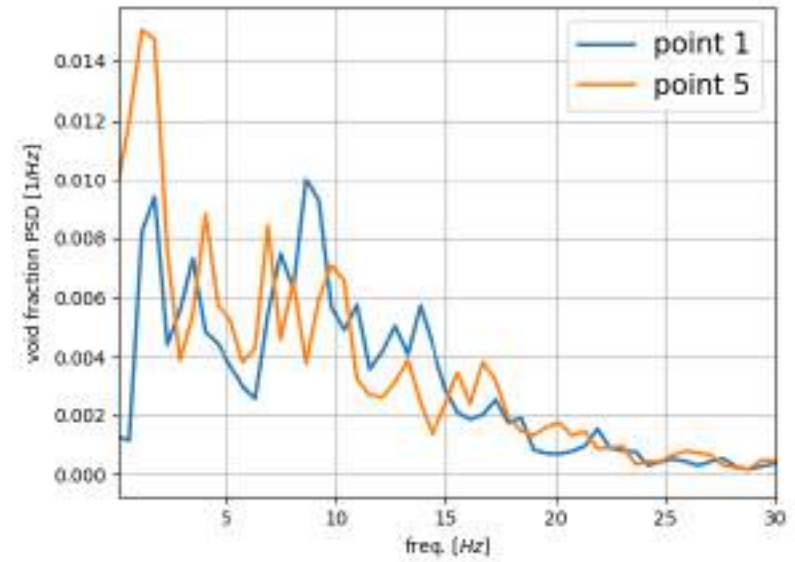
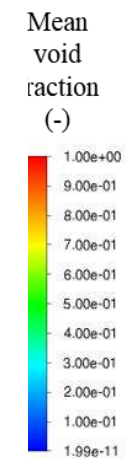
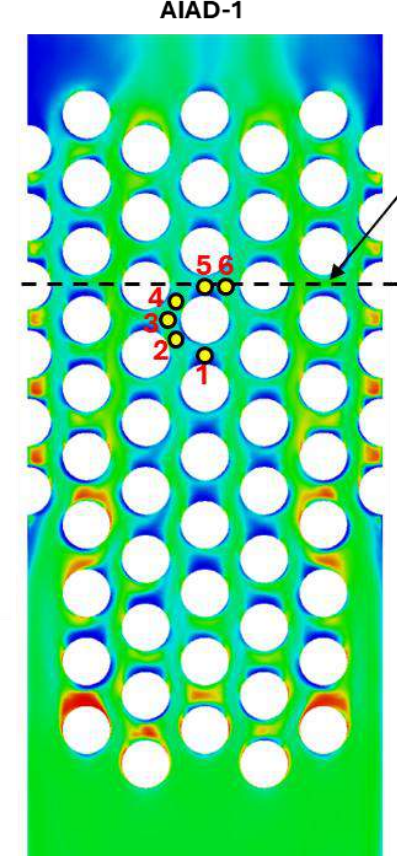
Air velocity



Water velocity

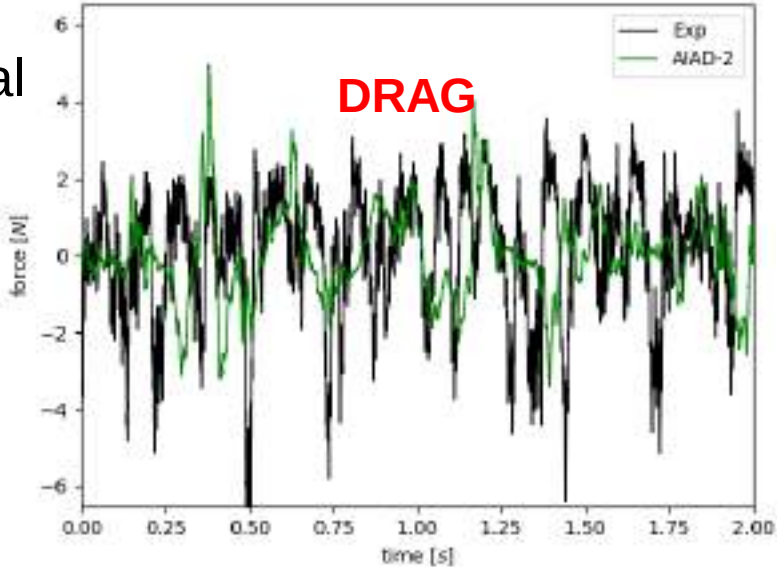
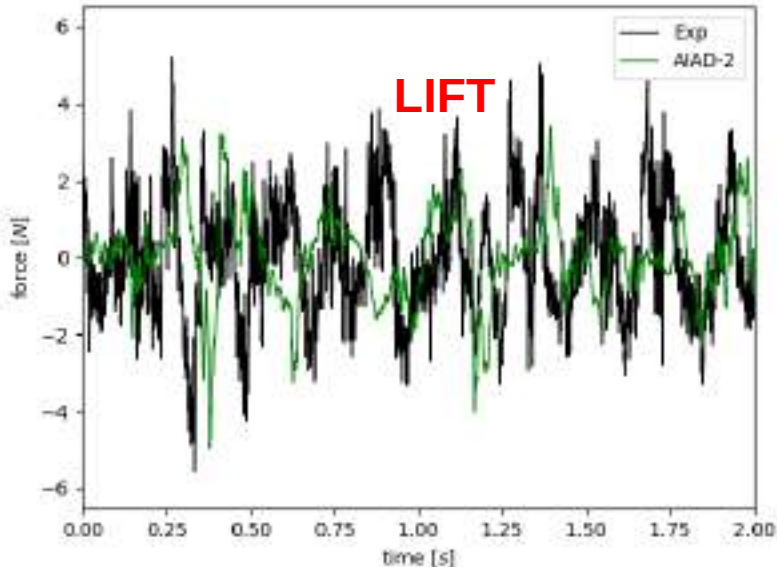


Results – void fraction spectra (3D) - TITAN case 2

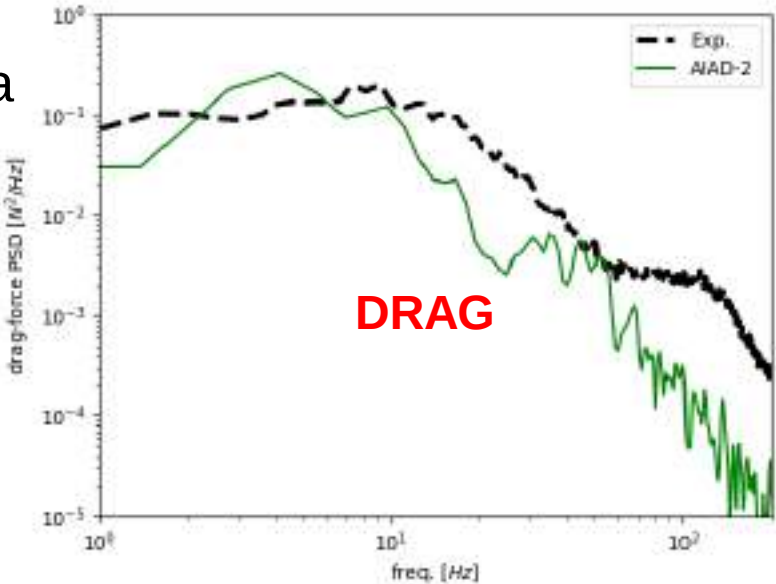
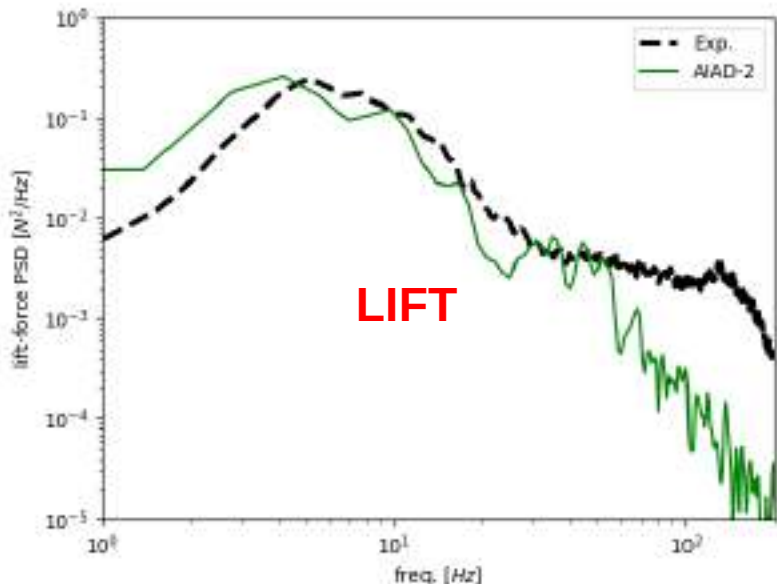
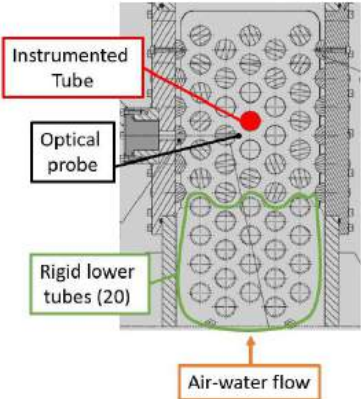


Results – force spectra (3D) - case 3

Force signal

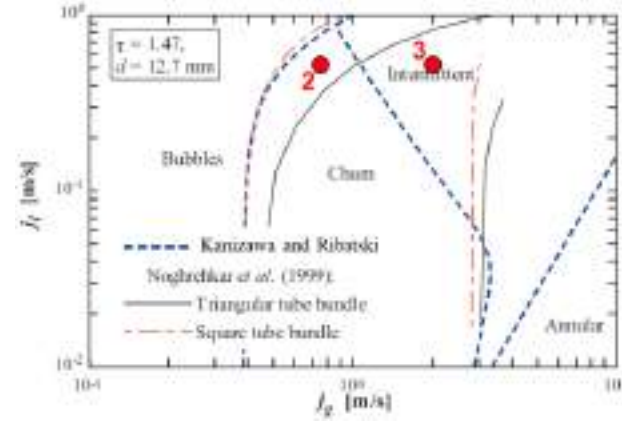


Force spectra

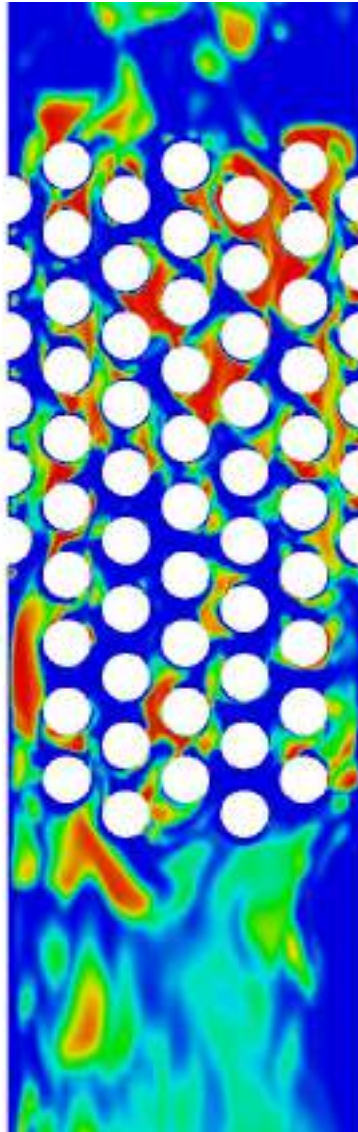


Results – flow regime blending functions - case 2

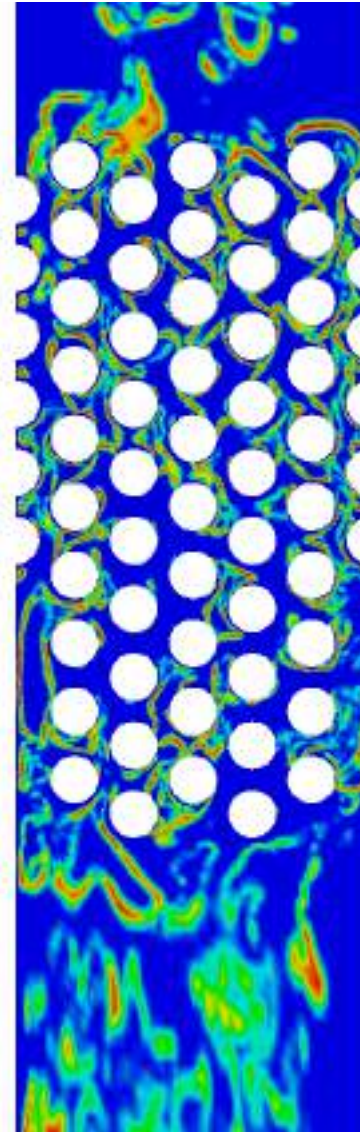
$\alpha_H=0.6$



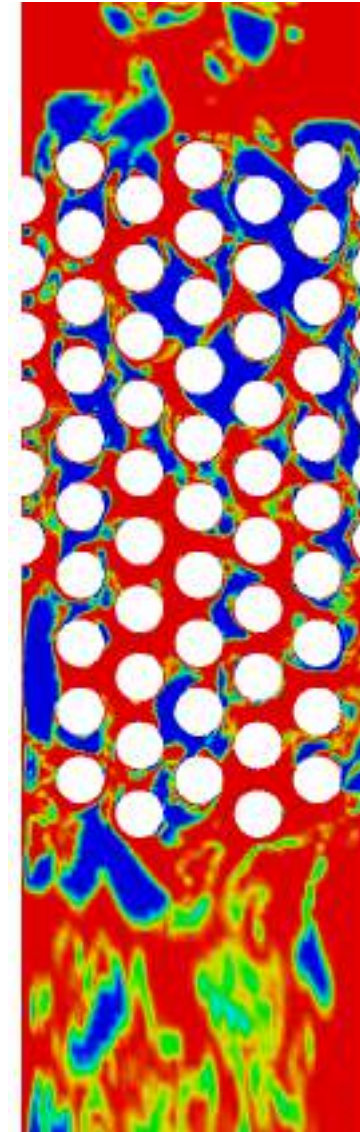
Void fraction (α)



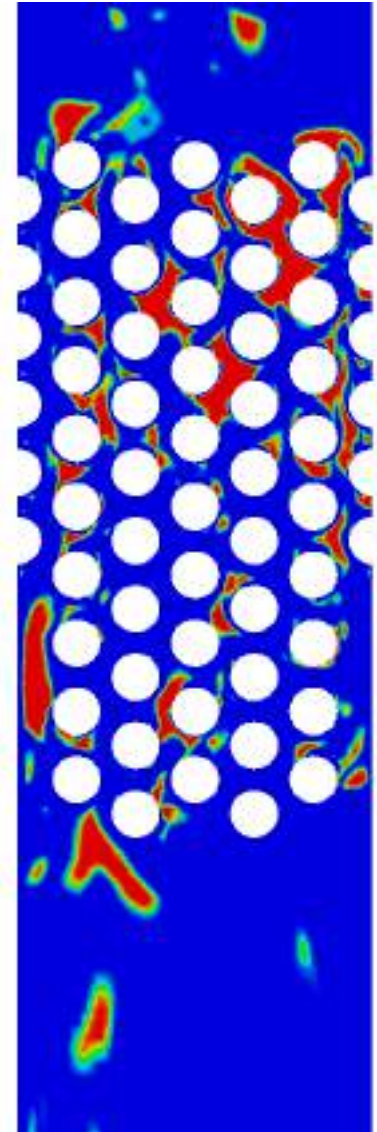
Free surf. ($\psi_{fs} = 0$)



Bubbly ($\psi_{fs} = 1$)

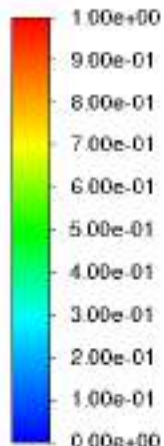


Droplet ($\psi_{fs} = -1$)

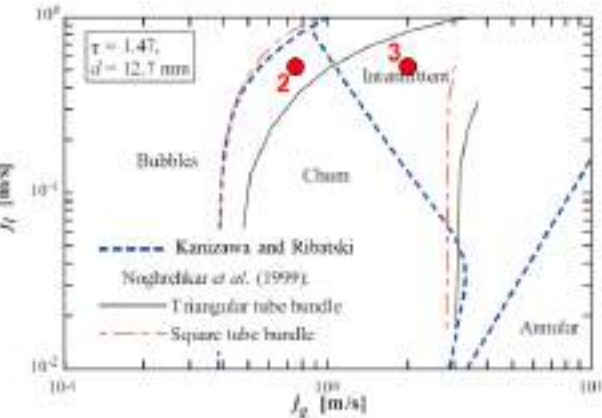


$$\psi_{fs} = \varphi_{fs}(f_B - f_D)$$

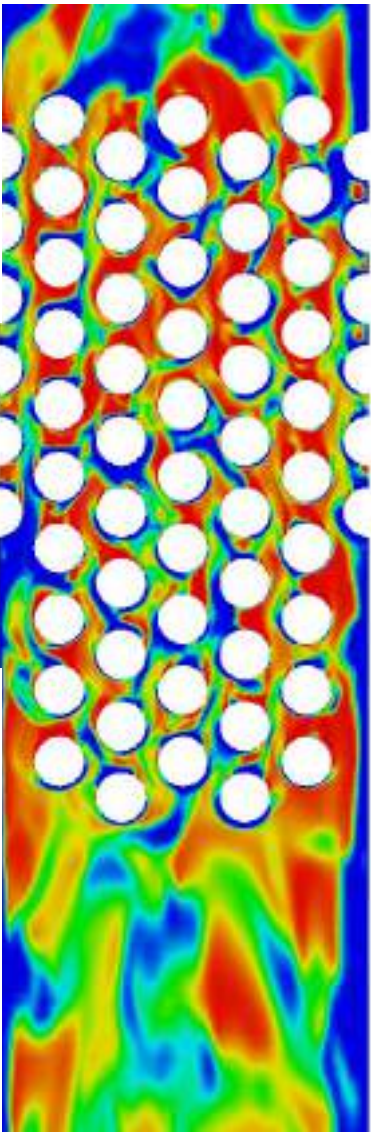
$$\psi_{fs} = \begin{cases} 1 & \text{bubbles} \\ 0 & \text{interface} \\ -1 & \text{droplets} \end{cases}$$



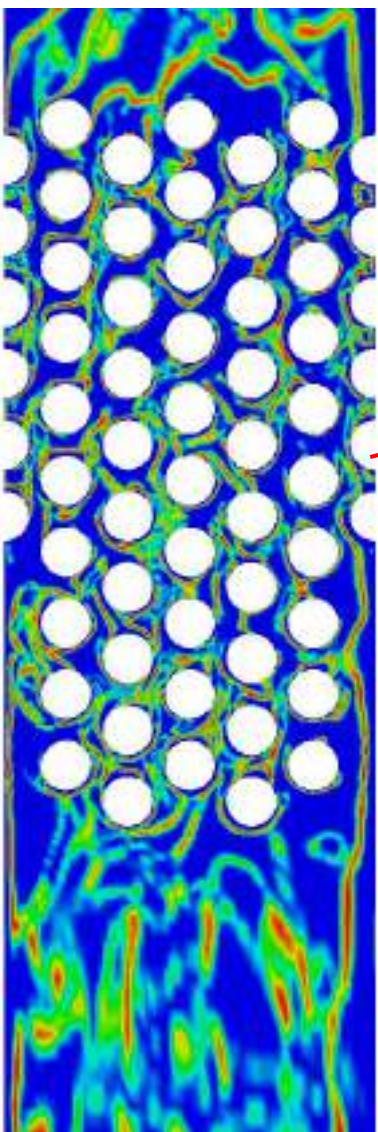
Results – flow regime blending functions - case 3



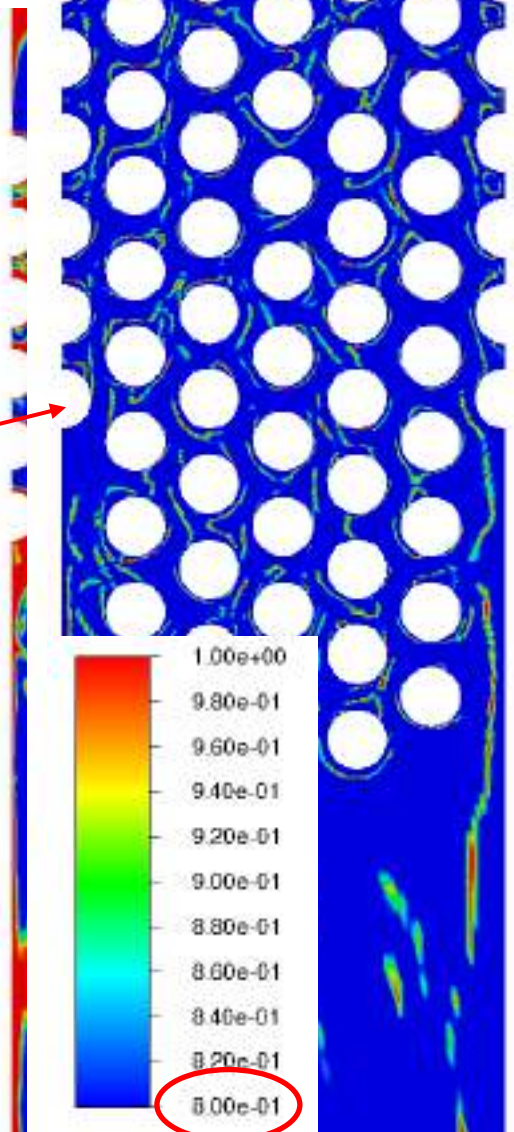
Void fraction (α)



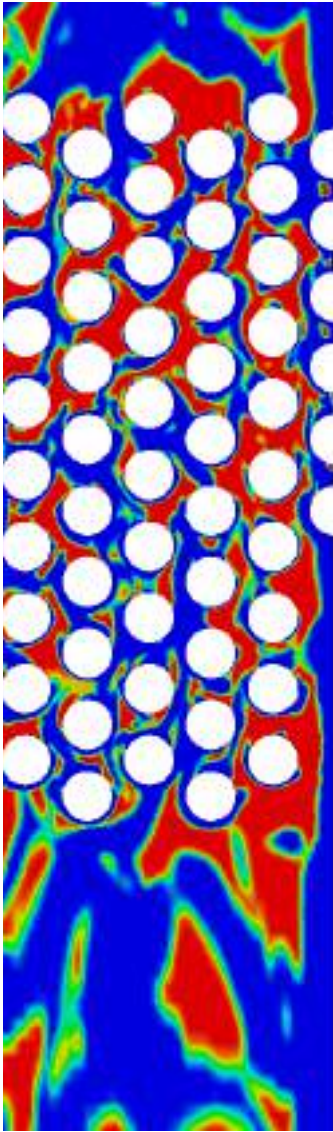
Free surf. ($\psi_{fs} = 0$)



Bubbly ($\psi_{fs} = 1$)

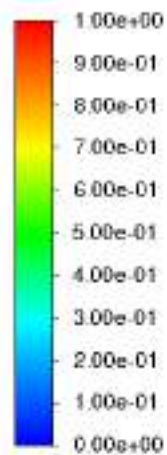
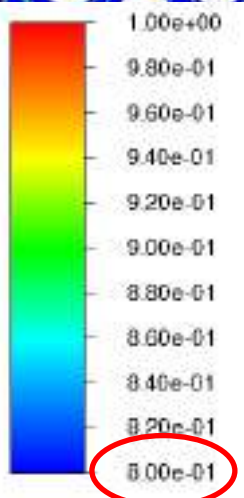


Droplet ($\psi_{fs} = -1$)



$$\psi_{fs} = \varphi_{fs}(f_B - f_D)$$

$$\psi_{fs} = \begin{cases} 1 & \text{bubbles} \\ 0 & \text{interface} \\ -1 & \text{droplets} \end{cases}$$



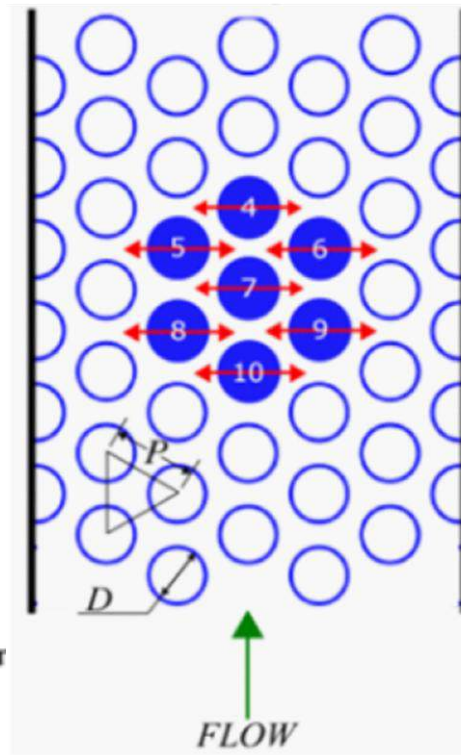
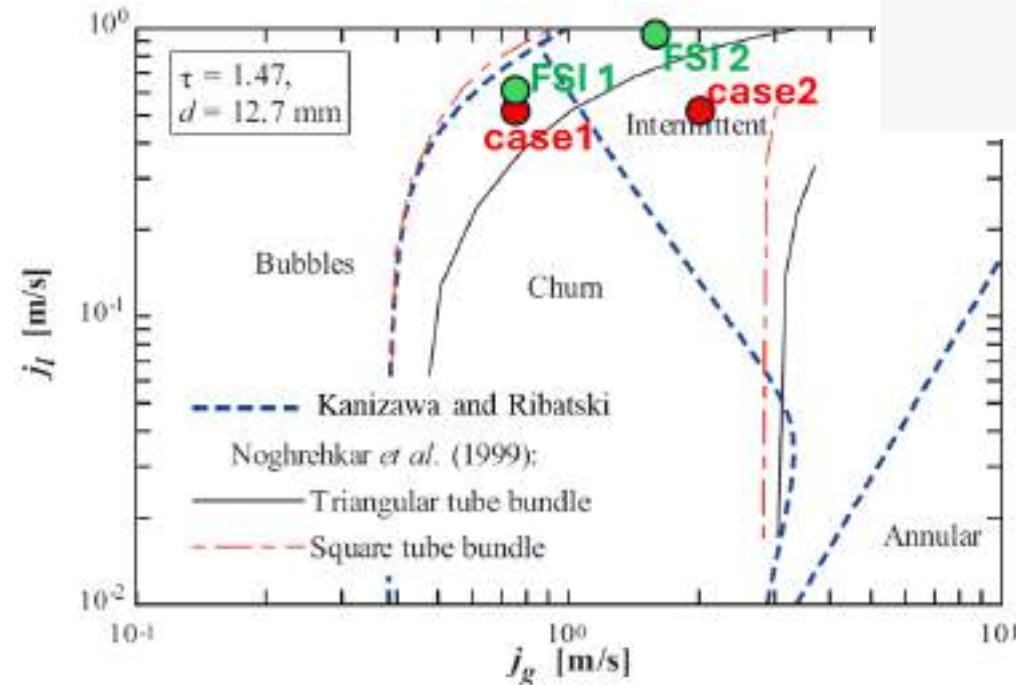
FSI simulations

7-flexible tube configuration

$$\alpha_H = 0.55$$

Important:

- Different flow rates compared to non-vibrating configuration!
- This limits the force validation relevance...



Numerical approach

Structure modeling:

Rigid body solver (1 DOF – pure translation):

- Implemented in Fluent with UDF (**Dolfen and Degroote, 2024**) → GO-VIKING collaboration 😊
- Newmark HHT method

Fluid-structure coupling:

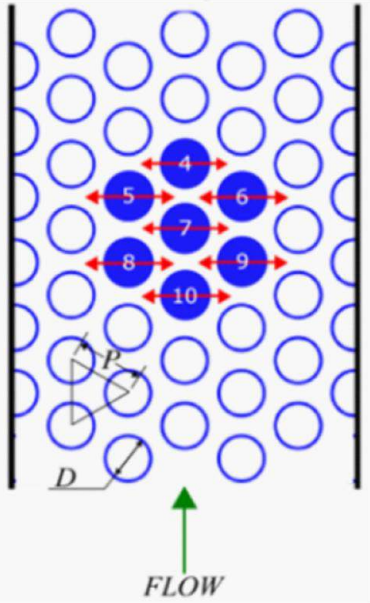
- **Two-way implicit** (Dolfen and Degroote, 2024)
 - ✓ Forced to run 2 sub-iterations per time-step, displ. convergence always reached!
- ALE with mesh deformation (Fluent standard model)

FSI simulations - results

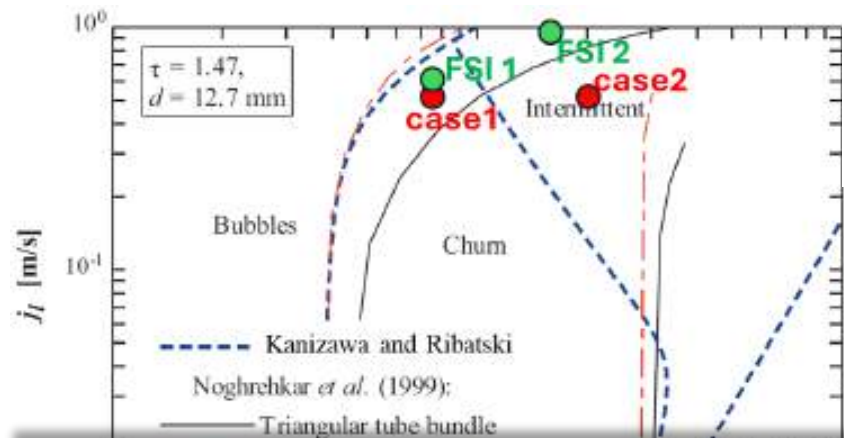
7-flexible tube configuration

Important:

- different flow rates compared to non-vibrating configurations!
- This limits the force validation relevance...



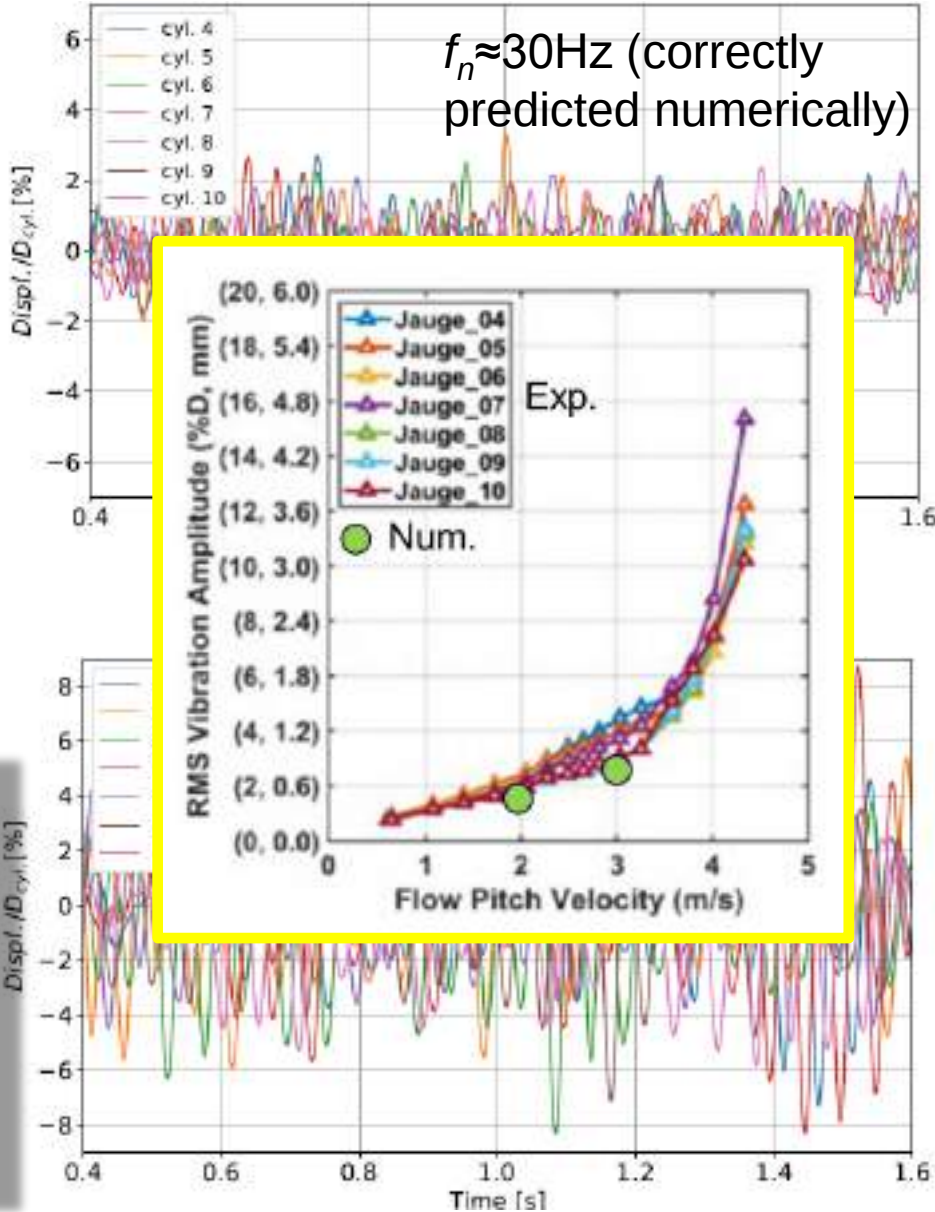
Underestimation
by a factor of
1.3-2.0



		rms cylinder displacement (%D)							
		Cyl. 4	Cyl. 5	Cyl. 6	Cyl. 7	Cyl. 8	Cyl. 9	Cyl. 10	
FSI 1	Exp.	1.52	1.64	1.56	1.45	1.42	1.40	1.41	
	Num.	0.83	0.95	0.86	0.78	0.92	0.94	0.78	
FSI 2	Exp.	3.37	3.22	3.03	2.92	2.44	2.41	2.44	
	Num.	1.87	2.42	2.58	2.34	1.86	1.94	2.28	

case FSI 1

$f_n \approx 30\text{Hz}$ (correctly
predicted numerically)



CONCLUSIONS

- Two-fluid morphology-adaptive AIAD indeed allows to achieve results closer to exp. values
- High CPU time required (HPC limited – not open source...)
- Two-phase numerical set-up in Fluent overall delicate
- Coupled FSI probably not required for the cases simulated (but will be for higher velocities → FEI)

PERSPECTIVES

- Run FSI simulations close to FEI
- AIAD in CFX meant to be more stable → more proper numerical and physical set-up achievable with CFX?
- Assess GLIM with last recent improvements (ongoing for TREFLE)

THANK YOU FOR YOUR ATTENTION



The TITAN benchmark: presentation of the numerical results

W. Benguigui, EDF



Funded by
the European Union

TITAN Benchmark



William Benguigui^{1,2}, Daniele Vivaldi³, Sina Tajfirooz⁴, Kevin Zwijsen⁴,
Constantinos Katsamis⁵, Ehimen Iyamabo⁵, Théo Ondoua¹, Fanny
Beltran¹, Samuel Gaucher¹

¹EDF R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatou, France

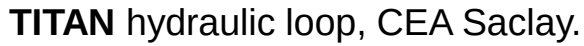
²IMSIA – UMR 9219 EDF/ENSTA/CNRS – 7 boulevard Gaspard Monge 91120 Palaiseau,
France

³ASNR, Cadarache, St Paul-lez-Durance, BP3, 13115, France

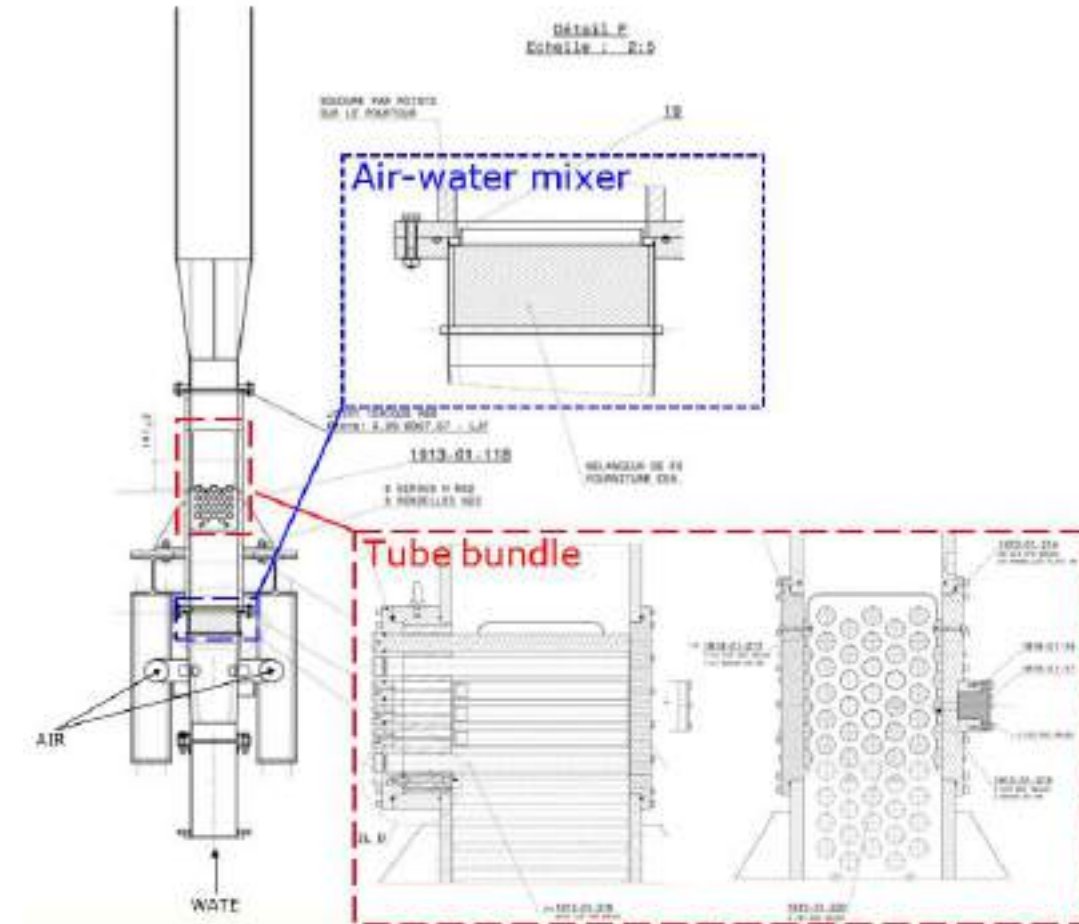
⁴NRG PALLAS, Westerduinweg 3, 1755 LE Petten, The Netherlands

⁵EDF Energy R&D UK, 11 Bressenden Place, London, SW1E 5BY, United Kingdom





Air-water crossflow at ambient conditions.



P. Piteau, X. Delaune, D. Panunzio, R. Lagrange, J. Antunes, Experimental investigation of in-flow fluidelastic instability for rotated triangular tube bundles subjected to single-phase and two-phase transverse flows, *Journal of Fluids and Structures*, Volume 123, 2023.

D. Panunzio, R. Lagrange, P. Piteau, X. Delaune, J. Antunes, Experimental investigation of out-of-flow fluidelastic instability for rotated triangular bundles subjected to single-phase and two-phase transverse flow. *Proceedings of the Flow Induced Vibration conference*, Paris-Saclay, 2022.

Numerical modeling



Partner	Software	Two-phase modeling	Turbulence model	Type of Inlet	Type of Outlet
EDF R&D	Neptune_cfd	Euler-Euler with Generalized Large Interface model	Rij-SSG with wall-law	Homogeneous	Averaged pressure condition
NRG	OpenFOAM 12 (Foundation branch)	Hybrid two-fluid model (HZDR model)	K-Omega SST for liquid phase and K-Omega SST Sato for gas phase	Uniform velocities and volume fractions are set based on the experimental superficial velocities and volume fractions	Fixed pressure, zero-gradient for k and Omega, Void fraction, pressureInletOutletVelocity for velocity,
ASNR	Fluent v2024	Euler-Euler AIAD	k-w SST – per phase	Homogeneous	Fixed pressure
EDF UK	StarCCM+	Volume of Fluid (VOF) Method	k-w SST	Homogeneous	Pressure

Calculation domains



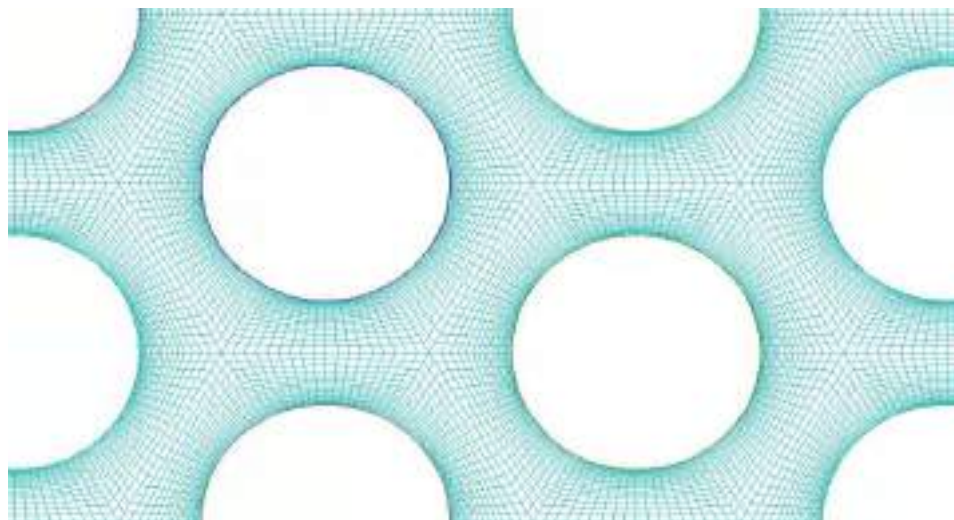
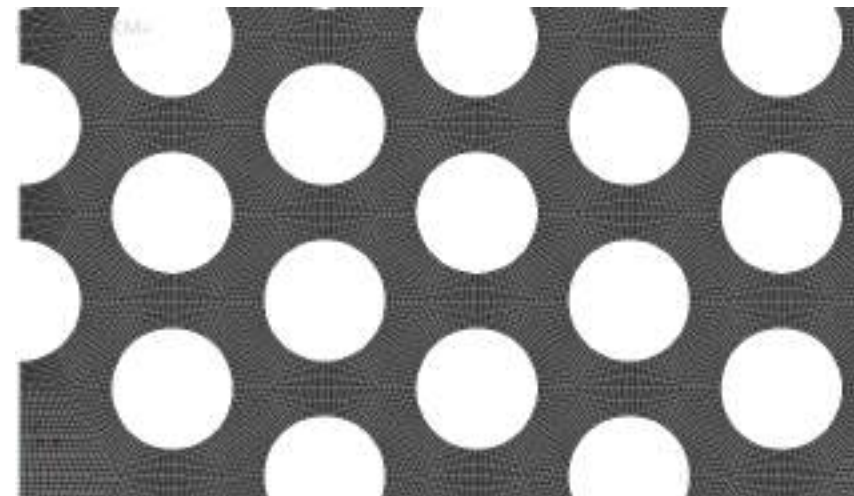
	2D or 3D	Tube Length	With upstream tube bundle	periodicity	Type of mesh	Total number of cells	Dr (mm) at cylinder wall	Dx_mean (mm)
EDF R&D	3D	2D and 10D	0.3 m	Yes (2D) and No (10D)	Hexahedral	8M (2D) 40M (10D)	$2.5 \cdot 10^{-4}$ m	
NRG	3D	10D	1.89 m	no	Unstructured hexagonal	1 M	$3,0 \cdot 10^{-4}$ m	3 mm
ASNR	3D	3*D	0.416 m	Symmetry	Hexa	4.1M	$1.5 \cdot 10^{-4}$ m	
EDF UK	3D	10D	0.3 m	No (10D)	Hexahedral - unstructured	5.8 M	$2.5 \cdot 10^{-4}$ m	1.5 – 3.0

Calculation domains

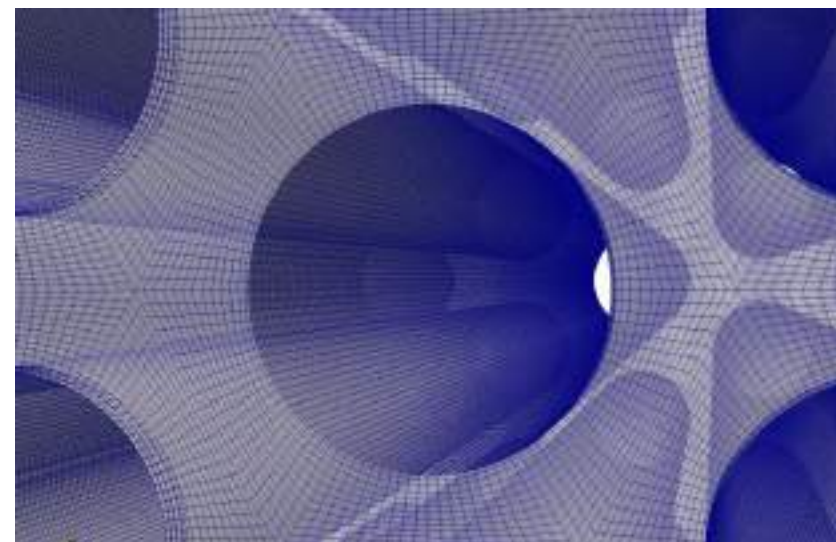
NRG



ASNR



EDF UK

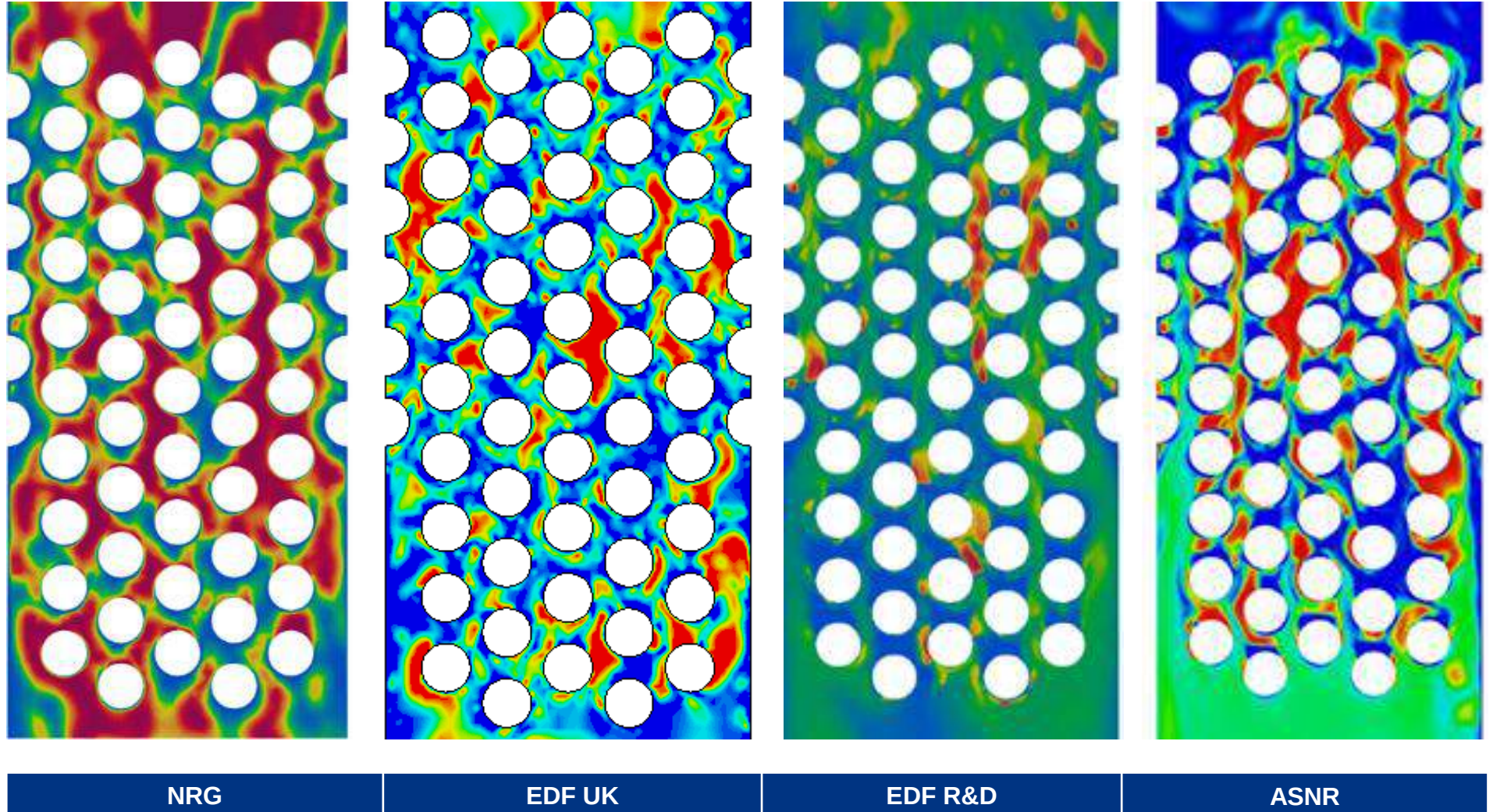


EDF

Numerical simulations



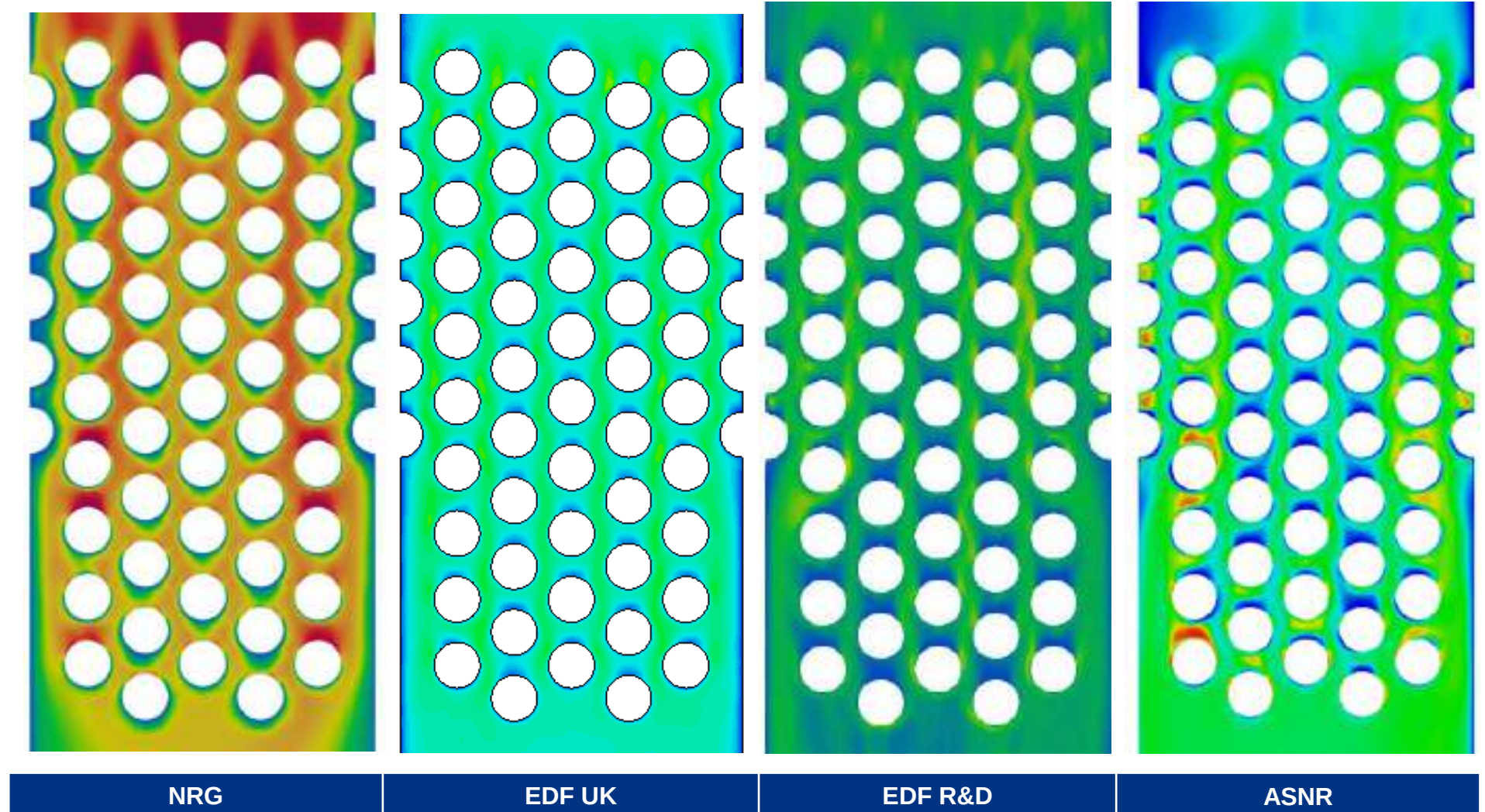
Case 2 - Instantaneous



Numerical simulations



Case 2 - Averaged

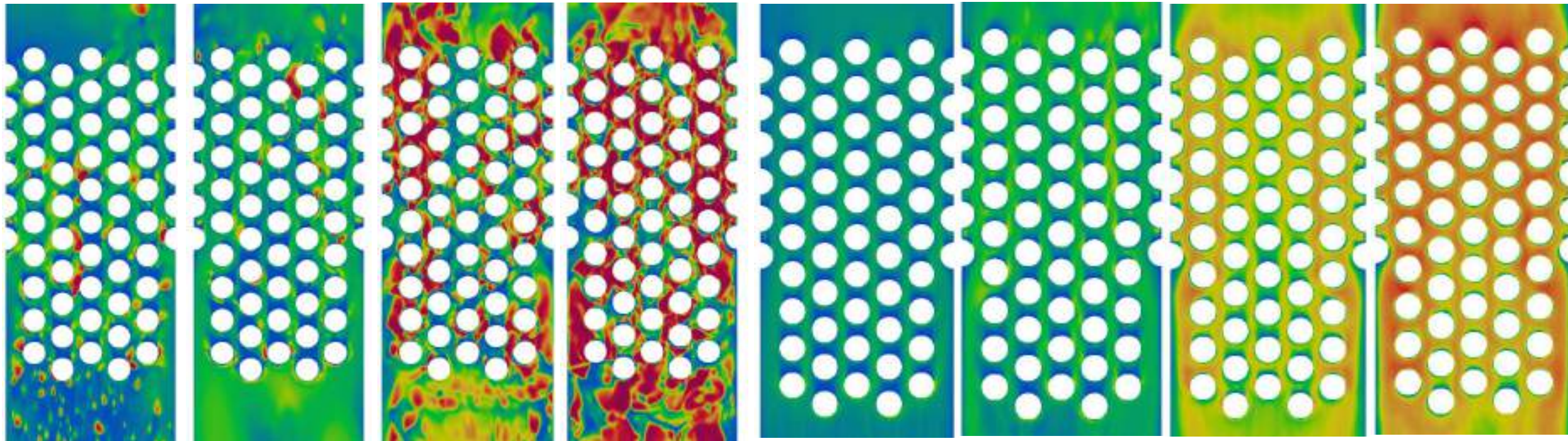


Numerical simulations



Instantaneous

Time-averaged



CASE 1

CASE 2

CASE 3

CASE 4

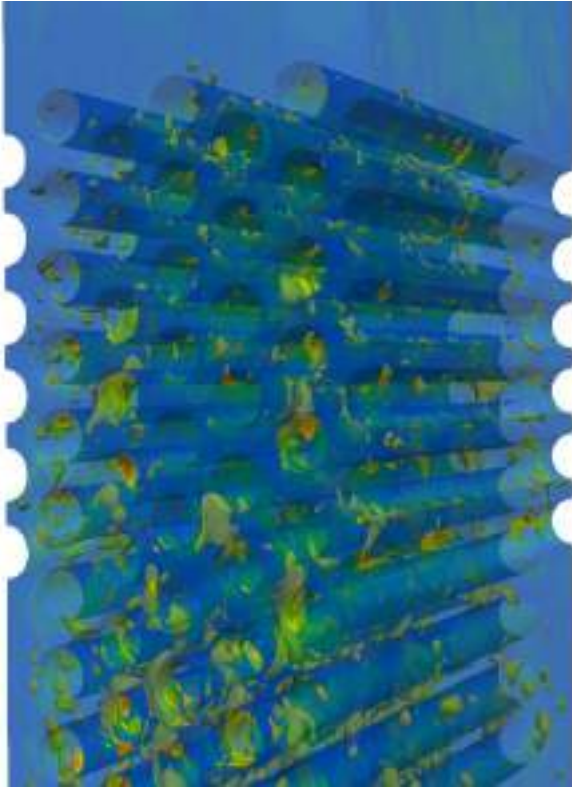
CASE 1

CASE 2

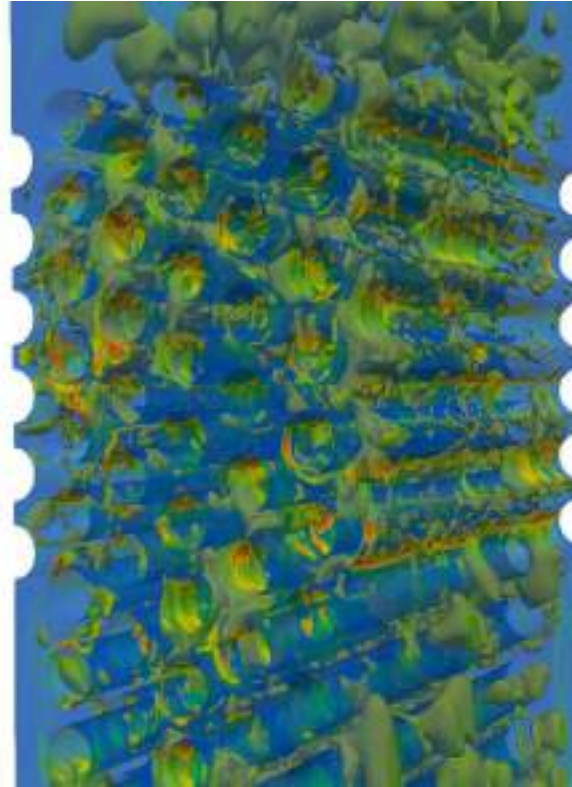
CASE 3

CASE 4

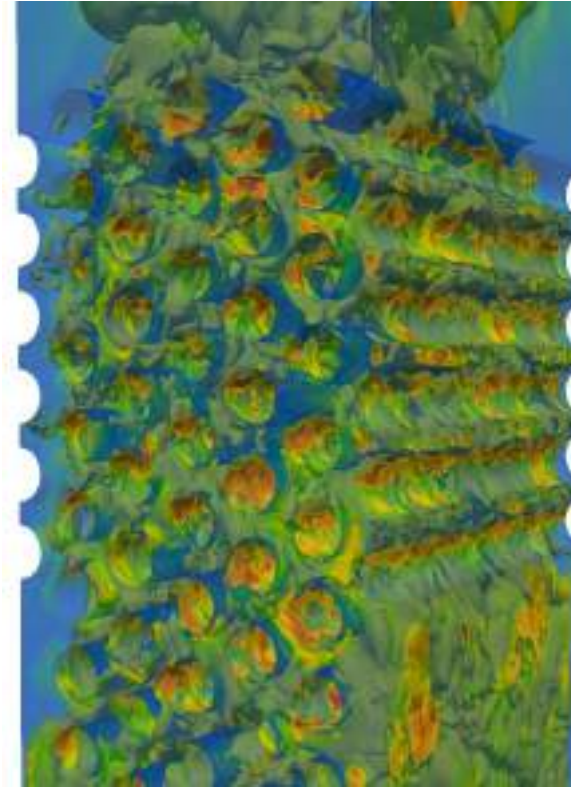
Numerical simulations



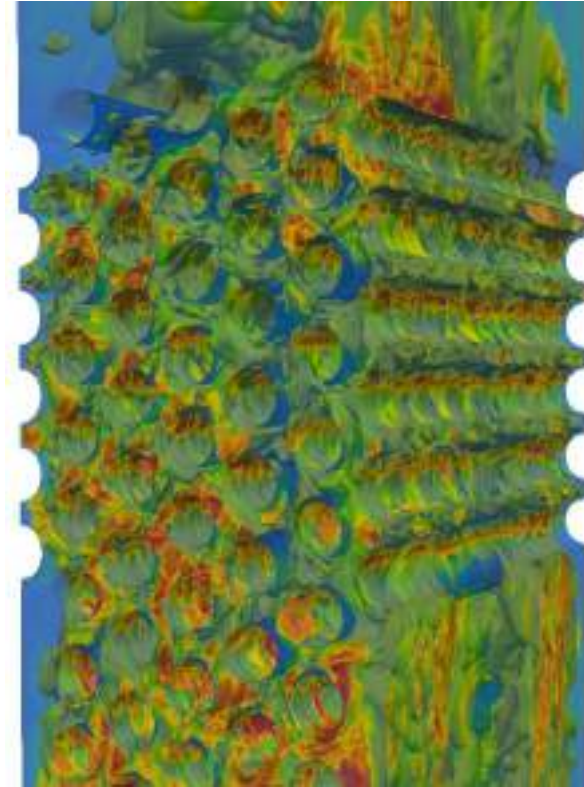
CASE 1



CASE 2



CASE 3



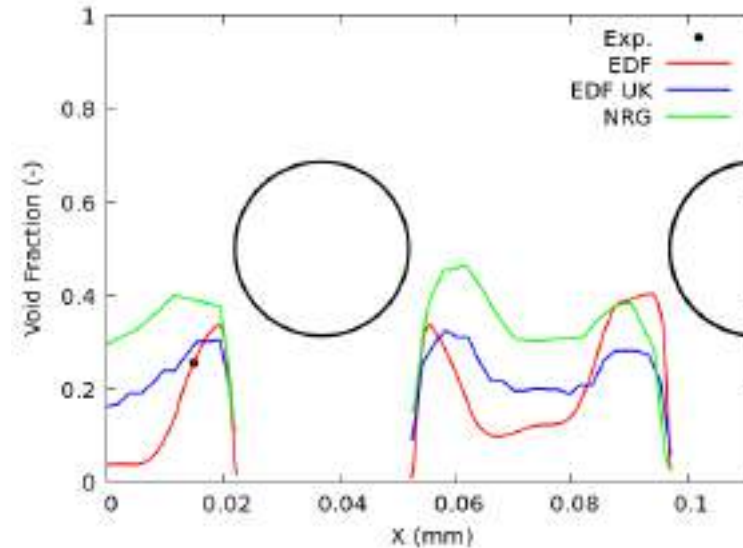
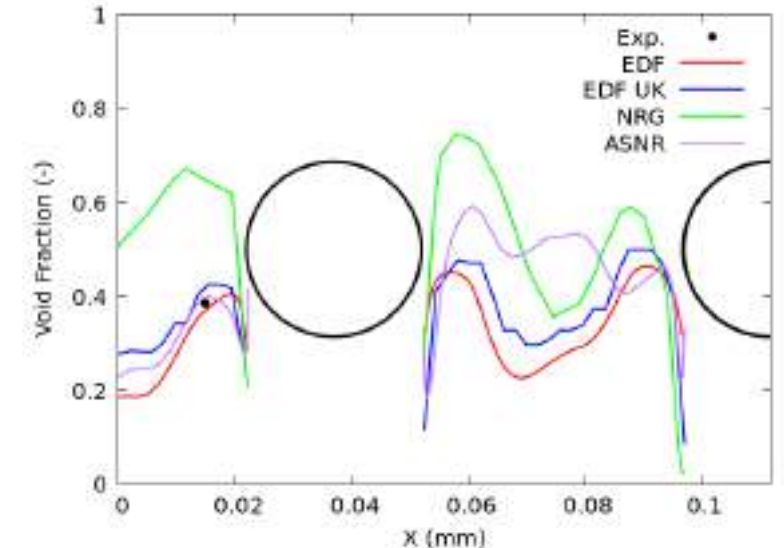
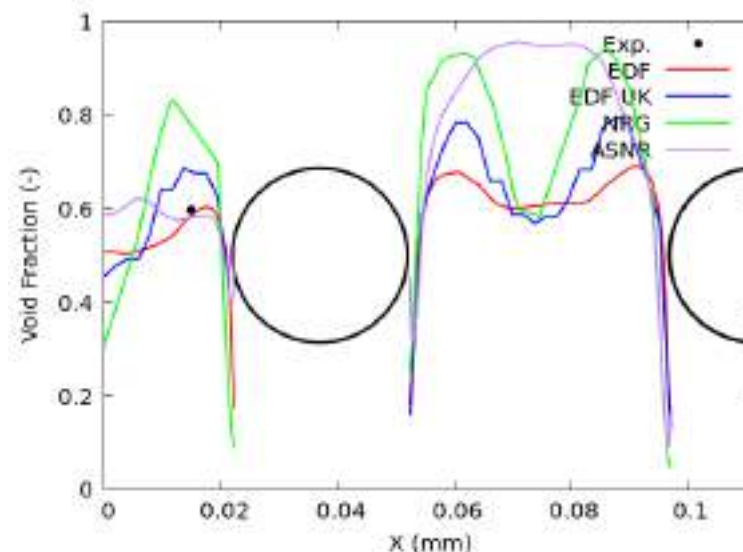
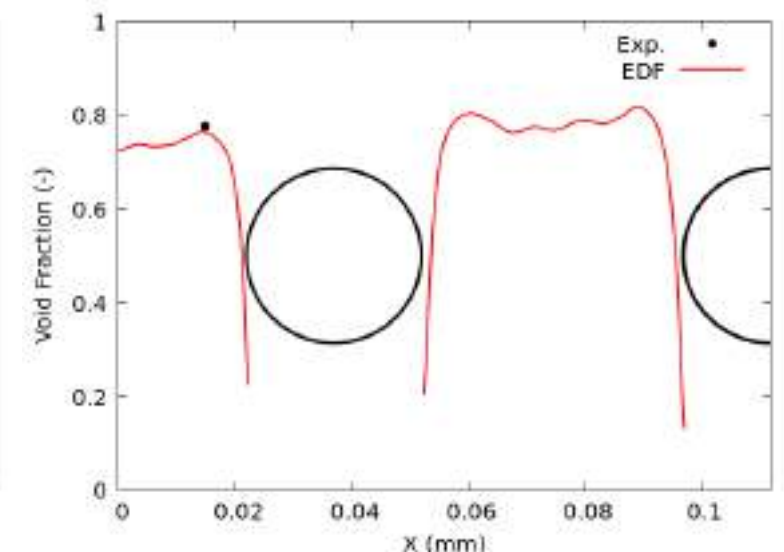
CASE 4

EDF contribution on the simulated length

Numerical simulations



- For Case 2, for which the void distributions are shown in, a reasonable agreement between EDF UK, EDF R&D and ASNR simulation at the location of the optical probe, and more generally across the tube bundle, can be seen.
- For Case 1 and 3, this is also true for EDF UK and EDF R&D simulations highlighting similarities in terms of two-phase flow predictions between the different numerical models. What is exactly meant here? Void distribution?

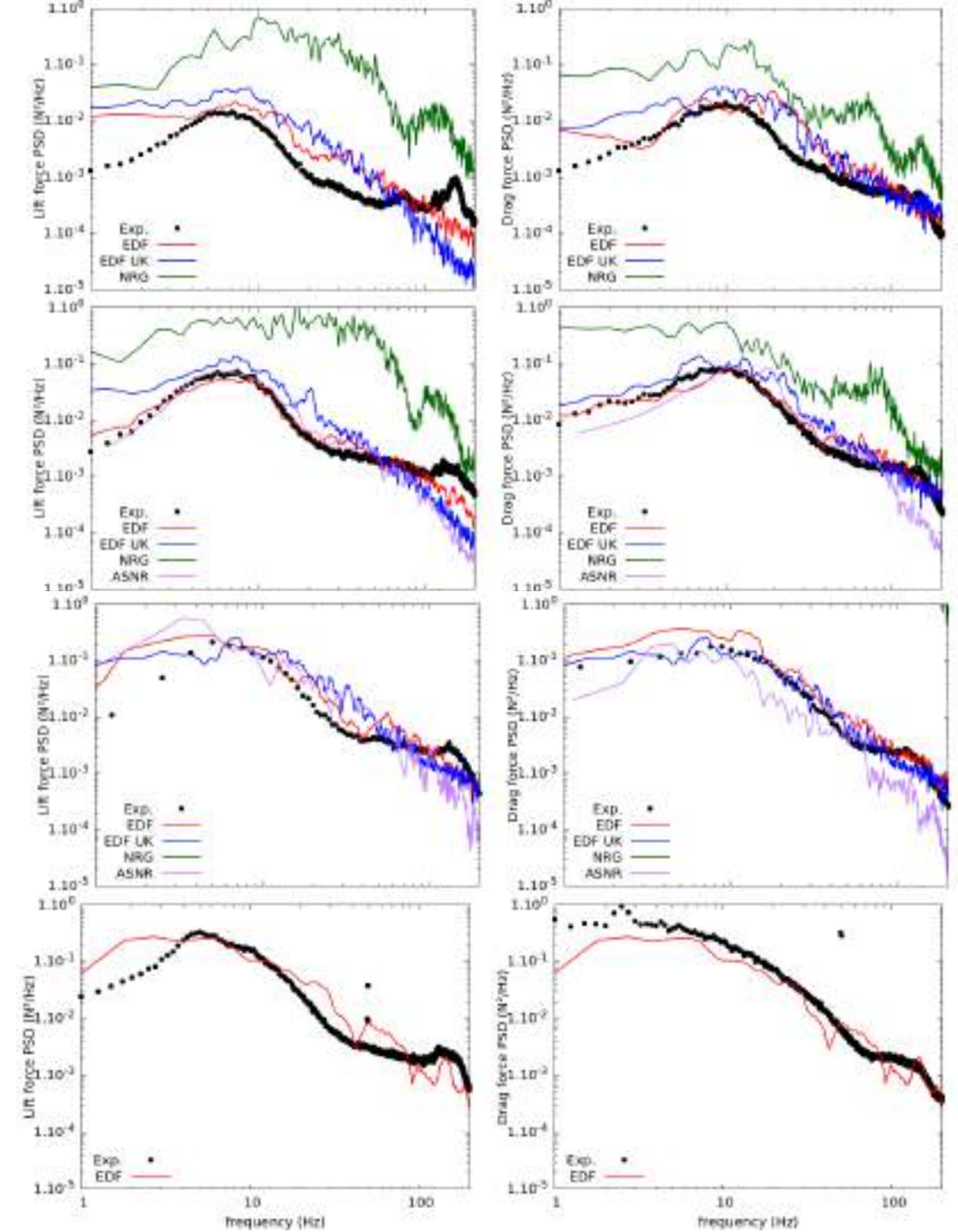


Numerical simulations

- Similarities observed between EDF R&D, EDF UK and ASNR with three different codes and set-ups are also confirmed by the force prediction where there is a correct agreement between simulations and experiment for the different flow patterns.
- Less true for Case 1, being a bubbly to slug flow. Probably due to the numerical. → resolved gas responsible of the main force fluctuations whereas for the dispersed phase, it is modelled.
- For case 1, EDF performed an inlet gas distribution and mesh sensitivity analysis from which it was found that the inlet bubble diameter should be at least 4 mm in order to have the correct force profile. For other cases, the sensitivity to the inlet gas distribution appeared to be much lower, probably due to the high gas flow rates leading to high coalescence and large gas structures.
- Triangular arrangement coupled to a slip between phases close to 1 are probably responsible for the stability of the flow and the agreement between simulation, since the flow is more restricted by the geometry than in a square arrangement.

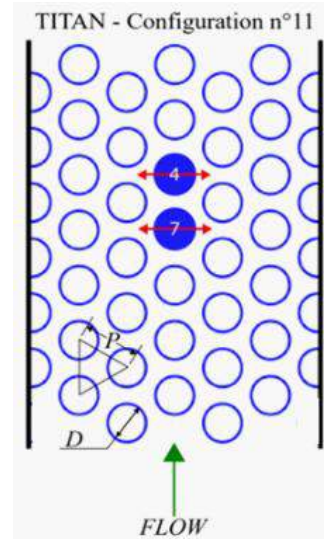
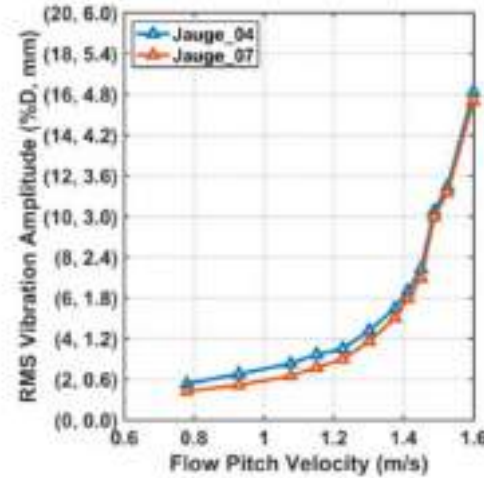
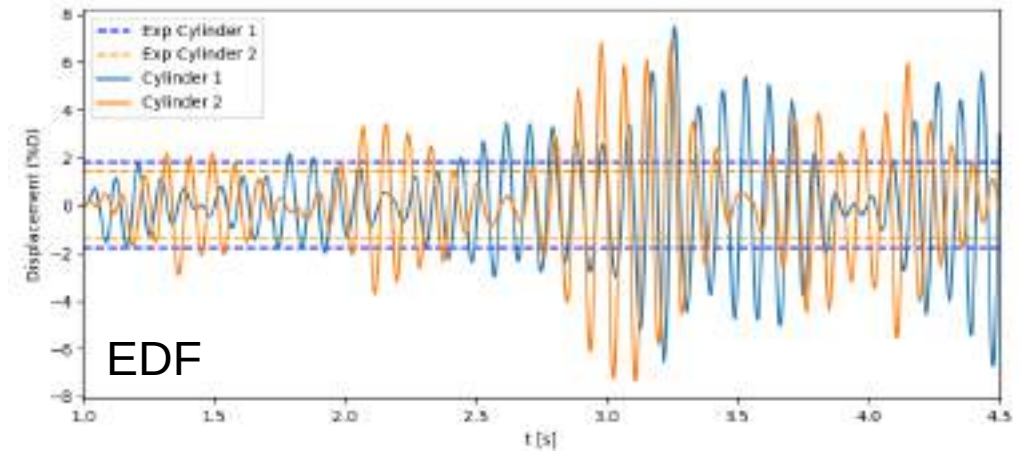
→ having a different inlet gas size distribution than the experiment appears to be less predominant after a couple of rows in the tube bundle than for the square arrangement.

→ Consequently, with 3 different numerical models and inlet conditions, having such similar results is even more interesting.

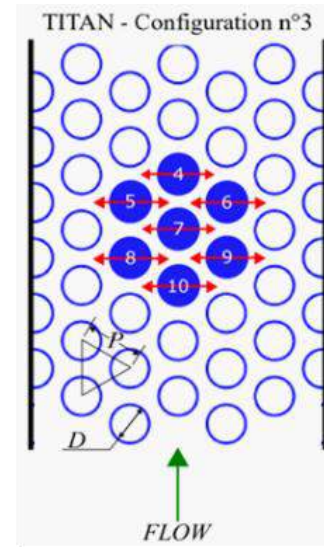
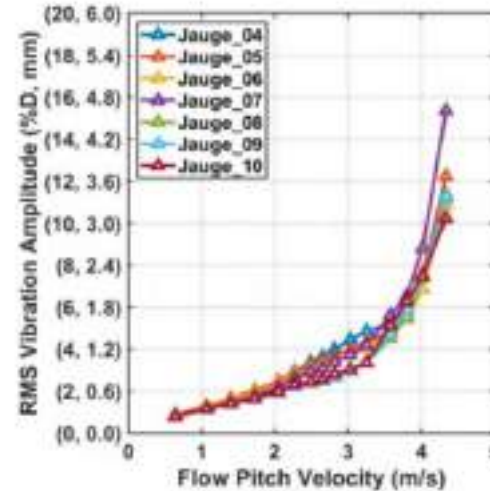
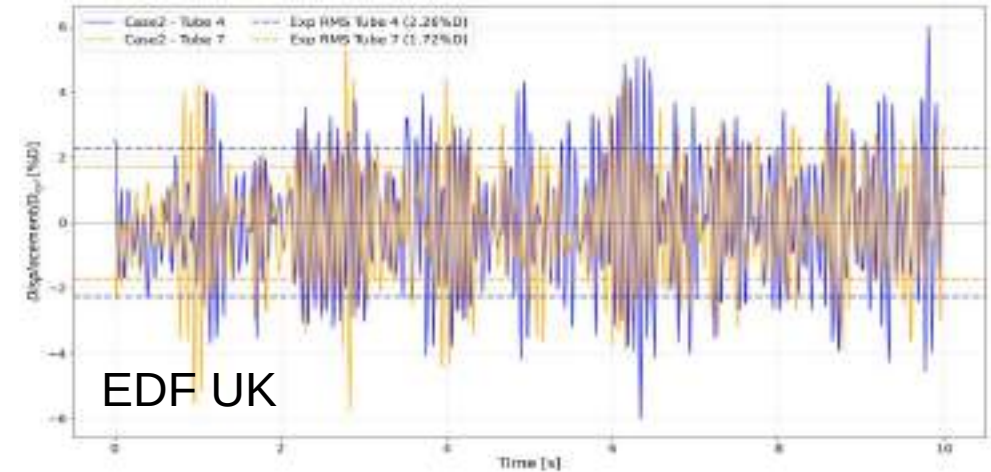
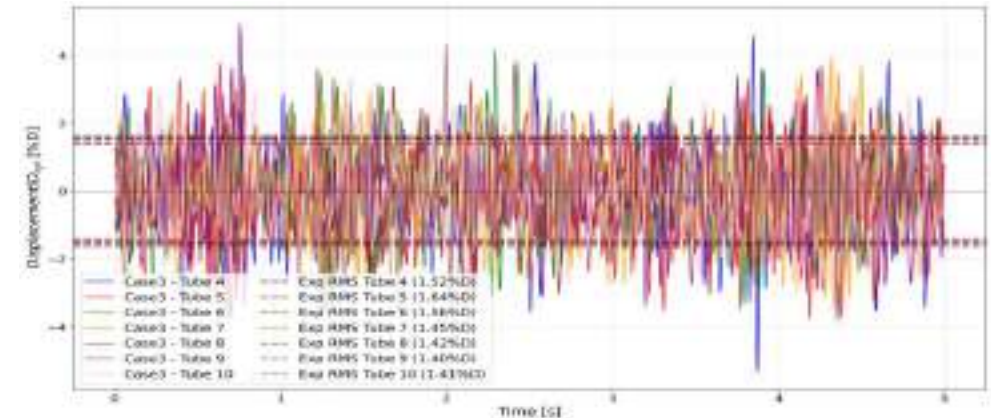


Numerical simulations

Conf. N°11 – 2 flexible tube with $f = 14\text{Hz}$



Conf. N°3 – 7 flexible tube with $f = 32\text{ Hz}$



Conclusions & Perspectives

- Variety of numerical models and calculation domains used, having such quantitative and qualitative similarities between simulations strengthens confidence in the numerical predictions.
- The triangular arrangement coupled to a slip between phases close to 1 are probably responsible of the stability of the flow and the agreement between simulations. The flow is more restricted by the triangular arrangement geometry than in a square arrangement. → having a different inlet gas size distribution than the experiment appears to be less predominant after a few rows in the tube bundle than for the square arrangement.
- the use of resolved approach for large gas structure appears to be the main parameter to get correct force spectra.
- The conclusions of the flexible configuration are in agreement and highlight that it is possible to predict two-phase flow-induced vibration in tube bundles. → To be continued..
- The Synthetic Bubble Model from UoGhent was tested by 3 partners in WP4, this kind of method appears of great interest for reducing the inlet calculation domain by having resolved structures at the inlet.



Thank you to the participants of the benchmark: Ehimen, Konstantinos, Daniele, Sina, Théo, Fanny, Samuel, Domenico and Romain

Communications



Analysis of cylinder force spectrum dependency to flow pattern, in a rotated triangular tube bundle subjected to air-water cross-flow. W. Benguigui & F. Beltran, FIV 2024.

Local-Scale Numerical Simulation of an Air-Water Flow in a Triangular Tube Bundle in to Predict Force Spectra, W. Benguigui & F. Beltran, PVP, 2025.

To come at Flow-Induced Vibration 2026 :

Numerical Investigation of Multiphase Flow-Induced Vibration in Triangular Tube Bundles Under Cross-Flow
Ehimen Iyamabo, Constantinos Katsamis, Ewan Harris

Two-phase churn-intermittent regime CFD simulations of air-water cross-flow in triangular array for the GO-VIKING project
Daniele Vivaldi, Joris Degroote

Euler-Euler Simulation of Flow-Induced Vibrations in Tube Bundles Subject to Two-phase Cross Flow. Sina Tajfirooz, Edo Frederix, Kevin Zwijsen

Partners



Associated Partners





GO-VIKING

Thank you!



William Benguigui



EDF



william.benguigui@edf.fr

www.go-viking.eu



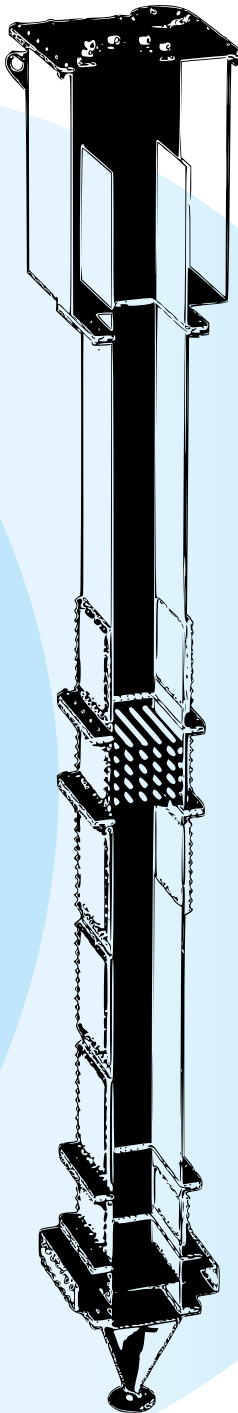
Funded by the
European Union

MAY 2025

TREFLE: EXPERIMENTAL SETUP AND CAMPAIGNS

GO-VIKING FINAL WORKSHOP

L. ROTILY – ASNR/SEREX/LETH



TREFLE EXPERIMENT OVERVIEW

TREFLE Characteristics :

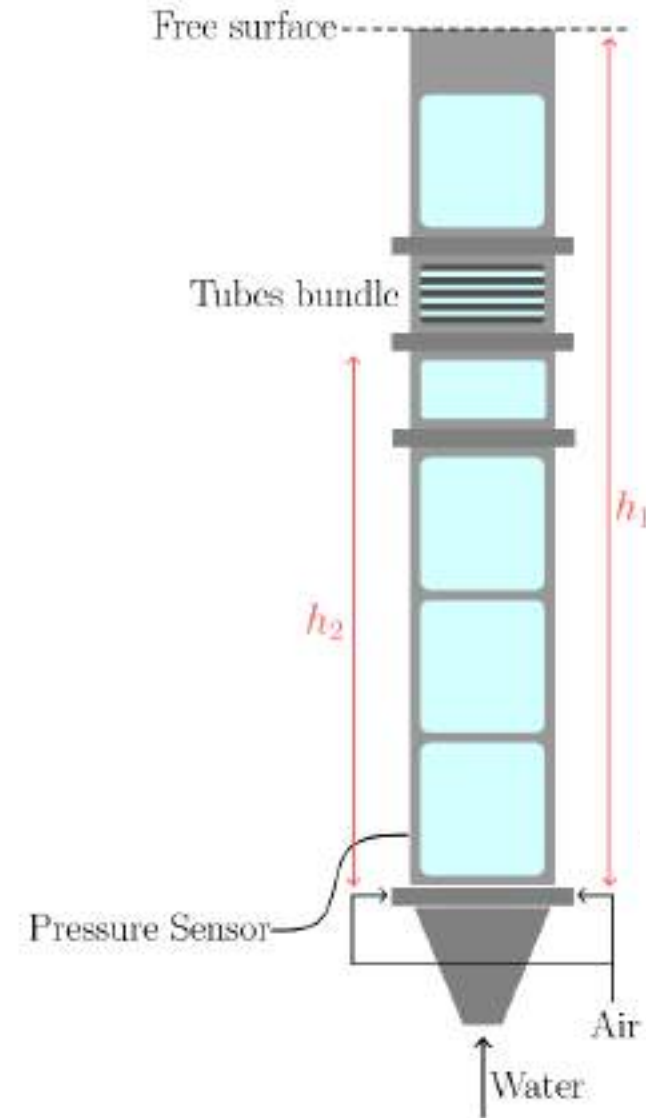
- Closed two phase loop air/water
- Ambient temperature
- Atmospheric pressure

TREFLE Dimensions :

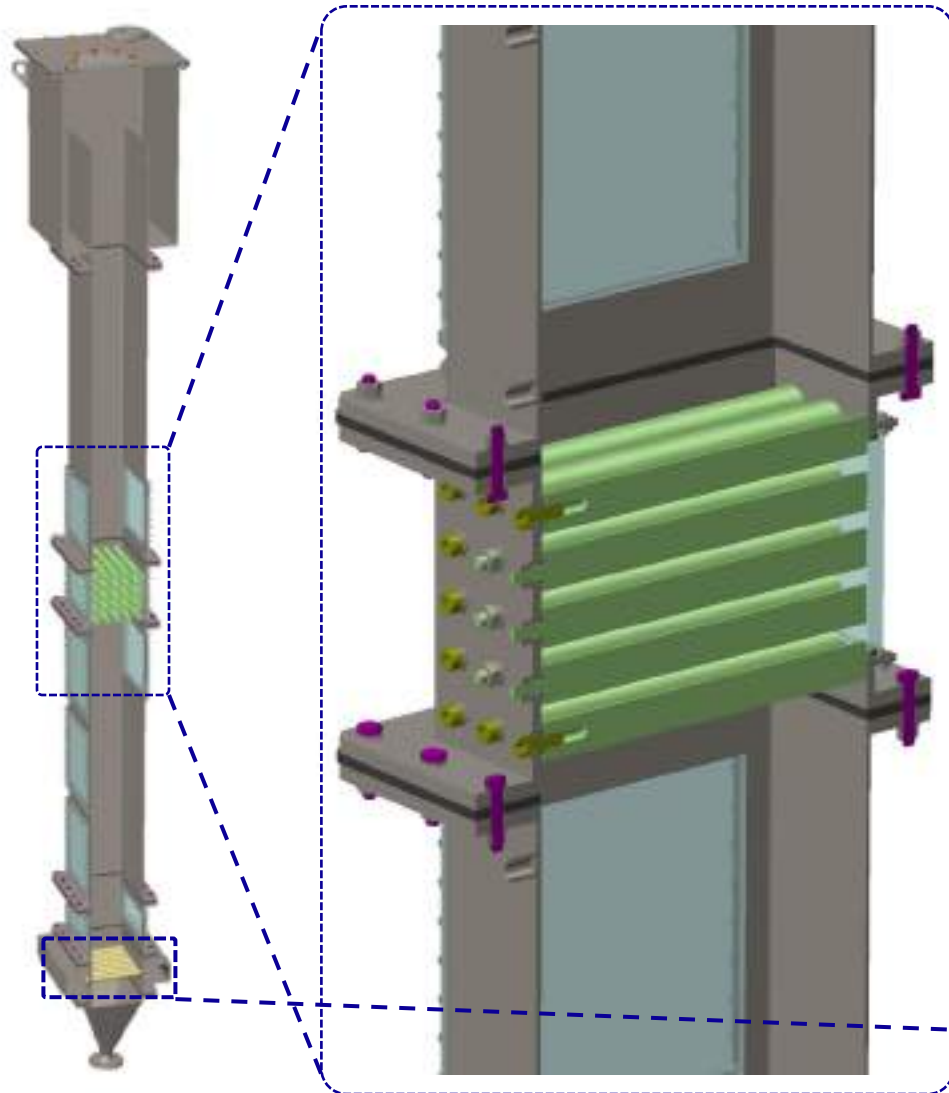
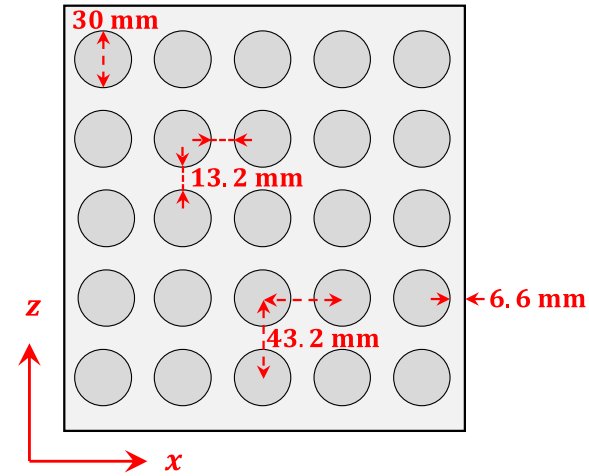
- Squared section length = 308 mm
- Squared section width = 224 mm
- Total water height $h_1 = 3.5$ m
- Homogenization height $h_2 = 1.4$ m

EXISTING LITTERATURE

- Chahine, A., Vivaldi, D., & Brillant, G. (2025). *New experimental insights into two-phase air–water bubbly and intermittent flow in large duct and cross-flow configurations*. International Journal of Multiphase Flow, 105266.
- Spina, G. (2023). *Innovative Insights into Experimental Investigation of Two-Phase Cross-Flow Induced Vibration and Numerical Simulation of Complex Two-Phase Flow Scenarios* (Doctoral dissertation, Institut National Polytechnique de Toulouse-INPT)
- G. Spina, W. Benguigui, G. Brillant, D. Vivaldi, C. Colin Frederix, A.H. van Zuijlen, *Local-scale experimental investigation of a two-phase cross-flow in a tube bundle and flow-induced vibration; Bubbly flow regime* - Nuclear Engineering and Design, Volume 433, March 2025, 113836 – ScienceDirect doi: [10.1016/j.nucengdes.2025.113836](https://doi.org/10.1016/j.nucengdes.2025.113836)
- Vivaldi, D, Chahine, A, & Rotily, L. "Experimental Study of the Two-Phase Flow and Induced Vibrations in a Square-Pitch Tube Bundle in Air-Water Cross-Flow." *Proceedings of the . Volume 4: Fluid-Structure Interaction; High Pressure Technology*. Montreal, Quebec, Canada. July 20–25, 2025. V004T04A018. ASME. <https://doi.org/10.1115/PVP2025-153890>

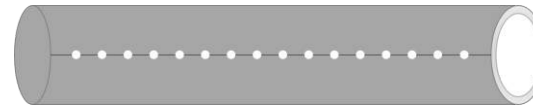


TREFLE EXPERIMENT OVERVIEW

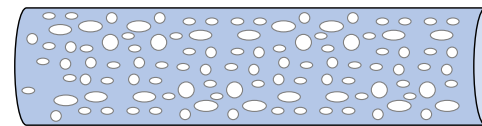


- $P/D = 0.44$
- 5x5 square tube bundle
- 7 air injectors
- Dimensions :
 $L = 308 \text{ mm}$
 $W = 224 \text{ mm}$

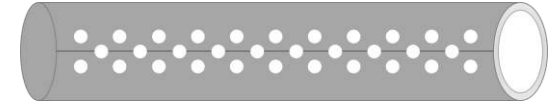
Type 1 : $0.1 < \dot{m}_g < 5 \text{ g/s}$



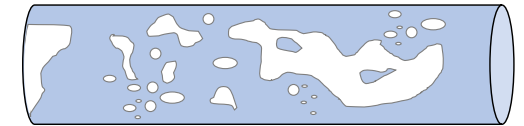
Bubbly flow



Type 2 : $5 < \dot{m}_g < 35 \text{ g/s}$



Intermittent flow



SUMMARY

01

**A WELL DEFINED AND
CHARACTERIZED TWO PHASE FLOW**

02

VIBRATIONS OF FLEXIBLE TUBES

03

FORCES ON FIXED TUBE

04

FUTURE WORK ON TREFLE

01

A WELL DEFINED AND CHARACTERIZED TWO PHASE FLOW

BUBBLES VISUALISATION

$$Q_{air} = 0.5 \text{ g/s}$$

W4



W3



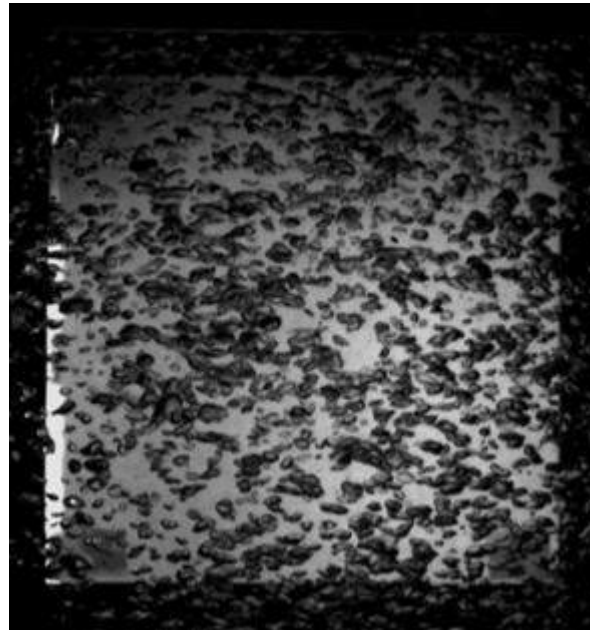
W2



W3



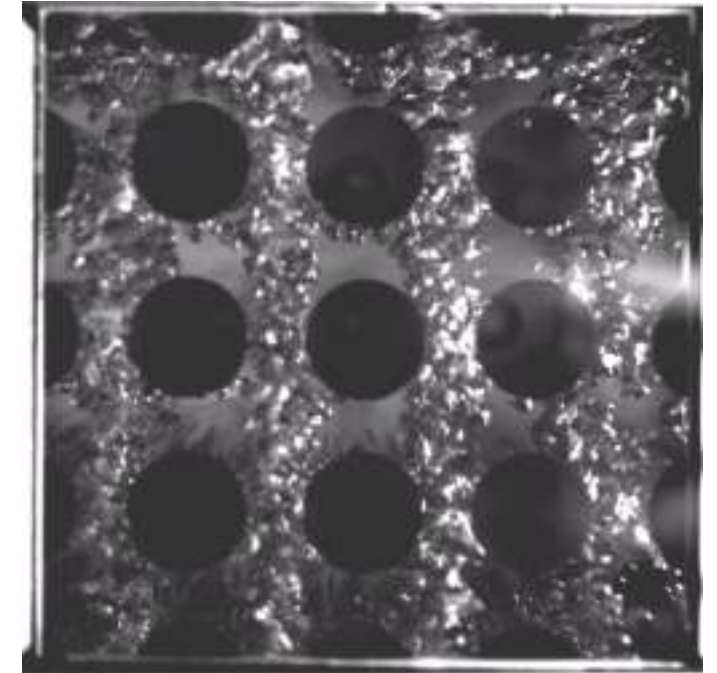
W2



W5



W5



BUBBLES VISUALISATION

$$Q_{air} = 5 \text{ g/s}$$

W4



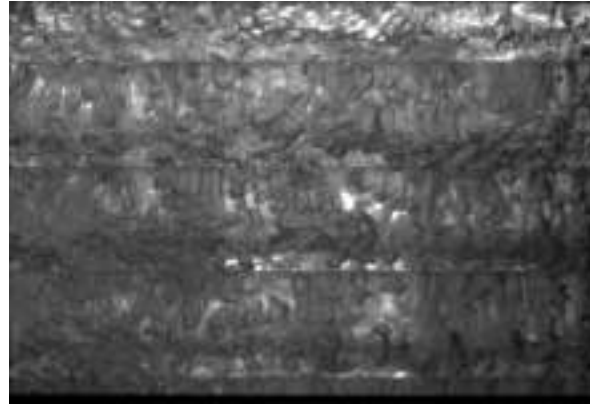
W3



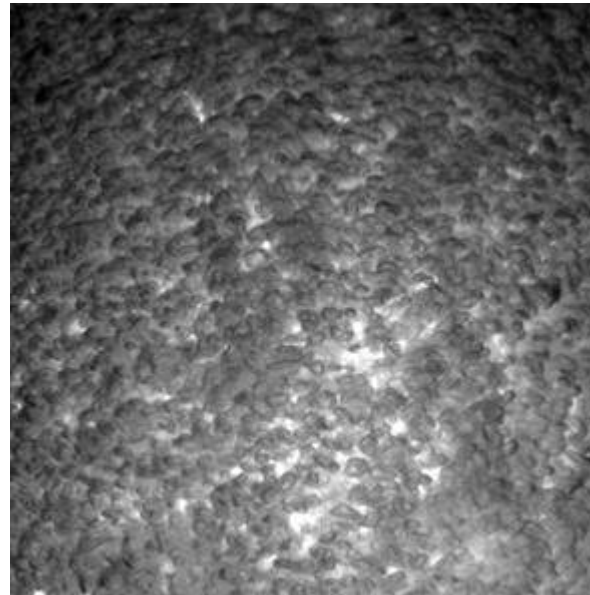
W2



W3



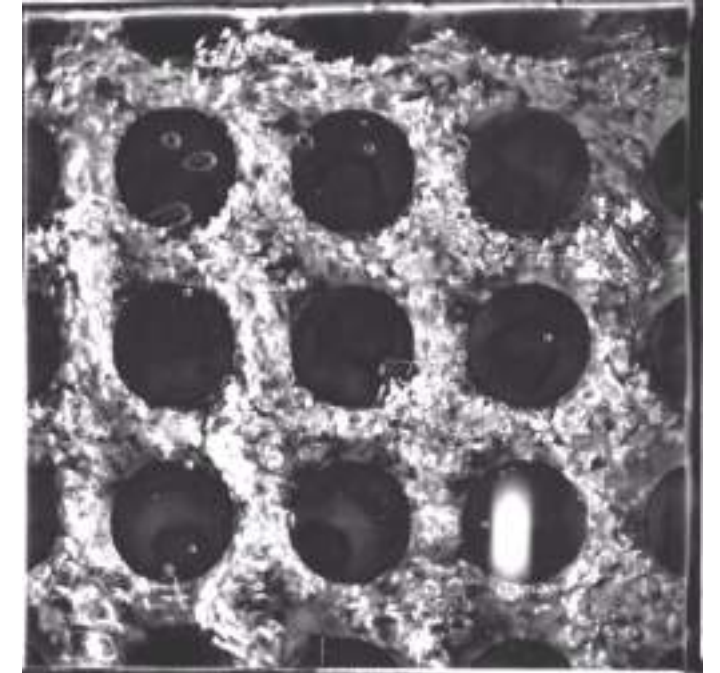
W2



W5



W5

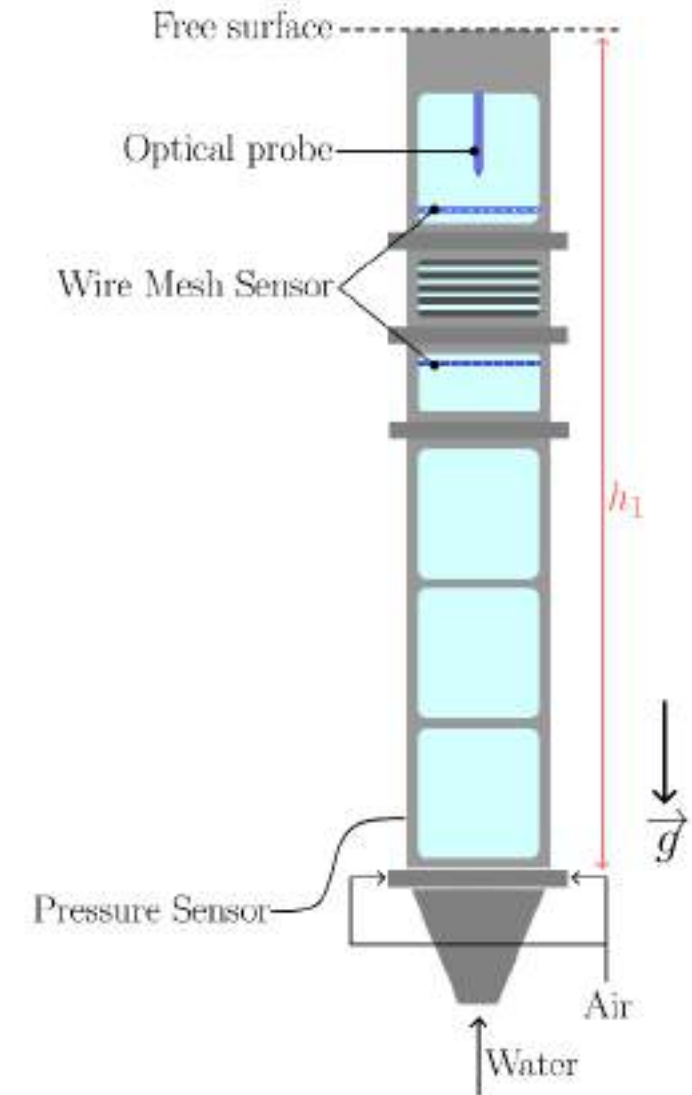


TREFLE INSTRUMENTATION FOR FLOW CHARACTERISATION

PRESSURE SENSOR

WIRE MESH SENSOR

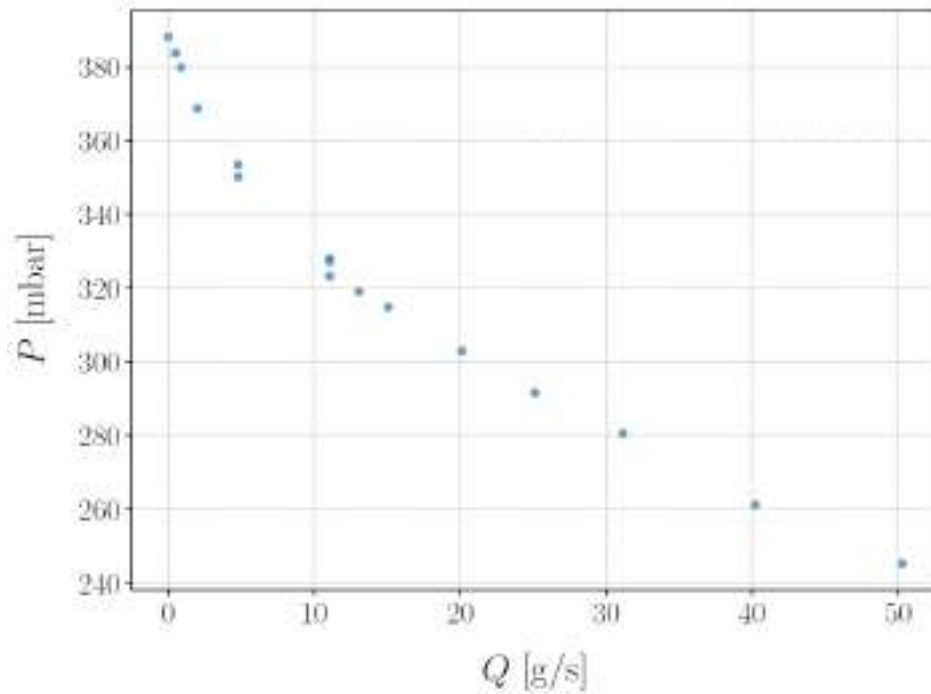
OPTICAL PROBE



TREFLE INSTRUMENTATION FOR FLOW CHARACTERISATION

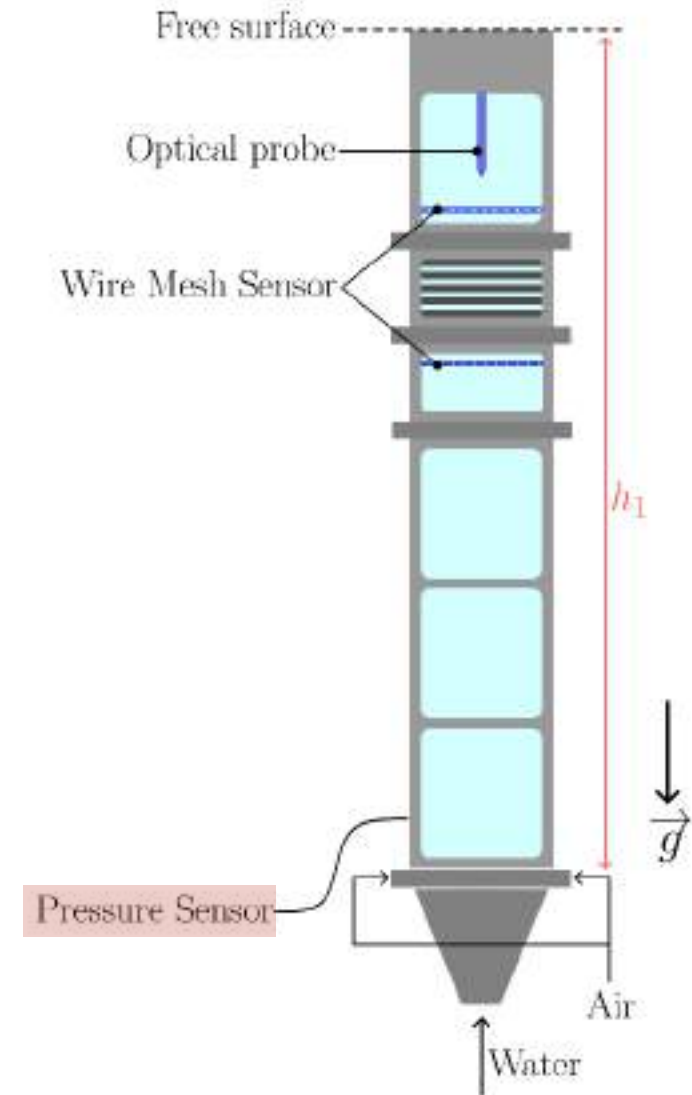
PRESSURE SENSOR

Differential pressure sensor at the bottom of the section for measurement of total gas volume in the test section



WIRE MESH SENSOR

OPTICAL PROBE



TREFLE INSTRUMENTATION FOR FLOW CHARACTERISATION

PRESSURE SENSOR

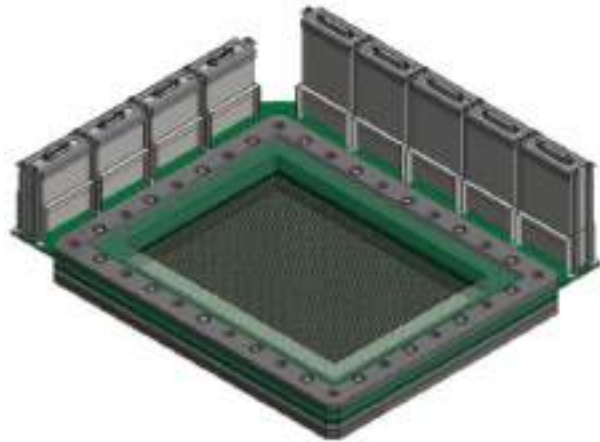
WIRE MESH SENSOR

Dimension of TREFLE section

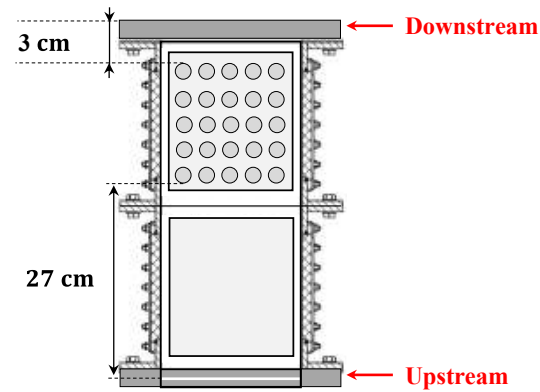
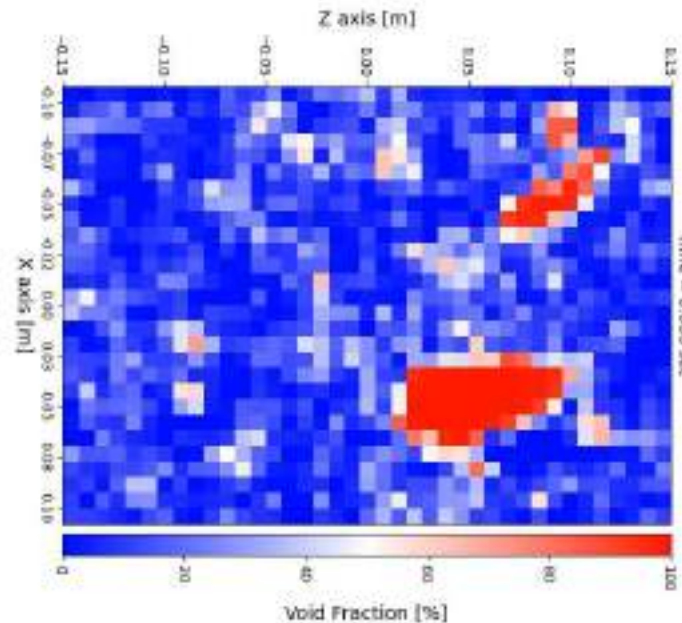
78x56 wires

Diam. wire 0.2 mm

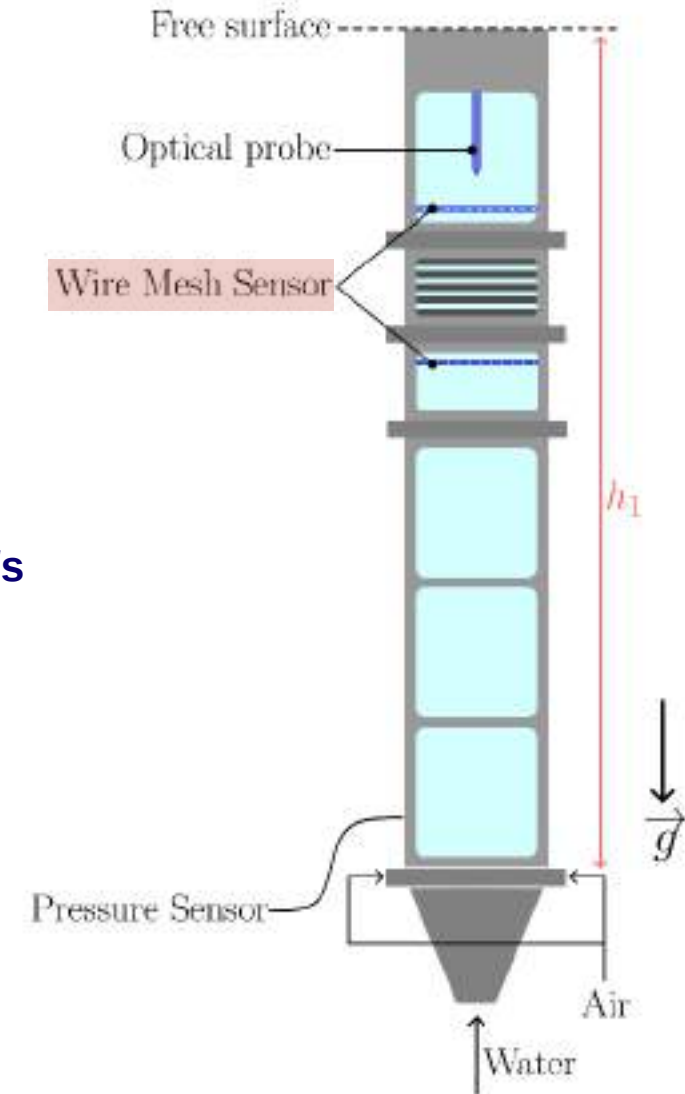
Acquisition frequency: $f=1000$ Hz



OPTICAL PROBE



$Q = 11$ g/s

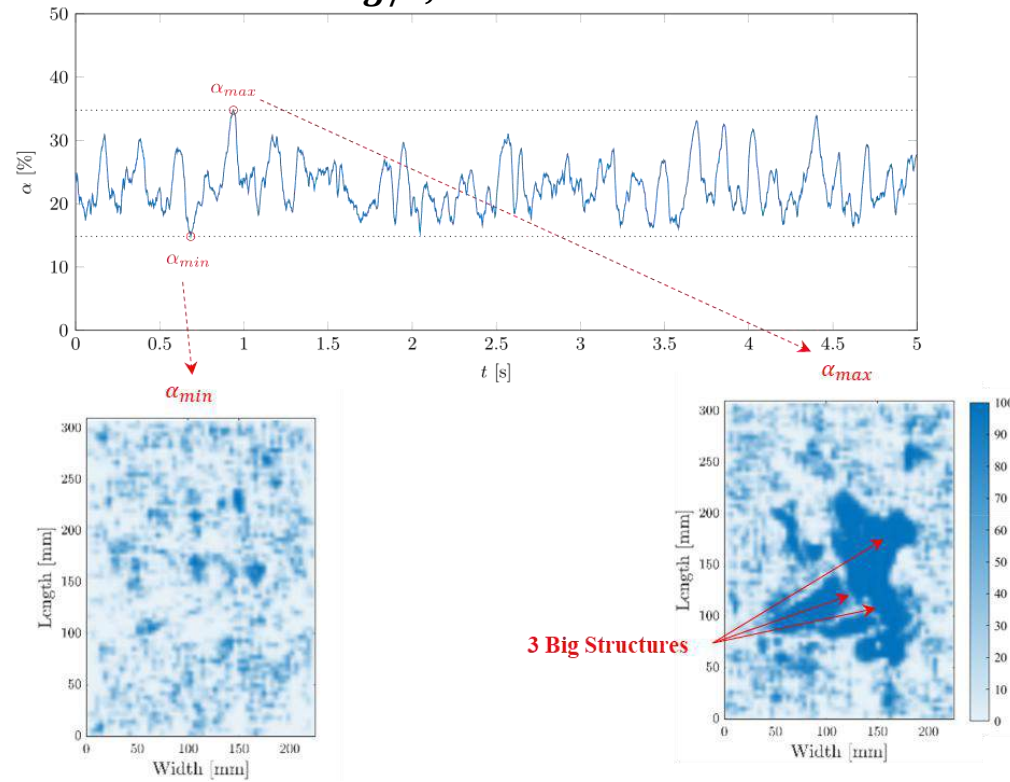


TREFLE INSTRUMENTATION FOR FLOW CHARACTERISATION

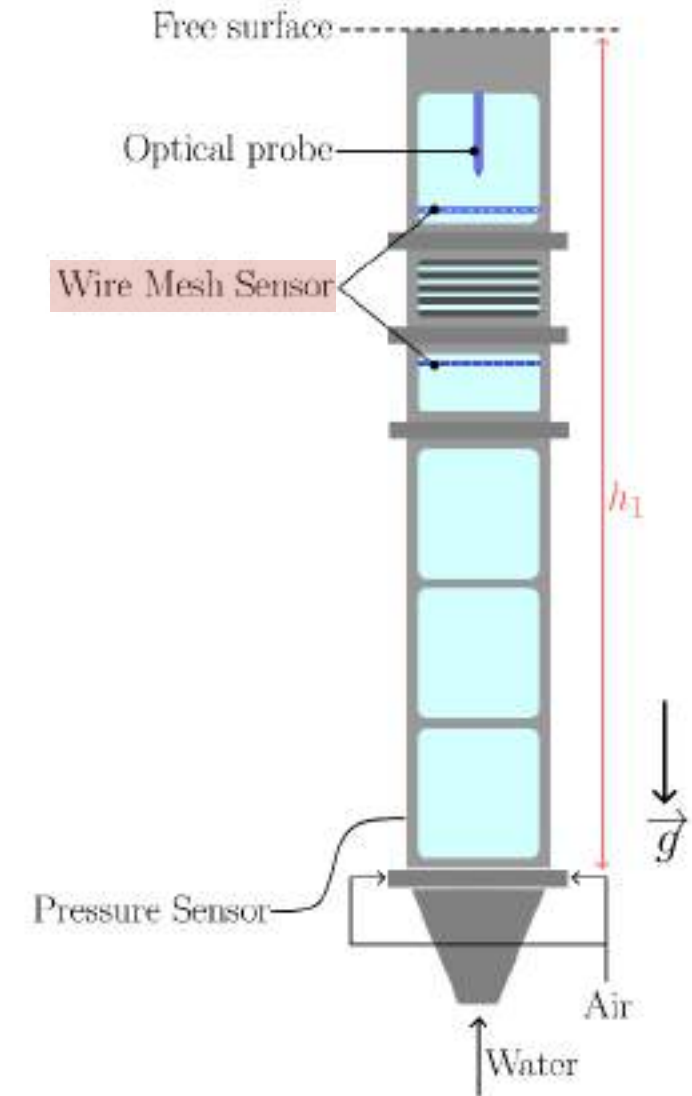
PRESSURE SENSOR

WIRE MESH SENSOR

Void fraction over time for air flowrate
11 g/s, Intermittent flow



OPTICAL PROBE

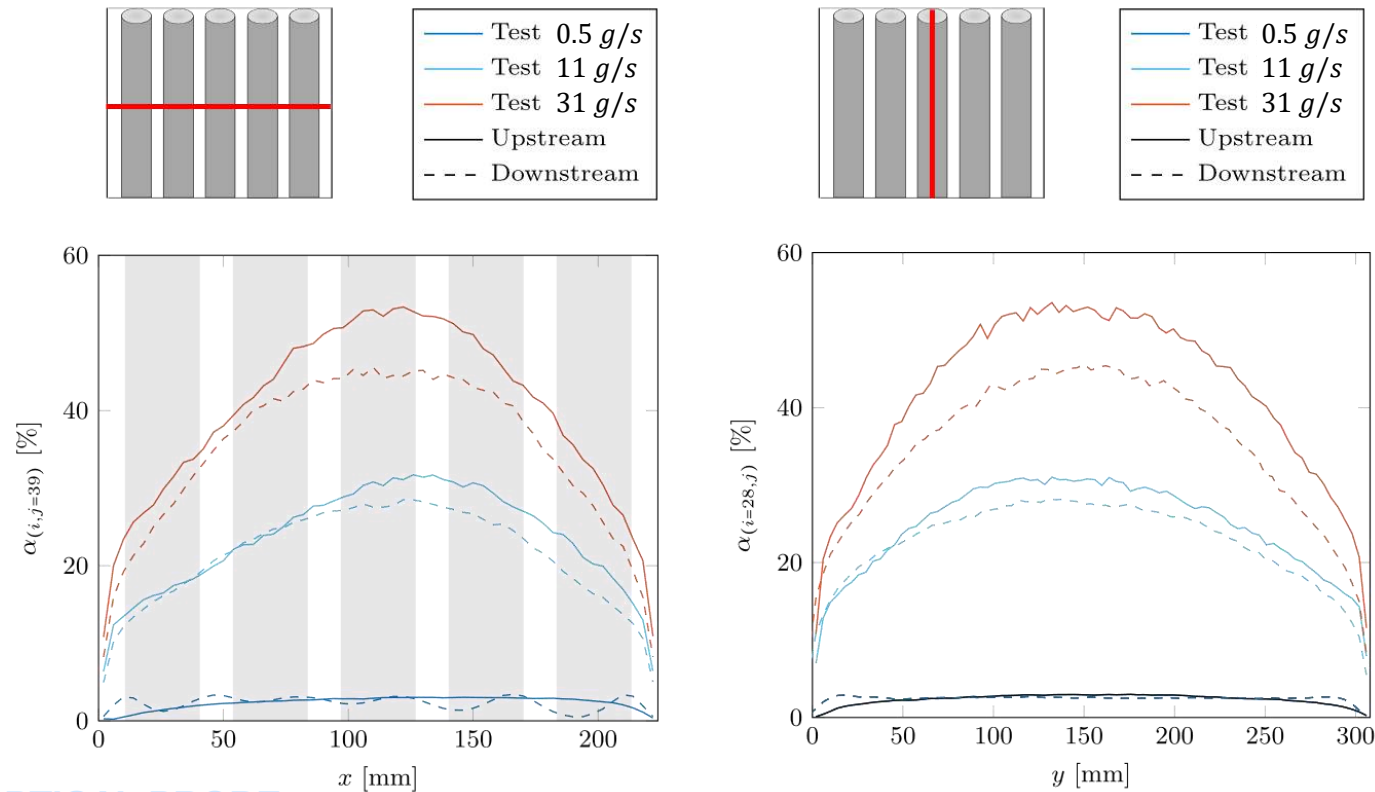


TREFLE INSTRUMENTATION FOR FLOW CHARACTERISATION

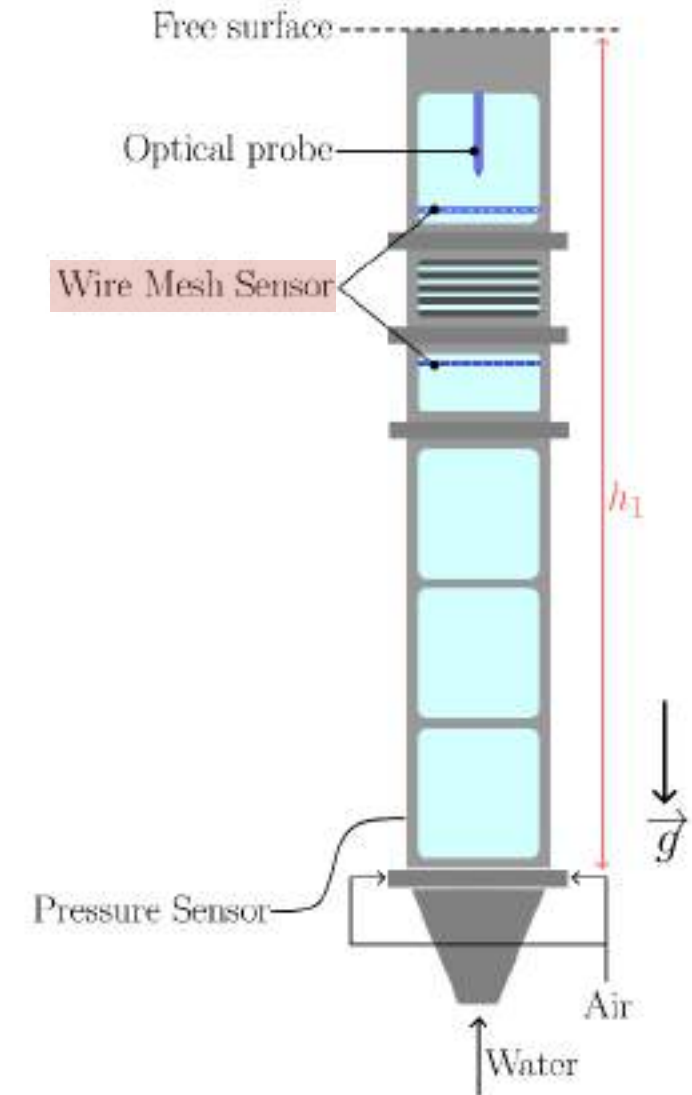
PRESSURE SENSOR

WIRE MESH SENSOR

Void fraction profiles



OPTICAL PROBE

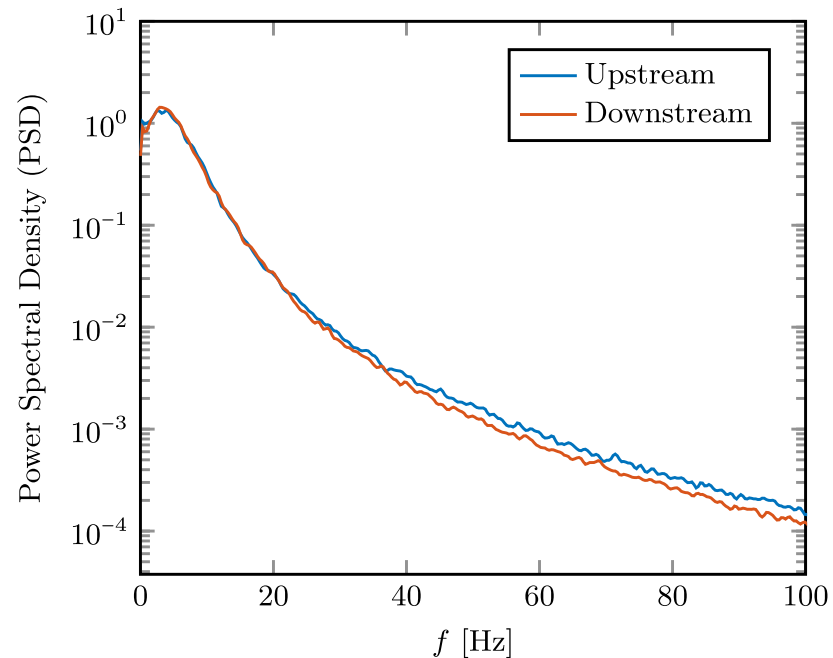
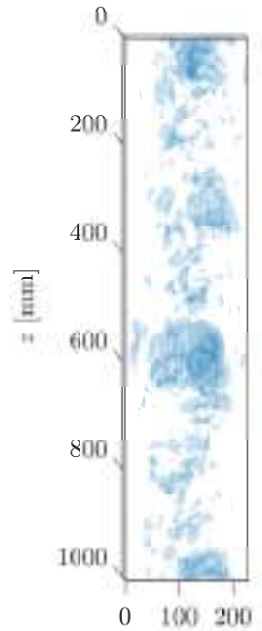


TREFLE INSTRUMENTATION FOR FLOW CHARACTERISATION

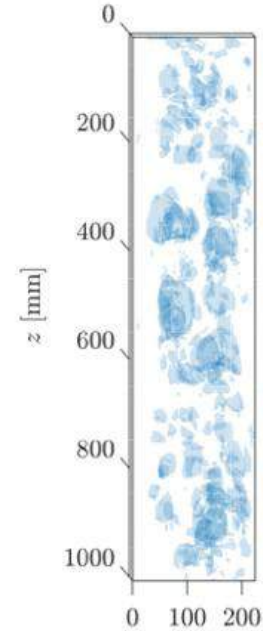
PRESSURE SENSOR

WIRE MESH SENSOR

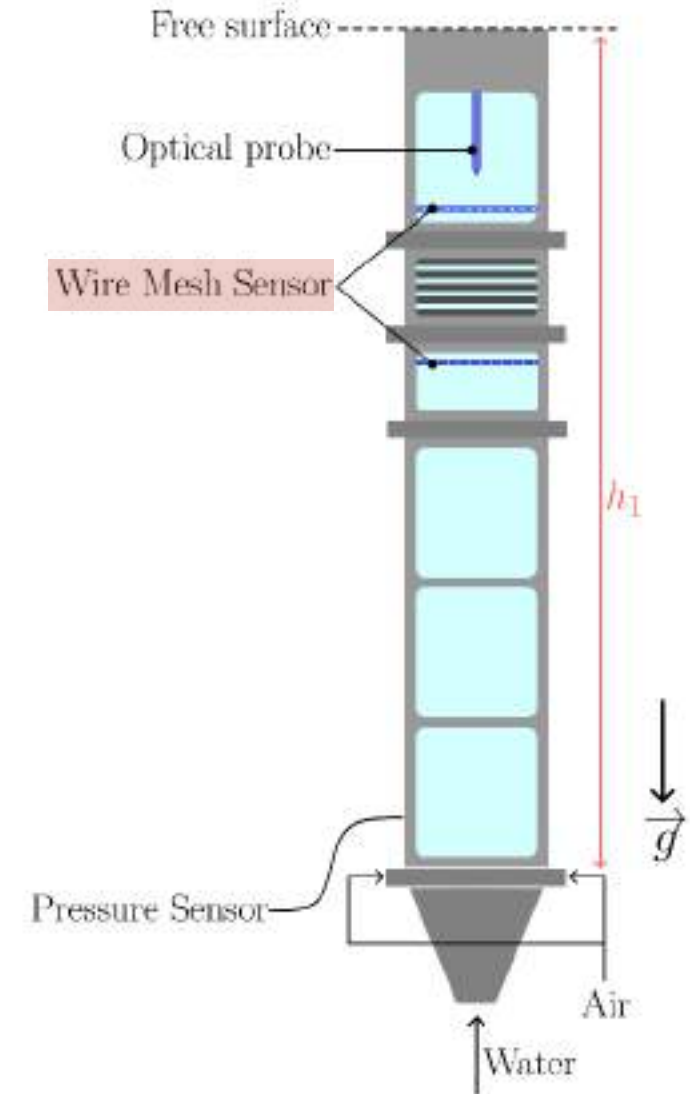
Void fraction frequency



Upstream



Downstream



OPTICAL PROBE

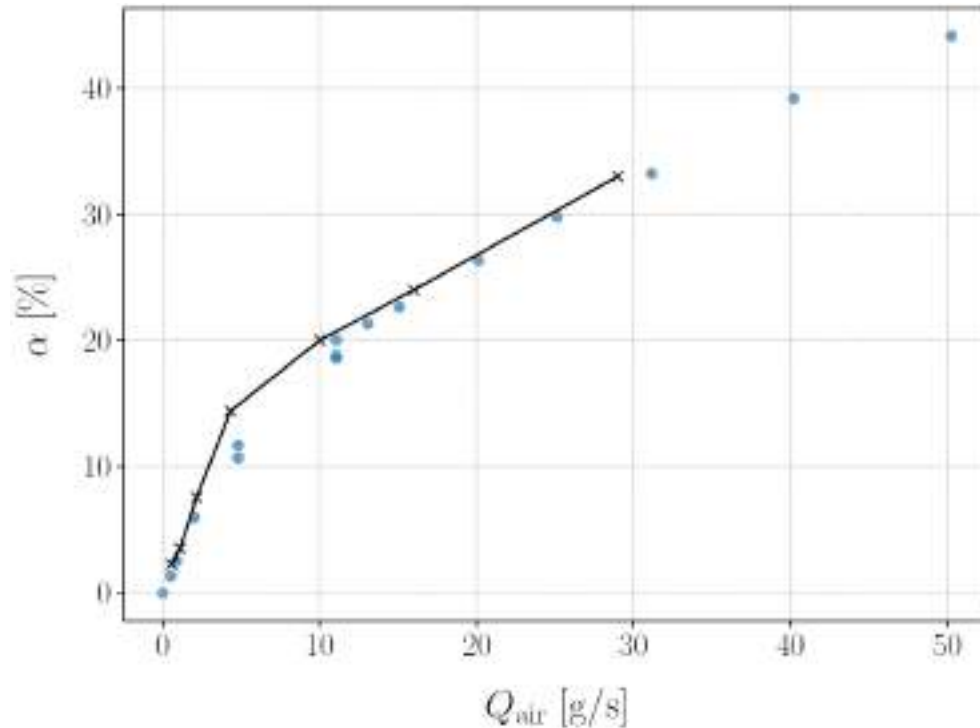
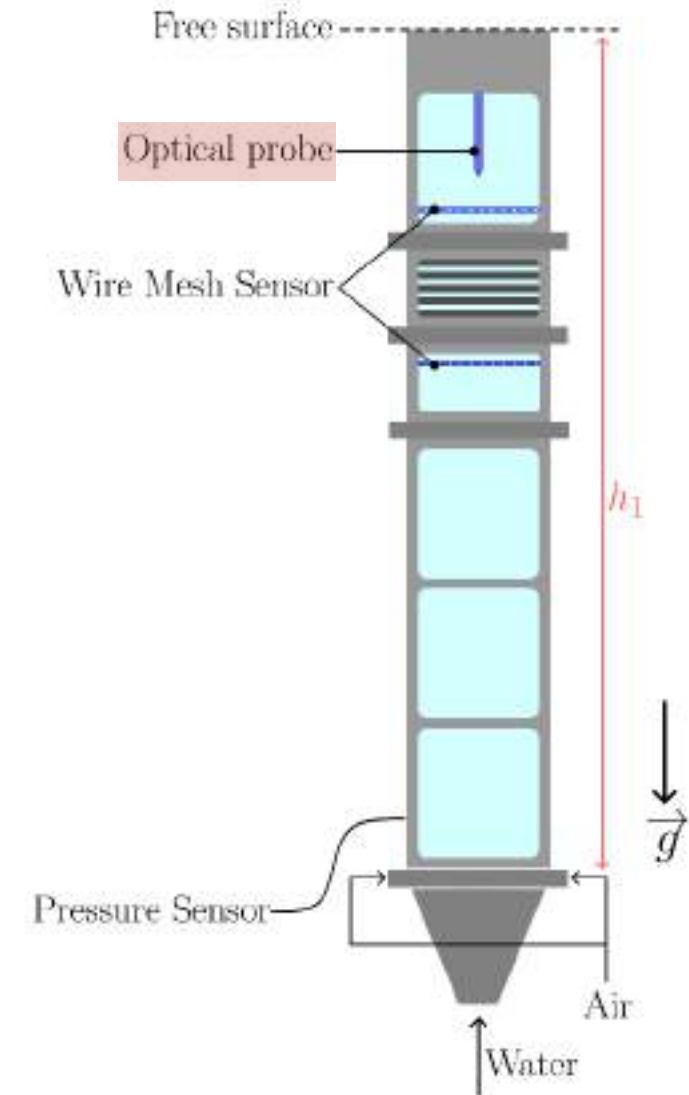
TREFLE INSTRUMENTATION FOR FLOW CHARACTERISATION

PRESSURE SENSOR

WIRE MESH SENSOR

OPTICAL PROBE

Optical probe is a dual tip probe, with results matching global void measurement in the bulk of TREFLE from pressure sensor.

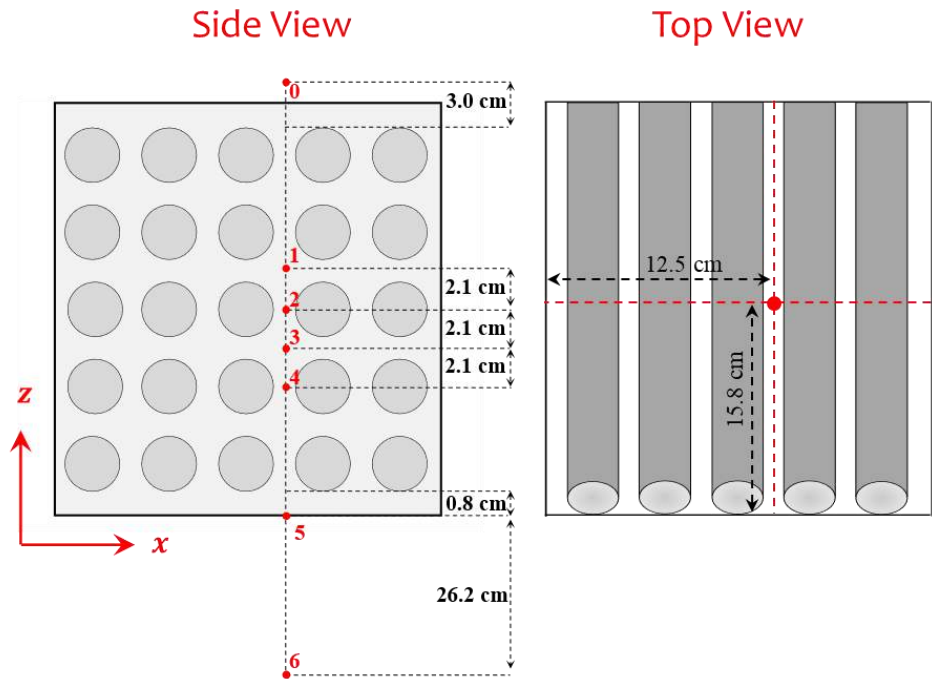


TREFLE INSTRUMENTATION FOR FLOW CHARACTERISATION

PRESSURE SENSOR

WIRE MESH SENSOR

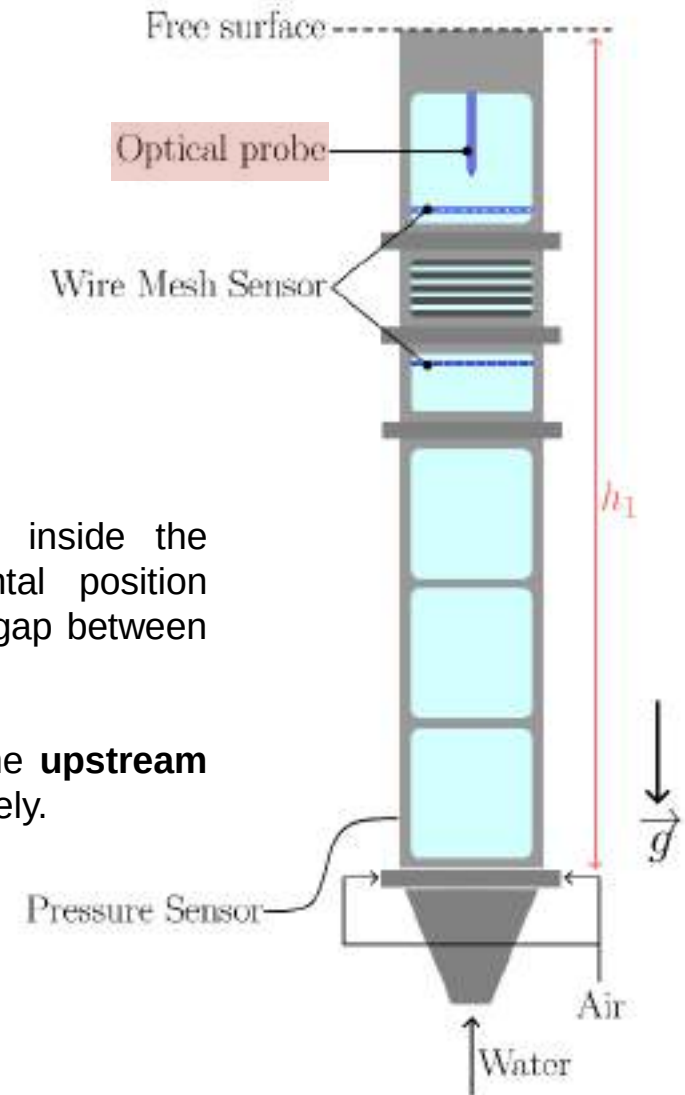
OPTICAL PROBE



The dual optic probe was fixed inside the experimental setup at a horizontal position corresponding to the middle of the gap between the 3rd and the 4th columns.

Side view:

Position 6 and 0 corresponds to the **upstream** and **downstream** of WMS respectively.

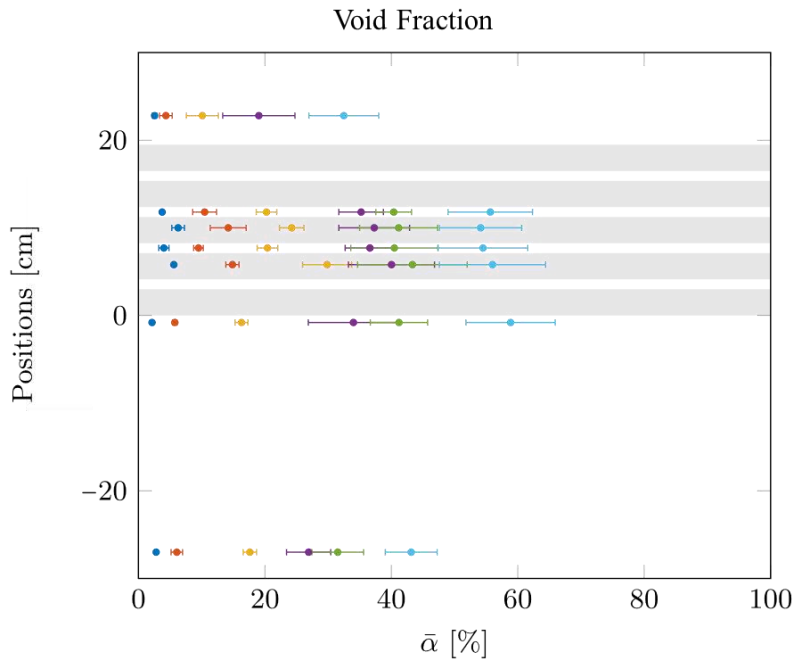
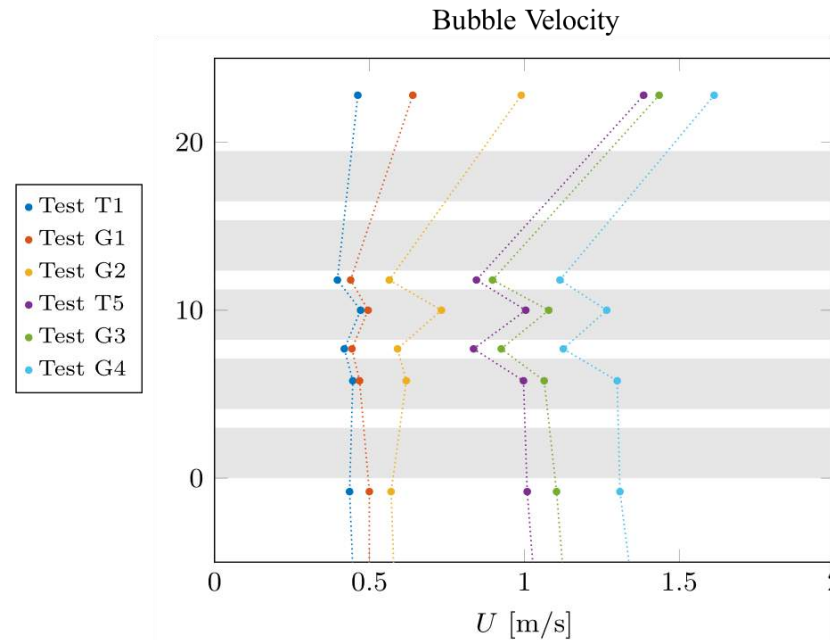
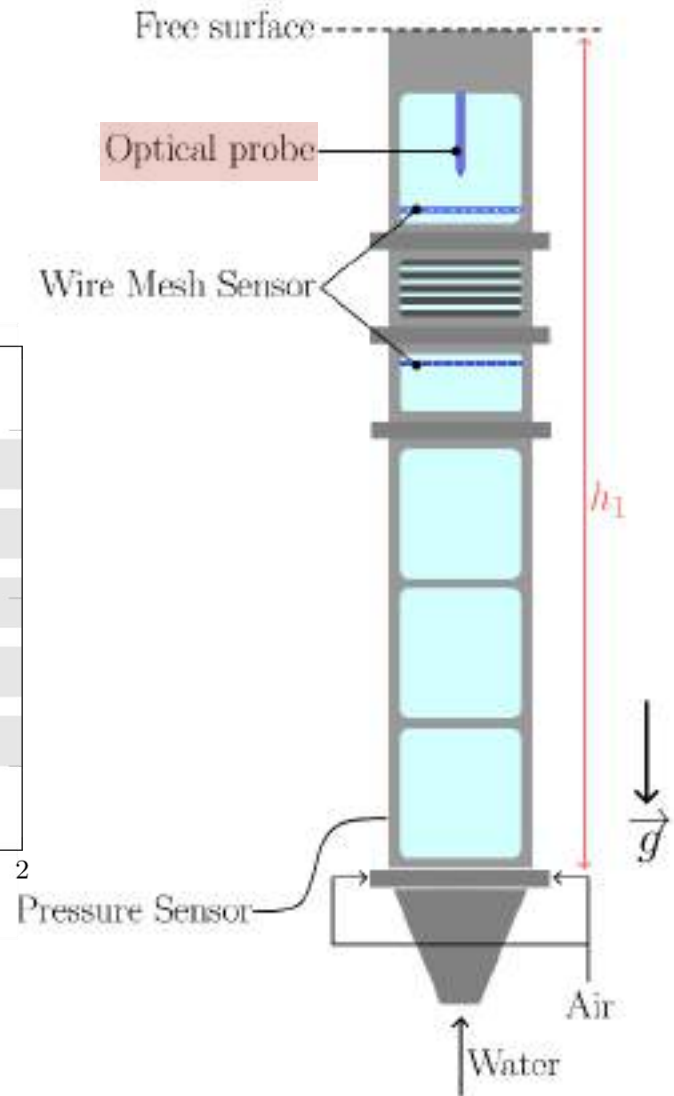


TREFLE INSTRUMENTATION FOR FLOW CHARACTERISATION

PRESSURE SENSOR

WIRE MESH SENSOR

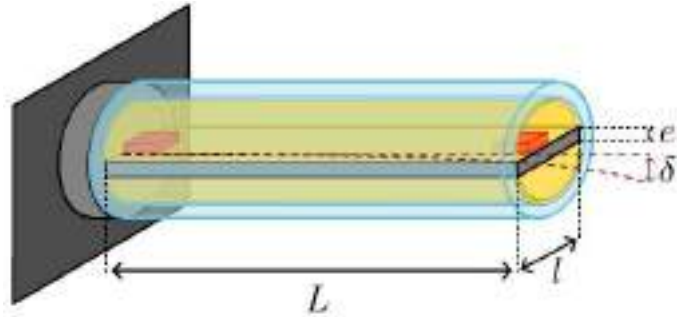
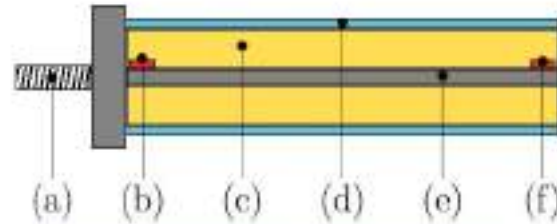
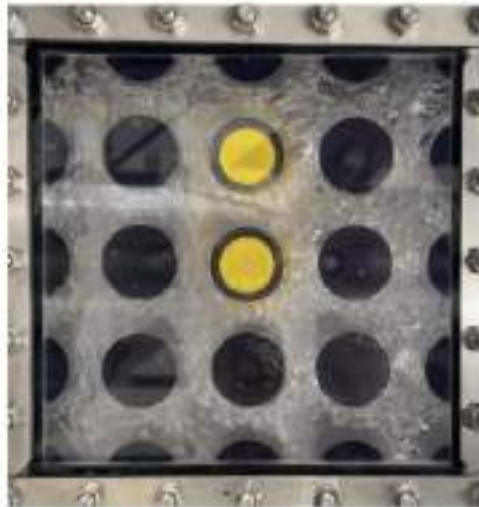
OPTICAL PROBE



02

VIBRATIONS OF FLEXIBLE TUBES

FLEXIBLE TUBE VIBRATING UNDER HYDRODYNAMIC FORCES



(a) Screw to fix in TREFLE section

(b) Strain sensor on the metallic blade

(c) Half cylinder plastic spacer

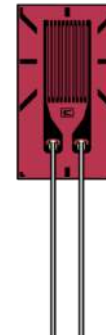
(d) Transparent hollow tube

(e) Metallic blade of approx. 30 cm

(f) Accelerometer

Three displacement measurement methods:

Strain sensor



$$\frac{\Delta E}{V} = S_g \cdot \varepsilon$$

with S_g pre-calibrated gauge factor

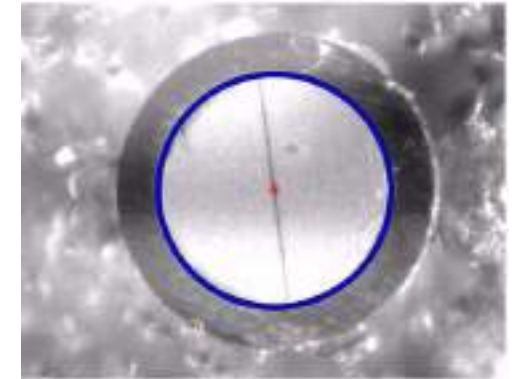
$$\delta = \frac{2 \varepsilon L^2}{3 e}$$

Accelerometer



$$x = \iint a \, dt^2$$

Fast camera



Immediate displacement measurement by tube center tracking

FLEXIBLE TUBE VIBRATING UNDER HYDRODYNAMIC FORCES

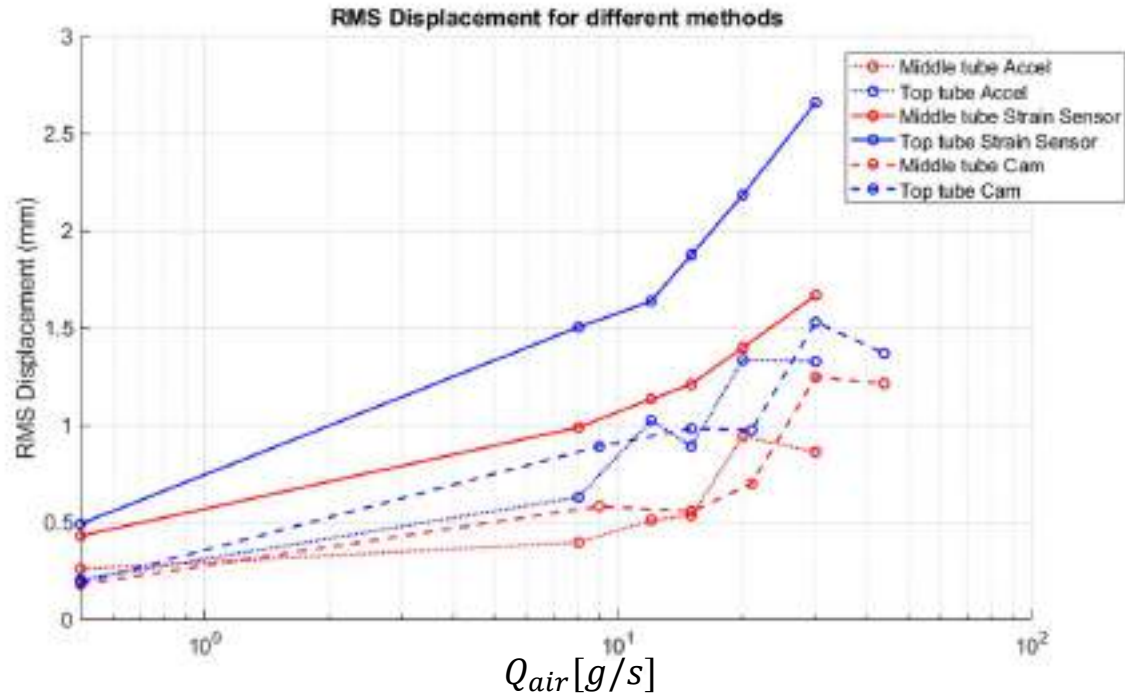
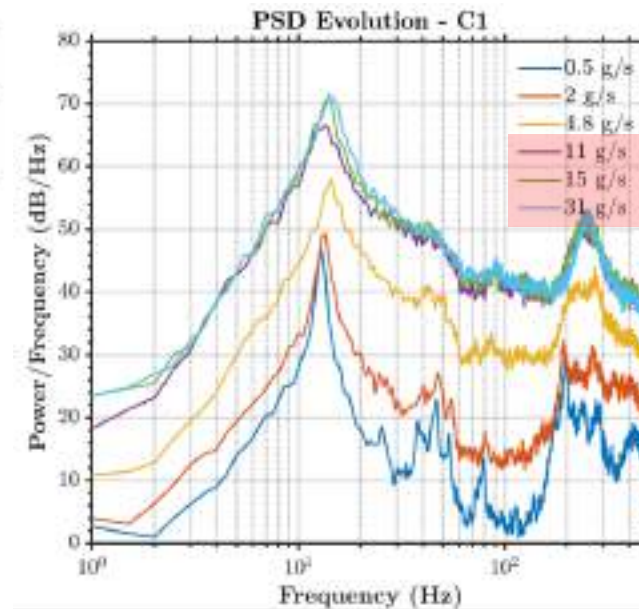
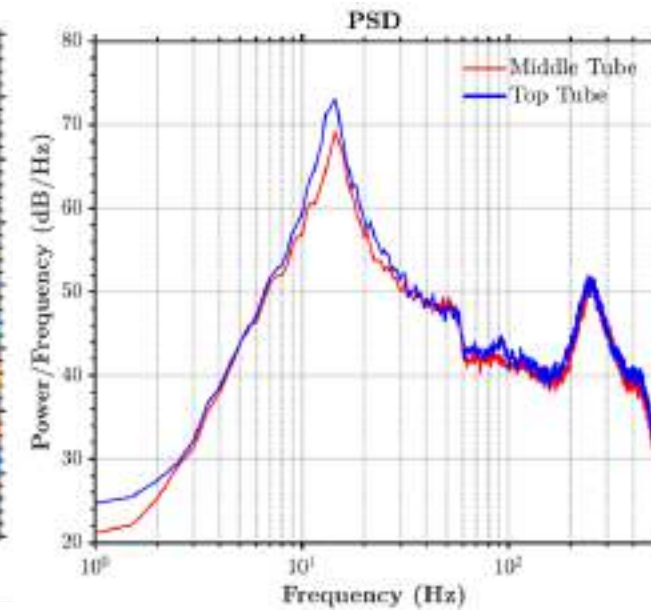
KEY CONCLUSIONS OF PREVIOUS WORK

Displacement amplitude grows with increasing air flowrate

Downstream tube exhibits higher displacement amplitude than upstream tube

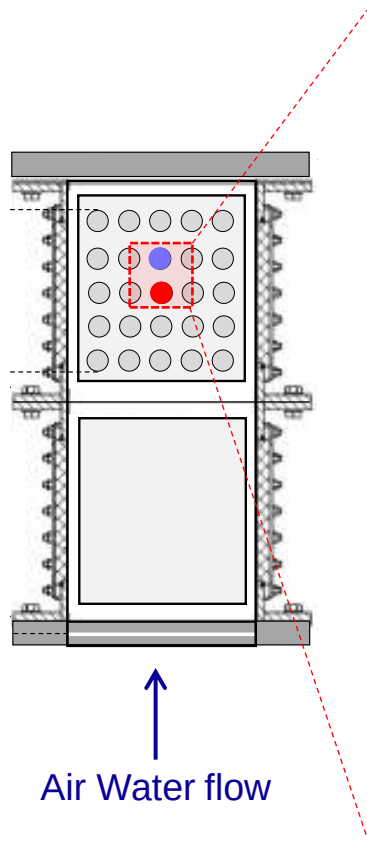
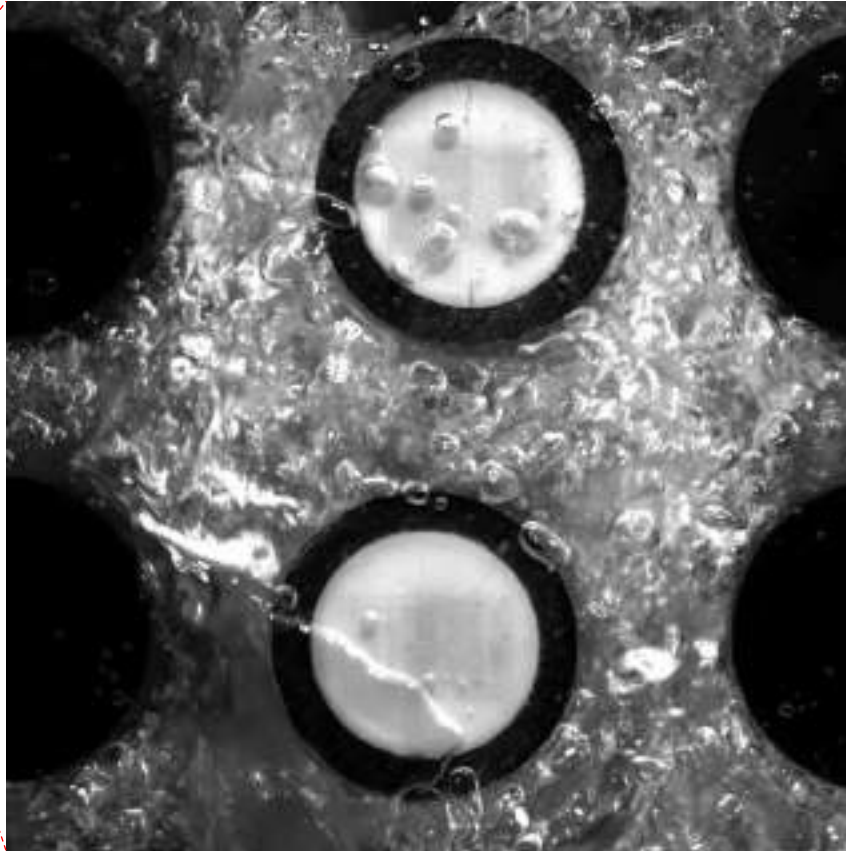
PSD amplitude increases with flowrate, drastically first in bubbly flow until reaching almost saturation in intermittent regimes

Both downstream and upstream tubes exhibit similar frequencies, dominantly at their natural frequency



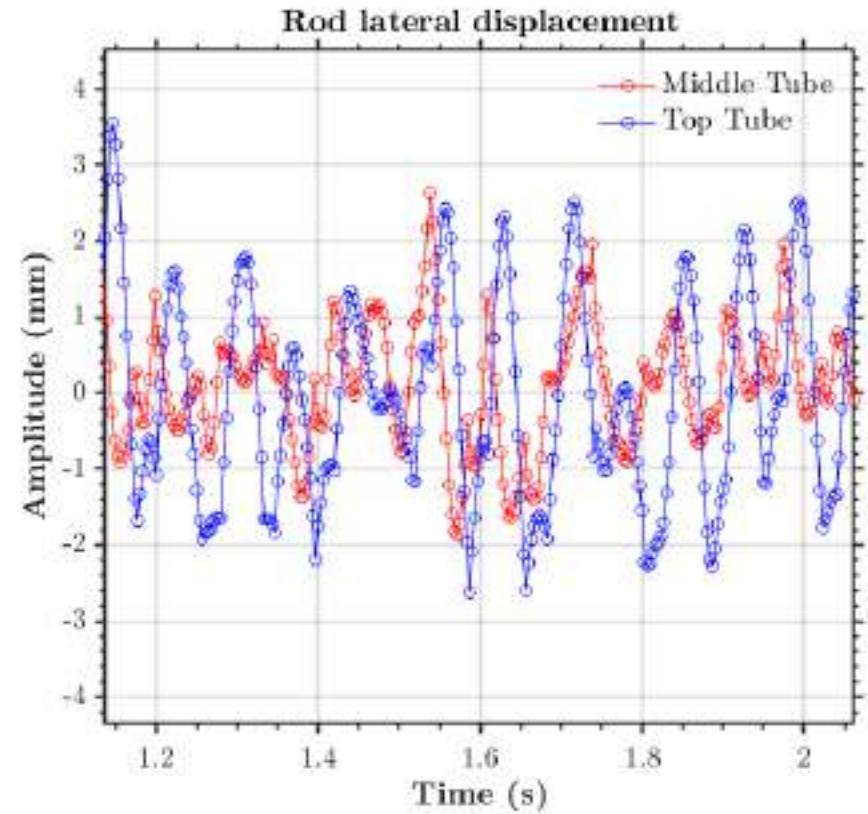
$$Q_{air} = 20 \text{ g/s}$$

FLEXIBLE TUBE VIBRATING UNDER HYDRODYNAMIC FORCES



Tube diameter 3 cm, FPS
300 (at play fps 30)

$$Q_{air} = 30 \text{ g/s}$$

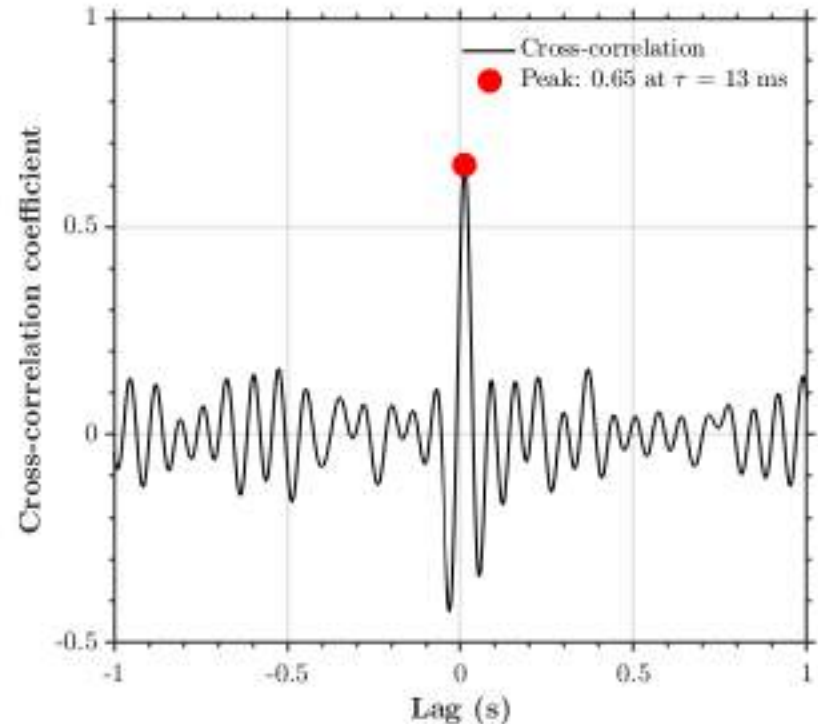
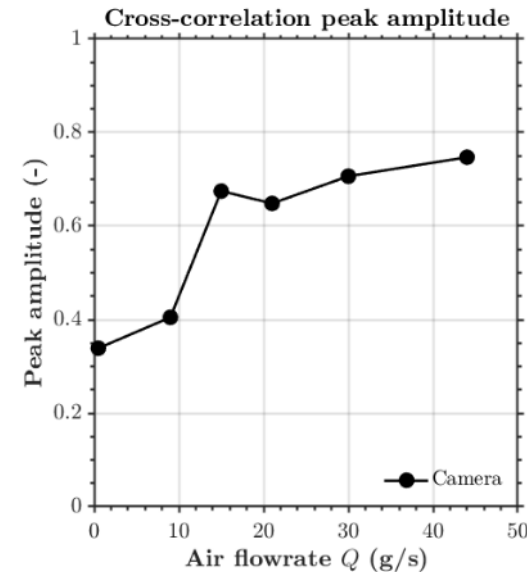
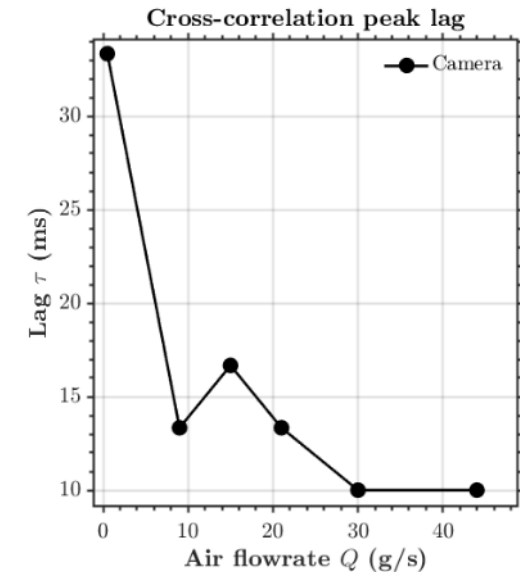


CROSS CORRELATIONS OF UPSTREAM AND DOWNSTREAM TUBES

$$R_{xy}(\tau) = \frac{\sum_t x(t) y(t + \tau)}{\sqrt{\sum_t x(t)^2 \cdot \sum_t y(t)^2}}$$

$x(t)$ = signal upstream
 $y(t)$ = signal downstream
 τ = temporal lag
 $R_{xy}(\tau) \in [-1,1]$ with 1 for total correlation

Example of cross correlation for $Q_{air} = 21 \text{ g/s}$

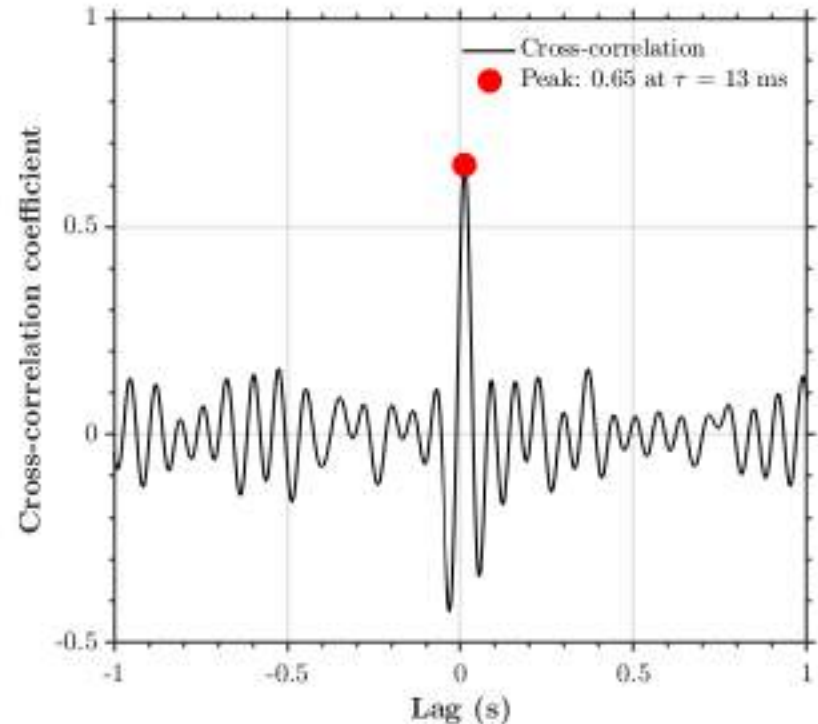
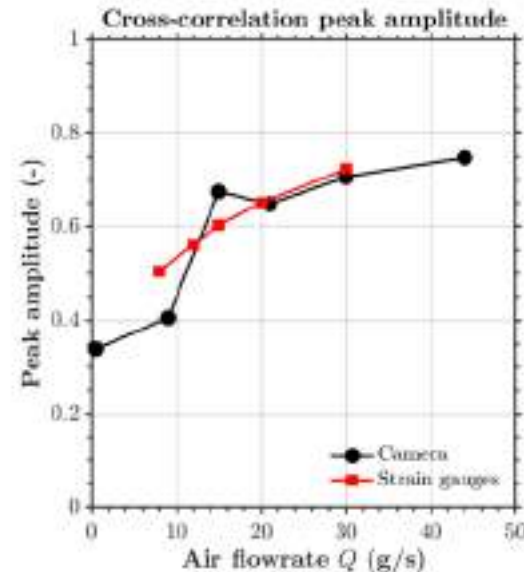
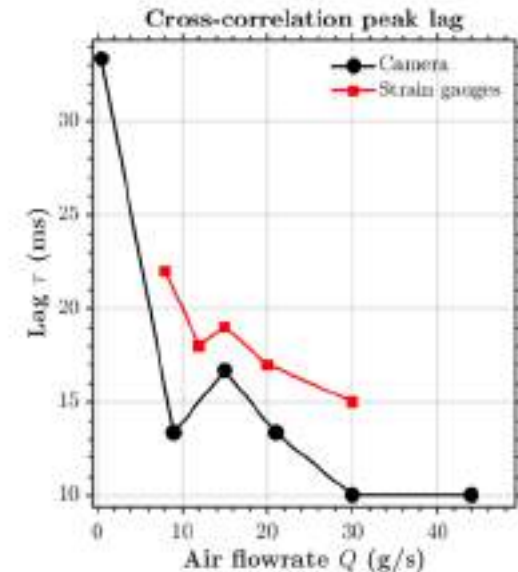


CROSS CORRELATIONS OF UPSTREAM AND DOWNSTREAM TUBES

$$R_{xy}(\tau) = \frac{\sum_t x(t) y(t + \tau)}{\sqrt{\sum_t x(t)^2 \cdot \sum_t y(t)^2}}$$

$x(t)$ = signal upstream
 $y(t)$ = signal downstream
 τ = temporal lag
 $R_{xy}(\tau) \in [-1,1]$ with 1 for total correlation

Example of cross correlation for $Q_{air} = 21 \text{ g/s}$



CROSS CORRELATIONS OF UPSTREAM AND DOWNSTREAM TUBES

Gaz superficial velocity: $J_g = \frac{\dot{m}_g}{\rho_g a n L}$

Liquid superficial velocity: $J_l = \frac{\dot{m}_l}{\rho_l a n L} = 0.13 \text{ m/s}$
(constant in experiment presented)

Mixture superficial velocity: $J_g + J_l$

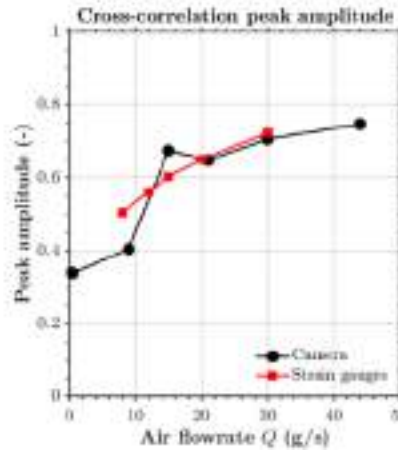
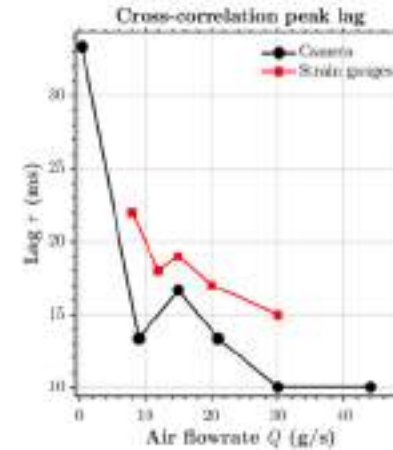
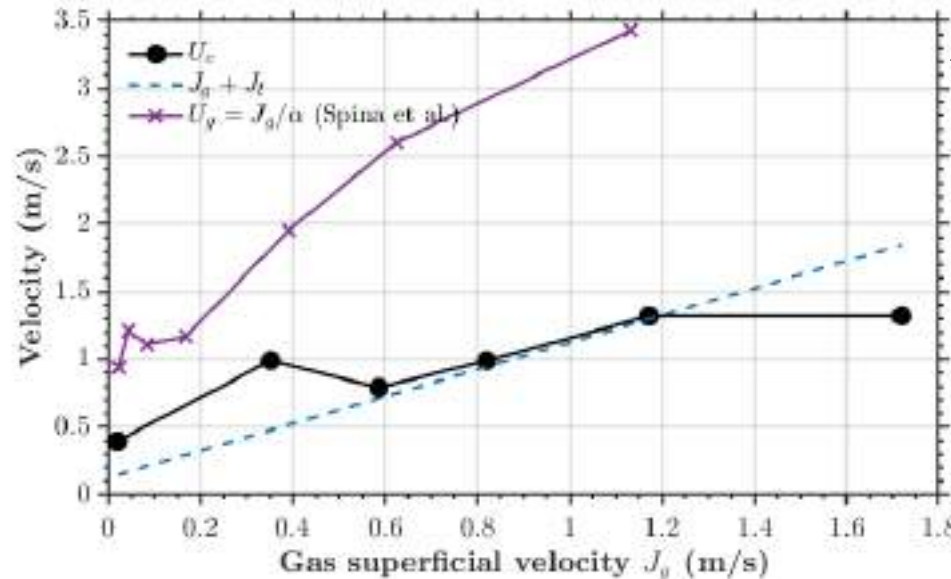
True gas velocity: $U_g = \frac{J_g}{\alpha}$
(with α void fraction)

$$U_c \propto J_g + J_l$$

perturbation transported by the mixture

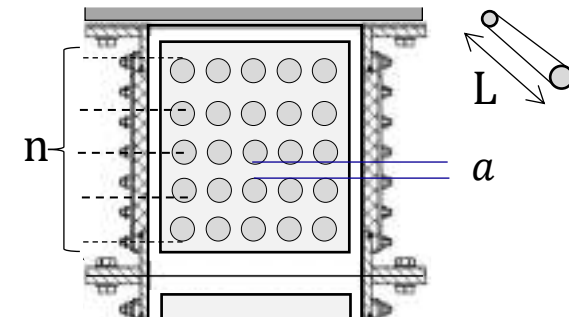
$$U_c \propto U_g$$

perturbation transported by discrete gas structures



Convection velocity

$$U_c = \frac{a}{\tau}$$

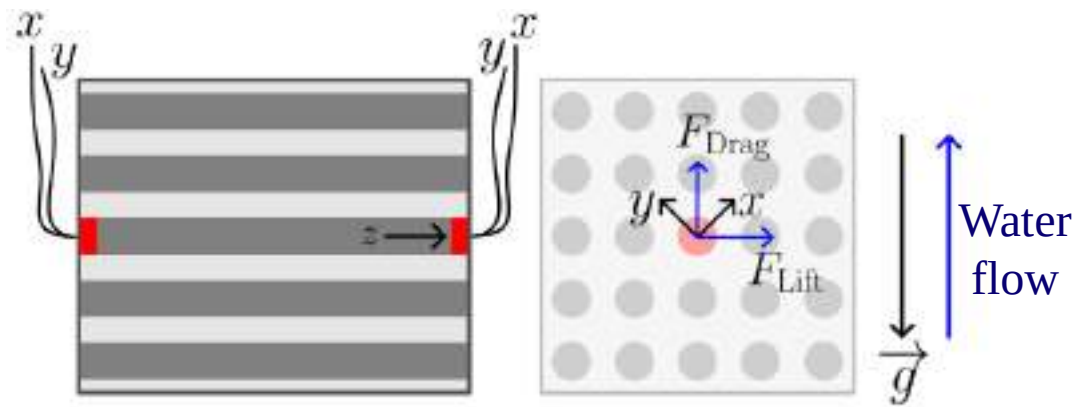
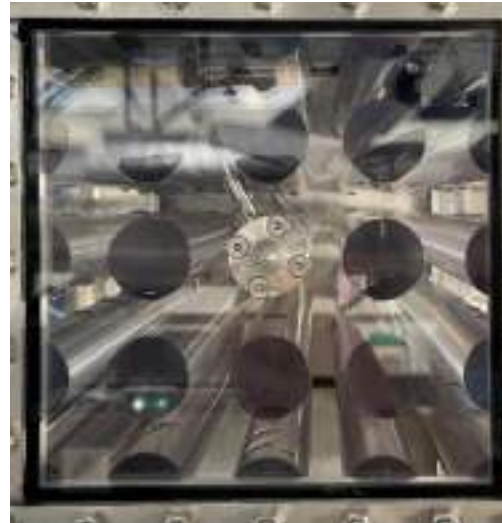


with $a = P - D = 13.2 \text{ mm}$, $n = 5$, $L = 0.3 \text{ m}$

03

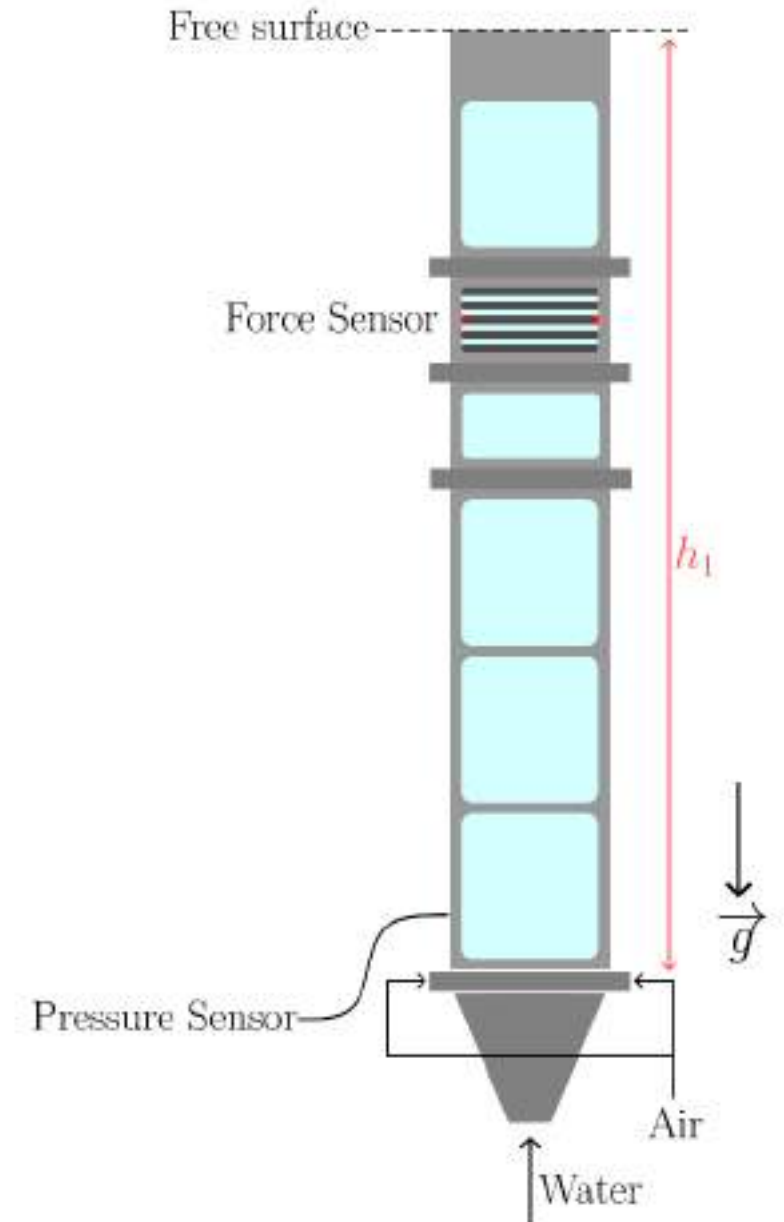
FORCES ON FIXED TUBE

TREFLE NEW EXPERIMENTS: FORCE SENSOR

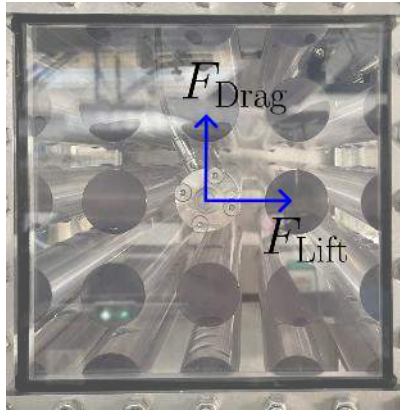


2x Kistler 9317C

- Triaxial piezoelectric sensor measuring two force components F_x , F_y
- Measuring ranges = ± 1.5 kN
- Sensitivities ≈ 25 pC/N
- Compact dimensions: $25 \times 25 \times 30$ mm
- Natural frequency (fixed-free) ≈ 5 kHz

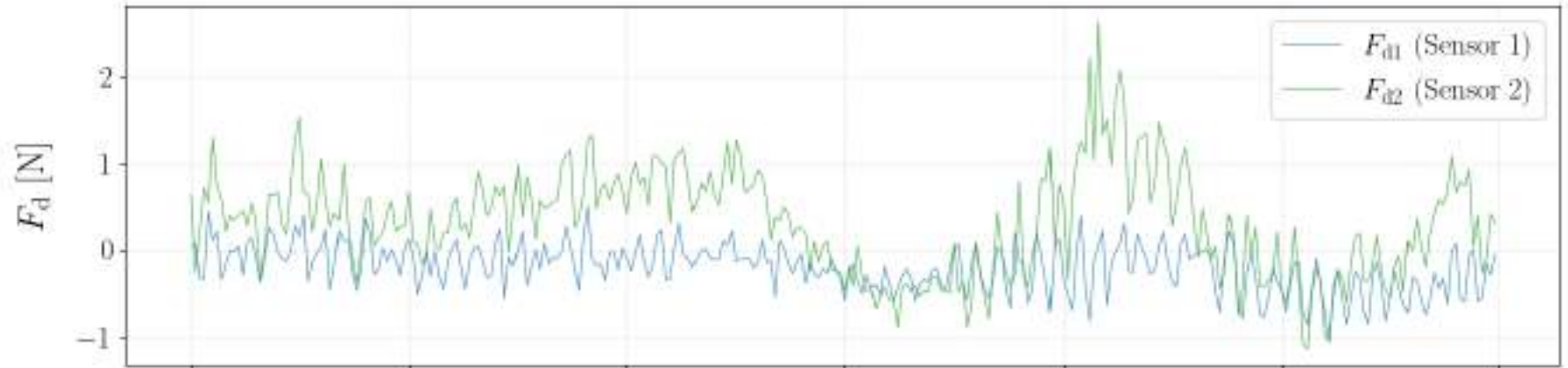


RAW SIGNALS

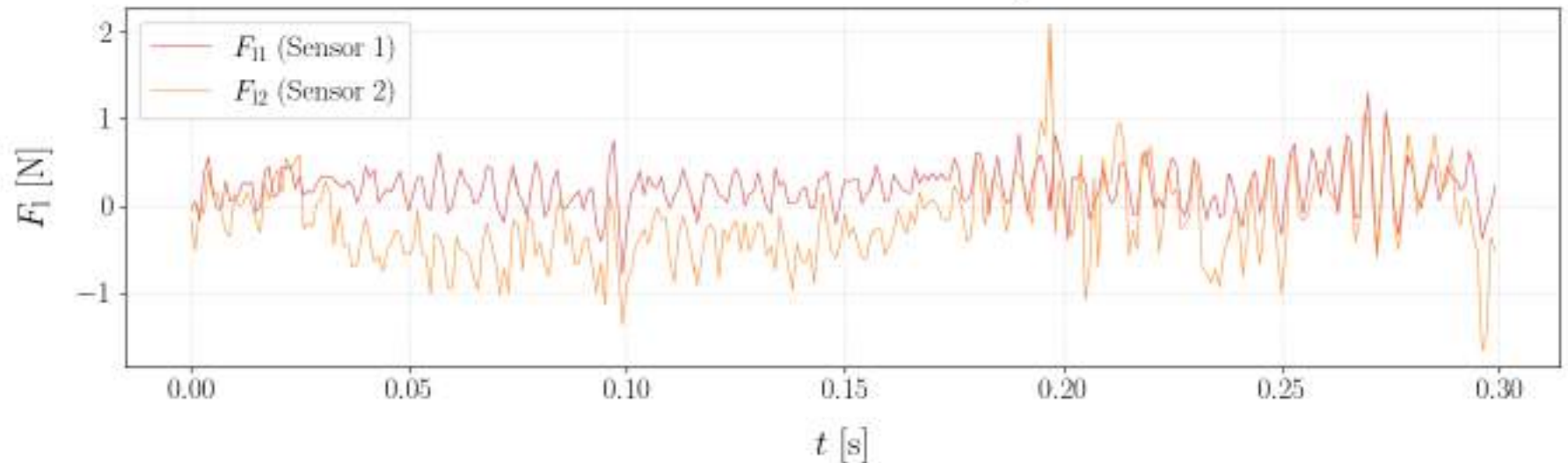


Raw drag oriented force signals

Example $Q_{\text{air}} = 5g/s$



Raw lift oriented force signals



Sensor drift requires mean removal; only fluctuations are experimentally reliable

$$F(t) = \cancel{\langle F \rangle} + \delta F(t)$$

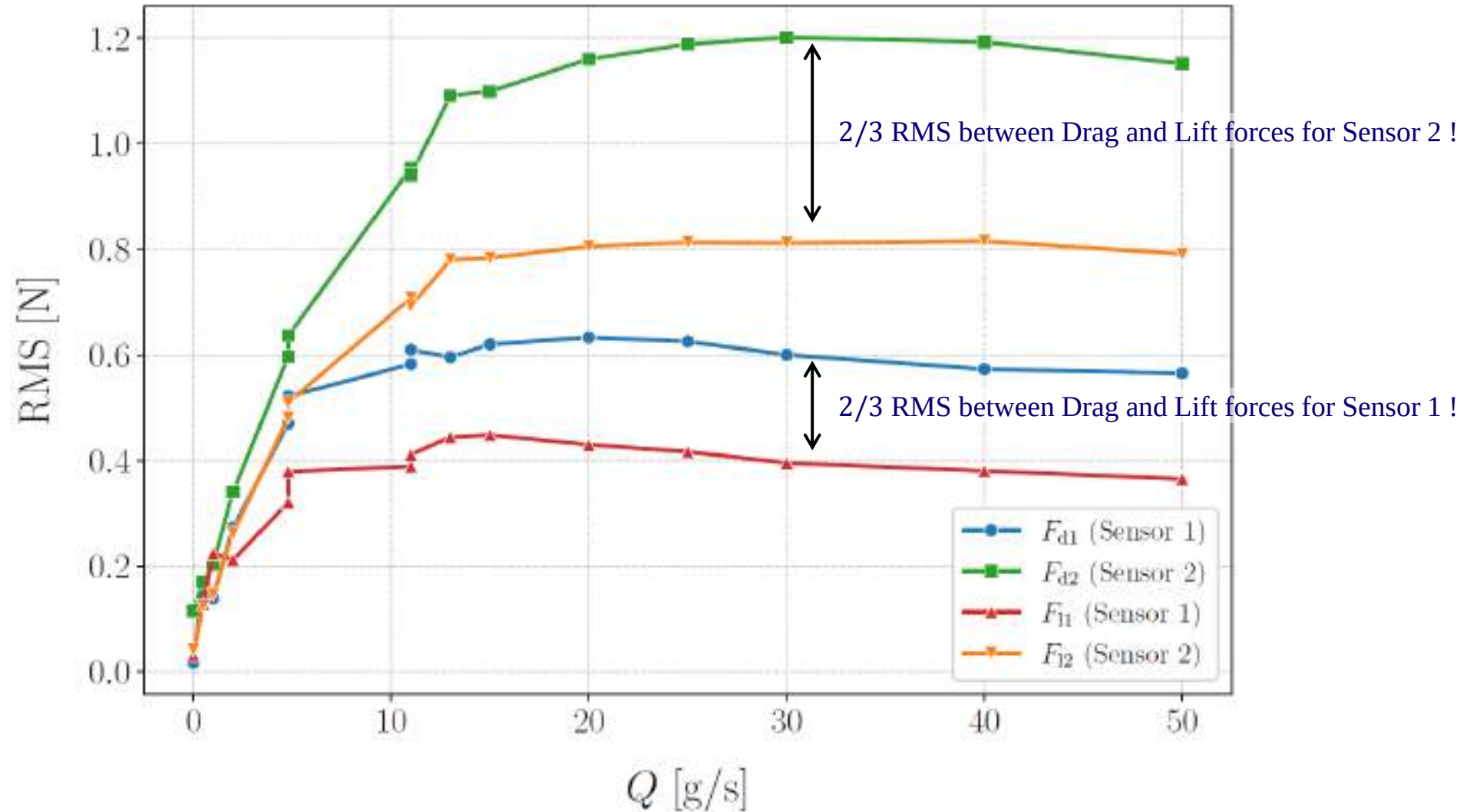
Tube fixation differences cause different amplitude responses at each sensor

$$\|F_{\text{Sensor 2}}\| > \|F_{\text{Sensor 1}}\|$$

RMS SIGNAL

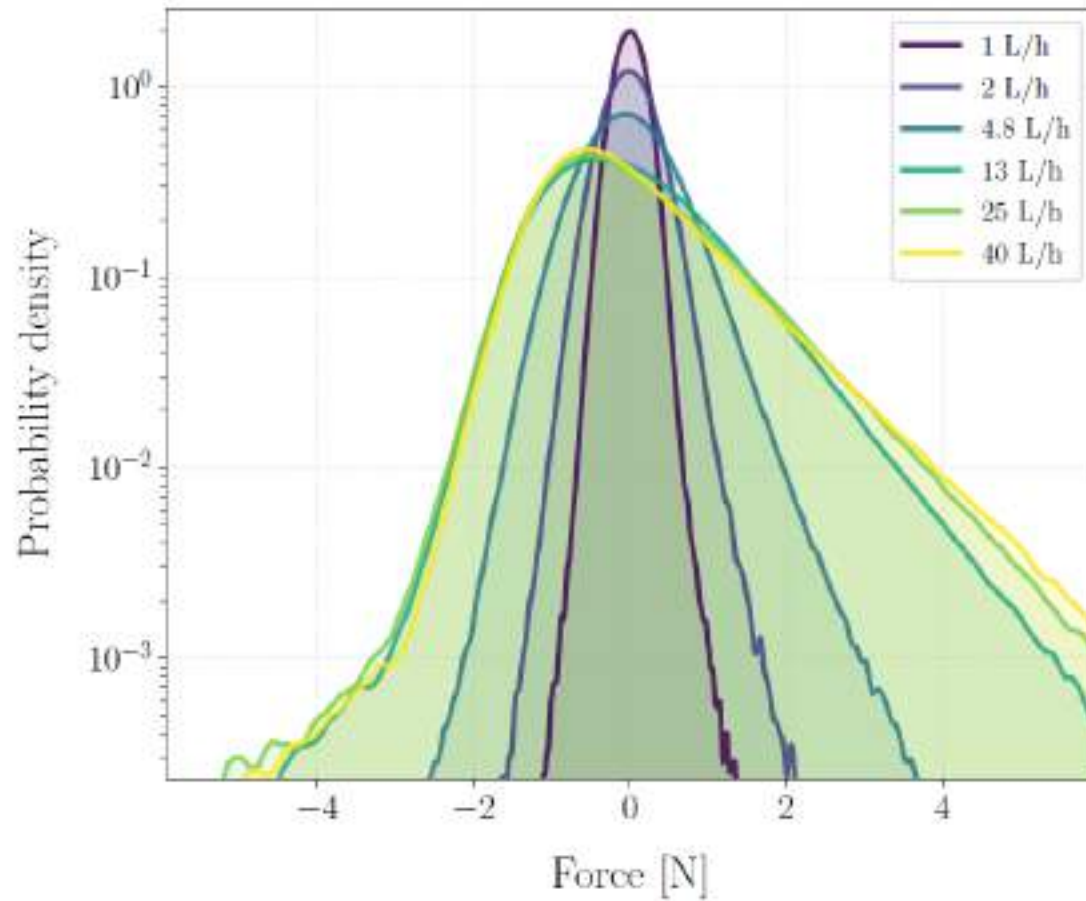
Fluctuation Root Mean Square
(signals pre-centered):

$$F_{RMS} = \sqrt{\langle F^2 \rangle} = \sigma$$

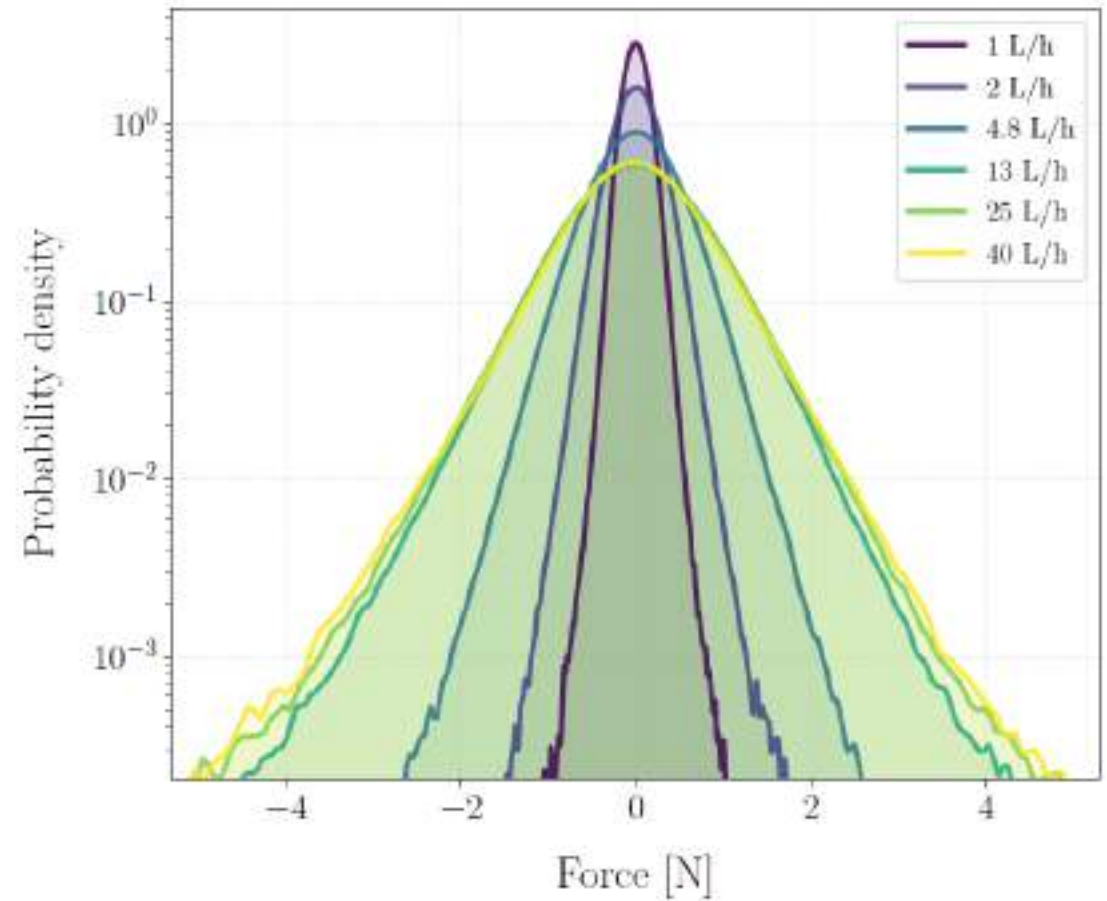


PROBABILITY DENSITY FUNCTION

Drag force PDF - Sensor 2



Lift force PDF - Sensor 2

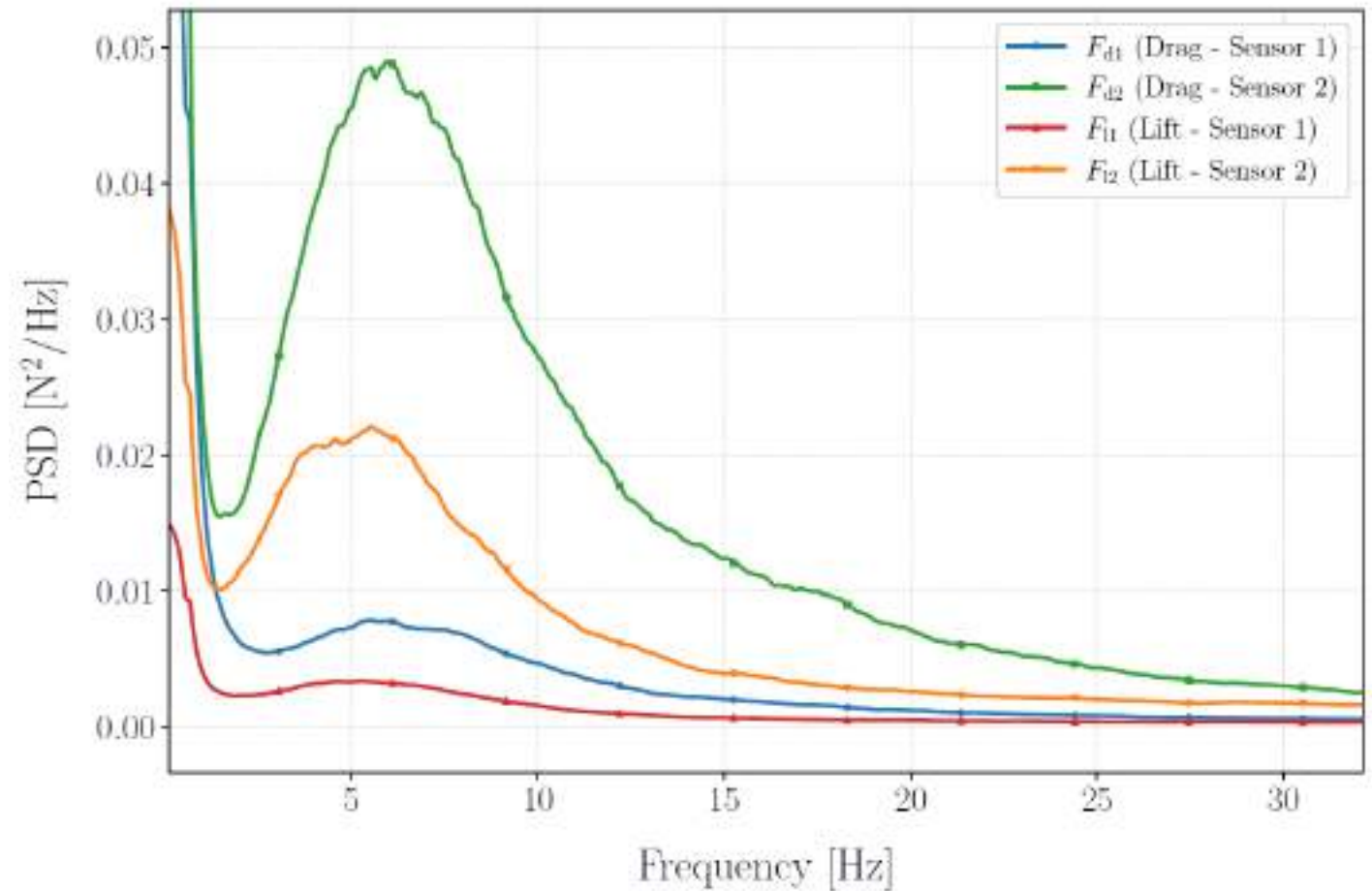


SPECTRAL ANALYSIS

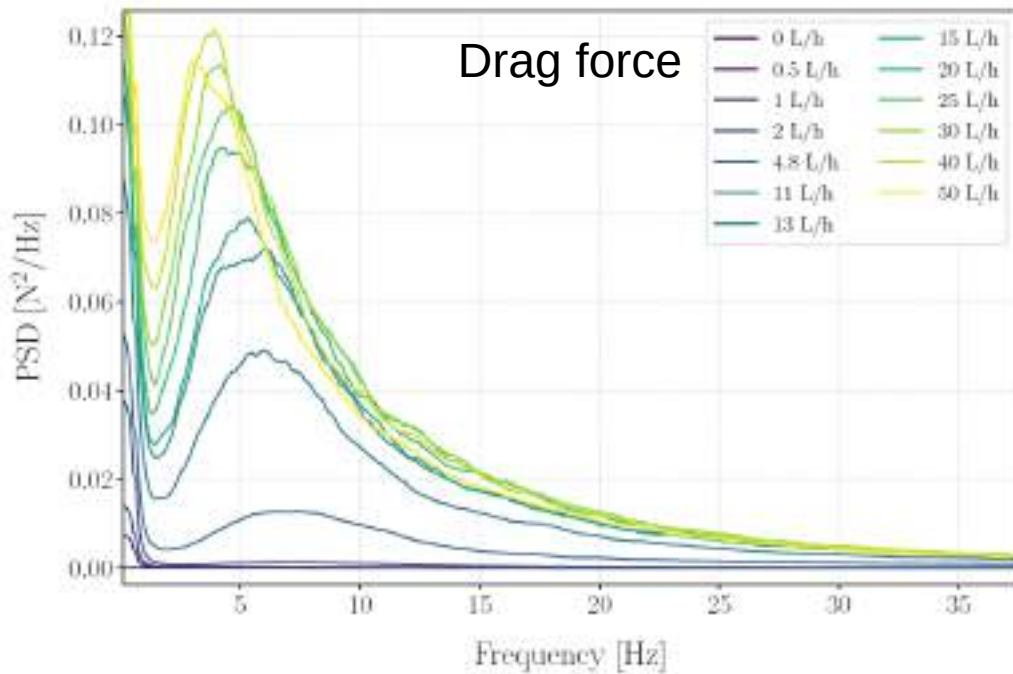
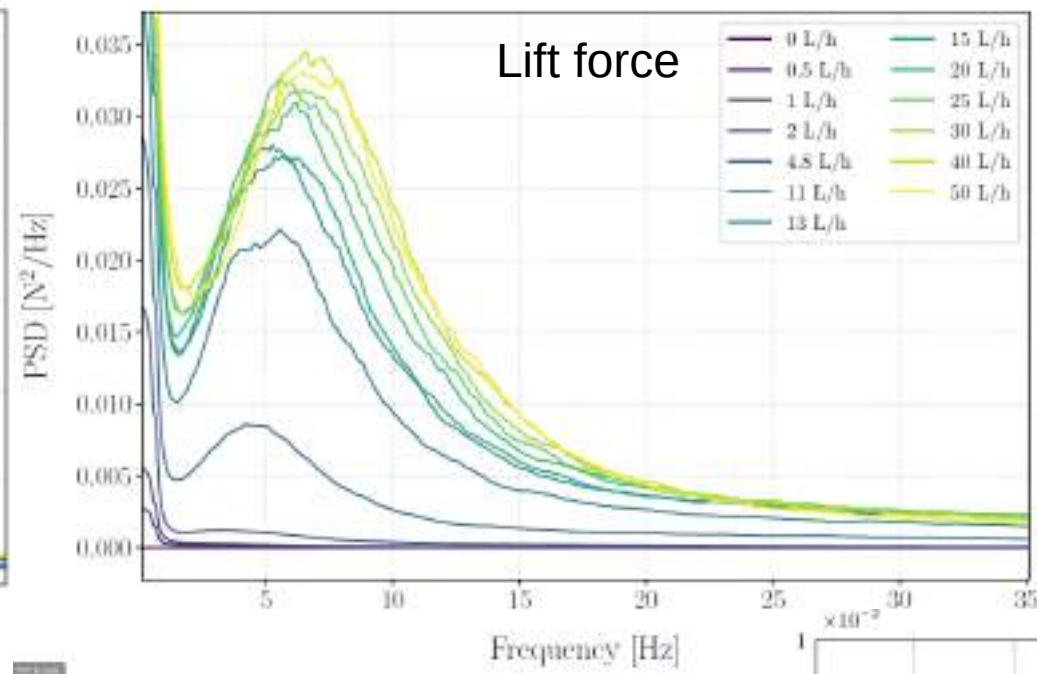
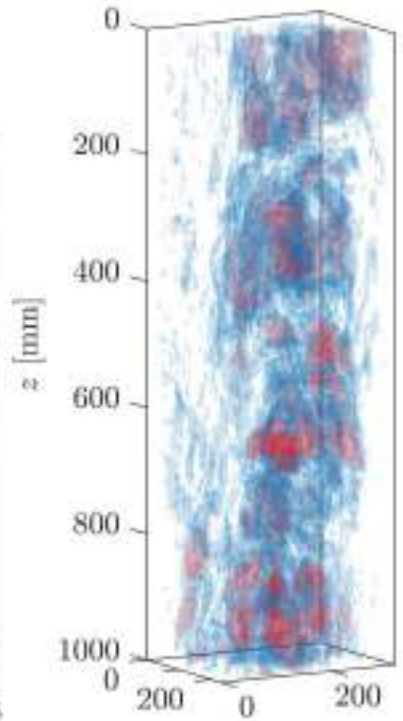
Spectral analysis characteristics:

- 30 minutes signal
- Welch's method with 16384-point segments (resolution $\Delta f = 0.061$ Hz, duration 16.4 s),
- Hann window
- 50% overlap, averaging ~ 220 realizations.
- A moving average smoothing over 20 frequency bins (1.2 Hz)

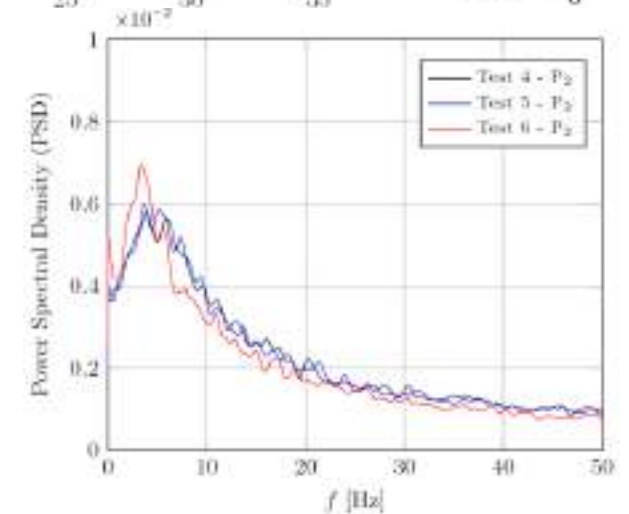
PSD comparison for both sensors - Flow rate: 11 g/s



SPECTRAL ANALYSIS



Vivaldi, D, Chahine, A, & Rotily, L. "Experimental Study of the Two-Phase Flow and Induced Vibrations in a Square-Pitch Tube Bundle in Air-Water Cross-Flow." Proceedings of the . Volume 4: Fluid-Structure Interaction; High Pressure Technology. 2025 - **Article conclusion: "The behavior of the intermittent rising gas structures was quantified; a characteristic frequency was measured of about 3-5 Hz"**





FORCES MEASUREMENTS CONCLUSIONS

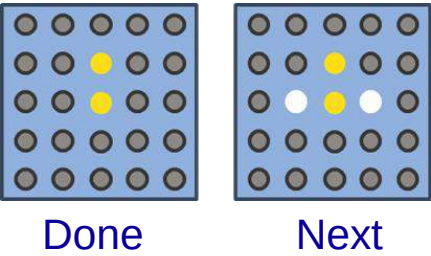
- **Both sensors measure the same forces simultaneously** and exhibit consistent trends, despite differences in the rod-to-structure mounting system that appear to **dampen** the sensor response at the beam tip.
- **RMS forces increase with flow rate**, particularly during bubbly flow regime, while the increase is less pronounced in the churn/intermittent flow regime.
- **Drag force** are predominantly oriented in the flow direction and **exhibits higher RMS** intensity than lift force.
- Spectral analysis reveals **low-frequency vibrations** consistent with bubble passage through the tube cross-section, with similar dominant frequencies observed for both drag and lift forces.

04

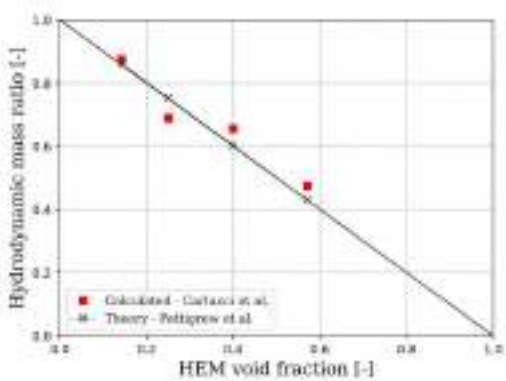
FUTURE WORK ON TREFLE

FUTURE EXPERIMENTS ON TREFLE

ADDITIONAL FLEXIBLE TUBES FOR CORRELATION WITH HORIZONTAL NEIGHBOUR



ADDED MASS CALCULATIONS AT HIGHER VOID FRACTION, DEVIATIONS FROM PETTIGREW?



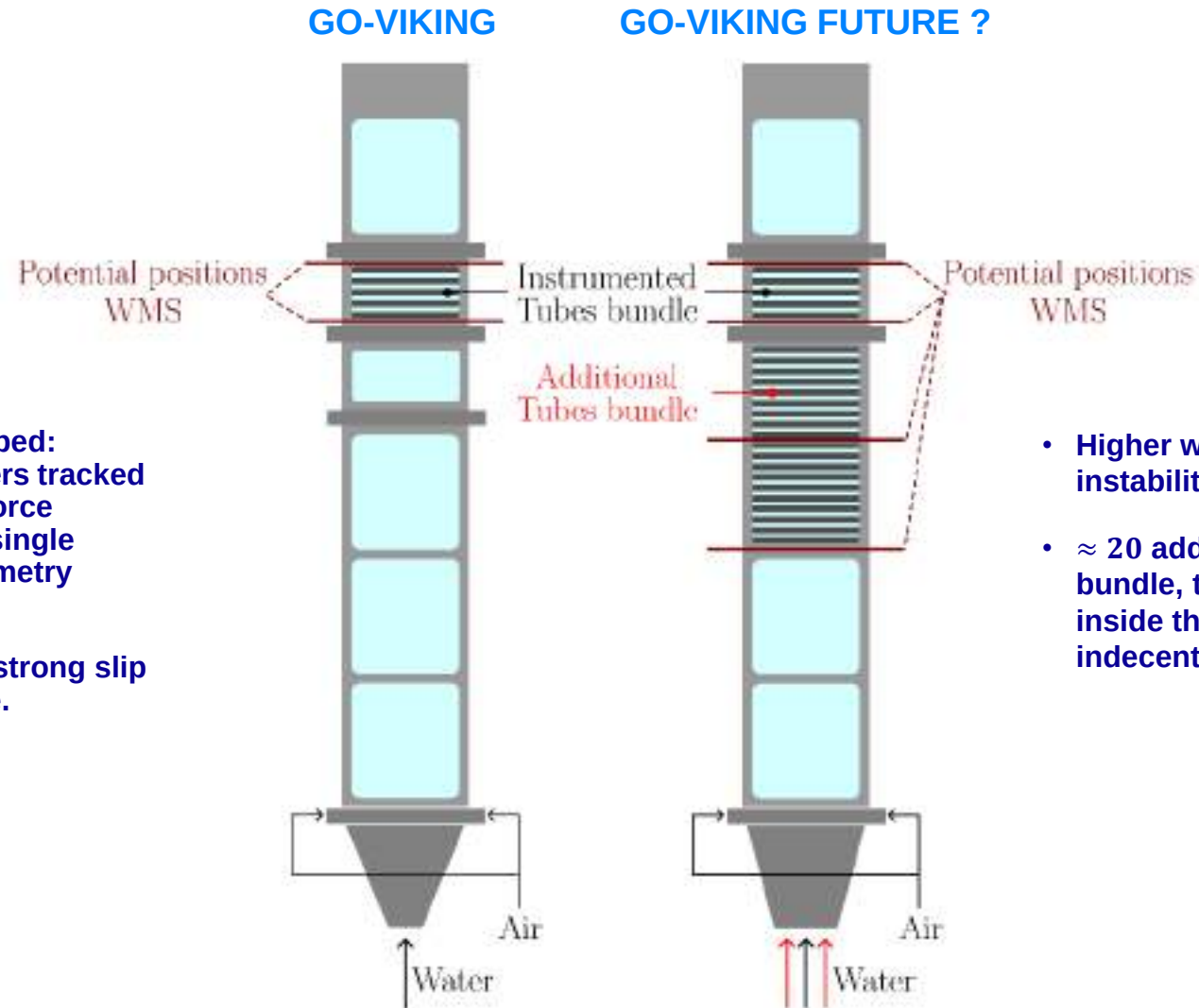
From



FORCE MEASUREMENT ON TWO TUBES SIMULTANEOUSLY FOR FORCES SIGNALS CORRELATION, AND FORCE MEASUREMENT DEPENDING ON BUNDLE POSITION.

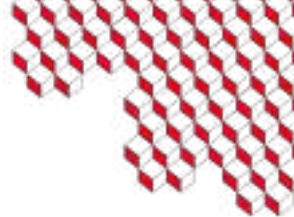


FUTURE EXPERIMENTAL MODIFICATION OF TREFLE



- Complete instrumentation developed: vibrating tubes with accelerometers tracked by fast camera, fixed tubes with force sensors, wire mesh sensor, dual/single optical probe with Doppler velocimetry measurement...
- Bubbly to intermittent flows with strong slip velocity between gas/liquid phase.

- Higher water flowrate to reach fluid elastic instability
- ≈ 20 additional arrays of tubes, in at least two bundle, to measure two phase flow properties inside the bundle, and to experiment indecently of initial conditions



Presentation of the TREFLE simulations at CEA

GO-VIKING FINAL WORKSHOP

May 21, 2026: Project Results

Clément Bazin (clement.bazin@cea.fr)

CEA DES/ISAS/DM2S/STMF SACLAY

Outline

Introduction

Numerical modelling of two-phase turbulent flows

TREFLE experimental data processing

TREFLE simulations at CEA

Conclusion

Outline

Introduction

Numerical modelling of two-phase turbulent flows

TREFLE experimental data processing

TREFLE simulations at CEA

Conclusion

Industrial context : tube vibrations in PWR

What?

- Pressurized Water Reactor (**PWR**) – Steam Generator (**SG**) : Inverse U-shaped bundle of tubes for heat exchange.
- **Upward two-phase transitioning flow** from pure liquid to pure vapour.

Why?

- Tube **vibrations** induced by **two-phase** flow.
- Structural wear, even **rupture** of some tubes.
- **Accident** scenario: contaminated water spills into the clean water circuit.





TREFLE simulations - C. Bazin

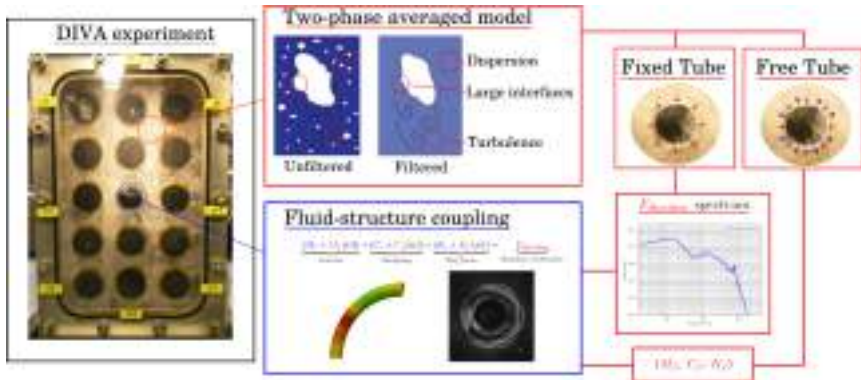
What we had?

- **Prior research** on the modelling.
- DIVA mockup at CEA.
- PhD thesis on how to simulate DIVA.

Differences and challenges of TREFLE compared to DIVA:

- **Very low liquid** injection (e.g. low liquid velocity) : high drift.
- High emphasis on gas accuracy (**not driven by liquid turbulence**).
- Tubes close to **lateral walls** (well-known numerical inaccuracy).

Research problem: numerical simulations



Fixed tube:

- Validation of the model.
- Force spectra and local data.

Free tube:

- Fluid-structure coupling : Immersed boundary method

Outline

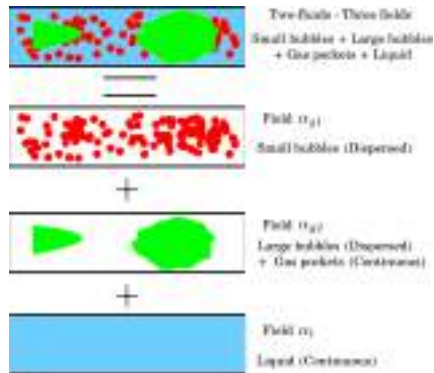
Introduction

Numerical modelling of two-phase turbulent flows

TREFLE experimental data processing

TREFLE simulations at CEA

Conclusion

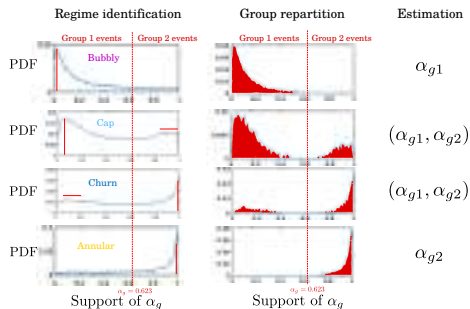
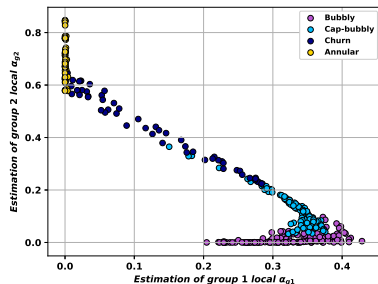


Euler-Euler URANS 3 fields.

- 1 **dispersed** field of quasi-spherical bubbles (2-group IATE from Smith et al., 2015)
- 1 **hybrid** field for large bubbles (2-group IATE from Smith et al., 2015) and resolved gas pockets (LIM from Coste, 2013)

- ## Euler-Euler URANS 3 fields.
- 1 **dispersed** field of quasi-spherical bubbles (2-group IATE from Smith et al., 2015)
 - 1 **hybrid** field for large bubbles (2-group IATE from Smith et al., 2015) and resolved gas pockets (LIM from Coste, 2013)

What should be modelled? Experimental data from DIVA



From the literature:

- Experiments: Jones & Zuber (1975), Schlegel et al. (2009), Julia et al. (2013), Zhang et al. (2020).
- 1D models: Hibiki & Ishii (2000), Hibiki et al. (2018).

What does this refer to?

- Two velocity groups with distinct behaviour: nearly spherical bubbles (**g1**) and highly distorted bubbles (**g2**).
- Clearly visible in void fraction and bubble diameter profiles.

Evidence for two-group behaviour

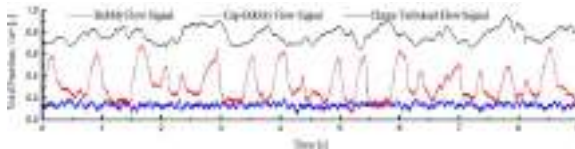


Figure 1: Schlegel et al. (2014)

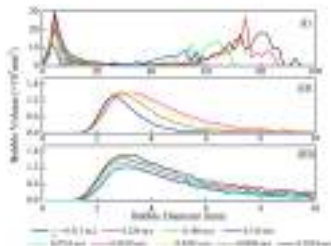


Figure 2: Zhang et al. (2020)

Key takeaways:

- Figure 2: α_{g1} (blue) remains roughly constant while α_{g2} (red peaks) varies.
- For small bubbles (**g1**): $ai_1 \approx 100 \text{ m}^{-1}$; for highly distorted ones (**g2**): $ai_2 \approx 10 \text{ m}^{-1}$, owing to D_{sm} (Figure 3).
- Easy to neglect **g2** since models scale with ai , but its dynamics cannot be ignored!
- **Dispersed** model for **g1** and **hybrid resolved interface** for **g2**.

Governing equations solved

3 Fields: Liquid, gas group 1 and gas group 2.

Mass conservation:

$$\frac{\partial}{\partial t} \rho_k \alpha_k + \nabla \cdot (\rho_k \alpha_k \mathbf{u}_k) = \underbrace{\Gamma_k}_{\text{Sources}},$$

Momentum conservation:

$$\frac{\partial}{\partial t} \rho_k \alpha_k \mathbf{u}_k + \nabla \cdot (\rho_k \alpha_k \mathbf{u}_k \otimes \mathbf{u}_k) = \underbrace{-\alpha_k \nabla P_k}_{\text{Pressure}} + \underbrace{\nabla \cdot (\alpha_k \tau_{\equiv k})}_{\text{Diffusion}} - \underbrace{\nabla \cdot (\alpha_k \tau_{\equiv k}^t)}_{\text{Turbulence}} + \underbrace{\rho_k \alpha_k \mathbf{g}}_{\text{Gravity}} + \underbrace{\mathbf{l}_k}_{\text{Sources}},$$

With IATE (2 groups) [Smith (2015)], turbulence (EBRSM) [Manceau (2015)], and interfacial force models [Ishii & Zuber (1979), Cai & Wallis (1964), Hayashi et al. (2021), Bazin (2023), Tomiyama (1998), Coste (2013)].

Equations solved with NEPTUNE_CFD

Phase	Conservation of	Unknown	Number of scalars
Liquid	Mass	α_l	1
Liquid	Momentum	$\mathbf{U}_l$	3
Liquid	Reynolds stress	R_{ij}	6
Liquid	Dissipation	ε	1
Liquid	Blending	β	1
1st gas group	Mass	α_{g1}	1
1st gas group	Momentum	$\mathbf{U}_{g1}$	3
1st gas group	Interfacial area concentration	a_{i1}	1
2nd gas group	Mass	α_{g2}	1
2nd gas group	Momentum	$\mathbf{U}_{g2}$	3
2nd gas group	Interfacial area concentration	a_{i2}	1

Total of **22 coupled differential equations** with non-linear source terms solved with Neptune_CFD.

Outline

Introduction

Numerical modelling of two-phase turbulent flows

TREFLE experimental data processing

TREFLE simulations at CEA

Conclusion

Need for experimental data for realism

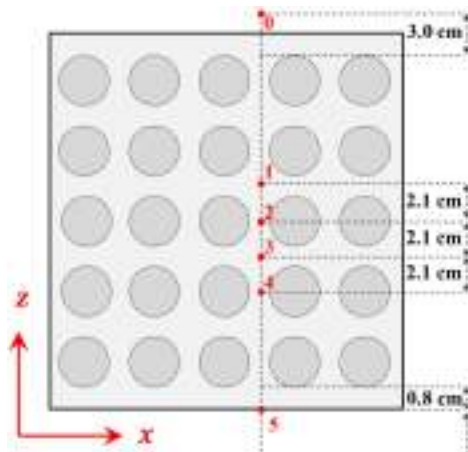
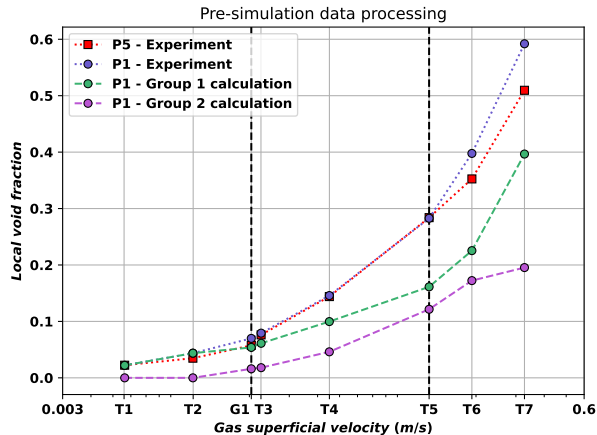


Figure 3: Optical probes locations in TREFLE

Data processing for inlet realism:

- Need for particular points to deduce inlet conditions, method learned from the DIVA experiment: **symmetries** (P1 - P5).
- **PDF** of void fraction and diameters available for group separation.
- **1D analytical** “by-hand” precalculation to check retro-compatibility.

A look at void fraction distribution



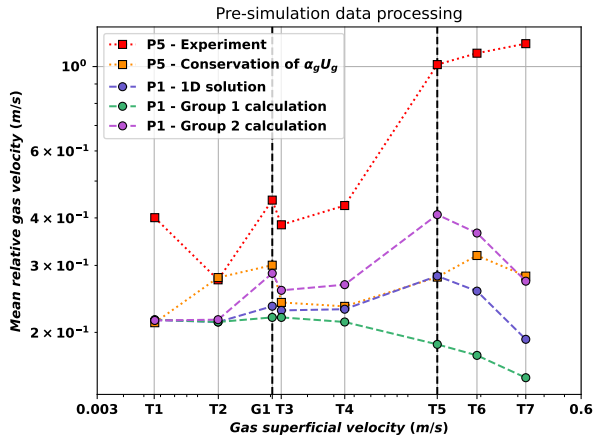
What we learn:

- Not many differences between P1 - P5 except for T6 and T7 (larger stagnant structures?).
- **G1**: mainly $g1$ but a little $g2$.
- **T5**: similar distribution of $g1$ and $g2$.

Pay attention:

- Checking the void fraction alone is not enough; the conserved quantity is $\alpha_g U_g$.
- $\langle \alpha_g U_g \rangle \neq \langle \alpha_g \rangle \langle U_g \rangle$, they are not independent.

1D solution of velocity based on void fraction and diameters



What we learn:

- **Unexpected measured velocities** on P5: very high relative velocity (maximum relative velocity for air-water bubbly flow ≈ 0.3 m/s).
- **Corrected velocity** compatible with gas superficial velocity and measured void fraction (orange).
- **G1**: even for corrected velocity, large discrepancy, but our model is consistent with **T3**.
- **T5**: good 1D representation.

Outline

Introduction

Numerical modelling of two-phase turbulent flows

TREFLE experimental data processing

TREFLE simulations at CEA

Conclusion

Numerical simulations at CEA



Figure 4: Coarse mesh with $y^+ \approx 1$ around cylinder and lateral walls with $y^+ \approx 12$ for G1 simulation [Fine 142,200 cells].

What we performed:

- Simulations for T2, G1, T4, T5, T6 with fixed cylinder in 2D, with common release agreement for **G1 and T5 (fixed and free tube)**.
- 3D is very expensive ($\approx 96 - 20\overline{M}$ cells) and the **common choice was made late**.

What is presented here:

- Most information is in the final document.
- Presentation of W. Benguigui about the common results on T5.
- **Focus on G1** (similar conclusions for all test cases).

G1 : void fraction qualitative results

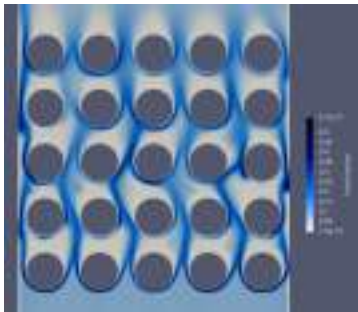


Figure 5: G1
time-averaged total void
fraction.

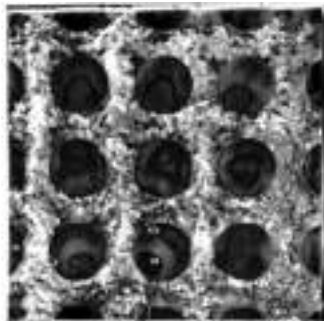


Figure 6: T3 experimental
snapshot (conditions close
to G1).

- **Similar** spatial features between time-averaged and instantaneous simulations and the experimental snapshot, with lateral paths.
- **Stable blocking effect:** little perturbation to destabilise the flow, since it is not driven by liquid turbulence (1 stable path in simulation, experimentally bi-stable).
- Could be improved in 3D: **fewer fluctuations** expected.

G1 fixed tube : void fraction quantitative results

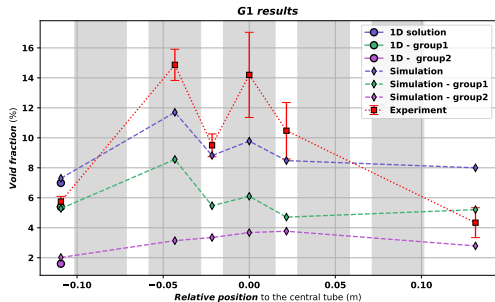


Figure 7: Local void fraction comparison.

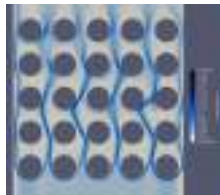


Figure 8: G1 time-averaged group 1 void fraction.

- The probe seems to measure **mainly group 1** bubbles.
- Due to the stable path, results are **very sensitive to probe location**.
- Lower void fraction at the bundle outlet may be due to **greater experimental fluctuations** (higher velocity and lateral fluctuations reduce the local void fraction).

G1 fixed tube : gas velocity quantitative results

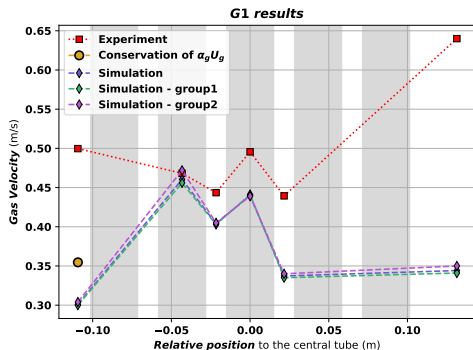


Figure 9: Local gas velocity comparison.

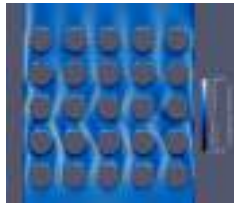


Figure 10: G1 time-averaged group 1 gas velocity.

- P5 (bundle inlet) shows unexpected behaviour.
- Results are **very sensitive to probe location**, but remain compatible with the experiment.
- High velocity at the outlet is consistent with the “**greater experimental fluctuations**” hypothesis.

G1 free tube : force spectra and acceleration

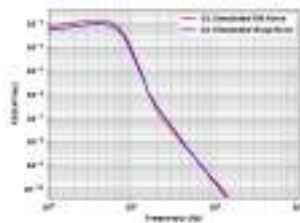


Figure 11: Lift and drag force spectra.

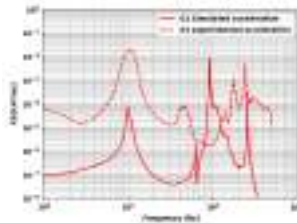


Figure 12: Acceleration spectra.

- Cylinder was initially blocked and then **released from its stable configuration** (to avoid numerical issues).
- Flow remains in a stable configuration; **fluctuations stay small** (visible in the height of the PDF), but the overall behaviour is similar to the literature.
- This limitation could be addressed by performing 3D computations.

Outline

Introduction

Numerical modelling of two-phase turbulent flows

TREFLE experimental data processing

TREFLE simulations at CEA

Conclusion

Conclusion

- Simulated T2, **G1**, T4, **T5** and T6 in 2D with a **fixed** cylinder.
- Simulated **G1** and **T5** in 2D with a **free** cylinder.
- Compared experimental results with **simulations and 1D solutions** to explain the differences and understand the flow behaviour.
- Similar conclusions across all test cases.
- **Fewer fluctuations** due to low liquid turbulence (as expected).
- RANS bubble fluctuation models (source terms) tend to homogenise the flow, further reducing fluctuations.
- This could be partially addressed by **performing 3D** simulations to capture more localised large structures (only a few resolved interfaces appear in these 2D simulations).

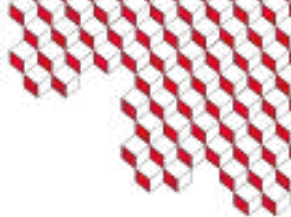
Perspectives

Short-term improvements:

- Perform **3D simulations** for G1 and T5 to better capture large-scale flow structures and increase fluctuation levels.
- Perform 2D simulations of G1 with newly available CEA **bubble induced fluctuations differential model** (Du Gluzeau et al.(2019))

Longer-term objectives:

- Develop and validate improved **bubble fluctuation models** that preserve flow heterogeneity.
- Apply method for **steam-water** with SG conditions (different flow topology than the low velocity TREFLE configuration).



Thanks! Any questions?



WP4 – TREFLE Benchmark

WP4 – Flow-induced vibrations in multiphase flows



Funded by
the European Union

TREFLE Benchmark



William Benguigui^{1,2}, Daniele Vivaldi³, Lucas Rotily³, Sina Tajfirooz⁴, Kevin Zwijsen⁴, Constantinos Katsamis⁵, Ehimen Iyamabo⁵, Antti Timperi⁶, Clément Bazin⁷

¹EDF R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatou, France

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

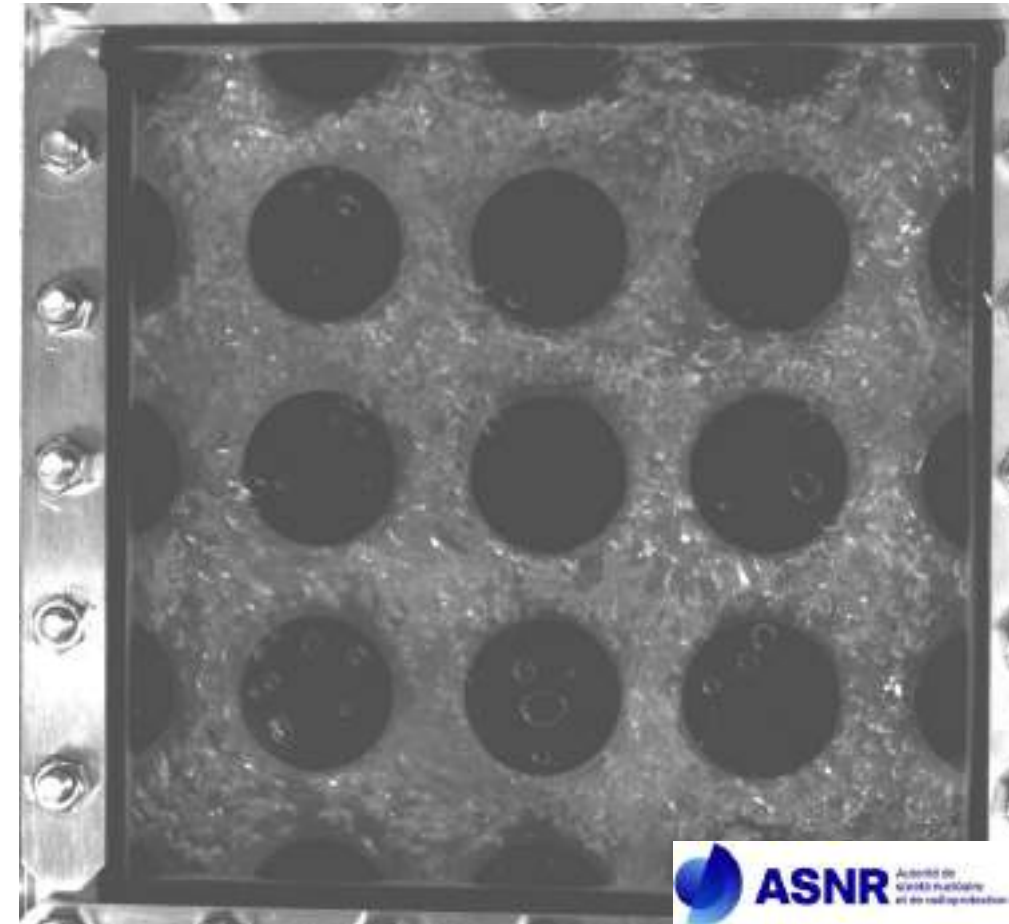
³ASNR, Cadarache, St Paul-lez-Durance, BP3, 13115, France

⁴NRG PALLAS, Westerduinweg 3, 1755 LE Petten, The Netherlands

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

⁶VTT Technical Research Centre of Finland, Kivimiehentie 3, Espoo, 02044 VTT, Finland

⁷CEA, CEA Site de Saclay 91191 Gif-sur-Yvette, France, France



TREFLE



TREFLE hydraulic loop, ASNR Cadarache.

Square tube bundle composed of 25 rigid tubes with $D = 30\text{ mm}$, $L = 300\text{ mm}$, and $P/D = 1.44$.

Air-water crossflow at ambient conditions.

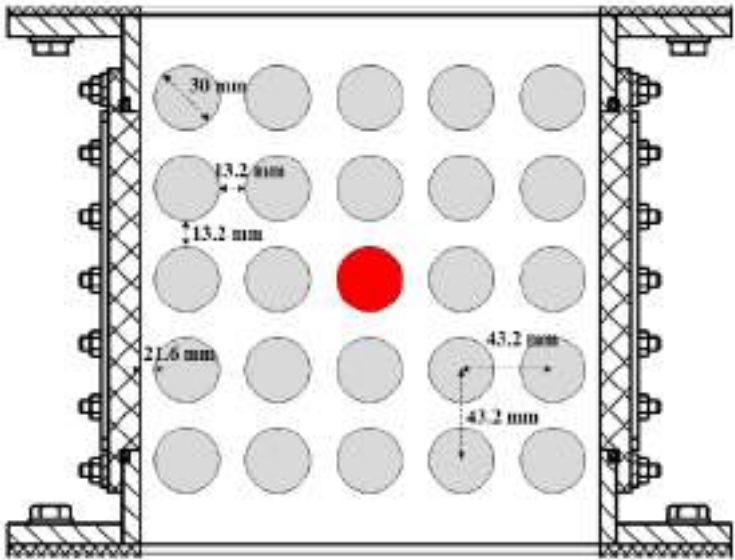
Optical probe measurements.

Wire mesh sensor

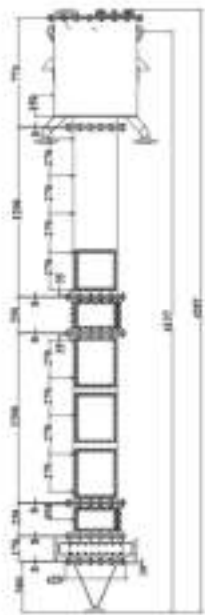
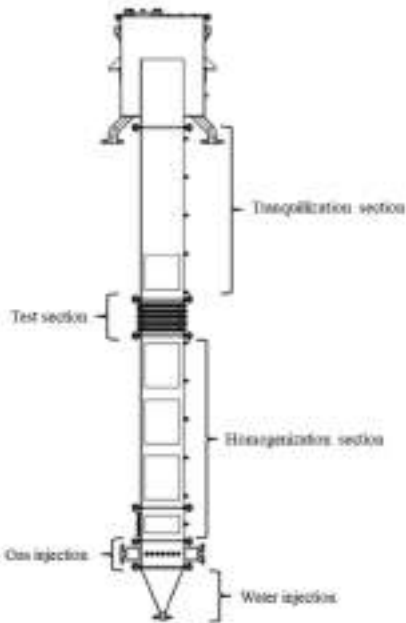
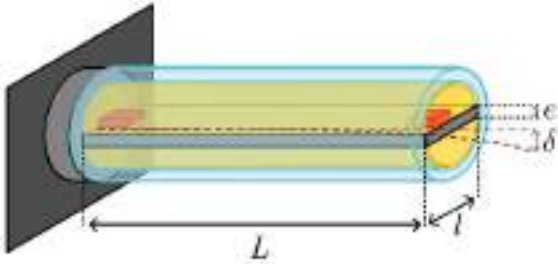
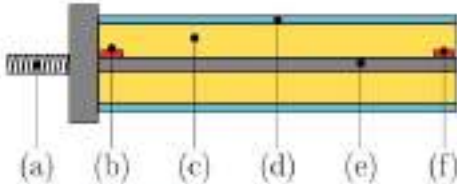
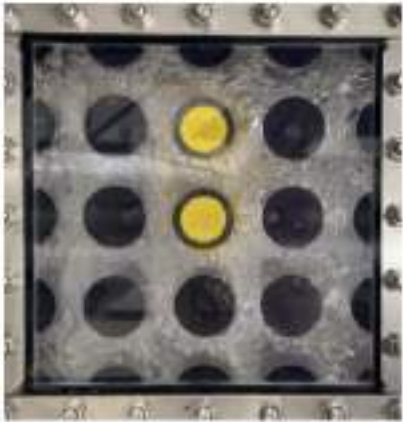
Vibration measurement of the center tube.

Recently: forces acting on the cylinder.

Test	Number of injector cylinders	Type of tubes	m_{i_f} [g/s]	m_{i_o} [g/s]
T1	2	Type 1	2500	0.5
G1	5	Type 1	2500	2
G2	7	Type 1	2500	4.8
T5	7	Type 2	2500	11
G3	7	Type 2	2500	15
G4	7	Type 2	2500	31



(a) characteristics of the tube bundle



Numerical modeling



Partner	Software	Two-phase modeling	Turbulence model	Type of Inlet	Type of Outlet	Simulated case
VTT	Ansys-Fluent	VOF	LES Smagorinsky	Velocity with SBM (Fluent UDF)	Pressure	T5
CEA	neptune_CFD	Averaged Euler-Euler	EBRSM (for liquid phase)	Homogeneous	Constant pressure	T5, G1
NRG	OpenFOAM 12 (Foundation branch)	Hybrid two-fluid model (HZDR model)	K-Omega SST for liquid phase and K-Omega SST Sato for gas phase	Uniform velocities and volume fractions are set based on the experimental superficial velocities and volume fractions	Fixed pressure, zero-gradient for k and Omega, Void fraction, pressureInletOutletVelocity for velocity,	T5
ASNR	neptuneCFD	Eulerian-EulerianGLIM	EBRSM (only for the liquid phase)	Single large bubble synthesized	Constant pressure	T5
EDF UK	StarCCM+ v202 4.6 Double Precision	Volume of Fluid (VOF) method	$k - \omega SST$	Flat Inlet with mixture velocity and void fraction=0.7822	Pressure	T5

Calculation domains

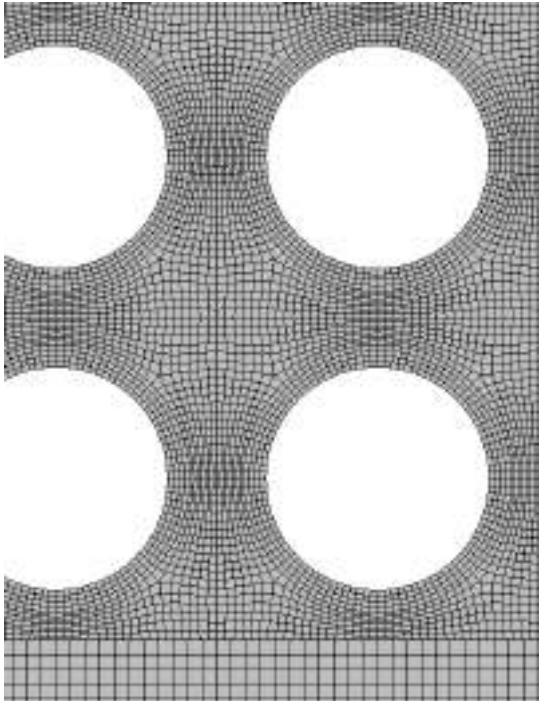
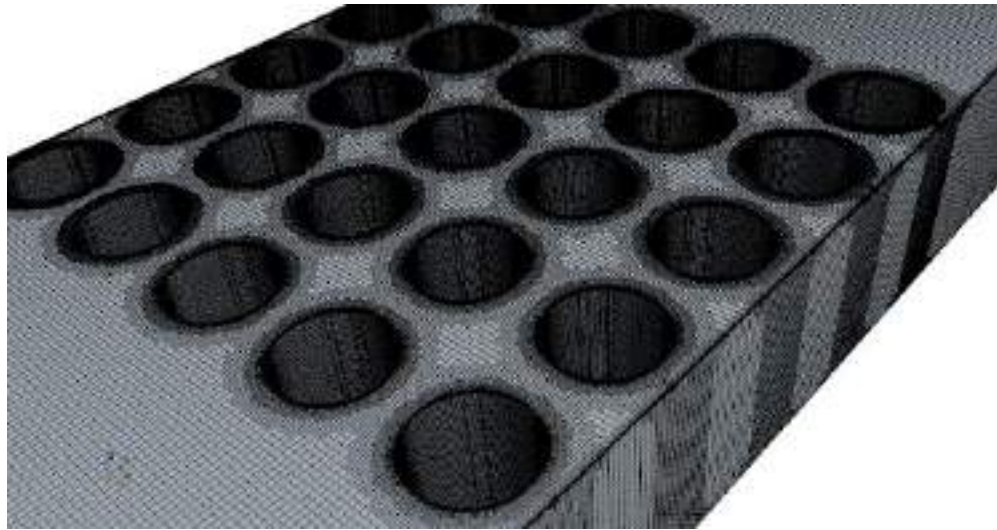


	2D or 3D	Tube Length	Widthupstream tube bundle	periodicity	Type of mesh	Total number of cells	Dr (mm) at cylinder wall	Dx_mean (mm)
CEA	2D (1 cell along cylinder)	10D	0.3 m	No	Hexahedral	G1: 142400 T5 : 320100	G1 : 0,12 T5 : 0,08	G1 : 1,459 T5 : 1,022
VTT	3D	10D	700 m	none	Hexahedral	16 M	0.75	1.0
NRG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

Calculation domains



VTT

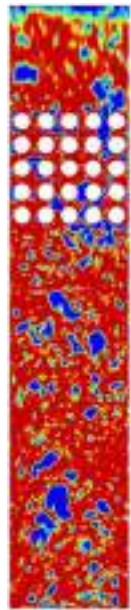
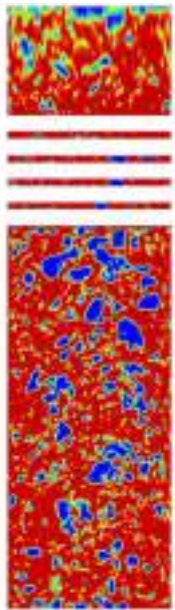
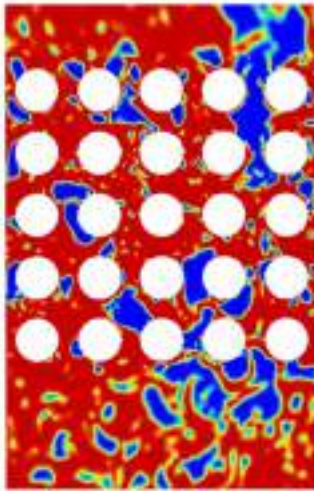


EDF UK

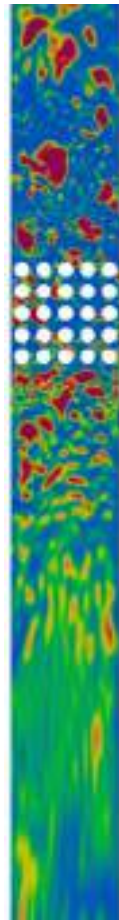
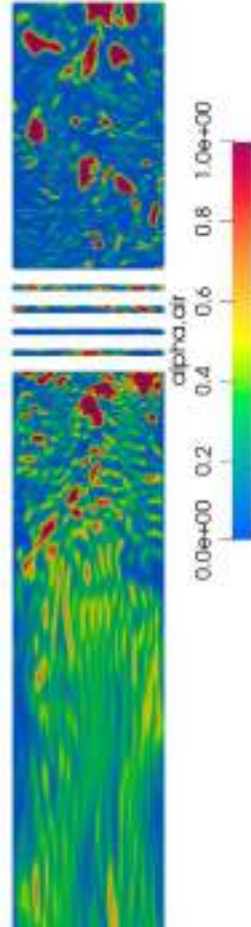


NRG

Numerical results - rigid



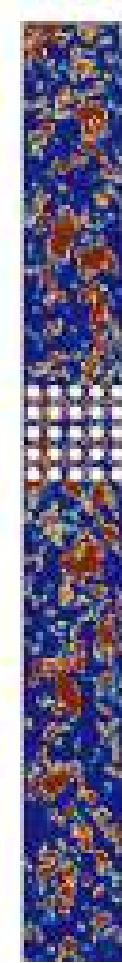
VTT



NRG

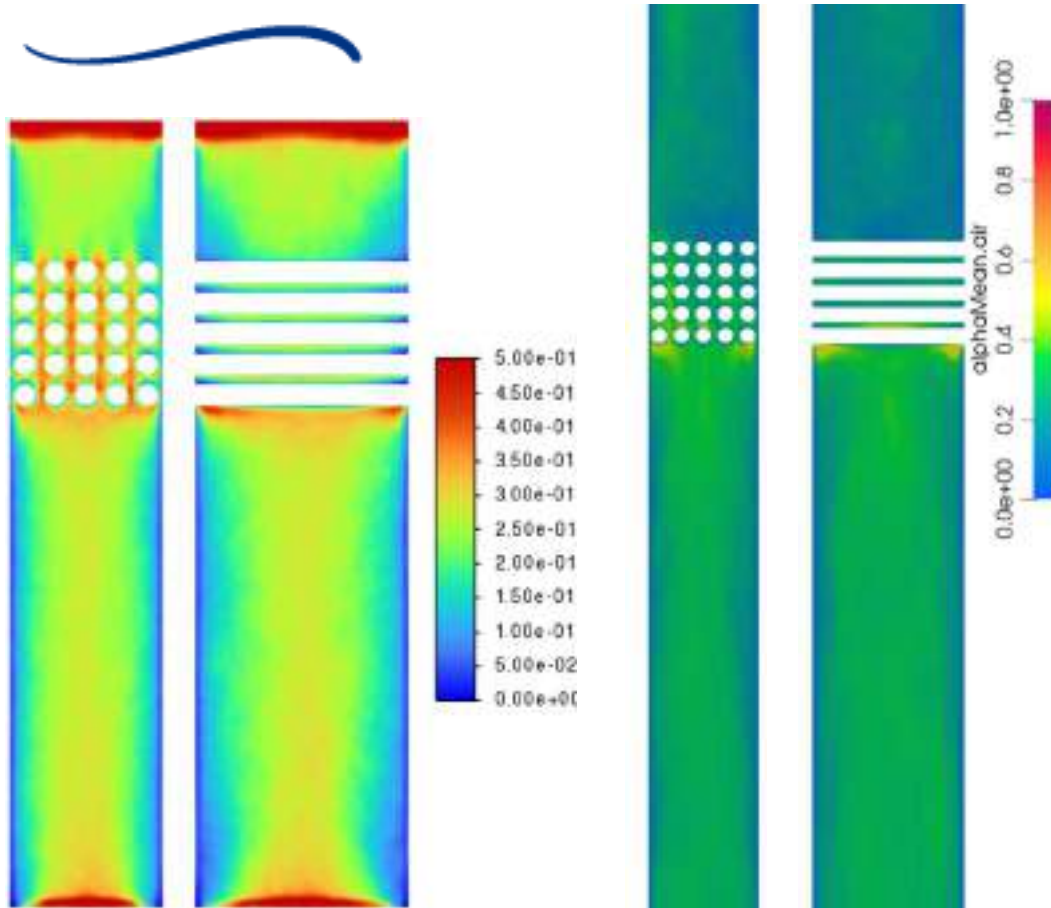


CEA



EDF UK

Numerical results - rigid

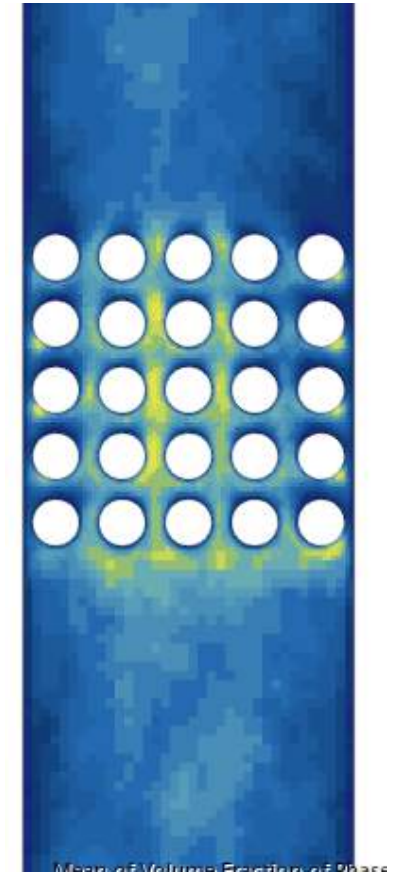


VTT

NRG

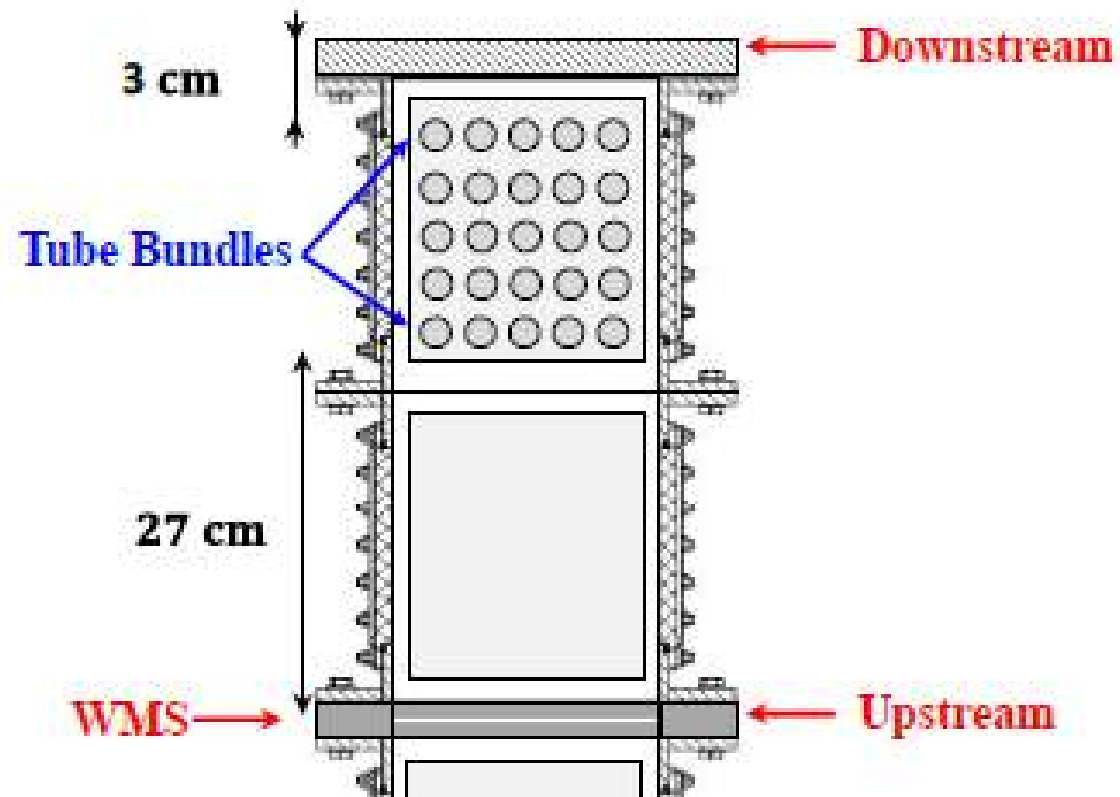
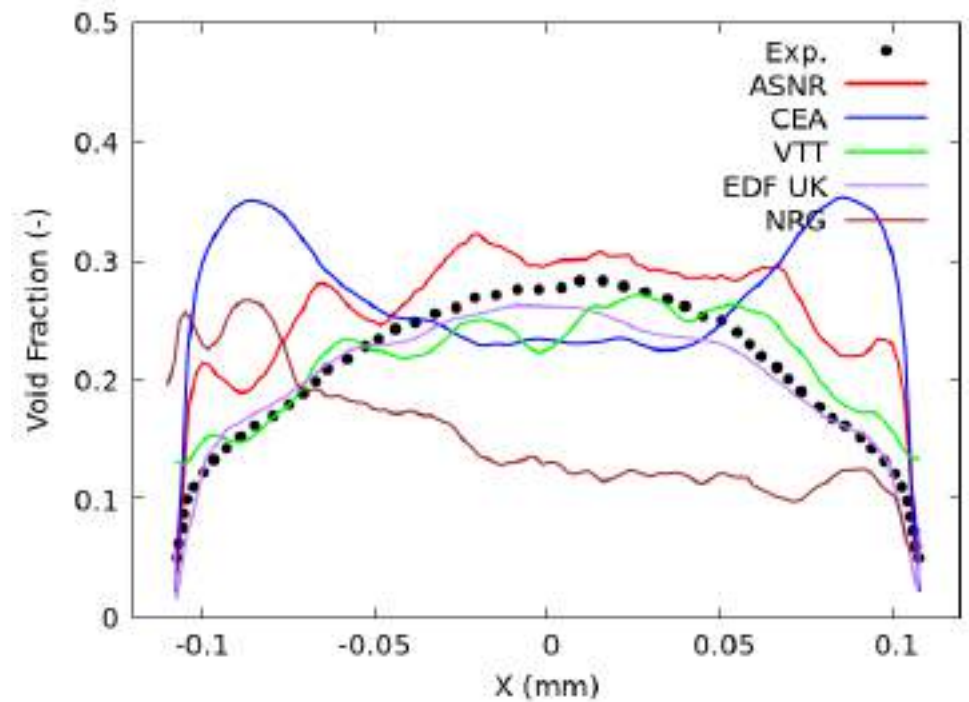
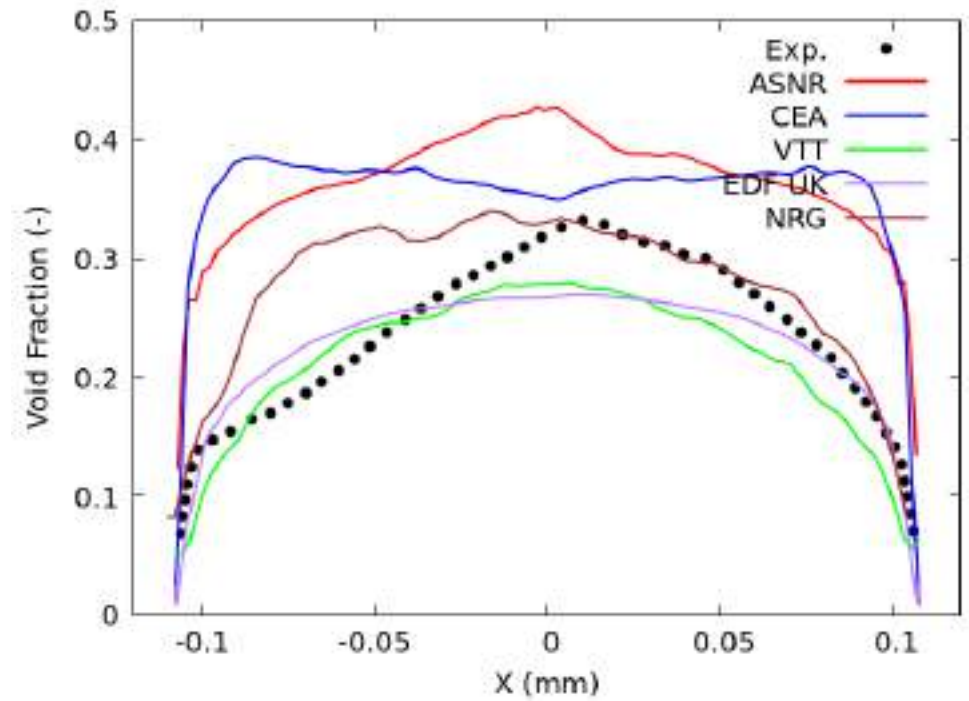


CEA

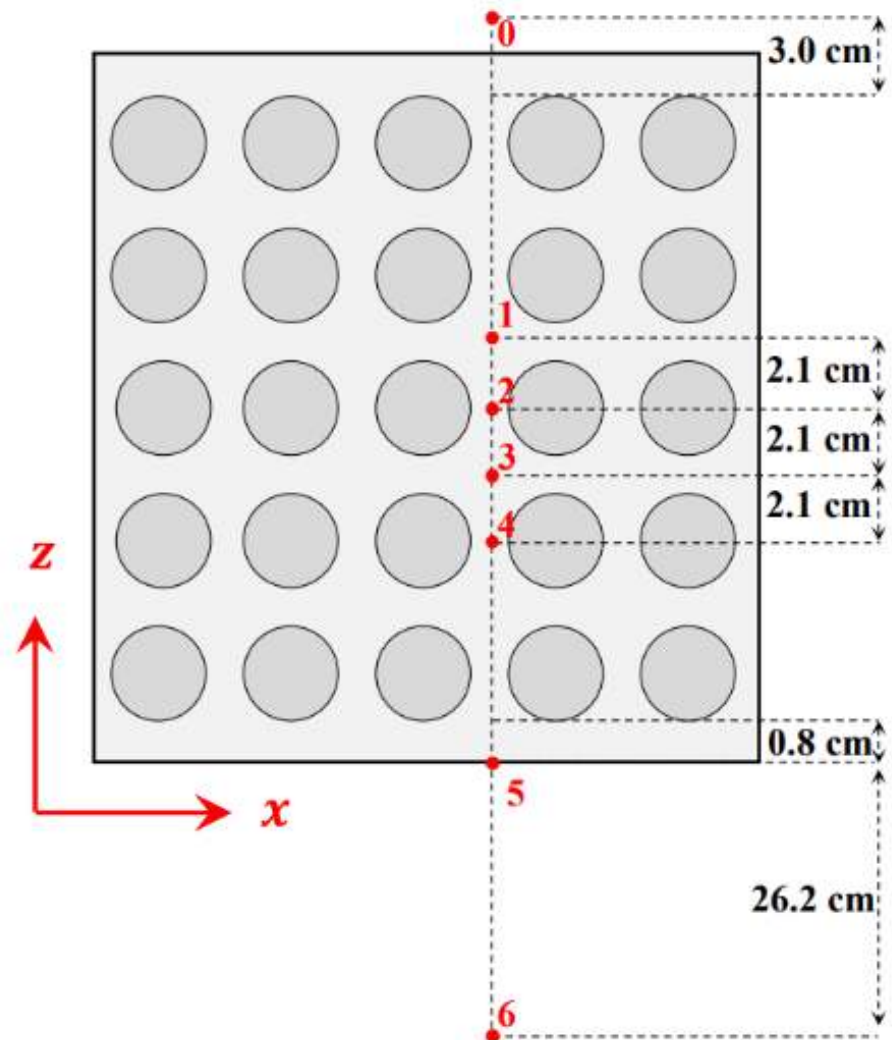
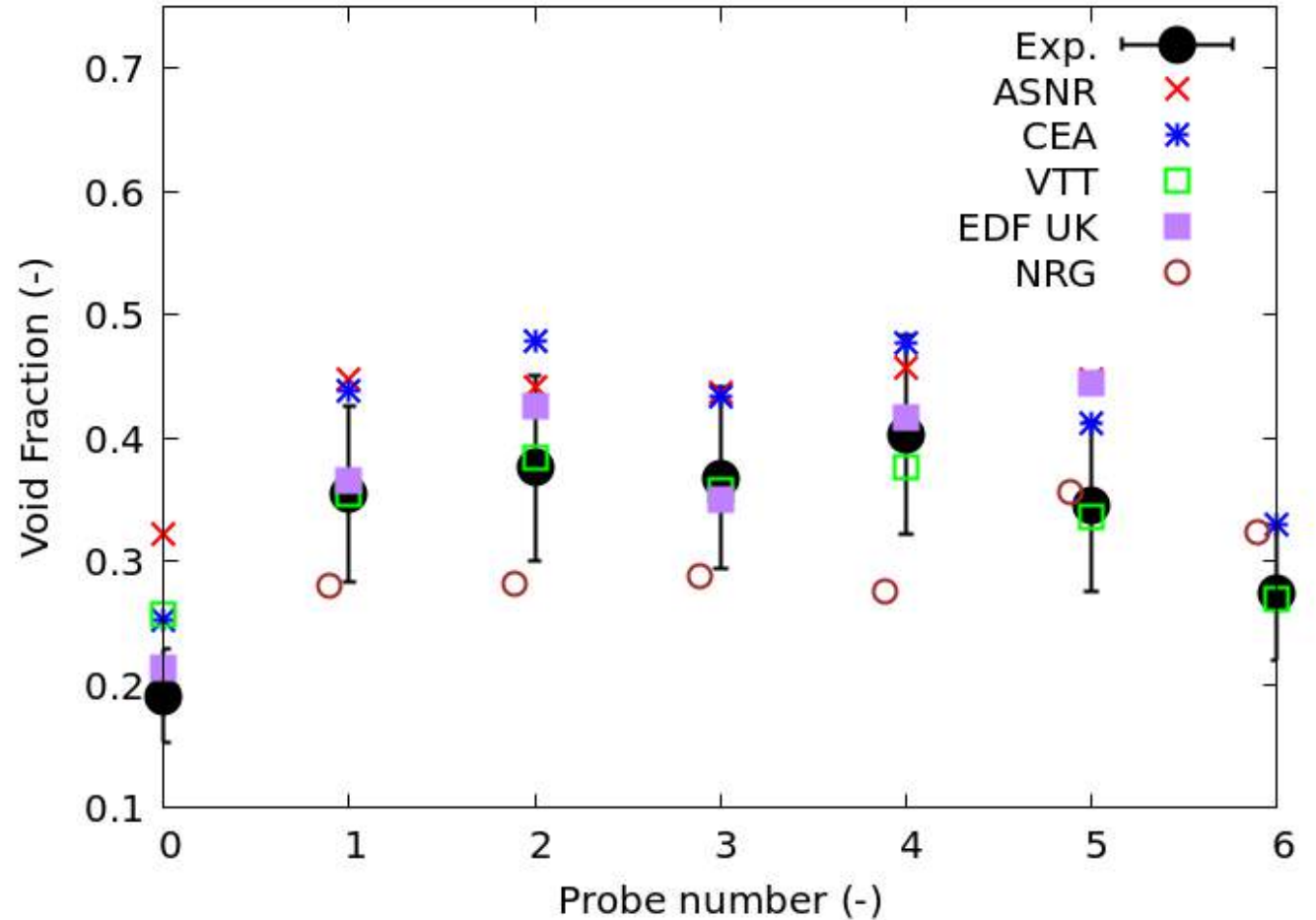


EDF UK

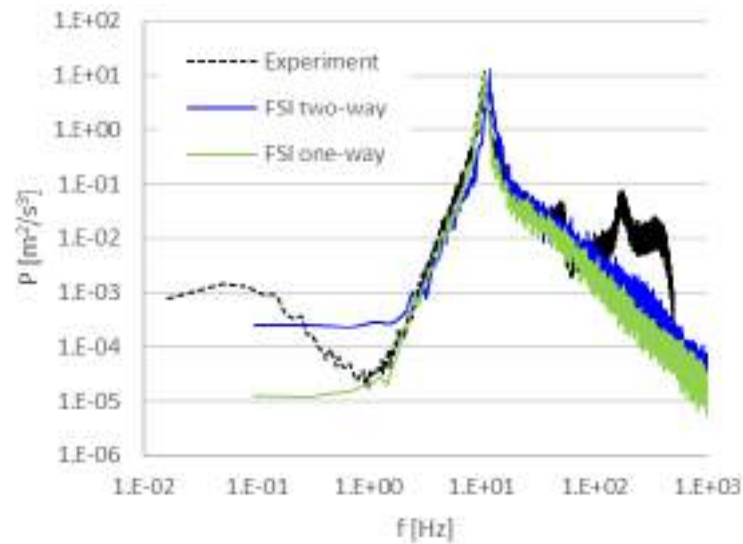
Numerical results - rigid



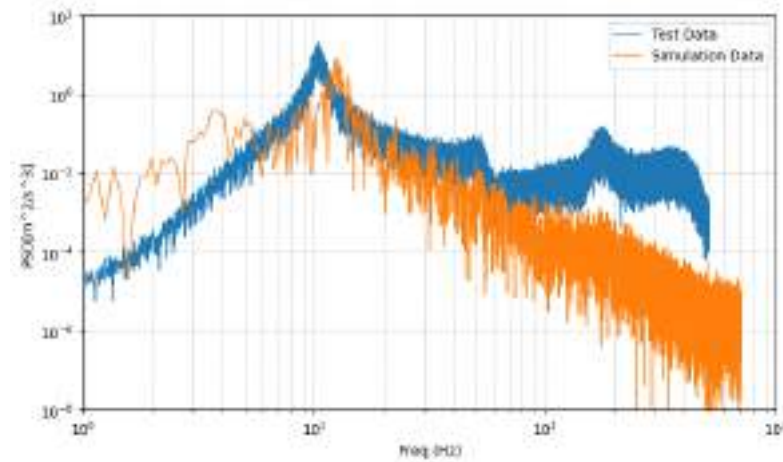
Numerical results - rigid



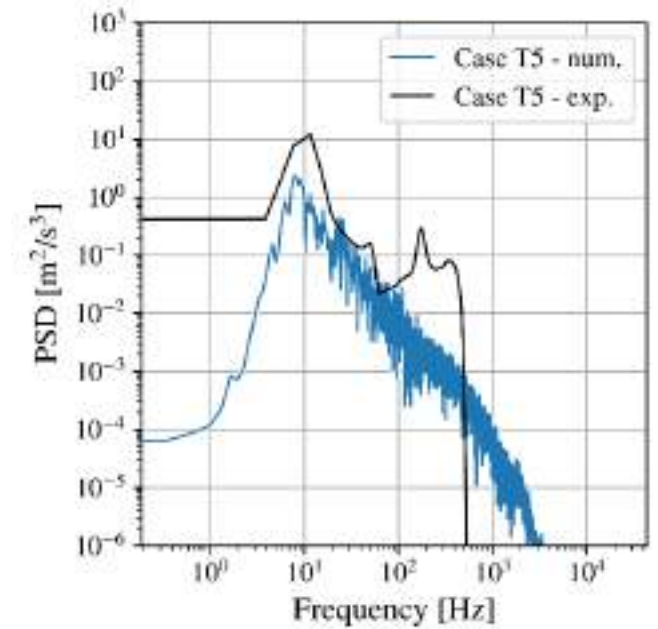
Numerical results - flexible



VTT

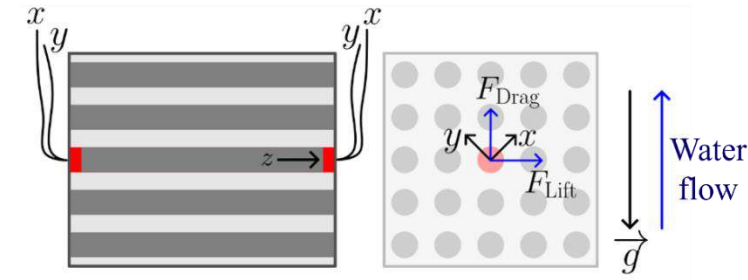


EDF UK

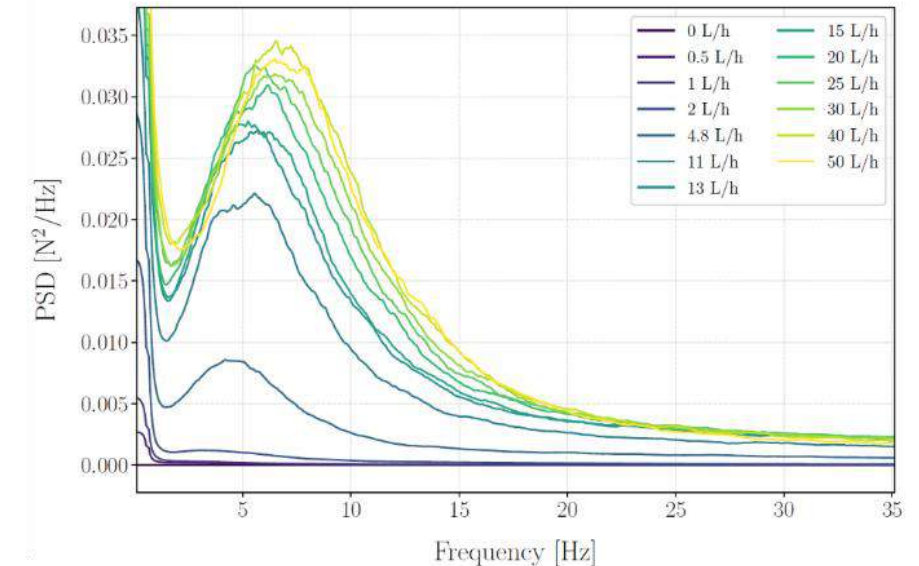


NRG

Conclusions & Perspectives



- Despite the range of numerical models and calculation domains, **quantitative and qualitative similarities between simulations strengthens confidence in the numerical predictions.**
- however **less agreement with the experiment than for the triangular arrangement** (see D4.6).
- **high slip between phases** in the present cases also pose a challenge for the two-phase flow prediction. Thus, having a different void distribution than the experiment appears to have a significant role in the tube bundle.
- similarly to the triangular case (D4.6), the **force fluctuation in two phase flow numerical simulation appears to be driven by the resolved gas structures**, highlighting correct prediction from VOF approaches without any treatment for the dispersed phase (oppositely to hybrid models).



- Still a lot of experimental data to use, force spectra or 2 flexible cylinders
- Partners worked a lot on inlet distribution within WP4, especially the SBM from UoGhent University, which is a promising way for boundary conditions.

Thank you to the participants and especially to the ASNR team for the discussion, answers and all the performed tests within Go Viking to improve CFD simulation: Daniele, Lucas, Giuseppe, Antonio and Emmanuel.

Communications



Antonio Chahine, Daniele Vivaldi, Guillaume Brillant, New experimental insights into two-phase air–water bubbly and intermittent flow in large duct and cross-flow configurations, International Journal of Multiphase Flow, 2025, <https://doi.org/10.1016/j.ijmultiphaseflow.2025.105266>.

Vivaldi, D, Chahine, A, & Rotily, L. "Experimental Study of the Two-Phase Flow and Induced Vibrations in a Square-Pitch Tube Bundle in Air-Water Cross-Flow." Proceedings of the . Volume 4: Fluid-Structure Interaction; High Pressure Technology. Montreal, Quebec, Canada. July 20–25, 2025. V004T04A018. ASME. <https://doi.org/10.1115/PVP2025-153890>

Giuseppe Spina, William Benguigui, Guillaume Brillant, Daniele Vivaldi, Catherine Colin, Local-scale experimental investigation of a two-phase cross-flow in a tube bundle and flow-induced vibration: Intermittent/Churn flow, Journal of Fluids and Structures, 2025, <https://doi.org/10.1016/j.jfluidstructs.2025.104390>.

Giuseppe Spina, William Benguigui, Guillaume Brillant, Daniele Vivaldi, Catherine Colin, Local-scale experimental investigation of a two-phase cross-flow in a tube bundle and flow-induced vibration; Bubbly flow regime, Nuclear Engineering and Design, 2025, <https://doi.org/10.1016/j.nucengdes.2025.113836>.

+ Conference papers in FIV2024, ICMF2025, PVP2025 and soon FIV2026 ☺ :

Two-phase flow-induced forces on tube bundles: experimental results from the Go-Viking project. Lucas Rotily, Daniele Vivaldi

Assessment of different interface tracking algorithms for morphology adaptive two-phase CFD: application to cross-flow. Peio Berrécochéa, Daniele Vivaldi, Dominique Legendre, Lucas Rotily

Numerical insights of the two-phase cross-flow inside an in-line 5×5 tube bundle configuration. Constantinos Katsamis, Bruce Kakimpa, Ehimen Iyamabo

Euler-Euler Simulation of Flow-Induced Vibrations in Tube Bundles Subject to Two-phase Cross Flow. Sina Tajfirooz, Edo Frederix, Kevin Zwijsen

Partners



Associated Partners



*EDF R&D UK Center



Thank you!



William Benguigui



EDF



william.benguigui@edf.fr

www.go-viking.eu



Funded by the
European Union



WP5: Development and validation of the Synthetic Turbulence Model for FIV analysis

Vladislav Filonov, [Yuliia Filonova](#)



Funded by
the European Union

GO-VIKING FINAL WORKSHOP 20-22 May 2026, Garching n. Munich, Germany

Content



- Starting point in development and implementation STG
- Considered methods of implementing STG for FIV analysis
 - SFA+STG
 - CFX+STG
- Validation on AMOVI experimental cases
- Main observations
- Computational efficiency
- Conclusions

Starting point in development and implementation STG



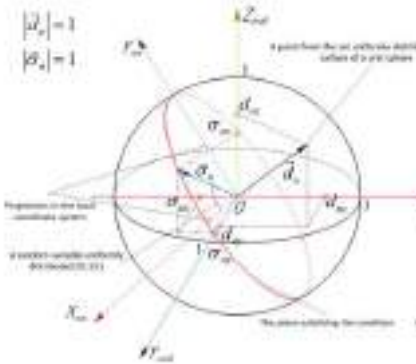
MAIN IDEA: creation of a special procedure for reconstructing the pulsation field based on Synthetic turbulence (based on RANS, generates a physically based field of turbulent pulsations)

- Procedure is based on local parameters that take into account only the averaged flow history
- the turbulent characteristics (correlations of velocity pulsations) are considered as isotropic
- the pulsation component of pressure is calculated based on a simple algebraic relationship, where it is assumed that pressure pulsation is a dynamic consequence of velocity pulsation

INSPIRED by the comparison of the velocity pulsation field generated using the synthetic turbulence method (1) with that calculated using the SRS method (2) from the literature (Adam'jan, 2011):

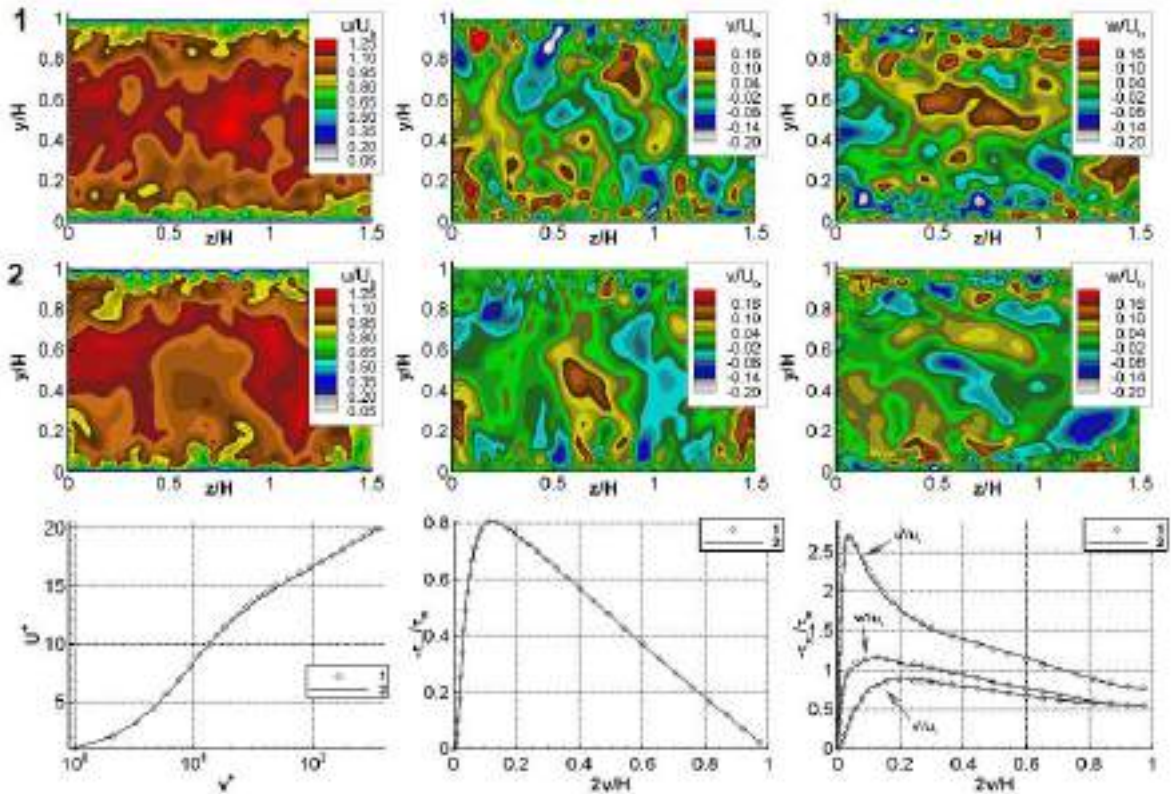
$$\vec{u}'(\vec{r}, t) = 2\sqrt{k_t(\vec{r})} \sum_{n=1}^N \sqrt{q_n(\vec{r})} \left(\vec{\sigma}_n \cos \left[k_n \vec{d}_n \vec{r} + \varphi_n + s_n \frac{t}{\tau} \right] \right)$$

Vector/scalar	Description
$\vec{d}_n = \{d_{nx}; d_{ny}; d_{nz}\}$	Basis vector, that determines the direction of the wave number, the direction of which is the radius vector of a point uniformly distributed on the surface of a unit sphere
$\vec{\sigma}_n = \{\sigma_{nx}; \sigma_{ny}; \sigma_{nz}\}$	Basis vector, that determines the direction of the velocity mode at a particular spatial point. The direction is determined by a random, uniformly distributed number in the interval $[0; 2\pi]$ provided that $\vec{d}_n \cdot \vec{\sigma}_n = 0$ (Fig. B.2)
φ_n	Mode phase number n , uniformly distributed in the interval $[0; 2\pi]$.
s_n	The dimensionless circular frequency of the mode with the number n , as a random variable with a normal distribution ($\mu = 2\pi; \sigma = 2\pi$)



Pressure pulsation:

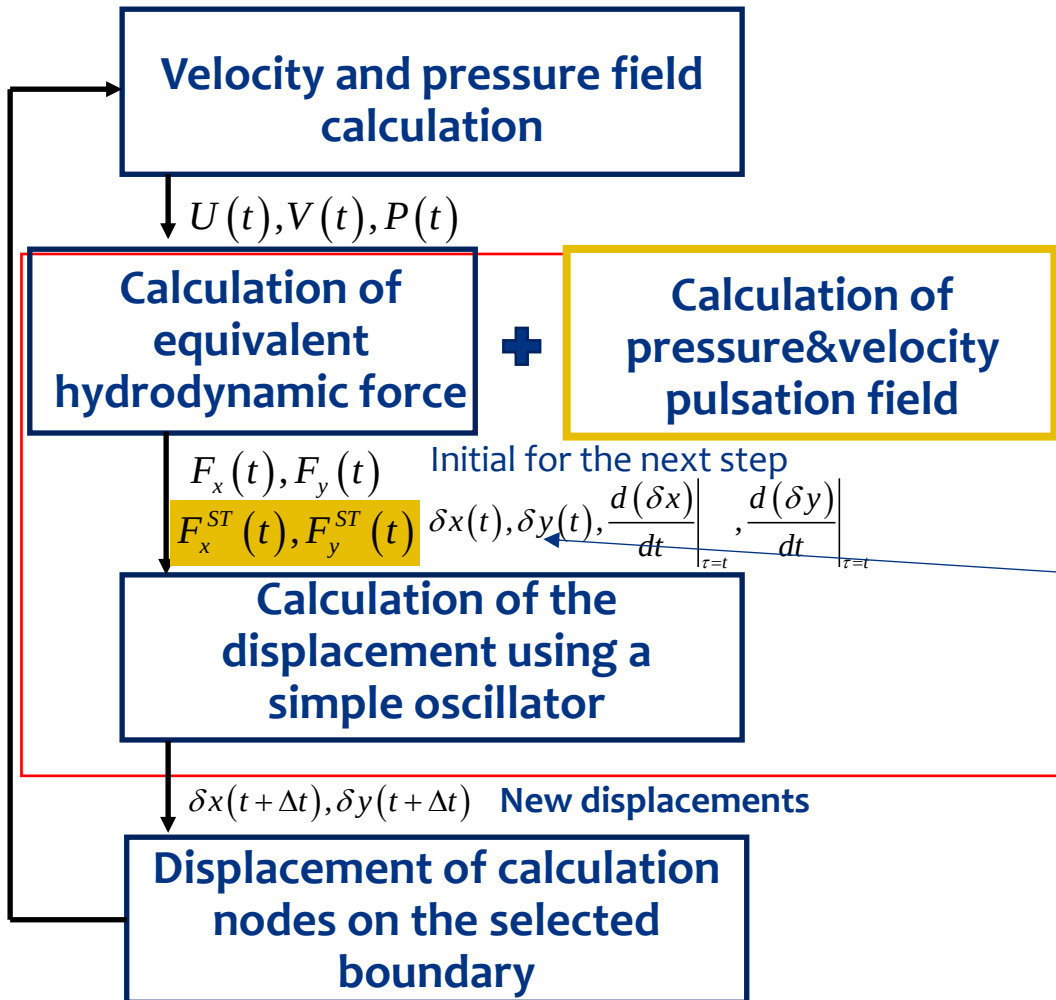
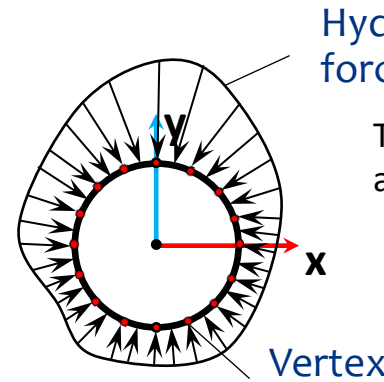
$$p'(r, t) - k_t = C \cdot \rho \left([u']^2 + [v']^2 + [w']^2 \right)$$



Starting point in development and implementation STG



Application of the Synthetic Turbulence procedure in order to generate turbulent pulsations, which can be taken into account when determining the **resultant force on the rod at FIV**.



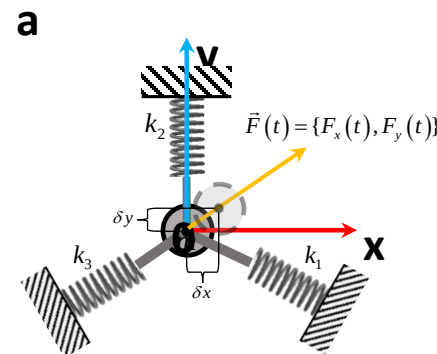
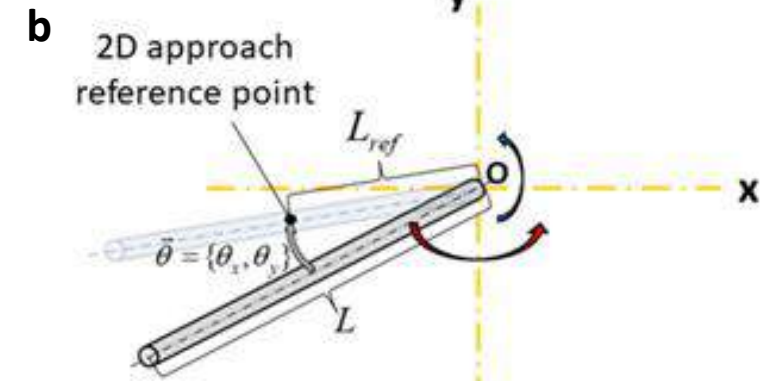
Hydrodynamic force profile

Calculation of equivalent force :

This form of equivalent force can be used for both the main field and the synthetic field

$$F(t) = \begin{cases} F_x = \sum_{i=1}^m \left(-p \cdot \cos(\phi_i) + \mu \left[2 \frac{\partial u}{\partial x} \cos(\phi_i) + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \sin(\phi_i) \right] \right) S_i^E \\ F_y = \sum_{i=1}^m \left(-p \cdot \sin(\phi_i) + \mu \left[\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \cos(\phi_i) + 2 \frac{\partial v}{\partial y} \sin(\phi_i) \right] \right) S_i^E \end{cases}$$

Oscillators (simple linear/ angular or 1D beam)



Considered methods of implementing STG for FIV analysis



ANSYS CFX + oscillator subroutine (FORTRAN) +STG (initially as external procedure)

STG – external procedure:

- the calculation itself was performed quite quickly, but more than 50-60% of the time was spent on retraining for a new cycle and updating the synthetic turbulence fields (the time step in CFX was taken to be smaller than the step for calculating displacements by the procedure)
- according to the results of test calculations, there was a need to synchronize iteration time steps in CFX with the updating of synthetic turbulence fields
- despite the 2D setting it **turned out to be not fast** and could be too much resource intensive for practical problems

problems with the RAM management when parallelizing

Implementation of **ST generation procedure** as an **internal ANSYS CFX library** based on User Fortran Function

SOLVED

OWN 2D code + reconstruction mesh

(potentially quick method)

- the special turbulence model, which is obtained by combining two RNG k-e equations into one equation for the transfer of eddy viscosity, was **successfully implemented**
- own 2D code with mesh reconstruction was adopted

fully coupled solver

(calculation step is the same for each of the subtasks including updating the mesh;
Time integration implicit of the first/second order.)

questions regarding the speeding up analysis

An idea of application the **Stable Fluid Algorithm** (usually used for physically based visualizations) extending its capabilities based on **deterministic vorticity source** and **Synthetic Turbulence**

SFA+STG for FIV analysis

The system of Reynolds equations for a **two-dimensional incompressible isothermal formulation**

$$\begin{cases} \frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0 \\ \frac{\partial U}{\partial t} + U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \eta_{eff} \left(\frac{\partial U^2}{\partial x^2} + \frac{\partial U^2}{\partial y^2} \right) + gn_x + S_U \\ \frac{\partial V}{\partial t} + U \frac{\partial V}{\partial x} + V \frac{\partial V}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \eta_{eff} \left(\frac{\partial V^2}{\partial x^2} + \frac{\partial V^2}{\partial y^2} \right) + gn_y + S_V \end{cases} \quad (1)$$

decomposed

$$\begin{cases} \frac{\partial U}{\partial t} = \frac{\partial u}{\partial t} = \alpha u + \beta u_0 + \gamma u_{00} = -\left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y}\right) + \eta_{eff} \left(\frac{\partial u^2}{\partial x^2} + \frac{\partial u^2}{\partial y^2} \right) + gn_x + S_U \\ \frac{\partial V}{\partial t} = \frac{\partial v}{\partial t} = \alpha v + \beta v_0 + \gamma v_{00} = -\left(u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y}\right) + \eta_{eff} \left(\frac{\partial v^2}{\partial x^2} + \frac{\partial v^2}{\partial y^2} \right) + gn_y + S_V \end{cases} \quad (2)$$

solved at time step using a physical decomposition
(stepping algorithm)

$$U(t) \xrightarrow{\text{source}} u_1 \xrightarrow{\text{advection}} u_2 \xrightarrow{\text{diffusion}} u_3 \xrightarrow{\text{projection}} U(t + \Delta t)$$

the advection terms (2) can be represented in the form of **Lagrangian self-advection**, so the system splits into two independent diffusion equations

$$u_2(x, y) = u_1^*(x - u_1 \Delta t, y - v_1 \Delta t) \cdot BC_{correction}$$

$$v_2(x, y) = v_1^*(x - u_1 \Delta t, y - v_1 \Delta t) \cdot BC_{correction}$$

the **diffusion in the velocity field** for each of the components is included by solving the equations

$$\begin{cases} u_3 - u_2(x, y) = \Delta t \cdot \eta_{eff} \left(\frac{\partial^2 u_3^2}{\partial x^2} + \frac{\partial^2 u_3^2}{\partial y^2} \right) \\ v_3 - v_2(x, y) = \Delta t \cdot \eta_{eff} \left(\frac{\partial^2 v_3^2}{\partial x^2} + \frac{\partial^2 v_3^2}{\partial y^2} \right) \end{cases}$$



Projection

$$\begin{aligned} \frac{\rho}{\Delta t} \nabla \cdot u_3 &= \Delta p \\ U(t + \Delta t) &= u_3 - \frac{\Delta t}{\rho} \nabla p \cdot BC_{correction} \end{aligned}$$



For ST

$$\frac{\partial \eta}{\partial t} + U \nabla \eta = \nabla \left(\frac{\eta}{Pr_k} \nabla \eta \right) + (\eta_{eff} - \eta) \sqrt{C_v^3} (C_{1\varepsilon}^* + C_{2\varepsilon}) S - \frac{4C_v^3 \eta^2}{S^2 Pr_k} (\nabla S)^2$$



A **significant restriction** is imposed on the calculation procedure:
It is assumed that the generation and dissipation of turbulence are in equilibrium.

$$(\varepsilon = \eta_t S^2).$$

In result, by performing relatively simple operations consecutively, the **basic parameters of a two-dimensional flow can be obtained.**



SFA+STG for FIV analysis



In contrast to classic **Stable Fluid Algorithm** the computational model is complemented by the **effective viscosity calculation**, that based on **theoretical one-parameter RNG turbulence model**

SYNTHETIC TURBULENCE GENERATION:

- The **fluctuation field** is generated at each computational point **based on probability values** that are uniform across the computational domain.
- To **transfer the velocity fluctuation fields**, a modified form is used (better corresponds to the correlation property of the generated vortices):

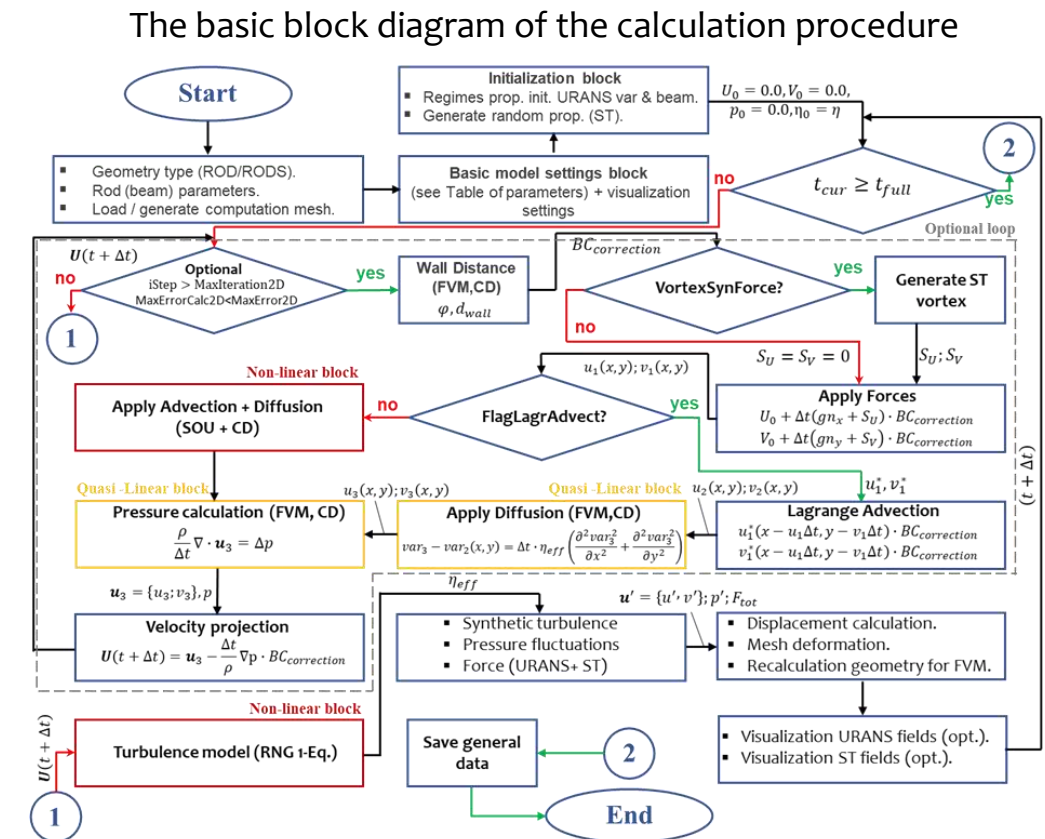
$$u' = \{u', v'\} = A[k]\sqrt{6} \sum_{n=1}^N \sqrt{q_n} \left[\sigma_n \cos \left(k_n d_n r + \varphi_n - \frac{2\pi t}{k_n L_{ref}} d_n U \right) \right]$$

- The definition of **static pressure pulsations** (the main source of unevenness in flow around a body) is based on the well-known Poisson equation (**governs induced pulsations from the moving boundary**)

$$\nabla^2 p' = -\rho [2\nabla U : (\nabla u')^T + \nabla \cdot \nabla (u' \otimes u' - R_{URANS})]$$

Relations	Key
Algebraic relations ¹	
$p' = C_{st}\rho(u' \cdot u')$	(K10)
$p' = C_{st}\rho \cdot sgn(u')^T (u')^2$	(K11)
Algebraic relations of gradient type²	
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} \nabla u' : (\nabla u')^T;$ $\max(k_{cut})^{-2} = L_{ref}^2$	(K20)
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} (\nabla u')^2$	(K21)
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} sgn(\nabla u')^T (\nabla u')^2$	(K22)
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} \nabla U : (\nabla u')^T$	(K30)
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} \nabla U \cdot \nabla u'$	(K31)
Notes: ¹ applied to the centers of control volumes adjacent to the wall. ² applied to boundary nodes located on the wall. The characteristic size L_{ref}^2 can be selected individually for each CV. C_{st} - amplification factor, for (K10)-(K11) the typical value is from 0.5 to 1.5. Generally, requires calibration.	

After all components of the average and synthetic fields have been calculated, the **forces acting on the rod determined**.



CFX+STG for FIV analysis



- **Two libraries** are developed – for **Synthetic Turbulence Generator** and for the **FIV analysis (USER BEAM LIBS)**
- Different **options for analysis of FIV** are available:
 - previously used analytical solution of the **Euler Bernoulli Beam**
 - analytical solution of the **Timoshenko Beam**
 - **FDM** implementation of the **Euler Bernoulli Beam** (no preliminary determined frequencies)

“rigid” or predetermined by the spectral feature*

Within the **FDM solver**, a **two-section** (each section is homogeneous) **Euler-Bernoulli beam** is implemented, which can move along a single selected direction

- **Synthetic Turbulence is generated in volume (principally 3D)** with an algebraic estimate of pressure pulsations
- The library supports **parallel calculations**
- The **user options for calculations configuring** are provided, which are accessible by declared names through User Input

The library provides for the **calculation of the force around the rod at a distance exceeding the momentum displacement layer**, which **makes it possible to overcome the issue of zero velocity pulsations near the wall**

*nonlinear interactions (especially vortex structure and separation points) can generally be “suppressed” by the frequency that is embedded in the oscillator

Validation on AMOVI experimental cases



The AMOVI experimental cases were considered for the validation of considered STG application strategies for FIV analysis

	Flow Rate (L/s)	Pressure (bar)	Temperature (°C)	PIV/LDV data available
Single flexible tube (lift direction)	0.5	2.19	18	-
	1.7	3.33	21	-
	2.9	3.20	25	-
	4.0	3.53	26	-
Flexible tube (lift direction) in a rigid tube bundle	0.5	1.48	17	-
	1.1	0.9	17	-
	1.6	1.07	17	PIV
	1.95	3.17	23	-

The **most interesting regimes** are those with **maximum flow rates**, since in these cases, the interaction between the moving wall and the flow is fundamentally coupled

SFT-170

SFT-290

SFT-400

FTRTB-160

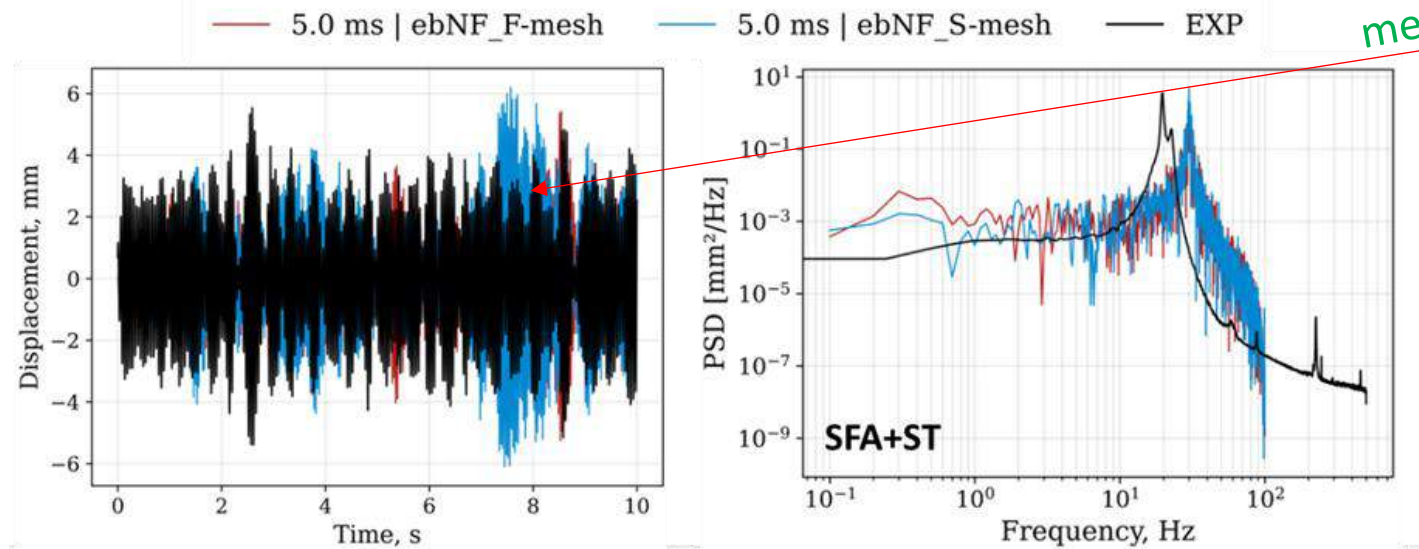
FTRTB-195

Both the **physical adaptation of the Stable Fluid Algorithm (SFA)** based on synthetic turbulence and the **User library in ANSYS CFX** with various algebraic expressions for pressure pulsations were tested in such regimes

- **SFA+ST simulations used default settings** selected based on preliminary testing (Lagrangian advection is used). **No calibration coefficients or correction parameters** were selected based on the considered or any other experiments
- **ANSYS CFX+ST** uses a similar calculation meshes, which additionally utilized prismatic meshing near the wall. The **SST turbulence model, first-order time integration**, and the **High Resolution advection** computational model were applied. Random parameters for the synthetic turbulence generator were predefined and uniform across all cases

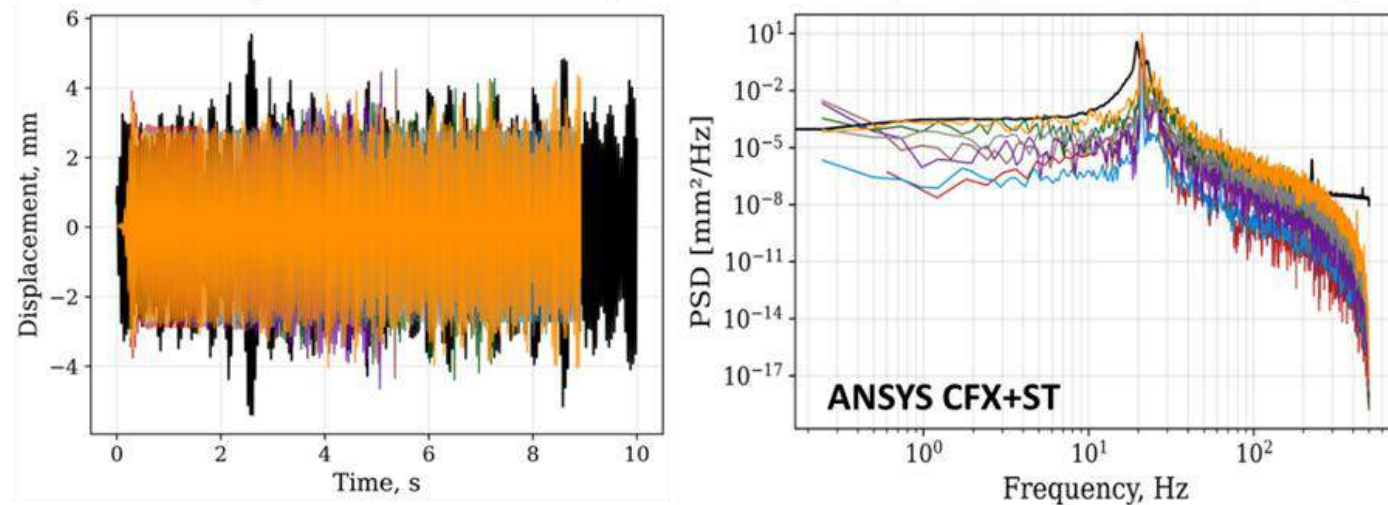
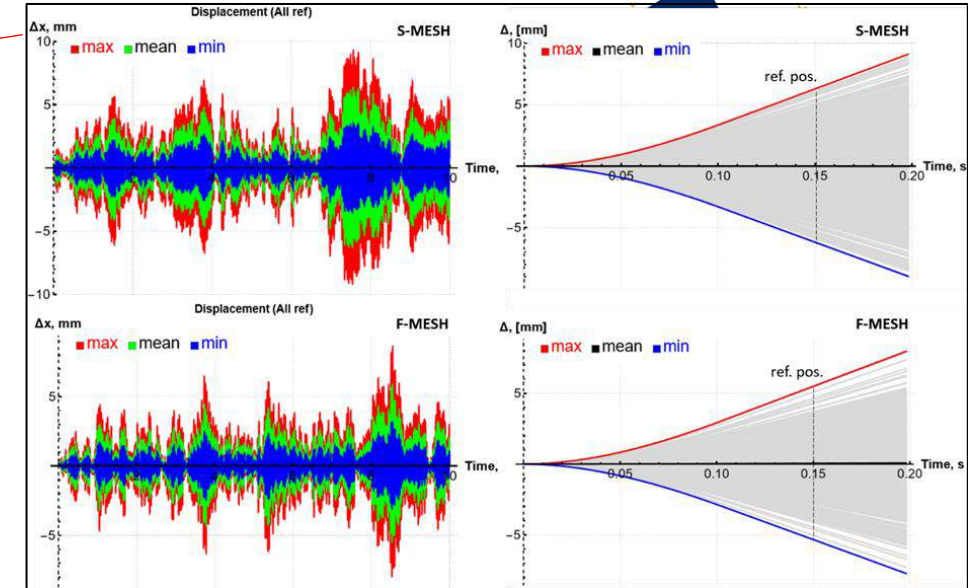
Validation on AMOVI experimental cases

SFT-400



EXP 1.0 ms | FDMK_(K11) 1.0 ms | FDMK_(K21) 1.0 ms | FDMK_(K30)

1.0 ms | FDMK_(K10) 1.0 ms | FDMK_(K20) 1.0 ms | FDMK_(K22) 1.0 ms | FDMK_(K31)



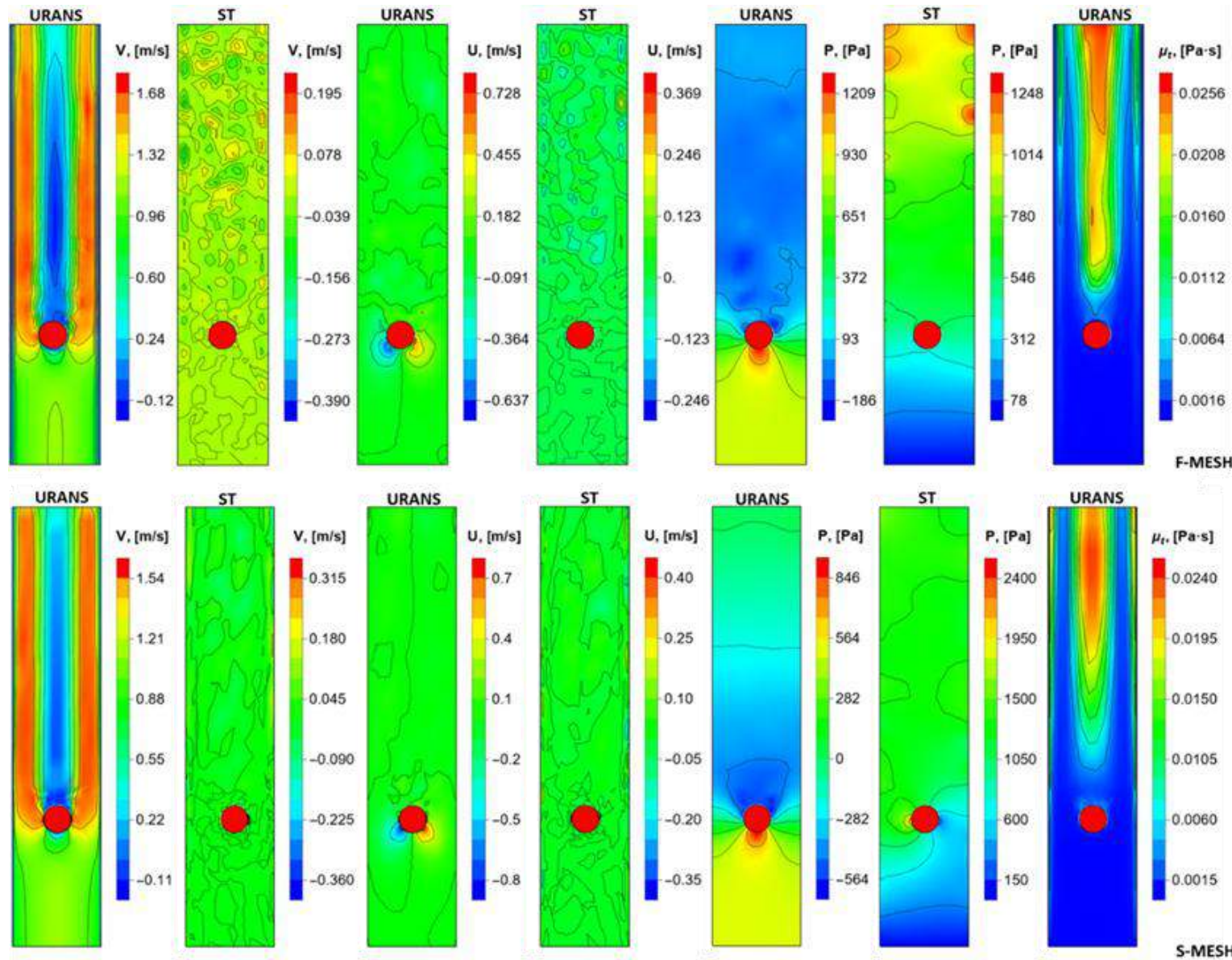
SFA+ST generally **satisfactorily predict** the amplitude values of the coupled vibrations. The **peak in the spectrum is slightly shifted toward higher frequencies** compared to the experiment, which is explained by the "effect" of pre-assigned frequencies when solving using the normal form method

ANSYS CFX + ST for various pressure pulsation estimation options also indicate **satisfactory vibration prediction** compared to experiment

Options related to **gradient algebraic expressions** (K20-K22; K30-K31) for **pressure pulsations** **turned out to be the most applicable**, although their physical behavior still requires careful interpretation.

SFT-400

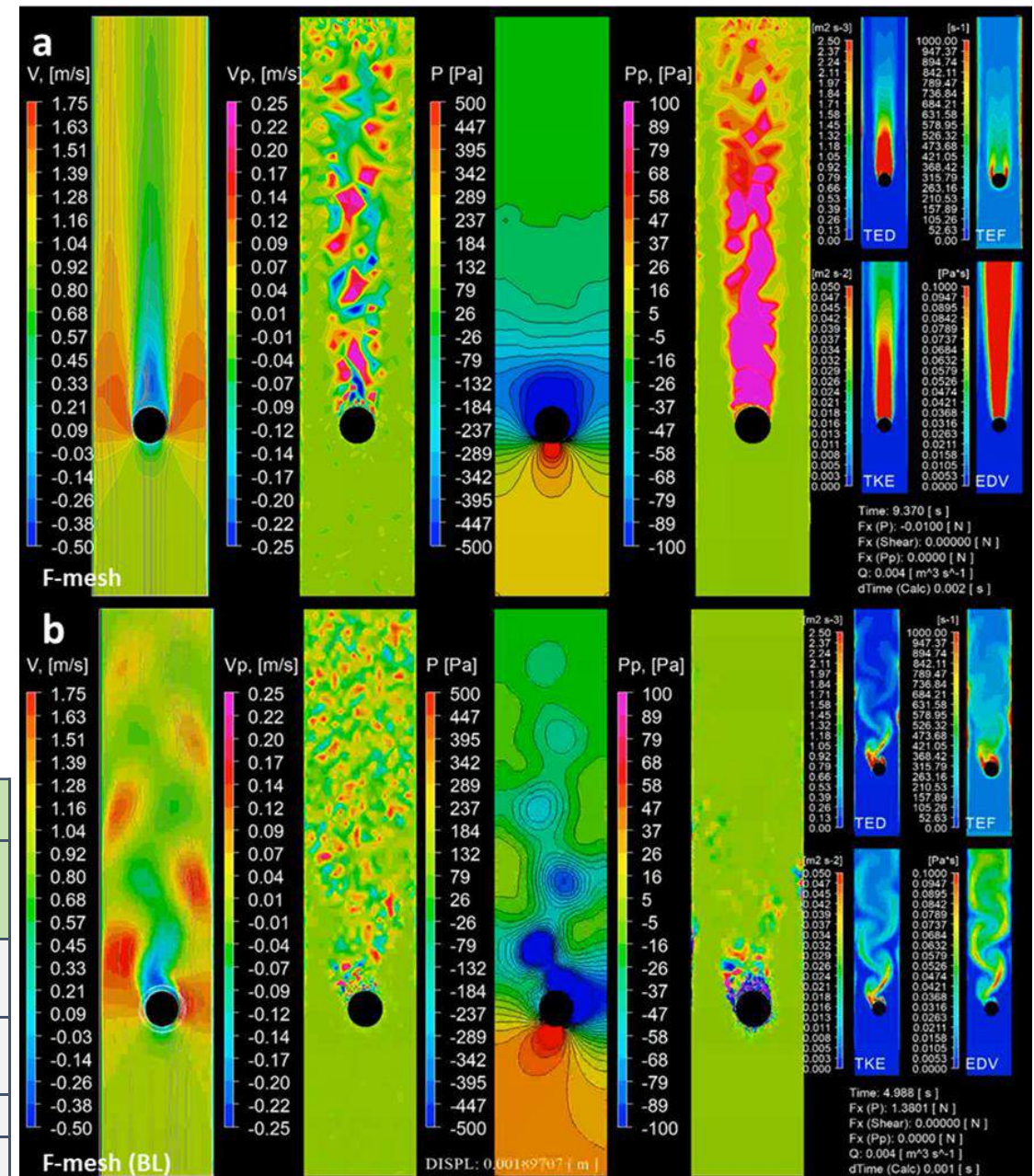
SFA+ST



- SFA+ST procedure the flow is predominantly diffusive due to the algorithm, but the **coupling with the turbulence model almost always excites complex oscillations**.
- In regimes with relatively high flow rates, the **force acting on the rod is mainly determined by the synthetic one-parameter field**.

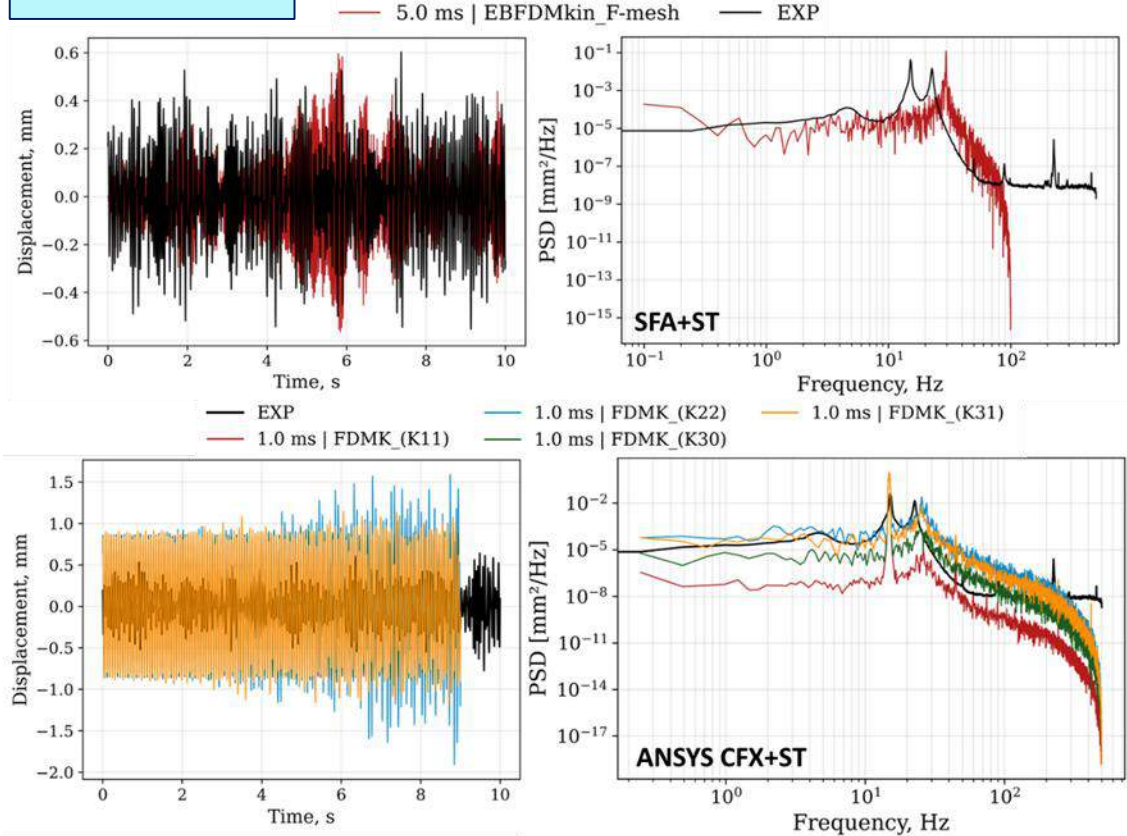
- Using in **ANSYS CFX+ST** the larger **time steps** (> 1.0 ms) results in a flow that is over-diffusive (**a**), as displayed by the eddy viscosity profile, which is similar to the one-parameter RNG turbulence model used in **SFA+ST**. In this case, **oscillations are practically not excited**.
- If the **integration time step is less than or equal to 1.0 ms** (**b**), then oscillations are excited and the eddy viscosity profile differs significantly from the calculation procedure.

	EXP	SFA+ST (EBNF)		ANSYS CFX+ST (FDMK)						
		F-mesh	S-mesh	K10	K11	K20	K21	K22	K30	K31
Displacement RMS, mm	1.945	1.37	1.89	2.04	1.92	1.97	1.98	1.93	1.93	1.98
Max. displacement, mm	5.4	5.3	6.07	3.91	3.59	4.53	4.46	4.27	3.61	4.37
Peak#1, Hz	19.53	29.9	29.9	21.16	20.99					
Peak#2, Hz	22.7	-	-	25.1	24.9					



Validation on AMOVI experimental cases

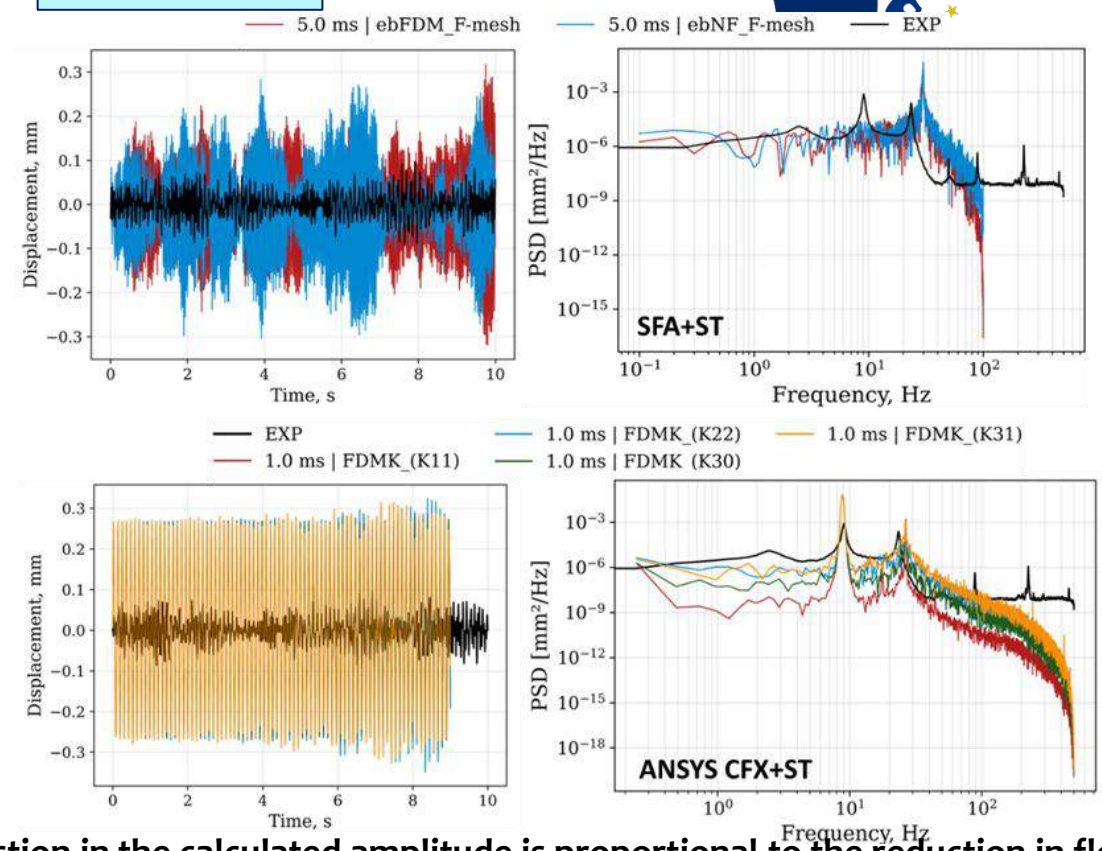
SFT-290



No proportional decrease in the RMS oscillation amplitude compared to the overall flow rate is **observed using both SFA+ST and ANSYS CFX+ST**, which generally indicates **sensitivity to fluid-elastic instability**

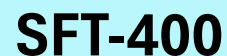
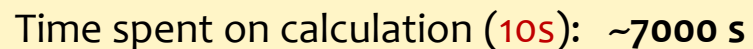
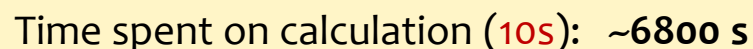
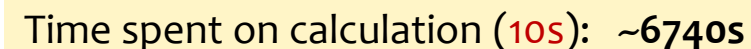
	EXP	SFA+ST (FDMK)	ANSYS CFX+ST (FDMK)			
			K11	K22	K30	K31
Displacement RMS, mm	0.205	0.202	0.600	0.623	0.601	0.611
Max. displacement, mm	0.6	0.6	0.86	1.59	0.96	1.15
Peak#1, Hz	15.14	-	14.89			
Peak#2, Hz	22.71	28.1	25.39			

SFT-170

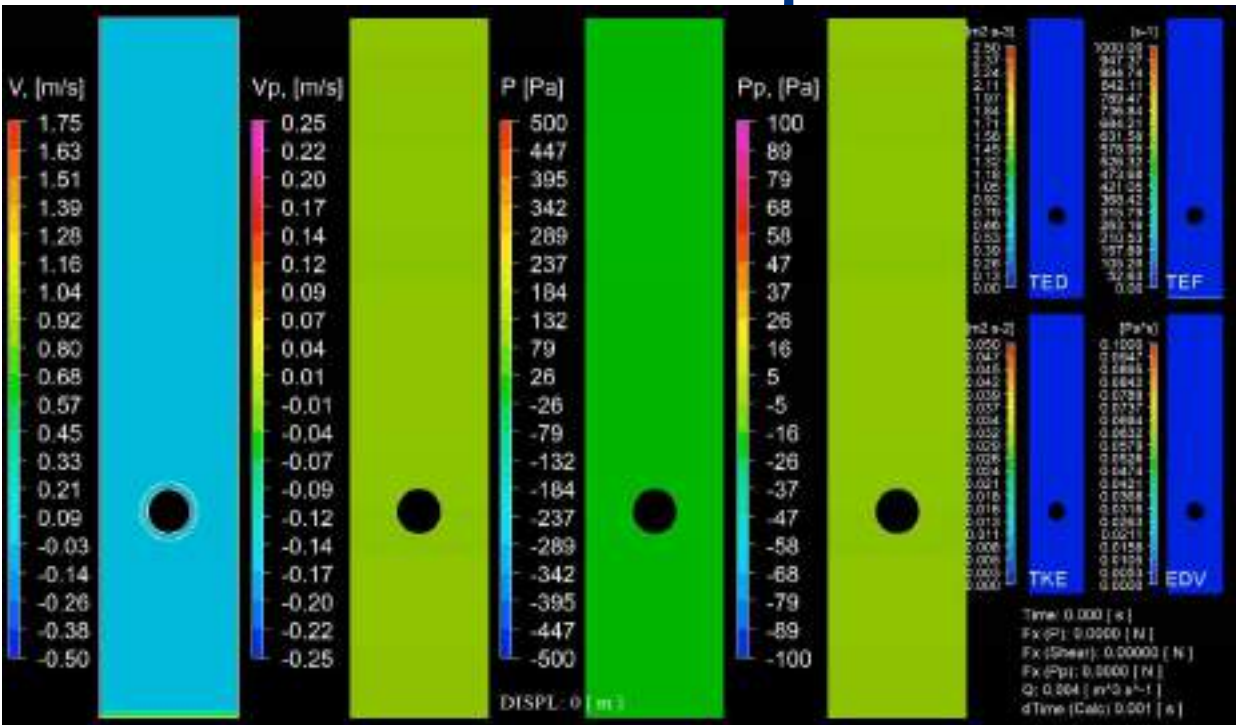


The **reduction in the calculated amplitude is proportional to the reduction in flow rate**, which indicates a sufficient distance from the region of possible resonance, although, **compared to the experiment, the rod displacements are overestimated**

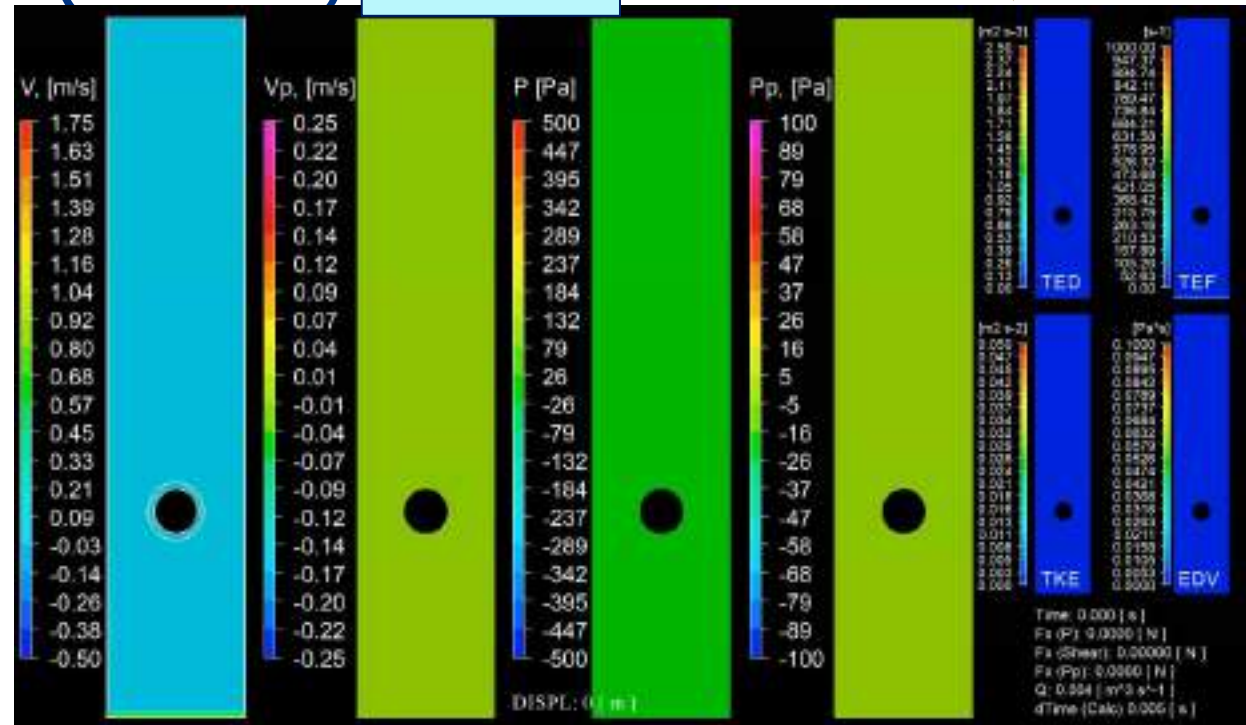
	EXP	SFA+ST		ANSYS CFX+ST (FDMK)			
		EBNF	FDMK	K11	K22	K30	K31
Displacement RMS, mm	0.028	0.118	0.089	0.165	0.166	0.165	0.166
Max. displacement, mm	0.1	0.29	0.31	0.27	0.32	0.29	0.31
Peak#1, Hz	9.03	-	-	8.79			
Peak#2, Hz	23.44	28.4	27.5	26.61			



Validation on AMOVI experimental cases (CFX+ST) SFT-400

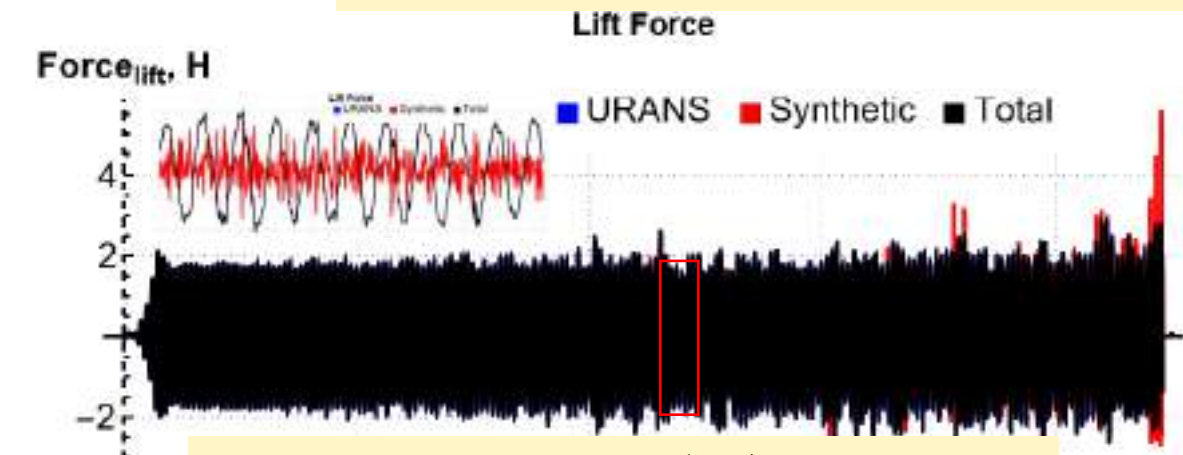


CFX+ST ($\Delta t \leq 1.0$ ms, FDMK, K31, $Q=4.0$ l/s)

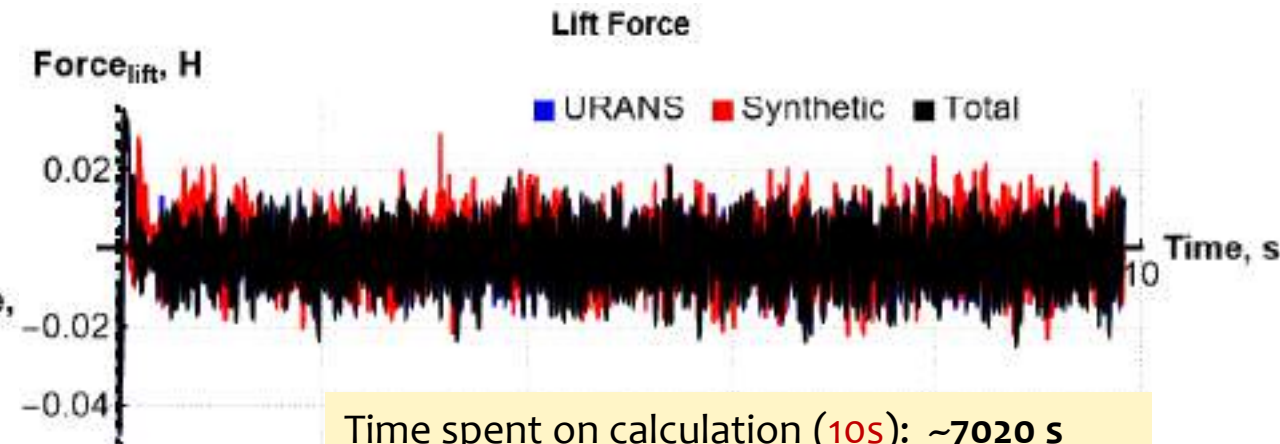


CFX+ST ($\Delta t \geq 5.0$ ms, FDMK, K10, $Q=4.0$ l/s)

Due to the large time step, the flow is too "diffusive," and oscillations are not excited.

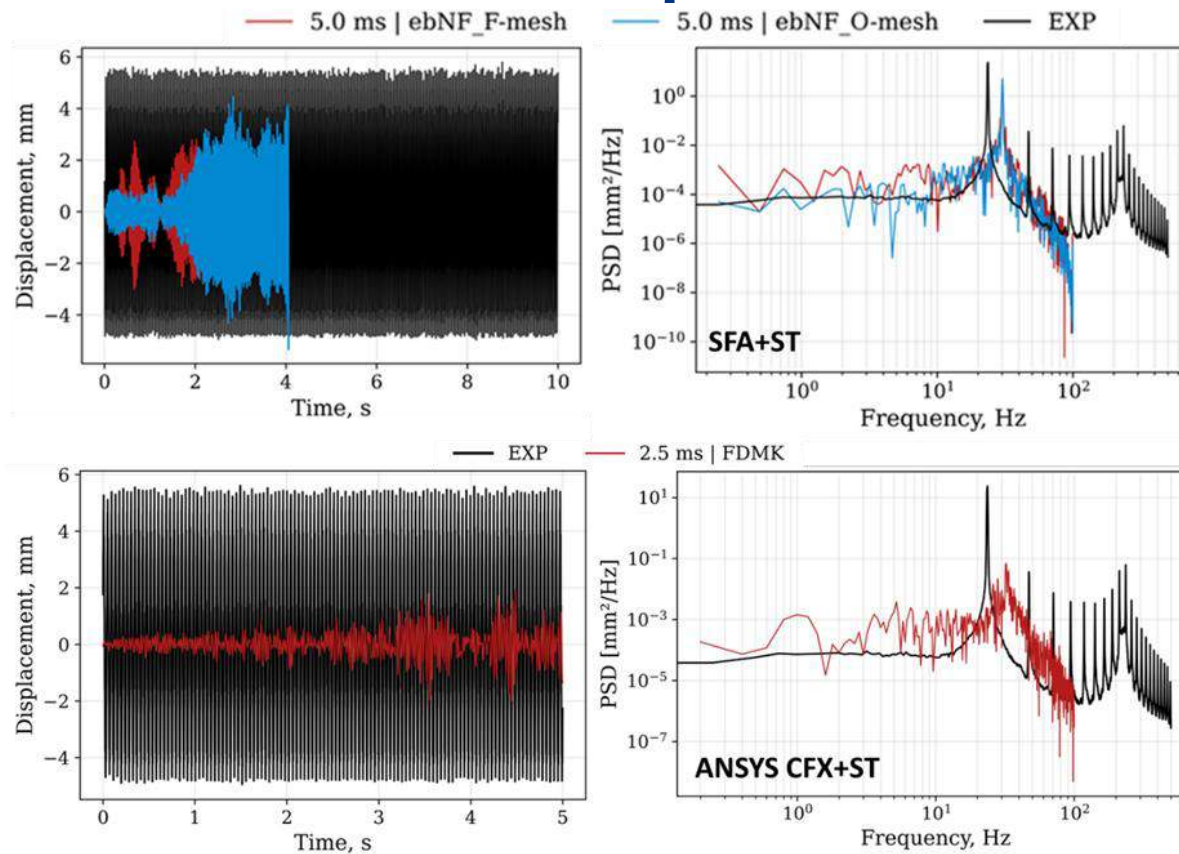


Time spent on calculation (10s): ~35100 s



Time spent on calculation (10s): ~7020 s

Validation on AMOVI experimental cases



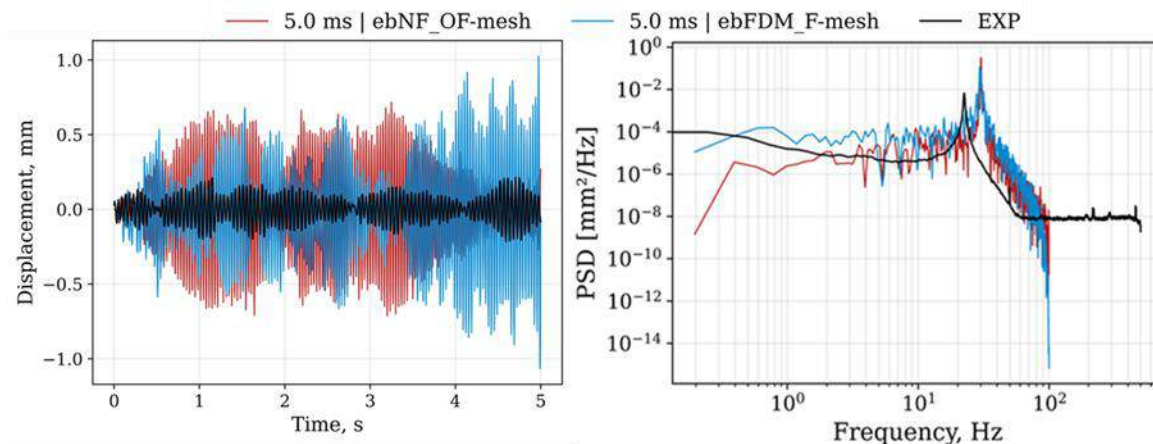
FTRTB-195

- SFA+ST was **able to predict large amplitudes**, but sensitivity to the set of random parameters pre-generated for the synthetic turbulence procedure was observed. This influenced by the physical time required to transition to oscillations with large displacements.
- SFA+ST **calculations were terminated** when a zero or negative control volume was reached.
- When applying **ANSYS CFX+ST**, vibrations were excited, but with a **significantly lower amplitude**.
- Moreover, **peak frequency shifts were observed in both approaches** relative to the experiment.

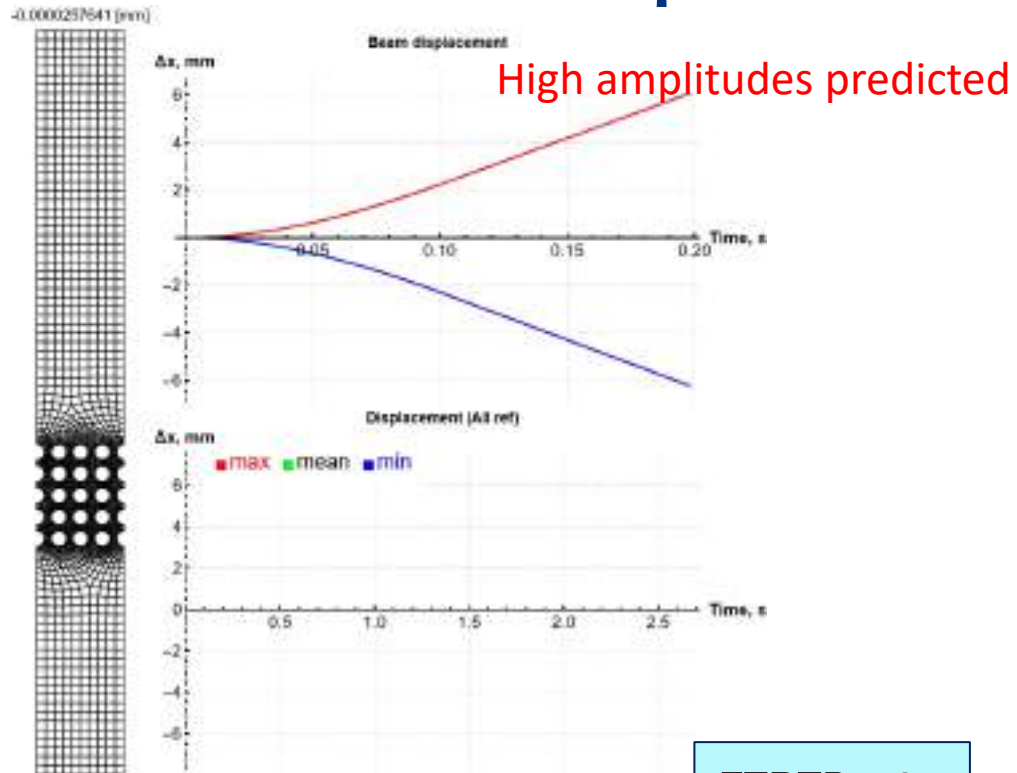
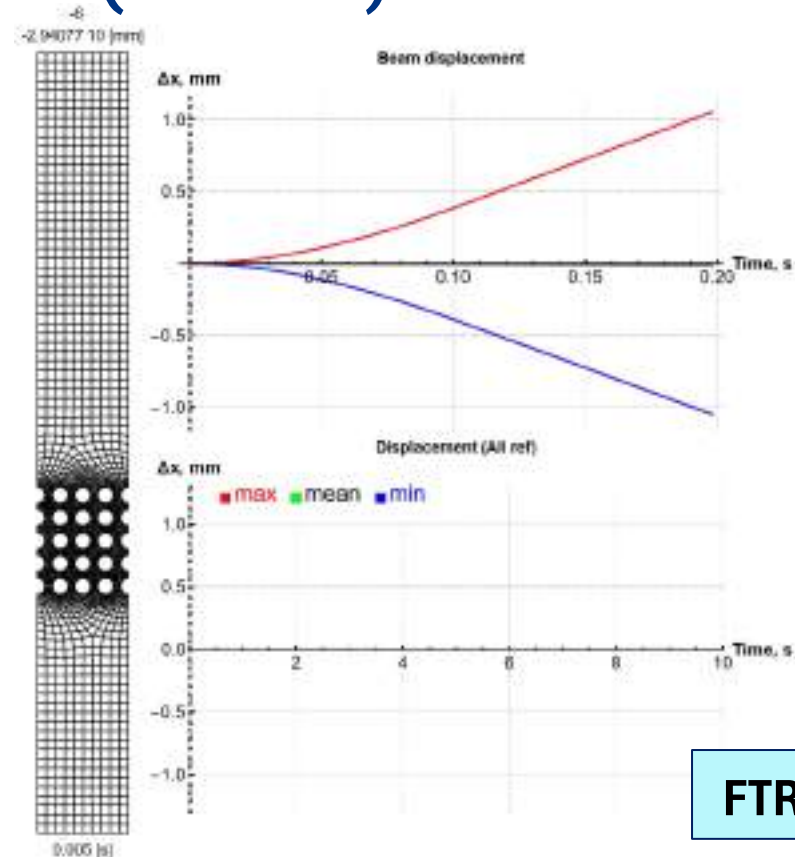
FTRTB-160

The predicted **amplitudes obtained for $Q = 1.6$ l/s are higher than the experimental ones**

However, while the flow rate through the assembly decreases, **the amplitude decreases nonlinearly, indicating sensitivity to resonant coupled oscillations**



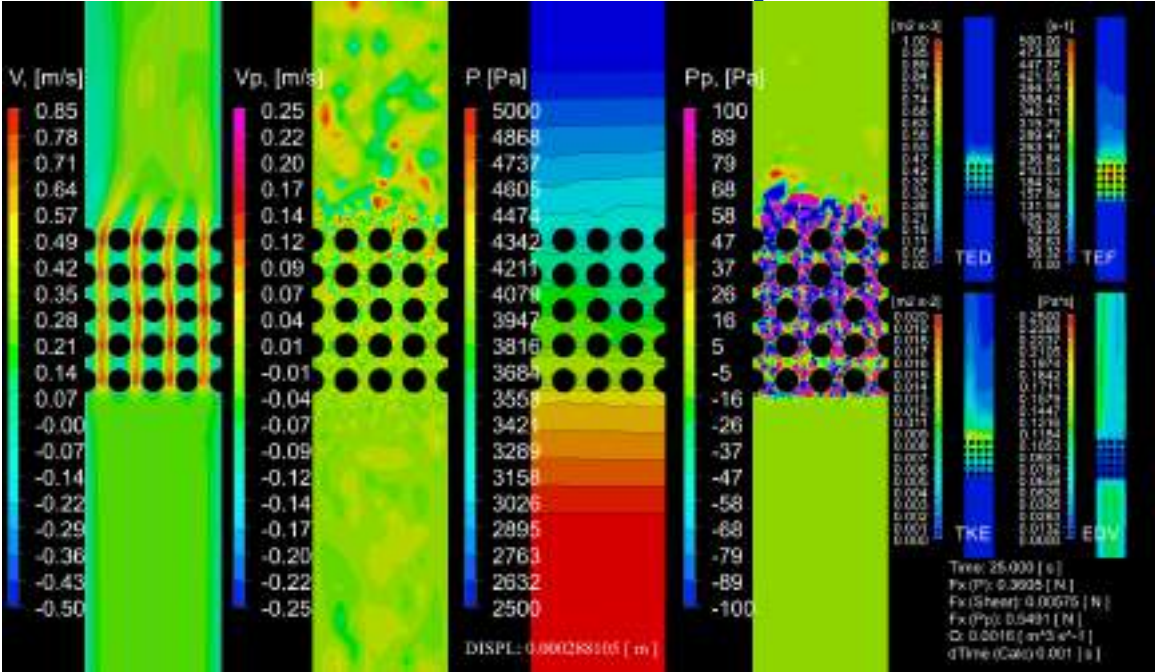
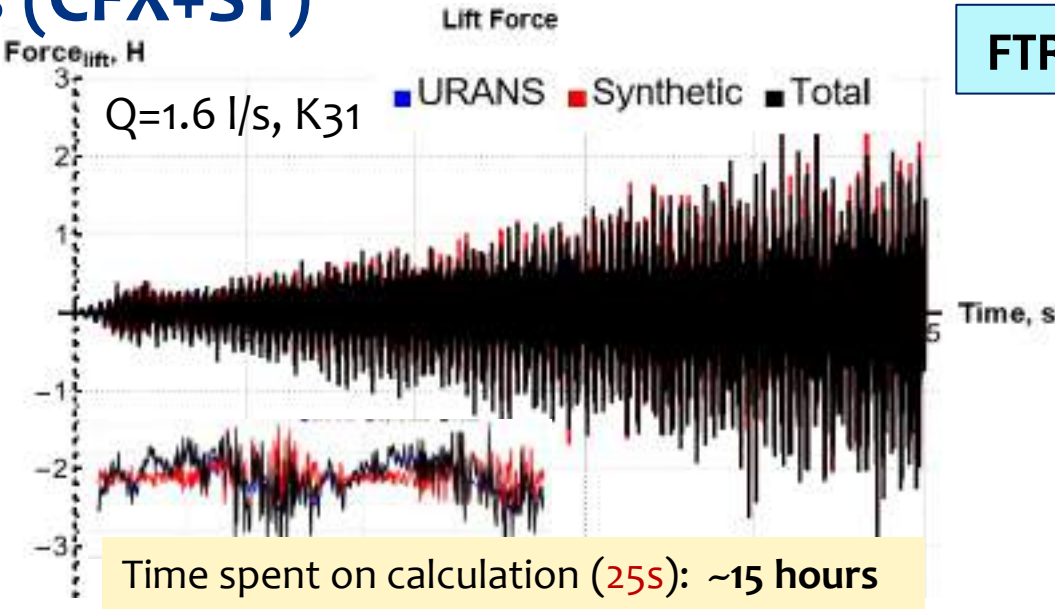
Validation on AMOVI experimental cases (SFA+ST)



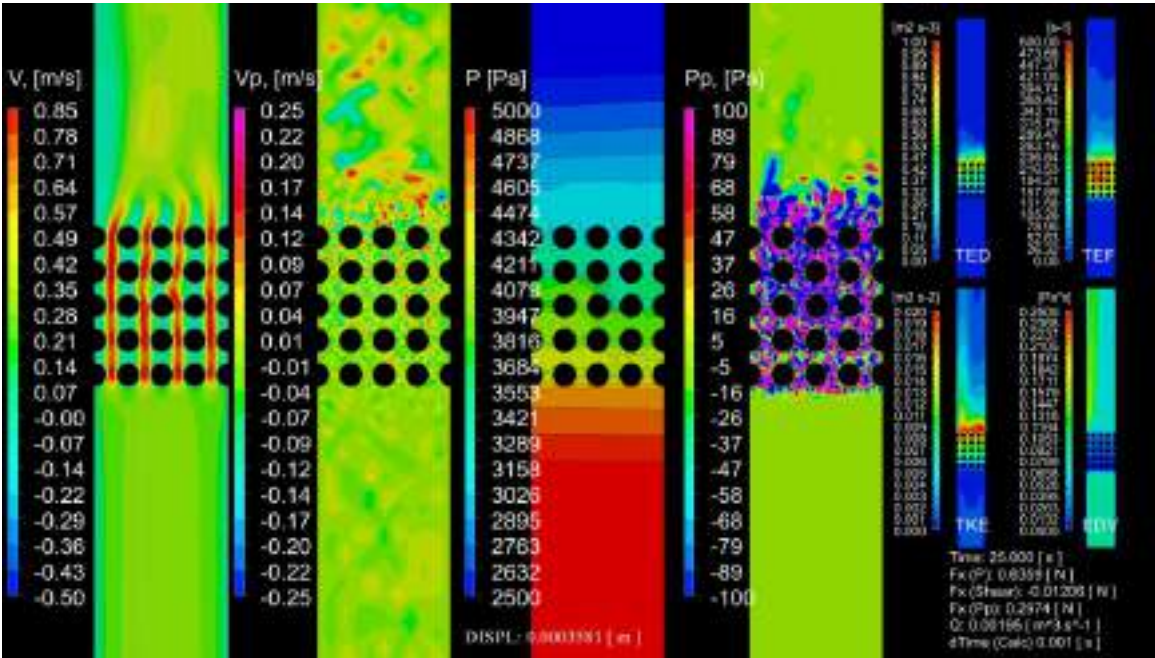
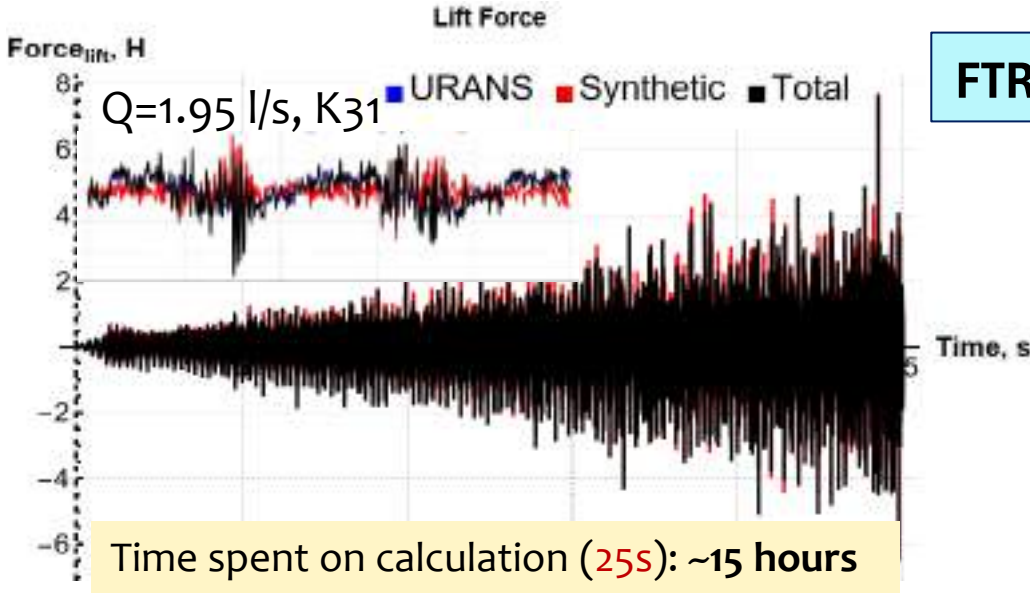
Time spent on calculation (10s): ~8937 s

Time spent on calculation (10s): ~11000 s

Validation on AMOVI experimental cases (CFX+ST)



FTRTB-160



FTRTB-195

The force is not stabilized after the 25s of transient – this means that the rod will continue to oscillate. The Synthetic force component is dominant

Main observations

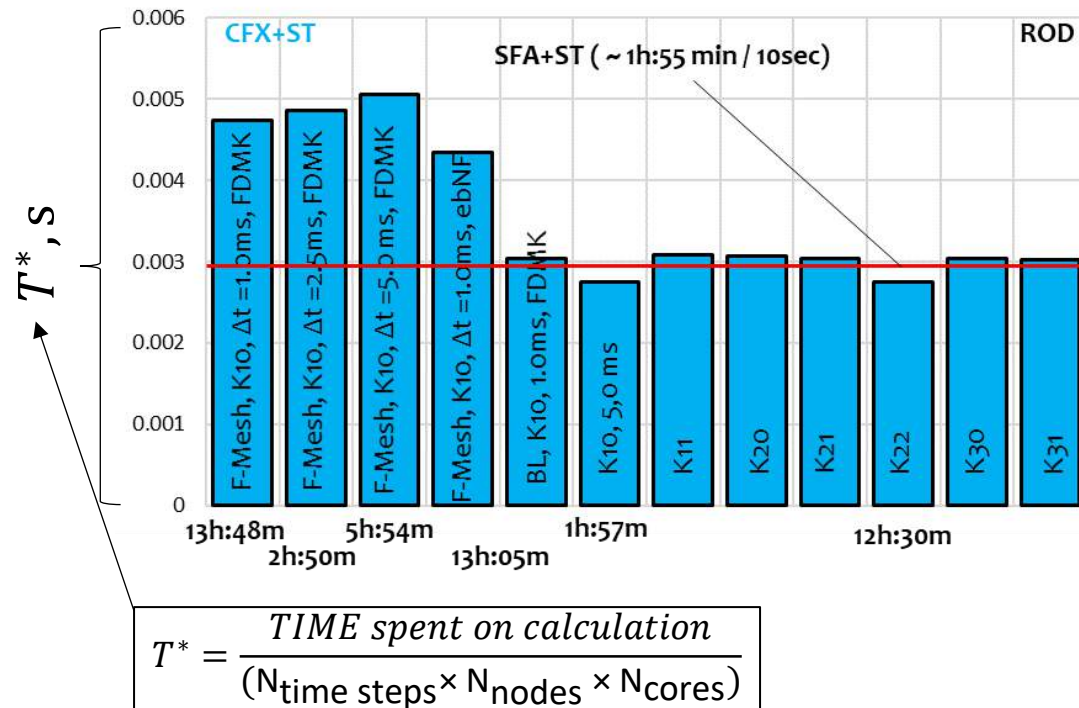
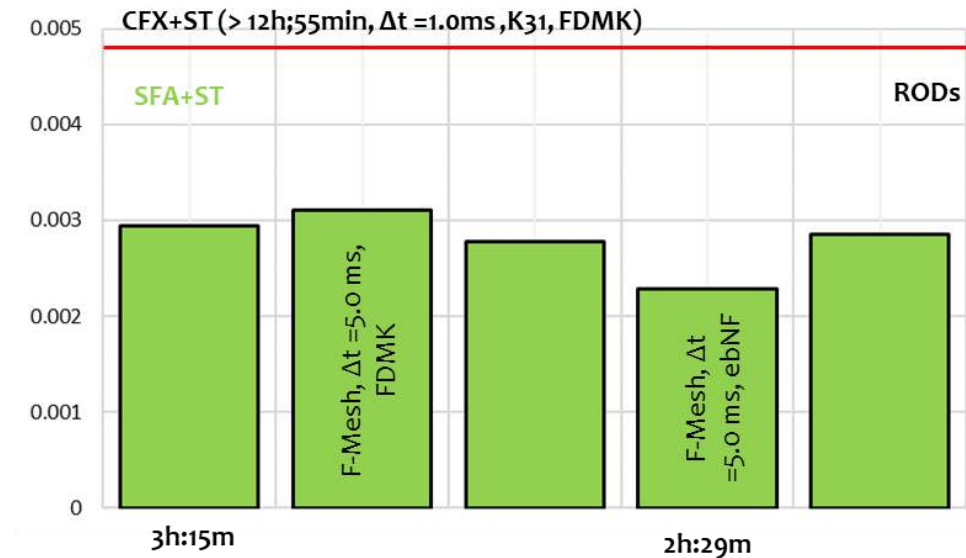
When comparing the results obtained for the same experimental AMOVI cases, a **fundamental difference is observed with the same discretization** of the computational domain, which is in the following:

- Oscillations are always excited in the SFA+ST computational procedure (at least within the timestep range of **0.1 to 30 ms** and even higher), which is most likely due to the assumption of equilibrium between turbulence generation and dissipation. In other words, high numerical diffusion does not interfere with the excitation and development of oscillations. In ANSYS CFX+ST oscillations at a calculation step greater than 1 ms may not occur or may have a significantly smaller amplitude.
- When coupled oscillations are established, the displacements amplitudes agree quite well for the cases which are close to resonances and FEI (e.g. Single Flexible Tube AMOVI cases with 2.9 and 4.0 l/s) where the flow unevenness (and hence the magnitude of the force) is determined directly by the moving boundary, and the oscillation modulation is determined by the turbulent nature.
- PSD prediction using ANSYS CFX+ST performs better than the SFA+ST procedure, which is likely **determined by the advection calculation method** (the SFA+ST procedure uses the Lagrangian approximation, which inherently does not distort the system's eigenvalues and possibly does not allow for the taking into account the interaction of the two systems - the flow path and the rod). The method for determining pressure pulsations may also influence this.

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

Computational efficiency

Intel Core i9-7900X 3.3GHz/64Gb (CFX+ST – 4 cores/SFA+ST – parallel only linear solver, approx. 10-20% all time)



The computational procedure **SFA+ST** is written in an interpreted programming language (essentially a prototype), which means it is several times slower than codes written with Fortran/C ++.

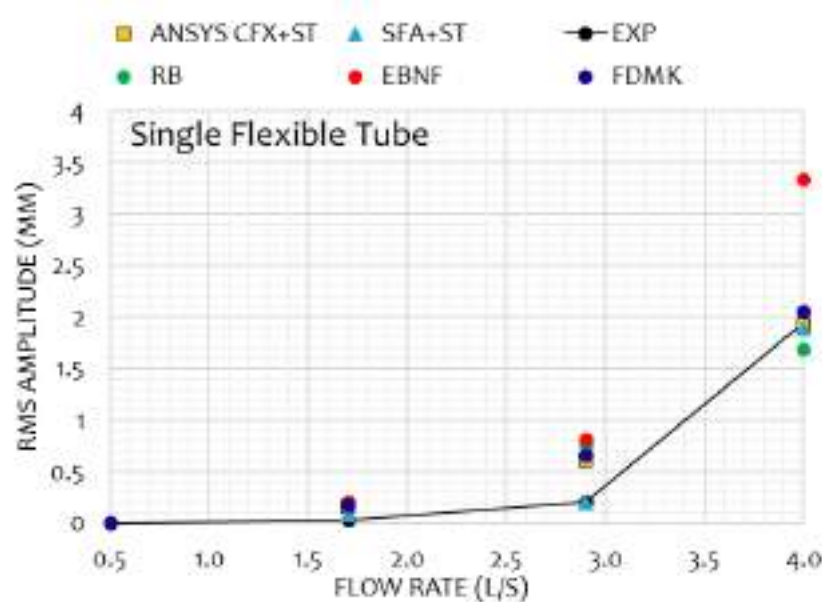
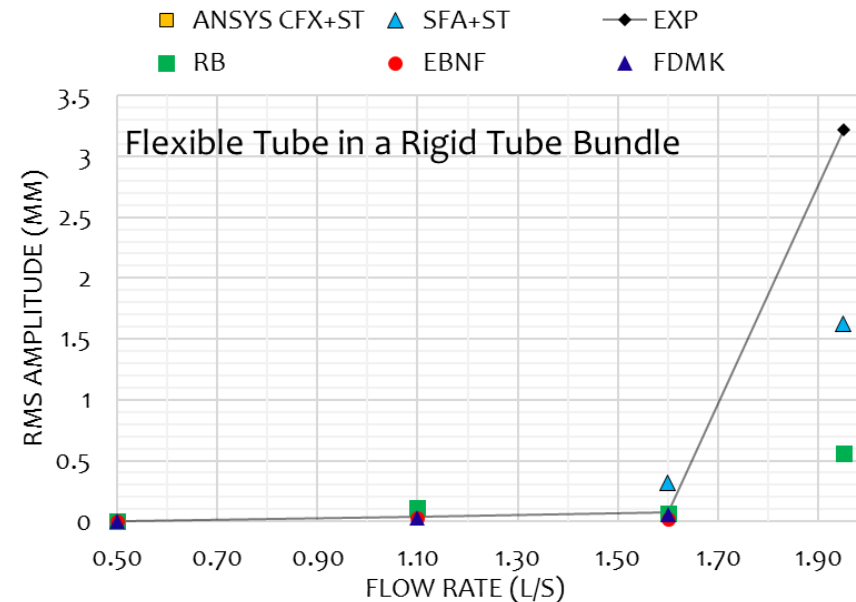
Therefore, there is significant potential for speeding it up!

Conclusions



In the case of **ANSYS CFX** - the synthetic turbulence generator was implemented in a three-dimensional setting, since it is possible to operate with the full Reynolds stress tensor (the solver is fundamentally three-dimensional). Moreover, **pressure pulsations are calculated based on a set of algebraic relations**, making the evaluation somewhat atypical, but **very fast and flexible**.

The **Stable Fluid Algorithm procedure** was developed specifically for the **fastest possible calculation of coupled vibrations**. This method is quite diffusion-based, so the use of a turbulence model is optional and necessary specifically for STG, which is implemented in 2D formulation.



In general, the **SFA+ST fast calculation procedure** can be calibrated on available experimental data, followed by validation on some of them (as it performed on AMOVI cases), **and then be applied with greater reliability to practical problems**.

Option	Value	Definition
TypeGeometry	ROD/RODS	Type of calculation geometry
MeshType	FLUENT/CFX/GENblock	Mesh generation type
dicrOption ¹	ZCA/MCA/OCA/ORA	Type of cross-diffusion correction: Zero correction algorithm (ZCA) Minimum correction algorithm (MCA) Orthogonal correction algorithm (OCA) Over-Relaxation algorithm (ORA)
intOption	CEN/CEN-F	Method of flow interpolation
gradBCzero	True/False	Calculating the gradient at the centers
divBCzero	True/False	Calculating divergence at the centers
BeamType	ebFDMkin/ebNF/TNF	BEAM solver: Euler-Bernoulli FDM – ebFDMkin Euler-Bernoulli (analytics) – ebNF Timoshenko Beam (analytics) - TNF
displFlag	Min/ Ave /Max	Reference displacement value for flow
LargeDeflFlag	True/ False	Accounting for large beam deformations
FlagLagrAdvect	True /False	Application of Lagrangian advection to U-V velocity variables
FlagLagrAdvectTurb	True /False	Application of Lagrangian advection to the RNG 1-eq turbulence model
LagrAdvType	True/ False	Using the internal step
LagrCFL	1.0	Courant number for the internal step
LagrMaxSubStep	10.0	Maximum number of time sub-steps for Lagrangian advection
LagrFlagBC	True/False	Forced correction of the boundary conditions at the stage of Lagrangian advection
VortexSynForce	True/ False	Synthetic vortex generator
WDCorrection	True/False	Wall Distance correction BC
dempFmin	0.0	Minimum multiplier on the wall
dempFcoef	5.0	Gradient multiplier of a function
dempFn	1.0	Damping coefficient
WDSTBL	True/ False	Damping function at the wall for synthetic turbulence
WDTurbBL	True/ False	Damping function at the wall for the turbulence model

TurbViscVelDiff	1.0	The coefficient of influence of turbulent viscosity on velocity diffusion
TurbViscToForce	0.0 /1.0	The influence coefficient of turbulent viscosity for the force calculation
ShearForce	1.0	Accounting for the shear stresses when calculating forces
uransForce	1.0	The coefficient of amplification/influence of the average flow field on the force
stForce	1.0	The coefficient of amplification/influence of the synthetic flow field on the force
signPF	-1.0/0.0/1.0	Auxiliary variable for the force from the static pressure field
uransEV	0.0 /1.0	Reynolds stress for synthetic turbulence: 0.0 – by kinetic energy, 1.0 – considering the velocity gradient.
MeanPressSourceCor	1.0	Contribution of the average source field in the calculation of pressure pulsations
StartTurbTime	Any (by default 0.0)	Beginning of accounting for the influence of synthetic turbulence, [s]
StartDisplTime	Any (by default 0.0)	Start of accounting of rod movements, [s]
StSource1	1.0	α_p , see (13)
StSource2	1.0	β_p , see (13)
StCoefPress	1.0	Pressure pulsation amplification factor
relaxWall, relaxU, relaxV, relaxUV, relaxP, relaxTurb, relaxMD	0.1	Relaxation coefficients for each variable
visualRANSfields	True/ False	Visualization of mean fields at each time step
visualSTfields	True/ False	Visualization of synthetic fields at each time step
visualMeshDispl	True/ False	Visualization of the deformed mesh at each time step
AutoRange	True /False	Automatic selection of limits for each variable
PrintSolverInfo	True/ False	Output of complete information about each solver and local algorithm

¹ – cross-diffusion correction is largely the reason for the presence of internal iterations (usually up to 4). If uniform meshes are used, then the solution for the linear part of the problem can be accelerated.

Default values are highlighted in bold in the table.





GO-VIKING

Thank you!



Yuliia Filonova



IPP-CENTRE



filonova-ys@ipp-centre.com.ua



Website:
www.go-viking.eu



Mail:
contact@go-viking.eu



Twitter:
[@GOVIKING_EU](https://twitter.com/GOVIKING_EU)



LinkedIn:
[GO-VIKING](https://www.linkedin.com/company/GO-VIKING)



Funded by the
European Union



GO-VIKING

Education, training, dissemination

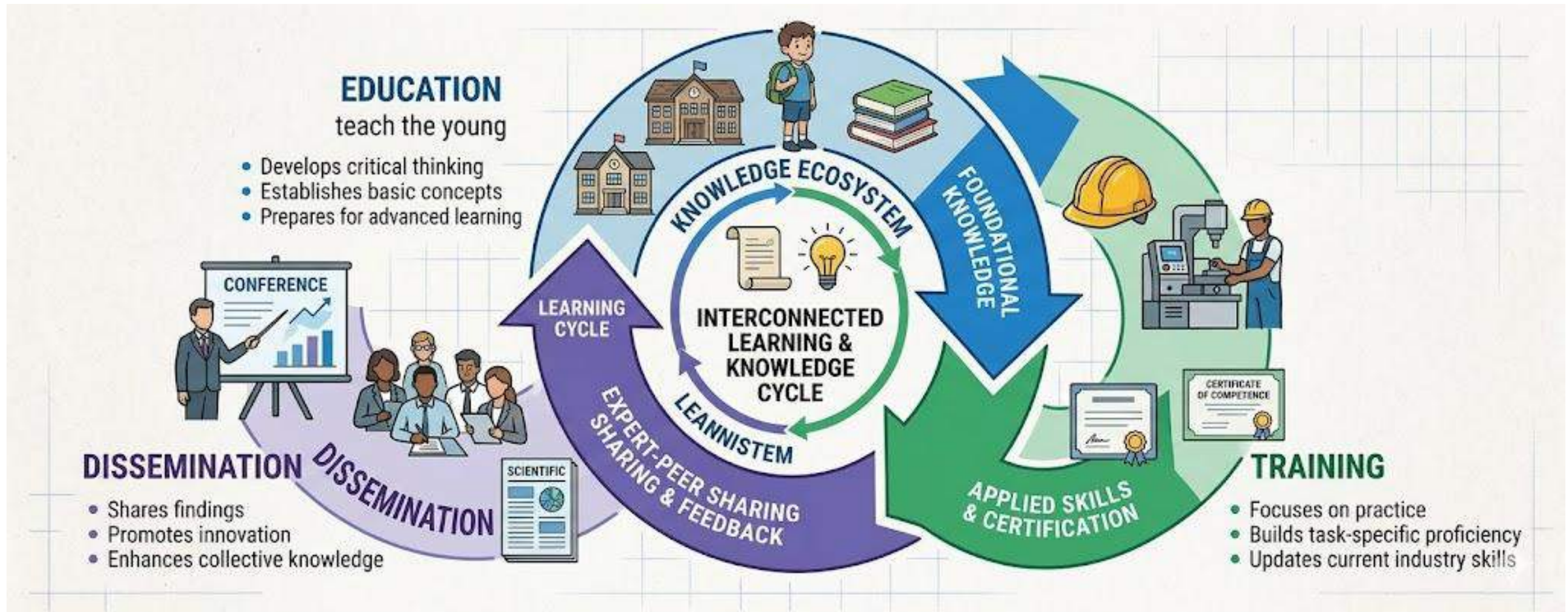
Final workshop

Philippe Planquart (VKI) – Lilla Koloszar (VKI) – Maria Teresa Ramandi (VKI)



Funded by
the European Union

Why we insist on this to have a separate WP in each European projects?





Education

Show what we are doing and preserve the data we created.

We have learned from the past, we are the present, we prepare the future



Funded by
the European Union

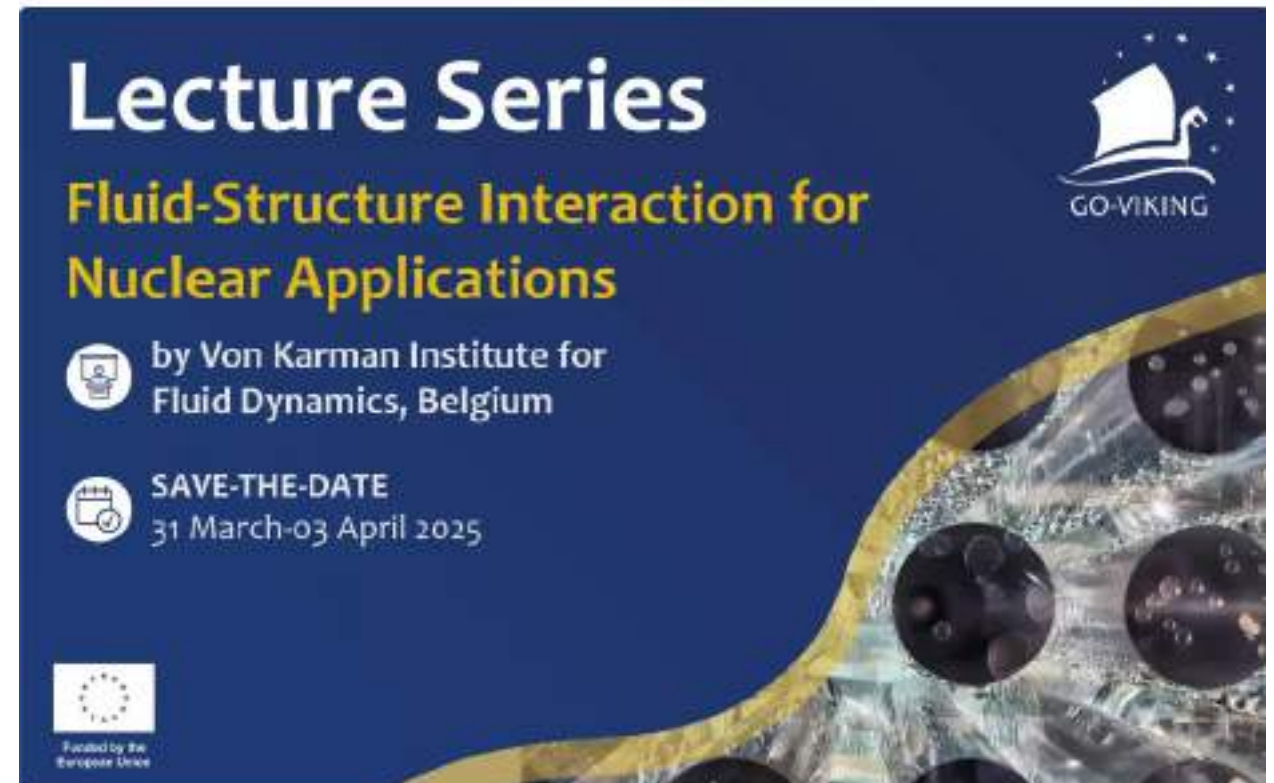
GO-VIKING Lecture Series



From the basics to the new state-of-the-art

- 10 lectures
- 1 industry panel session
- 1 facility visit
- 36 attendees in total, of which:
 - 11 were lecturers/panelists
 - 17 attended online
 - 14 were external to GO-VIKING

Target audience: Masters and PhD students, researchers, industrial engineers



GO-VIKING Lecture Series



VKI Lecture Series Proceedings



Lecture Notes available on VKI website and the GO-VIKING website.

GO-VIKING Open lectures



Simulation of a Single Flexible Tube Experiment – Tutorial



This tutorial presents a step-by-step numerical simulation of a single flexible tube exposed to cross-flow, based on the ANQVI experimental setup. Presented by Hemish Mistry (GRS), it guides viewers through key modelling steps, including assumptions, CAD geometry, mesh generation, simulation setup, and results analysis.

The session focuses on a rigid body fluid-structure interaction (FSI) approach to efficiently capture tube vibration behaviour and compares numerical outcomes with experimental results. It is well suited for engineers, researchers, and students interested in practical FSI simulations and flow-induced vibration analysis.

[Watch here](#)

GO-VIKING Open Lecture on Flow-Induced Vibration: The Numerical Approach



Watch the "GO-VIKING Open Lecture on Flow-Induced Vibration: The Numerical Approach" to delve into the critical role of flow-induced vibrations across multiple engineering fields. Presented by Alexander van Zuijlen, this lecture explores the intricate interactions between fluid flows and deformable structures through a detailed numerical lens. Learn about the three-field approach, which intricately links the dynamics of flow, structure, and mesh movement, and discover the impacts of various coupling methods on system behaviour. This session is essential for anyone interested in the advanced numerical modeling techniques that define fluid-structure interactions in real-world applications.

[Watch here](#)



- Widely advertised by ENEN and LGI
- Participants: OL1 - 63 , OL2 - 36
- Recording published on ZENODO

GO-VIKING Educational materials



Introduction to Fluid-Structure Interaction (FSI) / Flow-Induced



A comprehensive PDF guide that introduces the fundamentals of fluid-structure interaction (FSI) and flow-induced phenomena. It covers theory, governing equations, and illustrative examples. An ideal

Flow-Induced Vibrations in Nuclear Power Plants: A Literature Review.



Discover key insights in our document, "Flow-Induced Vibrations in Nuclear Power Plants: A Literature Review." This report addresses the significant impact of Flow-Induced Vibrations (FIV) on the operational safety and efficiency of nuclear power plants. As the global demand for low-carbon energy sources increases, understanding and mitigating FIV is crucial. Our review highlights the mechanisms of FIV, the associated risks, and the latest technological advancements to ensure continuous and safe nuclear power operations. Essential reading for industry professionals and researchers focused on enhancing nuclear energy reliability.

Fluid-Structure Interaction Effects Associated with Small Structural Deformations



This educational document, developed within GO-VIKING Work Package 8 (Education and Training), provides a focused introduction to fluid-structure interaction (FSI) effects associated with small structural deformations.

The material explains key physical mechanisms such as added mass, fluid-induced forces, and energy exchange between fluids and structures, with particular relevance to nuclear engineering applications. Analytical models and theoretical foundations are presented alongside practical insights, supporting the understanding of FSI phenomena in cases where structural motions remain in the elastic range.

Prepared by VTT Technical Research Centre of Finland, this resource is well suited for engineers, researchers, and students seeking a deeper theoretical understanding of FSI effects relevant to vibration analysis and nuclear power plant safety.

[Read more](#)

Item	Organization	Main author	Type	Description
1	TUD	Alexander Van Zuijlen	Lecture	Lectures from Master course
2	TUD / NRG	Alexander Van Zuijlen / Kevin Zwijsen	Literature report	State-of-the art review prepared in academic program
3	UGENT	Joris Degroote	Powerpoint Presentation	General academic introduction on FIV
5	VTT	Antti Timperi	Presentation	Commented PowerPoint
6	FRAMA TOME	Hidajet Hadžić	Lecture – Presentation	FIV in Reactor Fuel Assembly

GO-VIKING Educational materials



Demonstration video: in-plane fluid elastic instability in a full-scale steam generator mock-up with nonlinear supports



This video, presented by Angel Papukchiev (GRS), delves into the phenomenon of in-plane fluid elastic instability in a full-scale steam generator mock-up with nonlinear supports. Designed for educational purposes, this demonstration video serves as a valuable resource for understanding complex fluid-structure interaction in marine applications.

[Watch here](#)

Flow-Induced Vibration of a Tube in Square Tube Bundle



A video resource by David Lital de Franco (CEA) that demonstrates flow-induced vibration in a square tube bundle. Through simulations and visualizations, it delves into how fluid flow excites structural responses. Essential footage for comprehending complex flow-induced vibration.

[Watch here](#)

Two-Phase Flow Across a Square Tube Bundle with a Side View



A video resource presenting two-phase liquid-gas flow in a square tube bundle from a side perspective. The animation and accompanying commentary explore phase distribution, flow patterns, and structural impact—perfect for those studying multiphase flow phenomena in engineering systems.

[Watch here](#)

Vibration Test Demonstration in GOKSTAD



An educational video by Muhsin Can Akkurt (VKI), showcasing a vibration test conducted in the GOKSTAD facility. The footage provides insights into the experimental setup, equipment, behavior under laboratory loads, and data capture techniques used for vibration.

[Watch here](#)

Item	Organization	Main author	Type
1	GRS	Angel Papukchiev	Numerical results
2	NRG	Kevin Zwijsen	Numerical results
3	VKI	Muhsin Can Akkurt	Experimental results
4	CEA	Lucas Prevost	Experimental results
5	EDF	William Benguigui	Experimental results
6	Canadian Nuclear Labs	Salim El Bouzidi	Experimental results

Table 2: Status of proposed videos

GO-VIKING Education



Points of discussion

- Program: did we do too much or too few?
- Outreach: did we managed to attract the right audience?
- Outreach: are we satisfied with the attendance level?

- What and how to improve?



Training

Information is not knowledge. Knowledge is not expertise.

You can only learn (research), through doing (research)



Funded by
the European Union

Task 6.2 Training



Training sessions in OpenFoam and TrioCFD



Training on FSI with OpenFoam

22 - 23 October 2025

Garching, Germany

Participants: GRS, NRG PALLAS, VKI



Training on FSI with TrioCFD

9 - 10 October 2025

Saclay, France

Benchmark definition

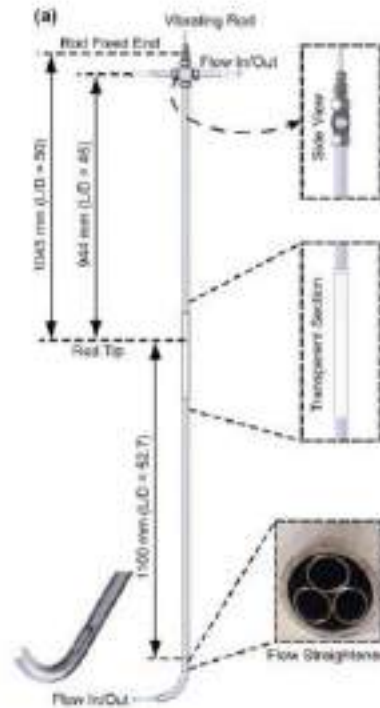


Benchmark definition

CANTILEVER ROD (CR)

Cantilever Rod (UoM)

- Geometry: available, problem of misalignment
- $Re = 21k$ and $40k$
- Approximate mesh size: $\sim 150M$ (for the $21k$ Reynolds, wall-resolved) and time step: $\sim 1e-5$
- inlet conditions: measured
- FSI movement: $\sim 0.5mm$



Partners for it: **VKI**, TU Delft, NRG Pallas, ASNR

Partners maybe: Framatome, UGent

Partners in WP2: EDF, NRG, EDF-E, PSU, IPP

Training on the benchmark case



Simulation of a Single Flexible Tube Experiment – Tutorial



This tutorial presents a step-by-step numerical simulation of a single flexible tube exposed to cross-flow, based on the AMOVI experimental setup. Presented by Hemish Mistry (GRS), it guides viewers through key modelling steps, including assumptions, CAD geometry, mesh generation, simulation setup, and results analysis.

The session focuses on a rigid body fluid-structure interaction (FSI) approach to efficiently capture tube vibration behaviour and compares numerical outcomes with experimental results. It is well suited for engineers, researchers, and students interested in practical FSI simulations and flow-induced vibration analysis.

[Watch here](#)

Tutorial: Fluid-Structure Interaction Simulation of a Cantilever Rod in STAR-CCM+



This tutorial, developed as part of the GO-VIKING project, provides a step-by-step guide to performing a fluid-structure interaction (FSI) simulation of a cantilever rod using Simcenter STAR-CCM+ (V2306). This resource is provided for educational purposes and was prepared by Mohammed Muazz M.D. Hussain (NRG PALLAS, Petten, The Netherlands), with project management by Kevin Zwijnen.

It is available for use in training and educational courses, with credit attributed to the authors and GO-VIKING.

[Access the tutorial here](#)

GO-VIKING Training



Points of discussion

- Program: did we do too much or too few?
- Outreach: did we managed to attract the right audience?
- Outreach: are we satisfied with the attendance level?
- Benchmark: was not much of a success.
- What and how to improve?



Dissemination

Show what we are doing and preserve the data we created.

At the end we are spending the money of the European taxpayers

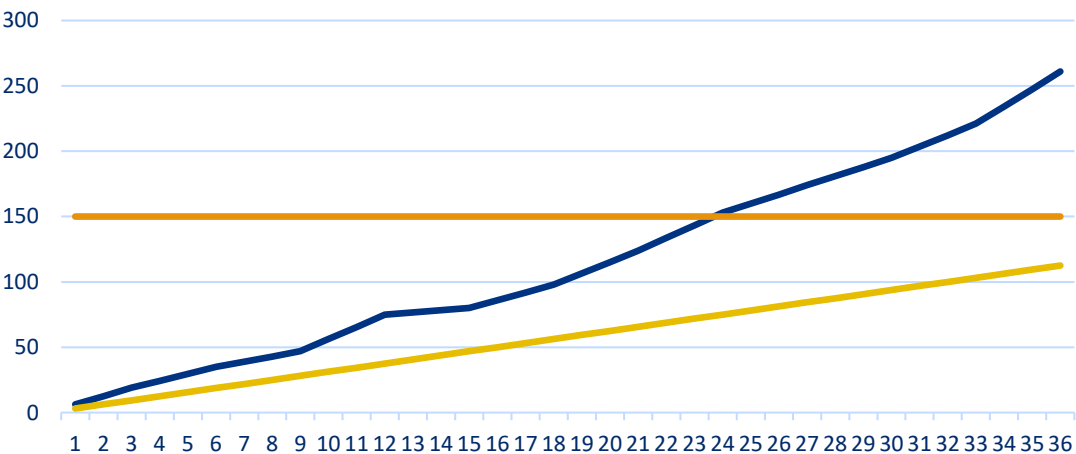


Funded by
the European Union

Wider public – reached our KPI

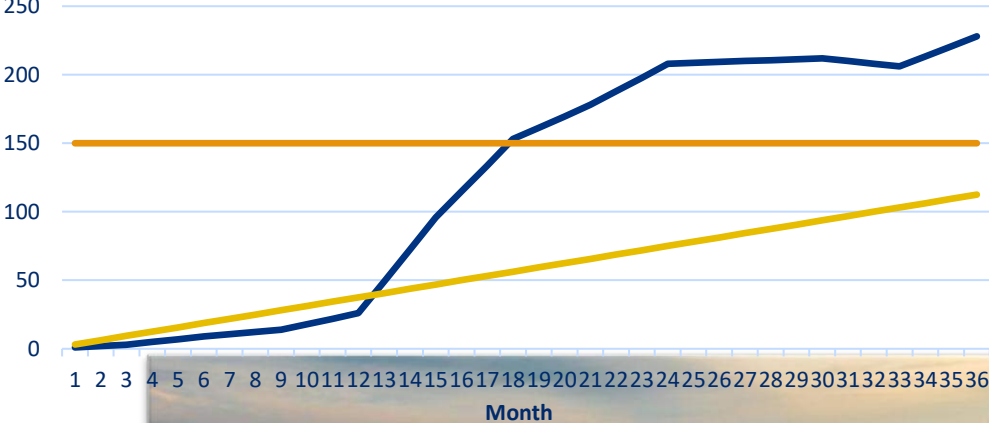


LinkedIn Followers
M1 - M36



GO-VIKING
The project aims to improve the safety of contemporary nuclear reactors and the design evaluation of newer models.
Research Services · Köln · 195 followers · 11-50 employees

X (Twitter) Followers
M1 - M36



GO-VIKING
@GOVIKING_EU
The project aims to improve the safety of contemporary nuclear reactors and the design evaluation of newer models.
Science & Technology go-viking.eu Joined October 2022
464 Following 212 Followers

Website



Most visited pages:

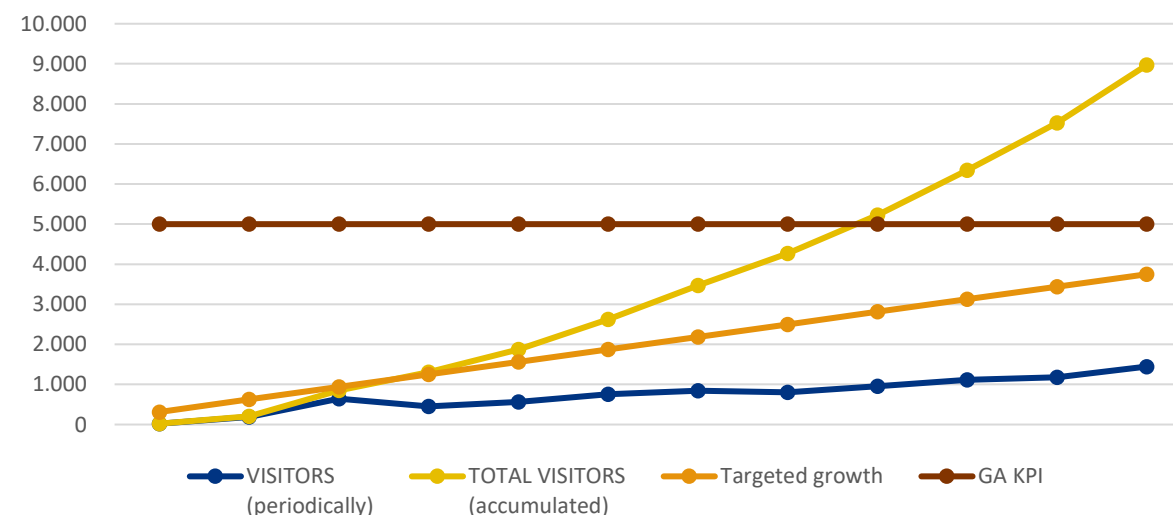
All times	Last 6 Months
Home page	Home page
News	Lecture Series VKI (news)
Training & Education	Training & Education
About	News
Resources	Open Lecture Series (news)



Website statistics

Total amount of visitors: 8.971

As of 16/05/2025 PROJECT WEBSITE VISITORS
M1 - M36

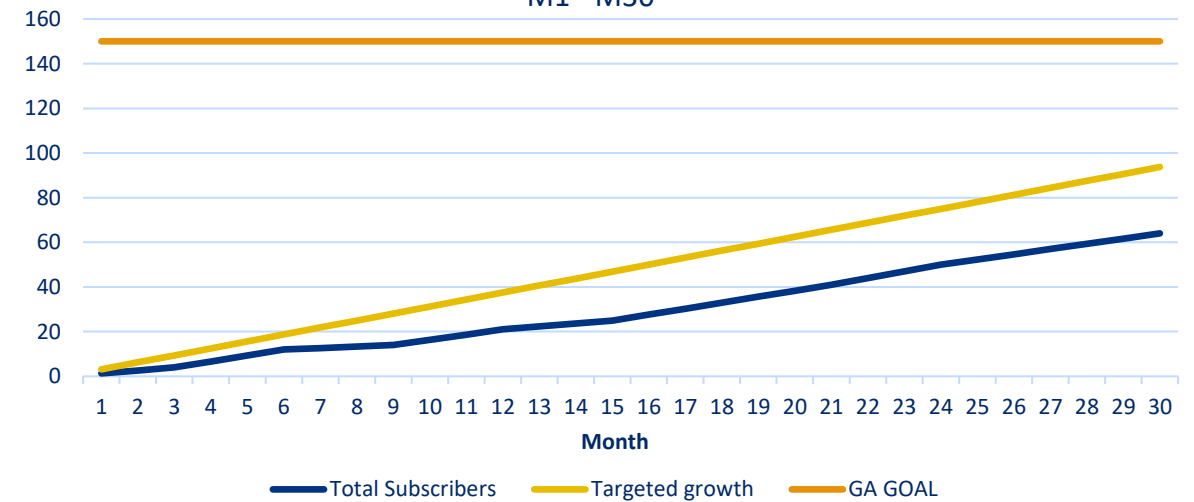


Newsletters



Newsletter Subscribers

M1 - M30



Is the newsletter the best way to communicate with the public?

Communication and identity



Communication materials:

Flyer

Roll-up

Factsheet



Visual Identity Handbook

The look and identity of GO-VIKING'S VISUAL IDENTITY

Version 01
05 / 2022

Edited by: Birgit Rieger, Carina Vack
Illustrated by: Carina Vack
Graphic design: Carina Vack



Activity	Indicator	Estimate at the end of the project
Public website	• Number page views by the end of the project	• 3000 page views
LinkedIn page	• Number of followers by the end of the project	• 150 followers
Twitter Page	• Number of followers by the end of the project	• 150 followers
E-newsletters	• Number of subscribers by the end of the project	• 150 subscribers
External events	• Number of events and conferences attended by the by the end of the project	• 3 events
Publications	• Number of papers published by the end of the project	• At least 5 papers published
Project events: Stakeholder engagement and final workshop	• Number of attendees per event • Number of events and conferences held by the by the end of the project	• Between 30 and 150 attendees • 6 events

LinkedIn / Twitter version (max. 120 letters):

The GO-VRING project aims to improve the safety of contemporary nuclear reactors and the design evaluation of newer models.

Short version:

The Gathering expertise On Vibrations Impact In Nuclear power Generation (GO-VRING) project will improve the safety of contemporary reactors by improving numerical approaches for the evaluation of Flow-Induced Vibration. In layman terms, GO-VRING will make Nuclear Power Plants more resilient against FIV phenomena. It will also improve the design evaluation of new nuclear power plants as well as disseminating results among current and future stakeholders, academics, and professionals in this field.

Visit the project website for more information at www.go-viking.eu

Medium version:

It is only possible to avoid inevitable and irreversible climate change if massive and immediate reductions in greenhouse gas emissions are made (IPCC, 2021). Nuclear safety is one of the top priority actions to be performed to accelerate the energy system transformation and to reach future low carbon energy system.

Dissemination towards the scientific community – 15 papers and 1 poster



2022

PAPER – K. Zuiljien, A. Popukhiev, D. Vivaldi, H. Hadji, S. Benhamadouche, M. Benguigui, and P. Planquart
GO-VIKING: A Horizon Europe Project on Flow-Induced Vibrations
FIV2022, Paris-Saclay, 2022 - [Download](#)

PAPER – H. Dolkens, D. Van Nieuwenhuijzen, A. Bui, J. Degroote
Fluid-structure interaction simulation of a wire-wrapped tube array using overset grids
ECCOMAS Congress 2022 - doi: [10.23907/eccomas2022.137](https://doi.org/10.23907/eccomas2022.137)

2023

PAPER – A. Girondeau, S. Zhang, M. R.A. Nabney, H. Li, D. Cooper, H. Girondeau
Experiments on axial-flow induced vibration of a free-clamped/clamped-free rod for light water nuclear reactor applications
Annals of Nuclear Energy, Volume 186, 15 September 2023 - [ScienceDirect doi: 10.1016/j.anucene.2023.109690](https://doi.org/10.1016/j.anucene.2023.109690)

PAPER – H. Dolkens, S. Vandewalle, & J. Degroote
Effect of stochastic deformation on the vibration characteristics of a tube bundle in axial flow
Nuclear Engineering and Design, 431, 112412, September 2023 - [ScienceDirect doi: 10.1016/j.nucengdes.2023.112412](https://doi.org/10.1016/j.nucengdes.2023.112412)

PAPER – K. Zuiljien, N. van den Bos, E.M.A. Frederix, F. Buxelofs, A.H. van Zuijlen
Development of an Anisotropic Pressure Fluctuation Model for the Prediction of Turbulence-Induced Vibrations of Fuel Rods
NURETH-20 – Proc. of the 18th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-20), August 20-25, Washington, USA, available [here](#) (participants and subscribers only)

PAPER – H. Li, S. Zhang, M. R. A. Nabney, H. Girondeau, A. Girondeau
Experiments on Axial-Flow Induced Vibration of a Free-Clamped Rod for Light Water Nuclear Reactor Applications
NURETH-20 – Proc. of the 18th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-20), August 20-25, Washington, USA, available [here](#) (participants and subscribers only)

PAPER – A. Popukhiev, K. Zuiljien, D. Vivaldi, H. Hadji, S. Benhamadouche, W. Benguigui, P. Planquart
The European GO-VIKING Project on Flow-Induced Vibrations
NURETH-20 – Proc. of the 18th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-20), August 20-25, Washington, USA, available [here](#) (participants and subscribers only)

PAPER – K. Mikulina, A. Popukhiev, K. Zuiljien, P. Planquart
Vibration impact in nuclear power generation: Go-viking advancements
NEM2023, Palermo, 2023 - [Download](#) - [View online](#)

PAPER – A. Popukhiev, K. Zuiljien, D. Vivaldi, H. Hadji, S. Benhamadouche, M. Benguigui, and P. Planquart
The European GO-VIKING project on flow-induced vibrations: overview and current status
Journal of Marine Technology - doi: [doi:10.2513/jmte-2023-0128](https://doi.org/10.2513/jmte-2023-0128)

2024

POSTER – H. Goffen, J. Degroote, S. Vandewalle
Effect of stochastic deformation on the vibration characteristics of a tube bundle in axial flow RMS
YG Nuclear Science Contest 2024 (2024 Nuclear Science Contest), 26 November 2024 - [Download](#)

PAPER – W. Benguigui, S. Benhamadouche, F. Beltrán, M. Hassan
Experimental and numerical contributions on flow-induced vibration in steam generator-like tube bundles: A review
Nuclear Engineering and Design Volume 424, 1 August 2024 - doi.org/10.1016/j.nucengdes.2024.112302

PAPER – H. Dolkens, J. Degroote
A synthetic bubble model as inlet for fluid-structure interaction simulations with two-phase flow
ECCOMAS Congress 2024, 3-7 June 2024, Lisbon, Portugal [Download](#) - [View Online](#)

PAPER – H. Dolkens, J. Degroote, S. Vandewalle
Effect of stochastic deformation on the vibration characteristics of a tube bundle in axial flow Nuclear Engineering and Design, Volume 431, September 2023, 112412 - DOI

PAPER – K. Zuiljien, A. Mineiro de Freitas, S. Tylkoot, E.M.A. Frederix, A.H. van Zuijlen
An Optimized Anisotropic Pressure Fluctuation Model for the Simulation of Turbulence-Induced Vibrations
Physics of Fluids 36, 125103 (2024) - doi.org/10.1063/1.6235793 - [Download Manuscript](#)

2025

PAPER – G. Spina, W. Benguigui, G. Brillen, D. Vivaldi, C. Collin Frederix, A.H. van Zuijlen
Local-scale experimental investigation of a two-phase cross-flow in a tube bundle
Nuclear Engineering and Design, Volume 431, March 2025, 115936 - [ScienceDirect](#)

PAPER – D. Vivaldi, A. Popukhiev, C. Viron, S. Graft, M. Kountzou
Fluid-structure interactions in nuclear power plants: Review of some main phenomena
Nuclear Engineering and Design, Volume 442, September 2025, 116121 - [ScienceDirect](#)

Best Poster Prize at the 2024 Nuclear Science Contest



GO-VIKING Dissemination



Points of discussion

- Program: did we do too much or too few?
- Outreach: did we managed to attract the right audience?
- Outreach: are we satisfied with the communication level?

- What and how to improve?

Partners



Associated Partners



*EDF R&D UK Center



Thank you!



Philippe Planquart
Lilla Kolozar
Maria Teresa Ramandi



VKI



philippe.planquart@vki.ac.be
maria.teresa.ramandi@vki.ac.be

www.go-viking.eu



Funded by the
European Union



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

Angel Papukchiev (GRS), K. Zwijsen (NRG PALLAS)

GO-VIKING Final International Workshop, 20-22.05.2026, Garching, Germany



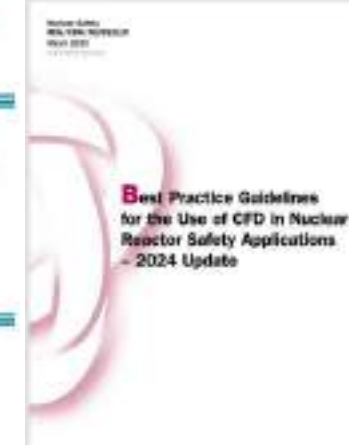
Funded by
the European Union

Outline



- Introduction
- Structure of the GO-VIKING Best Practice Guidelines (BPGs)
- Examples on BPGs for FIV in axial-flow
- Examples on BPGs for FIV in cross-flow
- Examples on BPGs for FIV in multiphase flow
- Examples on BPGs for uncertainty quantification (UQ) and fast-running methods
- Conclusions

Introduction



- Different Best Practice Guidelines relevant to CFD available
- ERCOFTAC BPGs:
 - “Industrial Computational Fluid Dynamics of Single-Phase Flows”; “Computational Fluid Dynamics of Dispersed Multi-Phase Flows”; “Computational Fluid Dynamics of Turbulent Combustion”
 - focused on numerical, convergence and round-off errors, turbulence modelling, application of uncertainties, user and code errors, validation and sensitivity
- OECD/NEA BPGs for the Use of CFD in Nuclear Reactor Safety Applications
 - deal with problem definition; selection of numerical tool and models; code and solution verification; code validation; user control of the numerical model
 - small part on FSI
- ASME ASME V&V Standard for CFD
 - deal with code and solution verification; effect of input parameters; experimental and validation uncertainties, examples

Introduction



- The BPGs structure should not be similar to the one of the existing BPGs
- Rather focus on own experience within GO-VIKING including advances in performing FIV simulations
- Focused on results from benchmarks and partners' experience - it is key to make the BPG unique like lessons learned
- Emphasis on the modeling strategies, also UQ added
- Most of the partners sent the topics they want to include in the BPGs
- Our BPGs are not prescriptive (teach how to design a component), but rather provide information that prevents other users dealing with similar problems from pitfalls

BPGs content



- Chapters:
 - General chapter on FSI: CFD, CSM, coupling approaches and data transfer (explicit, implicit, partitioned, monolithic, 1- and 2-way, etc.), interface tracking (Arbitrary Lagrangian-Eulerian (ALE), Immersed Boundary (IB)), etc.
 - BPGs for FIV in axial-flows
 - BPGs for FIV in cross-flows
 - BPGs for FIV in multiphase flows
 - BPGs for ROMs
 - Summary
- Content based on:
 - individual experiences of partners
 - results from work package benchmarks

FIV in axial-flows: single rods

- Vibration frequencies were often predicted with sufficient accuracy, while vibration amplitudes - underpredicted
- Rod vibration due to turbulence-induced vibrations (TIV) has small impact on the flow → one-way coupling sufficient
- **Fluid modelling:**
 - pay attention to the choice of turbulence model: it has the largest impact on the accuracy of the results and overshadows any potential gain of using Computational Structural Mechanics (CSM) over simplified beam model
 - the evolution of statistics is remarkably different near the rod tip and for a considerable distance along the rod wall, before returning to expected annular flow statistics behavior
 - use Reynolds stress turbulence models (RSM) to obtain good results at reasonable CPU cost
 - good velocity and Reynolds stress profiles can be predicted by CFD and FSI simulations
 - be aware that even if high-resolution methods provide good agreement in the flow field statistics with data, they still may underpredict the vibration amplitudes
 - axial-flow (up-flow or down-flow) introduces compressive or tensile forces that can further shift vibration frequencies
 - the calculation of forces on the structural surface might be sensitive to domain decomposition in parallel runs

FIV in axial-flows: single rods

- **Structural modelling:**

- utilize beam models in high-resolution simulations as it helps reducing high CPU costs
- use Euler-Bernoulli instead of Timoshenko beams, since the results are very similar

- **Validation:**

- validate structural and fluid models with vibration tests in vacuum and still water before moving to the cases with non-stagnant fluid
- validate extensively: the same numerical approaches that give good predictions for certain Re tend to underestimate/overestimate displacements for other Re numbers
- high-quality experimental data crucial for the validation of high-fidelity simulation approaches

FIV in axial-flows: bundles



- **Fluid modelling:**

- turbulence assumed to be the main mechanism driving the vibration → resolving turbulent structures is necessary to obtain the pressure fluctuations that induce the structural motion
- the turbulence model can have a significant influence on the acting fluid forces/momenta on the structures
- if the unsteady forces/momenta on the structures result from turbulent fluctuations, Unsteady Reynolds-Averaged Navier-Stokes (URANS) simulations are not appropriate
- use Large Eddy Simulation (LES), other scale-resolving methods to obtain the pressure fluctuations
- use non-dissipative numerical discretization schemes (e.g. *Central Differencing*) → be aware that these are much more prone to instabilities and are sensitive to the mesh quality
- wall-modelling instead of wall-resolving approach is the only practical option for larger structures like bundles
- still significant limitations for the general use of scale-resolving simulations (SRS) in industrial context → use fuel assembly spans to investigate turbulence and derive parameters

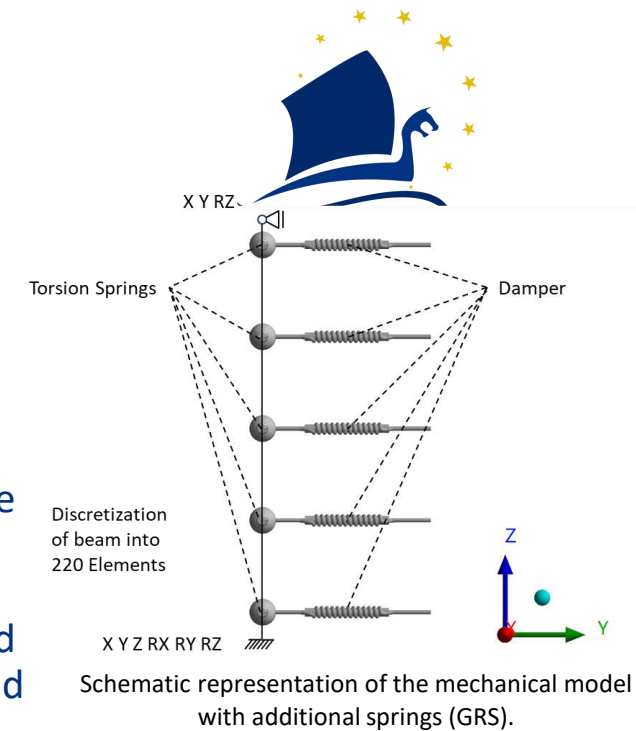
FIV in axial-flows: bundles

- **Structural modelling:**

- to simulate the fuel assembly dynamics, use a beam model with additional springs to model the stiffness introduced by the spacer grids, and fit against experimental values
- to fit the model, perform damped modal analysis using vibration mode, natural frequencies and logarithmic decrement for the fitting of the beam moment of inertia, stiffness of the springs and damping
- explicit modeling of the contacts between spacer grids and fuel rods not practical → use beam models; additional elements can be included to mimic the clamping of the fuel rods in the grids

- **Coupling strategy:**

- coupled FSI simulation can be very challenging and its necessity (due to the small displacements) should be checked
- use alternatively decoupled FSI, but should consider added mass and fluid damping



	1 st mode	2 nd mode	3 rd mode
Experiment	6 Hz	14.5 Hz	23.9 Hz
Model	5.97 Hz	14.9 Hz	23.55 Hz
Mode shape			

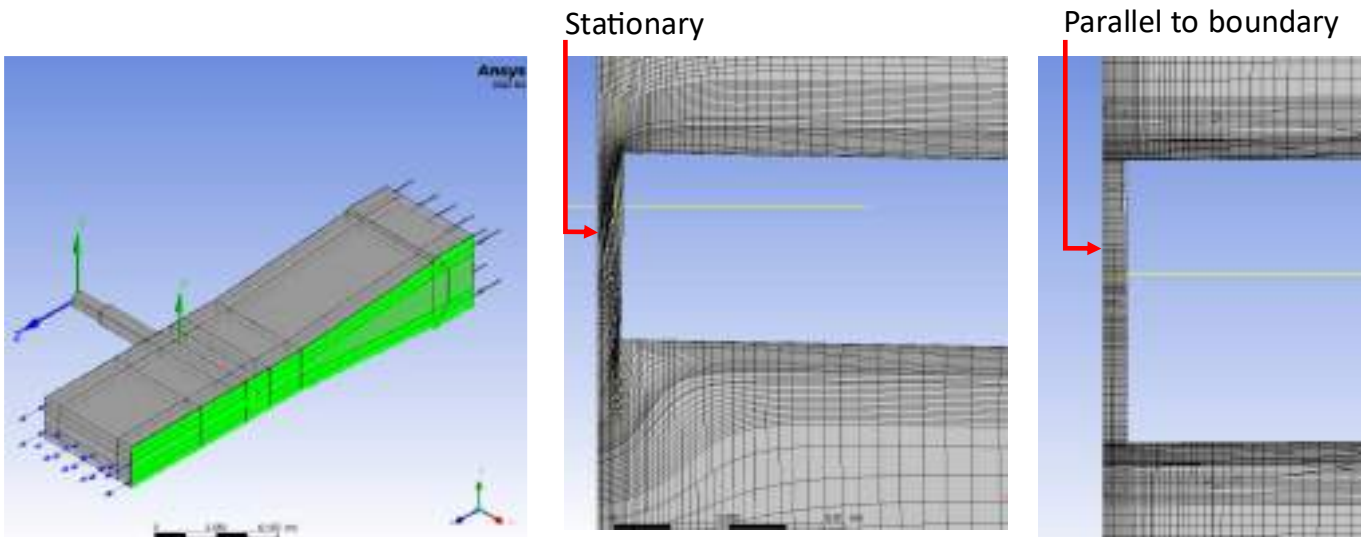
Natural frequencies and mode shapes after fitting (GRS).

FIV in cross-flows

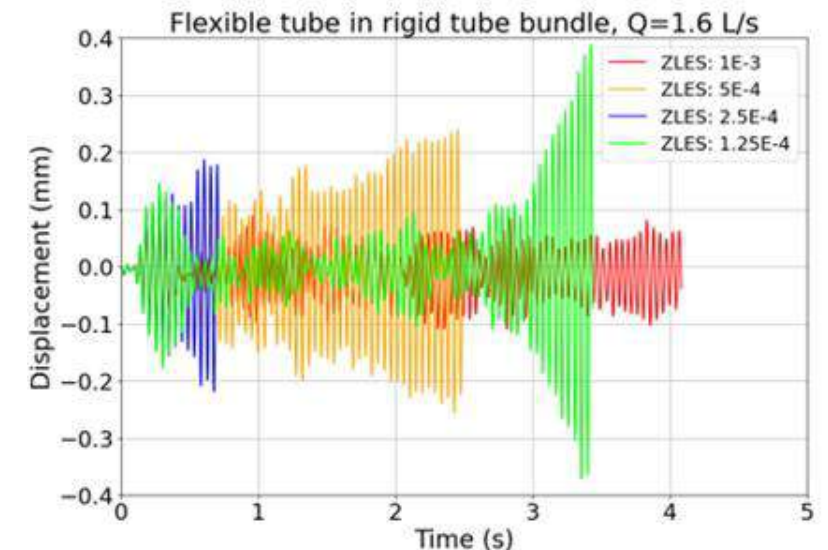


- **Mesh deformation:**

- needs special attention → appropriate boundary conditions (BC) may help a lot
- use *Parallel to Boundary* option in CFX to allow the tube bottom elements to slide along the stationary wall
- larger steps decrease displacements and stabilize the Rigid body Motion (RBM) solution → perform sensitivity studies



Mesh deformation when using “Stationary” and “Parallel to boundary” boundary conditions for the test section bottom wall (GRS).

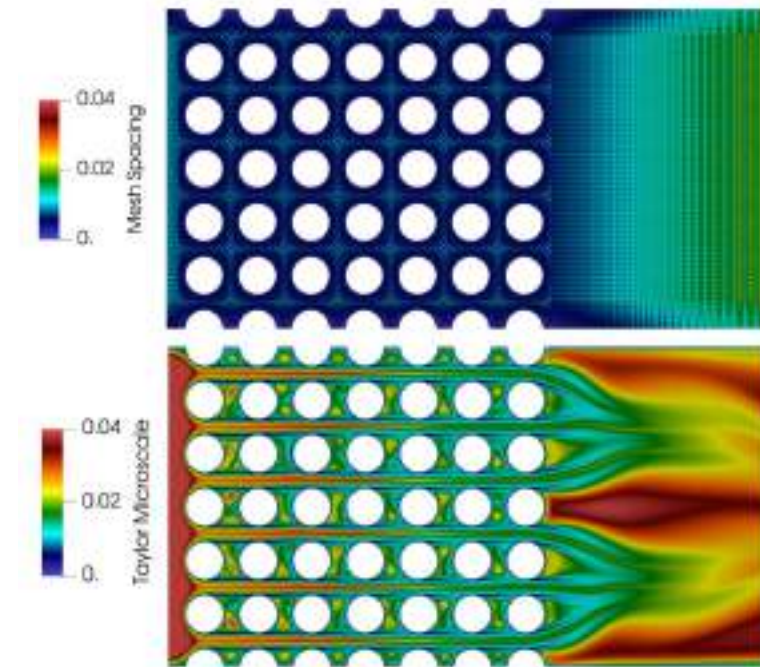


Dependence of the tube displacement on the time step size (GRS).

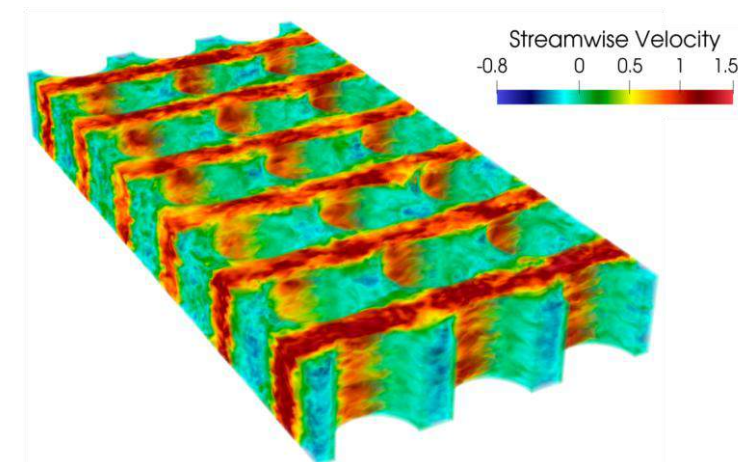
FIV in cross-flows

- **Fluid modelling:**

- URANS approach applicable for cross-flows, but generally underpredicts the structural displacement
- RSM and hybrid models are good and not too expensive alternative
- perform simplified simulations (RANS) before LES
- compute the Taylor microscale before running LES on the full bundle geometry
- compare the computed Taylor microscale to the local mesh spacing and assess whether the resolved turbulence is adequately captured when doing LES
- allow sufficient turbulence development in the inlet region of the LES domain before the zone of interest or use precursor domain
- first perform a small periodic tube bundle precursor simulation → reduced domain containing only a few repeating tube pitches and periodic boundary conditions allows rapid testing of mesh quality, numerical stability, and turbulence behavior
- ensure the periodic boundary conditions do not artificially constrain the flow: a min. spanwise domain extent of three tube diameters and a streamwise domain length of at least three tube pitches is recommended



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



Volume rendered streamwise velocity contour plot of a reduced domain of the GOKSTAD tube bundle (VCU).

FIV in cross-flows



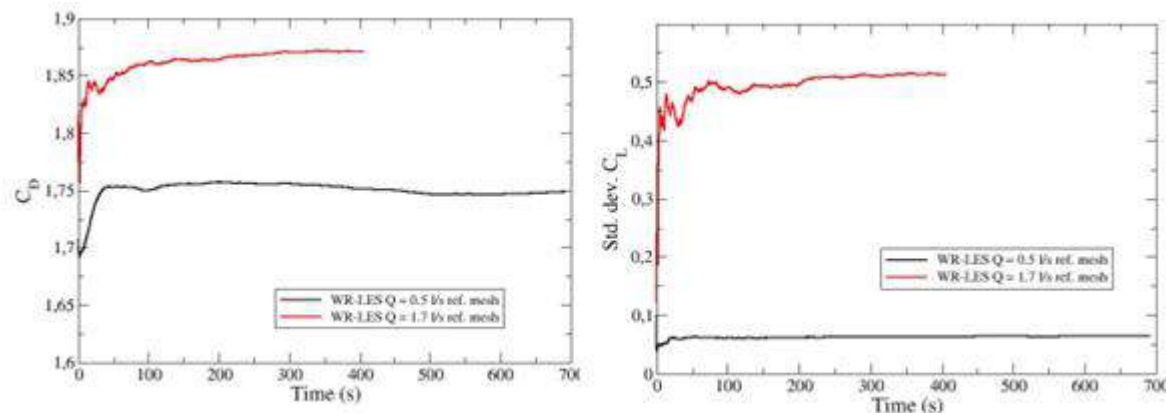
- **Fluid modelling:**

- not easy for all methods to predict lift and drag forces in some single rigid tube cases
- forces in rigid bundles predicted with greater accuracy than for single tube cases
- use 2D simulations to get first impression of the occurring phenomena and vibration response
- for high accuracy results use 3D high-resolution methods with sufficiently dense mesh
- medium-resolution models can still give reasonably accurate predictions → use a RSM or under-resolved LES on a URANS quality mesh
- hybrid models are a good compromise between rough 2D estimations and highly accurate calculations
- hybrid models showed advantages even on 2D setups compared to URANS models and performed well even in FEI cases

FIV in cross-flows



- Ensure sufficient mesh refinement in zones of interest (e.g. tube bundle)
- Proper inlet conditions important for accurate predictions → either use experimental data or allow sufficient flow development
- Outlet length important to avoid effect on the results upstream (region of interest)
- Use sufficient length of the moving tube when implementing mass-spring-damper system
- Perform long enough simulations to guarantee results are statistically steady: especially important due to bifurcation in flow



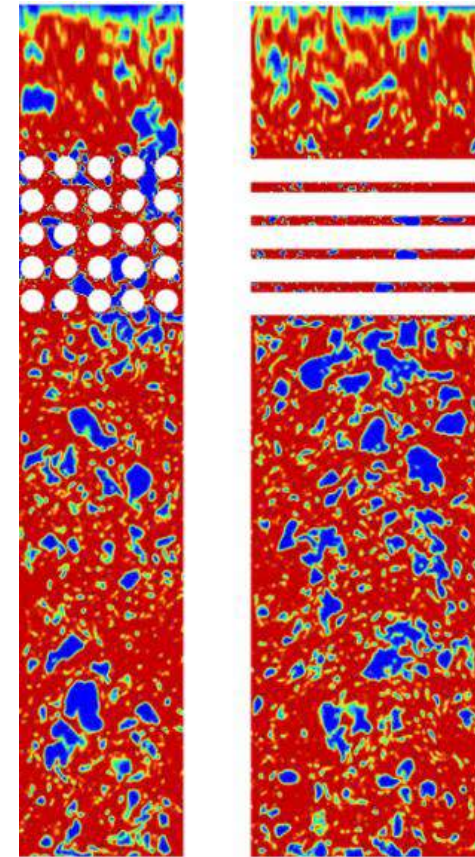
Time evolution of the mean drag coefficient (left) and the standard deviation of the lift coefficient (right) as obtained by EDF for the a single tube in a cross-flow (EDF).

FIV in multiphase flows



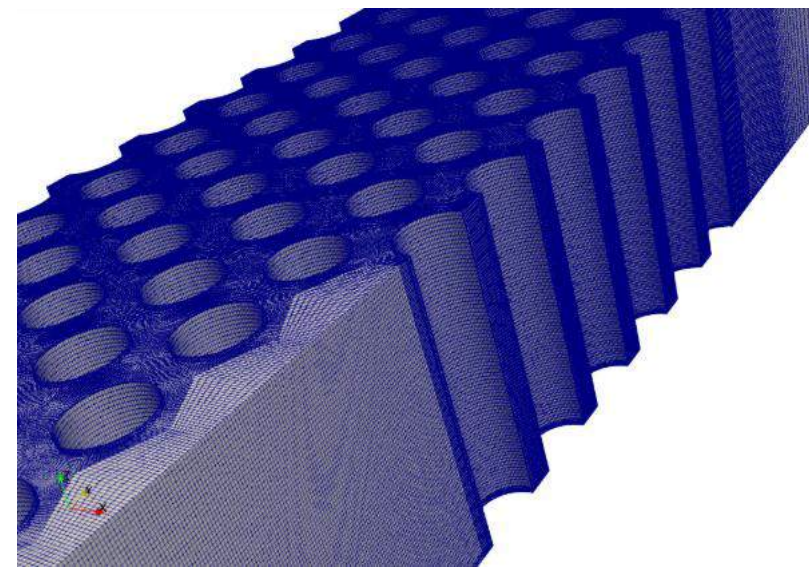
- **Fluid modelling:**

- Eulerian two-fluid approaches based on dispersed flow closure laws fail to predict experiments with higher voids. The reason: there is a larger spectrum of gas-liquid interface sizes when switching from bubbly to intermittent patterns → Use all-flow regime Eulerian approaches for improved results
- Algebraic Interfacial Area Density (AIAD) approach does not show an important sensitivity when a constant bubble diameter or a population balance is used, while the GENTOP approach is not easy to get converged.
- Volume of Fluid (VOF) seems to do a good job in the bundle simulations in terms of predicting void fraction profiles and force/vibration spectra, but still has some difficulties tracking the smallest bubbles
- the correct prediction of the force spectra on the structures seems be linked to the capability of the simulation to track and “resolve” large gas-liquid interfaces. Large gas structures seem to move inside the tube bundle without significantly breaking.
- numerical setups including compression methods for the void fraction seem to help the correct prediction of intermittent flows
- if the liquid velocity is too low, avoid averaged models and favor interface-resolving instead



Void distribution in TREFLE experiment, calculated with VOF (VTT).

FIV in multiphase flows



Numerical mesh of TITAN (EDF).

- **Inlet boundary conditions:**

- inlet boundary conditions of crucial importance for the FIV results in two-phase flows
- these play a major role especially in configurations with a limited number of tube rows, the flow pattern crossing the tube bundle region is mainly affected by the pattern found upstream
- using Synthetic Bubble Model (SBM) or dedicated gas injectors at the domain inlet might be useful

- **Numerical mesh:**

- mesh refinement is critical near the structure and in regions of strong void fraction gradients. Poor resolution leads to inaccurate force predictions and vibration amplitudes.
- ensure a well-resolved mesh at least 5 characteristic lengths upstream
- avoid abrupt mesh transitions and use progressive refinement
- deform the mesh only where necessary and optimize the smoothing algorithm settings for faster computations

FIV in multiphase flows



- **Numerical stability:**

- use iterative implicit FSI coupling schemes in case of numerical instabilities
- with the boundness factor in *Bounded Central Differencing (BCD)* schemes, one may increase the stability (numerical diffusion) at the expense of accuracy.
- velocity limiting may stabilize VOF (10-20 times the max. of mean mixture velocity)
- adaptive mesh refinement significantly increases computational cost, but can improve computational accuracy
- two-way coupling may become very expensive with two-phase flows, therefore, first check if one-way is feasible

- **Analysis and post-processing:**

- in low liquid superficial velocity regimes, fluctuations are often dominated by bubble dynamics rather than classical turbulence
- high void fraction cases easier for simulation than dispersed ones; for low void fraction, the dependency to the inlet gas size distribution is significant
- the largest structural vibrations mainly result from the large-scale interfaces → VOF likely to obtain the correct solution

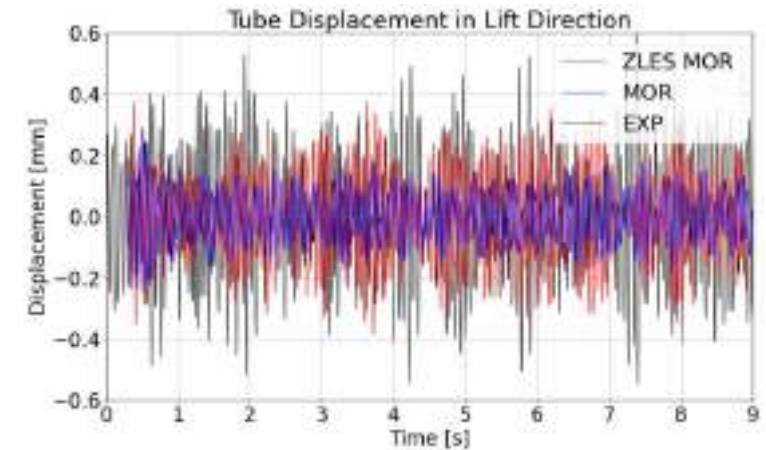
FLV in multiphase flows

- **Analysis and post-processing:**

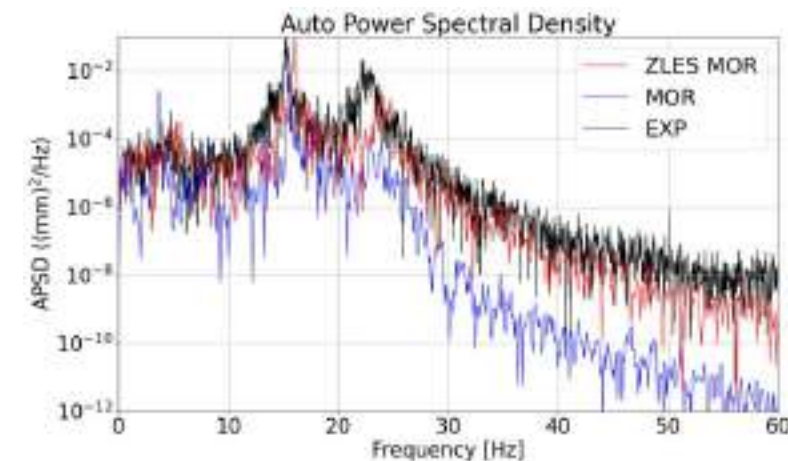
- to maintain a well-defined interface between liquid and gas phases, *the High-Resolution Interface Capturing (HRIC)* scheme within StarCCM+ shall be used
- spectral analysis is highly sensitive to: time step, total simulation time, windowing method, sampling frequency
- a meaningful comparison between experimental and numerical spectra requires strict consistency: same sampling frequency, time window, windowing technique → Do not use post-processing choices to artificially improve agreement.
- FSI analyses require longer sampling times for converged statistics (e.g. mean and RMS) compared to the flow field with fixed geometry
- monitoring only the void fraction is insufficient, the relevant conserved quantity is the phase flux
- simple 1D calculations are highly recommended to provide order-of-magnitude estimates for the simulation

Fast-running methods

- Reduced-order models (ROM) for the structural part can decrease the overall computational time (factor of 6), while negligibly affecting the accuracy
- Larger savings can be achieved when the ROM represents the fluid domain
- URANS and hybrid approaches in combination with structural ROM correctly predict the frequency response, but URANS underestimates the displacements
- Improve the prediction of displacements through local mesh refinement only in the vicinity of the FSI interface; the size of the first cell plays an important role
- Hybrid simulation may increase the displacements by 80% compared to SST on the same mesh
- Use hybrid model instead of URANS, if the mesh allows it, as the computational effort is similar

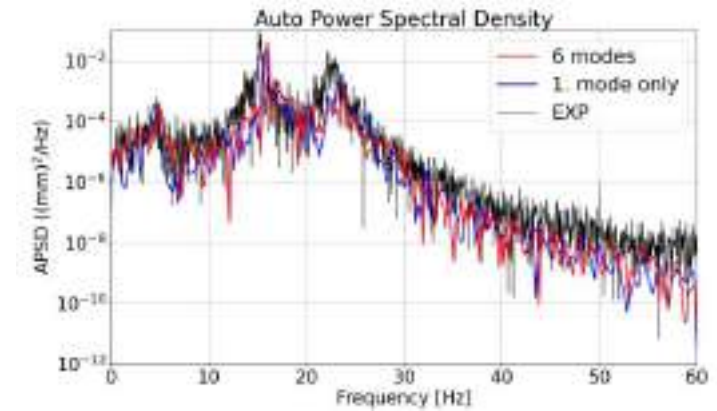


Tube displacements calculated with SST and ZLES turbulence models.



Frequency response calculated with SST and ZLES turbulence models.

Fast-running methods



APSD of simulation results based on the consideration of 1 and 6 modes (GRS).

- Pay attention to the FIV nature: if your structure vibrates primarily in its first mode, the consideration of higher modes will not significantly improve the results, but rather generate computational overhead
- Often, negative volumes appear during an FSI run with larger displacements. To improve the numerical stability in ANSYS CFX:
 - activate the option *Displacement Relative To = Initial Mesh*
 - increase the mesh stiffness, the option *Blended Distance and Small Volumes* provided best stability
- A mesh-independent CFD result does not necessarily mean that this is also true in terms of FSI
- Validate your model on different experimental runs by using the same setup with different boundary conditions
- The introduction of an extra damping or modification of the material properties (Young's modulus) or other non justified changes may improve the agreement with data, but does help the evaluation of the code's predictive capabilities.

UQ methods

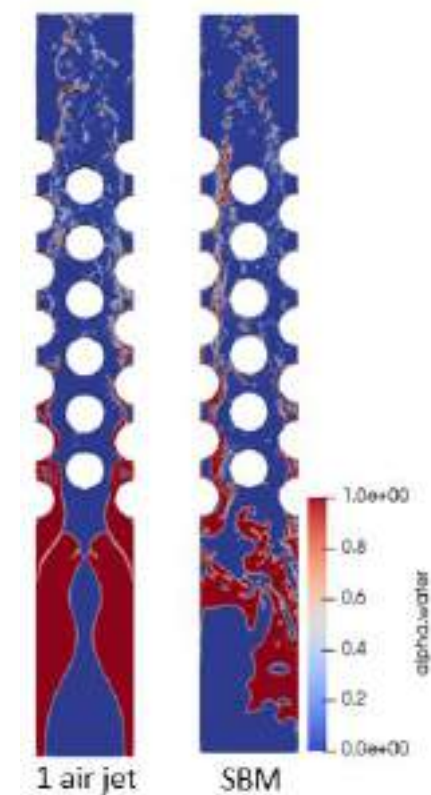


- UQ should not be treated as an additional step after CFD simulations, but as an integral part of the modelling process
- Sensitivity analysis at an early stage reveals which parameters actually matter and prevents unnecessary computational effort
- Surrogate models are indispensable in high-cost simulations
- First analyse simplified flow configurations to identify dominant parameters, then apply a reduced-parameter calibration to more complex geometries using surrogate models
- Predictive uncertainty should always be evaluated in terms of physical quantities of interest, not only in parameter space
- Calibration must be interpreted carefully. When predictive uncertainty bands fail to capture high-fidelity data, this is a clear indication of model-form error rather than insufficient parameter tuning.

UQ methods



- First validate fluid and structural solvers separately:
 - imposed deformation at the fluid-structure interface, using an analytical function for the motion in the expected mode and with the maximal expected amplitude
 - apply spatially and temporally varying load to the structure, as close as possible to what is expected in the FSI simulation
- Minimize the number of coupling iterations in case of partitioned simulation with coupling iterations → use quasi-Newton algorithms like IQN-ILS
- The structural supports in the simulations need special consideration
- Pay attention to the inlet boundary conditions, methods like SBM can be used

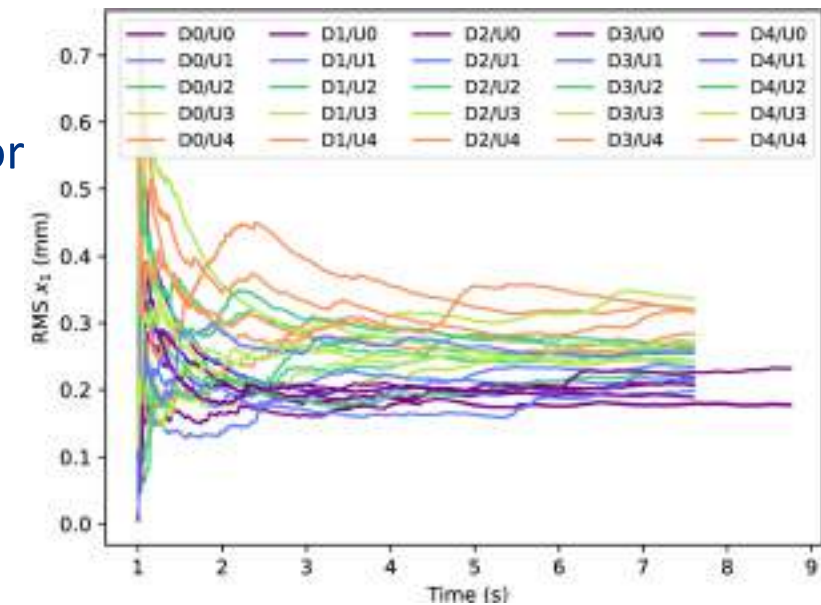


Difference between a single air jet and SBM as inlet for a tube bundle in multiphase cross flow (UGENT).

UQ methods



- Use efficient UQ methods (lower number of runs), as FSI is very expensive
- UQ techniques like Polynomial Chaos Expansion (PCE) are best suited for outputs that vary smoothly as a function of the inputs
- PCE is based on sampling of the input-output relation → incomplete convergence as a function of the simulation time can cause wiggles in the input-output relationship
- Hence, execute simulations for a sufficient number of vibration periods to ensure there are no wiggles in the output due to numerical parameters



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

Conclusions



- Best practices and lessons learned from the 4-year project duration were synthesized
- These shall help the stakeholders in the numerical analysis of FIV
- These are based on partners' experiences in the fields of FIV in axial and cross flows, multiphase flows, and the development and utilization of fast-running and UQ methods
- Guidelines and lessons learned are based on experiences with different codes and numerical approaches as well as on 7 dedicated experiments
- The BPGs are not prescriptive, but rather provide information that prevents other users dealing with similar problems from pitfalls
- The presented highlights are elaborated in more detail in GO-VIKING D5.5 "Best Practice Guidelines on the use of FSI methods for FIV evaluation"

Partners



Associated Partners



*EDF R&D UK Center



GO-VIKING

Thank you!



Angel Papukchiev, Kevin Zwijsen



GRS, NRG PALLAS



angel.papukchiev@grs.de,
kevin.zwijsen@nrgpallas.com

www.go-viking.eu



Funded by the
European Union