



GO-VIKING

Research and Innovation Action (RIA)

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

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



AMOVl model preparation and simulation results

Authors : Vladyslav FILONOV (IPP-Centre)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	AMOVl model preparation and simulation results
Author(s)	Vladyslav FILONOV
Number of pages	169
Document type	Deliverable
Work Package	WP3
Document number	D3.2
Issued by	IPP-Centre
Date of completion	2026-05-18 20:52:48
Dissemination level	Public

Summary

This work package 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. The GO-VIKING deliverable D3.2 is concerned with providing the results of the numerical simulations for the AMOVl 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. High-resolution AMOVl 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 in the corresponding Chapters. Each partner aimed to simulate the time dependent vibration amplitudes and forces acting on the rods for AMOVl 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 section 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 AMOVl 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.

Approval

Date	By
2026-05-19 08:00:05	Kevin ZWIJSEN (NRG)
2026-05-19 08:26:42	Papukchiev ANGEL (GRS)



D3.2 AMOVI model preparation and simulation results

Version N°2
May / 2026

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

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

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 Kevin Zwijsen, kevin.zwijsen@nrgpallas.com , NRG PALLAS
Project starting date (duration)	1st June 2022 – 31st May 2026 (48 Months)
Related Work Package	WP3
Related Task(s)	T3.1 Analysis of the AMOVI cross flow tube bundle
Lead Organisation	IPP
Contributing Partner(s)	IPP, GRS, EDF
Due Date	M42
Submission Date	18-05-2026
Dissemination level	PU

History

Date	Version	Submitted by	Reviewed by	Comments
05.05.2026	1	Y. Filonova	K. Zwijsen, A. Papukchiev	
18.05.2026	2	Y. Filonova	K. Zwijsen, A. Papukchiev	

Table of contents

1	Introduction	23
2	AMОВI benchmark.....	24
2.1	Experiment description	24
2.2	Selected benchmark cases	26
3	Simulations performed by GRS	28
3.1	Modelling of AMОВI Experiment	28
3.1.1	CAD Modelling of AMОВI Experiment	28
3.1.2	Numerical Mesh Generation	29
3.1.2.1	Force measurement cases.....	30
3.1.2.2	Displacement measurement cases	32
3.2	Generation of an inlet velocity profile	35
3.3	Analysis of the Forces Acting on the AMОВI Tube.....	36
3.3.1	Simulation setup and boundary condition.....	36
3.3.2	Single rigid tube.....	38
3.3.3	Rigid tube bundle	42
3.3.3.1	Analysis of force calculation for flow with 2.9 L/s	42
3.3.3.2	Analysis of force calculation for flow with 1.7 L/s	47
3.4	The Rigid Body Motion Approach	49
3.4.1	Angular Momentum Algorithm	49
3.4.2	Derivation of Numerical Parameter for the Rigid Body Motion Approach.....	50
3.4.2.1	Numerical modelling and structural analysis of the flexible tube	50
3.5	Analysis of AMОВI cases with mesh displacement.....	51
3.5.1	Simulation setup and boundary conditions	51
3.5.2	Single flexible tube	53
3.5.2.1	Analysis of FIV with 2.9 L/s flow rate	53
3.5.2.2	Analysis of FIV using 1.7 L/s flow rate.....	56
3.5.3	Flexible tube in a rigid tube bundle ($Q=1.6$ l/s)	58
3.6	Conclusions.....	62
4	Simulations performed by IPP	64
4.1	Physics and numerical modelling	64
4.1.1	Fluid modelling	64
4.1.1.1	CFD models for AMОВI Experiment	64
4.1.2	Mechanical modelling	70

4.1.2.1	Beam solution.....	71
4.1.2.2	Added mass effect.....	73
4.2	Numerical calculations results	75
4.2.1	Analysis of AMOVI cases with fixed rod (Rigid tube cases)	75
4.2.1.1	Single rigid tube - SRT-290 (Q=2.9 l/s)	75
4.2.1.2	Rigid tube bundle - RTB-170 (Q=1.7 l/s) and RTB-290 (Q=2.9 l/s)	76
4.2.2	Analysis of AMOVI cases with flexible Rod	79
4.2.2.1	Single flexible tube - SFT-50 (Q=0.5 l/s)	79
4.2.2.2	Single flexible tube - SFT-170 (Q=1.7 l/s)	80
4.2.2.3	Single flexible tube - SFT-290 (Q=2.9 l/s)	82
4.2.2.4	Single flexible tube - SFT-400 (Q=4.0 l/s)	84
4.2.2.5	Flexible tube in a rigid tube bundle - FTRTB-50 (Q=0.5 l/s)	86
4.2.2.6	Flexible tube in a rigid tube bundle - FTRTB-110 (Q=1.1 l/s)	87
4.2.2.7	Flexible tube in a rigid tube bundle - FTRTB-160 (Q=1.6 l/s)	88
4.2.2.8	Flexible tube in a rigid tube bundle - FTRTB-195 (Q=1.95 l/s)	89
4.3	Conclusions.....	90
5	Simulations performed by EDF	92
5.1	Single cylinder configuration computational domain and mesh generation	92
5.2	Tube bundle configuration computational domain and mesh	96
5.3	Modelling and numerical approach	99
5.4	Single tube configuration results	101
5.4.1	Single rigid tube configuration	101
5.4.1.1	Results at Q = 0.5 l/s and at Q = 1.7 l/s with the reference mesh	101
5.4.1.2	Results at Q=1.7 and Q = 2.9 l/s with coarser meshes.....	108
5.4.2	Single Flexible tube configuration results	112
5.4.2.1	Results at Q = 0.5 l/s and at Q = 1.7 l/s with the reference mesh	112
5.4.2.2	Results at Q = 2.9 l/s and Q = 1.7 l/s with a coarser mesh – sensitivity to the mesh refinement	114
5.5	Tube bundle configuration	116
5.5.1	Rigid cylinder in a tube bundle	116
5.5.1.1	Results at Q = 0.5 l/s, Q=1.7 l/s and Q = 2.9 l/s with the reference meshes	116
5.5.1.2	Results at intermediate flow rates	122
5.5.2	Single flexible tube in a rigid tube bundle.....	124
5.5.2.1	Results at Q = 0.5 l/s, Q = 1.6 l/s with the reference meshes	125
5.5.2.2	Fluid elastic instability thanks to coarser meshes ?	128
5.6	Conclusions.....	133
6	Simulations results summary: the AMOVI benchmark.....	135

6.1	Introduction.....	135
6.2	Single Rigid Tube cases.....	135
6.2.1	SRT-290 (Q=2.9 l/s). Simulations summary.....	138
6.3	Single Flexible Tube cases	140
6.3.1	SFT-170 (Q=1.7 l/s). Simulations summary	142
6.3.2	SFT-290 (Q=2.9 l/s). Simulations summary	143
6.4	Rigid Tube Bundle cases	145
6.4.1	RTB-170 (Q=1.7 l/s). Simulations summary	147
6.4.2	RTB-290 (Q=2.9 l/s). Simulations summary	151
6.5	Flexible Tube in a Rigid Tube Bundle cases.....	154
6.5.1	FTRTB-160 (Q=1.6 l/s). Simulations summary.....	156
6.5.2	FTRTB-195 (Q=1.95 l/s). Simulations summary.....	159
6.6	Overall Benchmark Summary.....	161
	Conclusion	165
	Bibliography.....	167

List of figures

Figure 1: AMOVI test section with two different configurations: single tube (left) and tube bundle (right) in a flow channel	24
Figure 2: AMOVI cross-sectional view parallel to flow direction: Flexible tube configuration for displacement measurements through a strain gauge	25
Figure 3: AMOVI cross-sectional view parallel to flow direction: Rigid configuration for force measurements through a tri-axial force sensor.	26
Figure 4: CAD models: Single rigid tube (left) and Rigid tube bundle (right).	29
Figure 5: CAD models: Single flexible tube (left) and Flexible tube in a rigid tube bundle (right) ..	29
Figure 6: Single rigid tube mesh: Full domain (top) and a sectional view (bottom).	30
Figure 7: Mesh study for the single rigid tube case.....	31
Figure 8: Rigid tube bundle mesh.....	31
Figure 9: Detailed sectional mesh view of the rigid tube bundle with central rigid tube	32
Figure 10: Mesh study for the rigid tube bundle case	32
Figure 11: Isometric mesh view of single flexible tube case.....	33
Figure 12: Mesh sectional view of single flexible tube including the fluid gap in the AMOVI domain	33
Figure 13: Mesh independence study for single flexible tube case	34
Figure 14: Isometric mesh view of flexible tube in rigid tube bundle.....	34
Figure 15: Isometric mesh view of the outer cylinder of flexible tube in the rigid tube bundle.	35
Figure 16: Sectional mesh view of the flexible tube.	35
Figure 17: Computation domain of small sector for development of inlet velocity profile.....	36
Figure 18: Fully developed velocity profile at the inlet for the AMOVI configuration with a single flexible tube.....	36
Figure 19: LES (red) and URANS (blue) Zones: Single rigid tube (top) and rigid tube in rigid tube bundle (bottom).....	38
Figure 20: APSD of lift force for single rigid tube	39
Figure 21: APSD of drag force for single rigid tube.....	40
Figure 22: Velocity contour in a longitudinal plane showing flow acceleration and deceleration around the tube as well as re-circulation zones (left) and formation of a von Kármán vortex street behind the rigid tube (right), visualized with the turbulence kinetic energy	41

Figure 23: Resolved structures for single rigid tube in cross flow (SAS model)	41
Figure 24: Resolved structures for single rigid tube in cross flow (SST model)	42
Figure 25: Calculated instantaneous velocity for 2.9 L/s at $t = 10$ s on middle ZX (left) and YZ Planes (right).	43
Figure 26: Recirculation zones visualized with velocity vectors on middle ZX plane for 2.9 L/s case.	44
Figure 27: Resolved structures for rigid tube bundle in cross flow (ZLES)	44
Figure 28: Measured (top) and calculated (bottom) time-averaged streamwise velocity in the tube bundle for 2.9 L/s case.....	45
Figure 29: APSD of lift force for rigid tube bundle for 2.9 L/s	46
Figure 30: APSD of drag force for rigid tube bundle for 2.9 L/s.....	46
Figure 31: Calculated instantaneous velocity for 1.7 L/s at $t = 10$ s on middle ZX (left) and YZ planes (right).	47
Figure 32: Measured (top) and calculated (bottom) time-averaged streamwise velocity in the tube bundle for 1.7 L/s case.	48
Figure 33: APSD of lift force for rigid tube bundle for 1.7 L/s	48
Figure 34: APSD of drag force for rigid tube bundle for 1.7 L/s.....	49
Figure 35: Flexible tube model: Numerical mesh of structural model (left); Boundary condition and coordinates system (right)	51
Figure 36: Measured displacement under application of external force	51
Figure 37: Modelling of rigid body for vibrating cases	52
Figure 38: LES (red) and URANS (blue) zones: flexible tube in rigid tube bundle (left) and Single flexible tube (right)	53
Figure 39: Velocity magnitude distribution in a longitudinal plane (water flows bottom-top) through the AMOVI test section.	54
Figure 40: Velocity colored vectors in a longitudinal plane showing flow acceleration and deceleration around the tube as well as re-circulation zones (top) and formation of a von Kármán vortex street behind the vibrating tube (bottom), visualized with the turbulence kinetic energy.	54
Figure 41: Comparison of tube displacements based on turbulence models	55
Figure 42: APSD of displacement for single flexible tube (2.9 L/s).....	56
Figure 43: Velocity magnitude distribution in a longitudinal plane (water flows bottom-top) through the AMOVI test section.	57

Figure 44: Displacement over time for single flexible tube (1.7 L/s).....	57
Figure 45: APSD of displacement for single flexible tube (1.7 L/s)	58
Figure 46: Calculated instantaneous velocity at $t = 5.5$ s in ZX (left) and YZ planes (right).....	59
Figure 47: Recirculation zones visualized with velocity vectors.....	59
Figure 48: Measured (top) and calculated (bottom) time-averaged streamwise velocity in the tube bundle.	60
Figure 49: Comparison of tube displacements based on Simo Wong (Rigid Body Motion algorithm accuracy) with experimental data.....	61
Figure 50: Comparison of tube displacements based on First Order Backward Euler (Rigid Body Motion algorithm accuracy) with experimental data	61
Figure 51: APSD of flexible tube in rigid tube bundle case with 1.6 L/s	62
Figure 52: General view and schematic representation of boundary conditions applied in CFD models for single rod (left) and rod bundle (right) cases	66
Figure 53: Calculation meshes applied for single rod CFD model.....	67
Figure 54: Mesh independence study for the single rigid tube case	68
Figure 55: Mesh independence study for the single flexible tube case	68
Figure 56: Calculation meshes applied for rod bundle CFD model.....	69
Figure 57: Mesh independence study for the rigid tube bundle case	70
Figure 58: Mesh independence study for the flexible tube in a rigid tube bundle case	70
Figure 59: Beam with Rigid body (up), Two Beam model (bottom)	72
Figure 60: Added mass: a) flexible cylinder oscillating in infinite fluid, b) flexible cylinder oscillating in rigid bundle	74
Figure 61: Added mass: effect for the flexible cylinder oscillating in infinite fluid	74
Figure 62: Comparison of resulting APSD of lift force (left) and drag force (right) with experimental APSDs for the single rigid tube with $Q=2.9$ l/s	75
Figure 63: Simulated fields of the longitudinal velocity component and vorticity picture obtained for the single rigid tube with $Q=2.9$ l/s	76
Figure 64: Comparison of the simulated lift force (left) and drag force (right) in time with experimental forces measured for the rigid tube bundle with $Q=1.7$ l/s.....	77
Figure 65: Comparison of resulting APSD of lift force (left) and drag force (right) with experimental APSDs for the rigid tube bundle with $Q=1.7$ l/s	77

Figure 66: Comparison of the simulated lift force (left) and drag force (right) in time with experimental forces measured for the rigid tube bundle with $Q=2.9$ l/s	78
Figure 67: Comparison of resulting APSD of lift force (left) and drag force (right) with experimental APSDs for the rigid tube bundle with $Q=2.9$ l/s.....	78
Figure 68: Comparison of resulting time-averaged streamwise velocity for RTB-170 (left) and RTB-290 (right) with the measured in experiment (bottom left/right)	79
Figure 69: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=0.5$ l/s	80
Figure 70: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=1.7$ l/s (Rigid Body approach).....	81
Figure 71: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=1.7$ l/s (User Beam lib)	82
Figure 72: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=2.9$ l/s (Rigid Body approach; LowRe mesh)	83
Figure 73: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=2.9$ l/s (Rigid Body approach; ICEM mesh)	83
Figure 74: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=2.9$ l/s (User Beam lib; ICEM mesh).....	84
Figure 75: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=4.0$ l/s (Rigid Body approach; LowRe mesh)	85
Figure 76: Comparison of tube displacements (left) and corresponding APSDs (right) for the single flexible tube with $Q=4.0$ l/s (Rigid Body approach; ICEM mesh)	85
Figure 77: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=4.0$ l/s (User Beam lib; ICEM mesh).....	86
Figure 78: Comparison of tube displacements (left) and appropriate APSDs (right) for the flexible tube in a rigid tube bundle with $Q=0.5$ l/s (Rigid Body approach).....	87
Figure 79: Comparison of tube displacements (left) and appropriate APSDs (right) for the flexible tube in a rigid tube bundle with $Q=1.1$ l/s (Rigid Body approach)	88
Figure 80: Comparison of tube displacements (left) and appropriate APSDs (right) for the flexible tube in a rigid tube bundle with $Q=1.6$ l/s (Rigid Body approach)	89
Figure 81: Comparison of tube displacements (left) and appropriate APSDs (right) for the flexible tube in a rigid tube bundle with $Q=1.95$ l/s (Rigid Body approach)	90
Figure 82: Comparison of simulated by IPP and experimental values of RMS value of the displacement as a function of the flow rate	91

Figure 83: The computational domain and the boundary conditions for the single cylinder configuration of Amovi. The frame axes are the one used in this figure, the origine being at the center of the cylinder.....	93
Figure 84: The main parameters for the mesh generation for the single cylinder configuration ...	95
Figure 85: The computational domain and the boundary conditions for the tube bundle configuration. The frame axes is the one used in this figure, the origine being at the center of the central cylinder.....	98
Figure 86: The main parameters for the mesh generation for the tube bundle configuration.....	98
Figure 87: The stream-wise instantaneous velocity for $Q = 1.7$ l/s showing how the precursor computation (left part of the domain) feeds the main computational domain (right part) containing the tube bundle	100
Figure 88: Instantaneous y^+ (top), time averaged y^+ (bottom left) and standard deviation of y^+ (bottom right) for the reference mesh at $Q = 1.7$ l/s – overview on the sidewalls.....	102
Figure 89: Instantaneous y^+ (left), time averaged y^+ (middle) and standard deviation of y^+ (right) on the tube wall for the reference mesh at $Q = 1.7$ l/s – top view on the cylinder, the flows goes from left to right.....	103
Figure 90: Instantaneous and time averaged non-dimensional stream-wise velocity in the center plane at $Q = 0.5$ l/s (left) and $Q = 1.7$ l/s (right) with the reference meshes - u/U_∞ (top), u/U_∞ (bottom)	103
Figure 91: Standard deviation of the velocity components at $Q = 0.5$ l/s (left) and $Q = 1.7$ l/s (right) with the reference meshes, u'^2/U_∞ (top), v'^2/U_∞ (middle), w'^2/U_∞ (bottom).....	104
Figure 92: Time averaged (top) and standard deviation (bottom) of the pressure at $Q = 0.5$ l/s (left) and $Q = 1.7$ l/s (right) with the reference meshes - $p/.5\rho U_\infty^2$ (top), $p'^2/0.5\rho U_\infty^2$ (bottom)	105
Figure 93: Standard deviation of the non-dimensional pressure $p'^2/0.5\rho U_\infty^2$ on the cylinder surface for the reference mesh at $Q = 0.5$ l/s (left) and $Q = 1.7$ l/s (right) – top view on the cylinder, le flows goes from left to right.....	105
Figure 94: Time evolution of the mean drag coefficient CD and the standard deviation of the lift coefficient $CL'2$ for the references meshes at $Q = 0.5$ l/s and 1.7 l/s – single rigid cylinder configuration	107
Figure 95: Time evolution of $Err(CD)$ and $Err(CL'2)$ for the references meshes at $Q = 0.5$ l/s and 1.7 l/s – single rigid cylinder configuration.....	107
Figure 96: Power Density Spectra (PDS) of Lift (left) and drag (right) forces per unit of length for the references meshes at $Q = 0.5$ l/s and 1.7 l.s – comparison with the experimental data....	108
Figure 97: Time averaged non-dimensional stream-wise velocity in the centre plane at $Q = 1.7$ l/s (top) and $Q = 2.9$ l/s (bottom).....	109
Figure 98: Time averaged non-dimensional pressure standard deviation in the centre plane at $Q = 1.7$ l/s (top) and $Q = 2.9$ l/s (bottom).....	110

Figure 99: Time evolution of the mean drag coefficient CD and the standard deviation of the lift coefficient $CL'2$ at $Q = 1.7$ l/s for the reference and coarse (a) meshes and at $Q = 2.9$ l/s for coarse (a) and coarse (b) meshes	111
Figure 100: Power Density Spectra (PDS) of Lift (left) and drag (right) forces per unit of length at $Q = 1.7$ l/s for the reference and coarse (a) meshes and at $Q = 2.9$ l/s for coarse (a) and coarse (b) meshes – comparison with the experimental data	111
Figure 101: Time evolution of the cylinder displacement for the references meshes at $Q = 0.5$ l/s and 1.7 l/s – single flexible cylinder configuration.....	113
Figure 102: Time evolution of cylinder displacement standard deviation for the references meshes at $Q = 0.5$ l/s and 1.7 l/s – single flexible cylinder configuration	113
Figure 103: Power Density Spectra (PDS) of the tube displacement at $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes – flexible single cylinder configuration – comparison with the experimental data	113
Figure 104: Time evolution of the cylinder displacement for the references meshes at $Q = 1.7$ l/s and 2.9 l/s – single flexible cylinder configuration.....	115
Figure 105: Time evolution of cylinder displacement standard deviation for the references meshes at $Q = 1.7$ l/s and 2.9 l/s – single flexible cylinder configuration	115
Figure 106: Power Density Spectra (PDS) of the tube displacement at $Q = 1.7$ l/s and $Q = 2.9$ l/s – flexible single cylinder configuration – comparison with the experimental data	116
Figure 107: Instantaneous views of the non-dimensional distance at the wall – overview of the sidewalls (top) – view on the 5 central tubes the flow going from left to right (left: view from the bottom, right: view from the top)	118
Figure 108: Instantaneous and time averaged non-dimensional stream-wise velocity in the center plane at $Q = 1.7$ l/s (left) and $Q = 2.9$ l/s (right) with the reference meshes - u/U_∞ (top), u/U_∞ (bottom)	119
Figure 109: Standard deviation of the non-dimensional stream-wise velocity component at $Q = 1.7$ l/s (left) and $Q = 2.9$ l/s (right) with the reference meshes	119
Figure 110: Time averaged (top) and standard deviation (bottom) of the pressure at $Q = 1.7$ l/s (left) and $Q = 2.9$ l/s (right) with the reference meshes – rigid tube bundle configuration – $p/0.5\rho U_\infty^2$ (top), $p'/20.5\rho U_\infty^2$ (bottom).....	120
Figure 111: Time evolution of the drag and lift forces standard deviation per unit of length (F'_D and F'_L) as a function of time for $Q = 0.5$ l/s with the reference mesh	121
Figure 112: Time evolution of the drag force standard deviation per unit of length (F'_D) as a function of time for $Q = 1.7$ l/s and $Q = 2.9$ l/s with the reference meshes	121
Figure 113: Time evolution of $Err(F'_D)$ and $Err(F'_L)$ for $Q = 0.5$ l/s with the reference mesh	121
Figure 114: Time evolution of $Err(F'_D)$ for $Q = 1.7$ l/s and $Q = 2.9$ l/s with the reference mesh.....	121

Figure 115: Power Density Spectra (PDS) of the lift forces per unit of length at $Q = 0.5$ l/s, $Q=1.7$ l/s and $Q=2.9$ l/s for the reference meshes – comparison with the experimental data	122
Figure 116: Power Density Spectra (PDS) of the drag forces per unit of length at $Q = 0.5$ l/s, $Q=1.7$ l/s and $Q=2.9$ l/s for the reference meshes – comparison with the experimental data	122
Figure 117: The time average of the drag coefficient of the central tube for the different flowrates of Table 40	124
Figure 118: The time average of the lift coefficient of the central tube for the different flowrates of Table 40	124
Figure 119: The standard deviation of the lift and drag coefficients for the different flowrates given in Table 40 – comparisons with the experimental data	124
Figure 120: The standard deviation of the lift and drag forces per unit of length for the different flowrates given in Table 40 – comparisons with the experimental data	124
Figure 121: Time average of the non-dimensional mean stream-wise velocity (top) and standard deviation of the non-dimensional pressure (bottom) at $Q = 1.7$ l/s for the rigid tube bundle configuration (left) and at $Q = 1.6$ l/s for the single flexible cylinder in a tube bundle (right) – reference meshes – rigid tube bundle configuration	126
Figure 122: 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 - flexible cylinder in a rigid tube bundle configuration	127
Figure 123: $\text{Err}(y')$ as a function of time for $Q = 0.5$ l/s and $Q = 1.6$ l/s – reference meshes - flexible cylinder in a rigid tube bundle configuration	127
Figure 124: Power Density Spectra (PDS) of the tube displacement at $Q = 0.5$ l/s and $Q = 1.6$ l/s with the reference meshes – flexible cylinder in a rigid tube bundle configuration – comparison with the experimental data	128
Figure 125: Time averaged non-dimensional stream-wise velocity and pressure in the center plane at $Q = 1.95$ l/s with the reference (left) and the coarse (right) meshes – rigid tube bundle ...	130
Figure 126: 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	131
Figure 127: Standard deviation of the tube displacement for different flowrates compared with experimental data – flexible cylinder in a rigid tube bundle configuration	132
Figure 128: Mean drag force per unit of length on the central moving tube for different flowrates – comparison between “the flexible cylinder in a rigid tube bundle” configuration and “the rigid tube bundle” one	132
Figure 129: Standard deviation of the drag force per unit of length on the central moving tube for different flowrates – comparison between “the flexible cylinder in a rigid tube bundle” configuration and “the rigid tube bundle” one	133

Figure 130: Standard deviation of the lift force per unit of length on the central moving tube for different flowrates – comparison between “the flexible cylinder in a rigid tube bundle” configuration and “the rigid tube bundle” one.....	133
Figure 131: Comparison of the lift force simulated by the participants with experimental data for the single rigid tube with $Q=2.9$ l/s.....	138
Figure 132: Comparison of resulting APSD of lift force with experimental APSD for the single rigid tube with $Q=2.9$ l/s.....	138
Figure 133: Comparison of the drag force simulated by the participants with experimental data for the single rigid tube with $Q=2.9$ l/s	139
Figure 134: Comparison of resulting APSD of drag force with experimental APSD for the single rigid tube with $Q=2.9$ l/s.....	139
Figure 135: Comparison of the tube displacements simulated by the participants with experimental data for the single flexible tube with $Q=1.7$ l/s.....	142
Figure 136: Comparison of the resulting displacements APSD calculated by the participants with experimental data for the single flexible tube with $Q=1.7$ l/s	143
Figure 137: Comparison of the tube displacements simulated by the participants with experimental data for the single flexible tube with $Q=2.9$ l/s.....	144
Figure 138: Comparison of the resulting displacements APSD calculated by the participants with experimental data for the single flexible tube with $Q=2.9$ l/s.....	144
Figure 139: Comparison of the lift force on the central tube simulated by the participants with experimental data for the rigid tube bundle with $Q=1.7$ l/s	147
Figure 140: Comparison of resulting APSD of lift force for the central tube with experimental APSD for the rigid tube bundle with $Q=1.7$ l/s	148
Figure 141: Comparison of the drag force on the central rod simulated by the participants with experimental data for the rigid tube bundle with $Q=1.7$ l/s	148
Figure 142: Comparison of resulting APSD of drag force for the central rod with experimental APSD for the rigid tube bundle with $Q=1.7$ l/s	149
Figure 143: Comparison of the measured in experiment (EXP) and calculated by participants (EDF, GRS, IPP) time-averaged streamwise velocity in the rigid tube bundle with 1.7 l/s flow rate.	150
Figure 144: Comparison of the lift force on the central rod simulated by the participants with experimental data for the rigid tube bundle with $Q=2.9$ l/s.....	151
Figure 145: Comparison of resulting APSD of lift force for the central rod with experimental APSD for the rigid tube bundle with $Q=2.9$ l/s	151
Figure 146: Comparison of the drag force on the central rod simulated by the participants with experimental data for the rigid tube bundle with $Q=2.9$ l/s.....	152

Figure 147: Comparison of resulting APSD of drag force for the central rod with experimental APSD for the rigid tube bundle with $Q=2.9$ l/s	152
Figure 148: Comparison of the measured in experiment (EXP) and calculated by participants (EDF, GRS, IPP) time-averaged streamwise velocity in the rigid tube bundle with 2.9 l/s flow rate	153
Figure 149: Comparison of the central rod displacements simulated by the participants with experimental data for the flexible tube in a rigid tube bundle with $Q=1.6$ l/s	157
Figure 150: Comparison of the resulting central rod displacements APSD calculated by the participants with experimental data for the flexible tube in a rigid tube bundle with $Q=1.6$ l/s	157
Figure 151: Comparison of the measured in experiment (EXP) and calculated by participants (EDF,GRS,IPP) time-averaged streamwise velocity in the rigid tube bundle with central flexible rod and 1.6 l/s flow rate.....	158
Figure 152: Comparison of the central rod displacements simulated by the participants with experimental data for the flexible tube in a rigid tube bundle with $Q=1.95$ l/s.....	159
Figure 153: Comparison of the resulting central rod displacements APSD calculated by the participants with experimental data for the flexible tube in a rigid tube bundle with $Q=1.95$ l/s	160
Figure 154: Comparison of simulated and experimental values of RMS value of the turbulent force per unit length as a function of the flow rate (Single Rigid Tube cases)	163
Figure 155: Comparison of simulated and experimental values of RMS value of the turbulent force per unit length as a function of the flow rate (Rigid Tube Bundle cases).....	163
Figure 156: Comparison of simulated and experimental values of RMS value of the displacement as a function of the flow rate	163

List of tables

Table 1: Overview of available test cases with flow conditions and an indication of those selected for simulations	27
Table 2: Fluid properties of force calculation cases	37
Table 3: Comparison of calculated and measured RMS of forces	40
Table 4: Comparison of calculated and measured RMS of forces for rigid tube bundle 2.9 L/s case	46
Table 5: Fluid properties of displacement calculation cases.....	53
Table 6: Comparison of calculated and measured tube displacements and vibration frequencies for single flexible tube case.....	56

Table 7: Comparison of calculated and measured tube displacements and vibration frequencies for single flexible tube case (1.7 L/s)	58
Table 8: Comparison of calculated and measured tube displacements and vibration frequencies for flexible tube in rigid tube bundle case (Q=1.6 l/s)	62
Table 9: Comparison of natural frequencies (in Air)	73
Table 10: Comparison of RMS of forces obtained in calculations for a single rigid tube (Q=2.9 l/s) with the measured values	76
Table 11: Comparison of RMS of forces obtained in calculations for rigid tube bundle (Q=1.7 l/s) with the measured values	77
Table 12: Comparison of RMS of forces obtained in calculations for rigid tube bundle (Q=2.9 l/s) with the measured values	78
Table 13: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube (Q=0.5 l/s).....	80
Table 14: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube (Q=1.7 l/s)	82
Table 15: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube (Q=2.9 l/s).....	84
Table 16: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube (Q=4.0 l/s)	86
Table 17: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle (Q=0.5 l/s)	87
Table 18: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle (Q=1.1 l/s)	88
Table 19: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle (Q=1.6 l/s).....	89
Table 20: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle (Q=1.95 l/s).....	90
Table 21: The lengths of the computational domains for the single cylinder configuration of AMOVI (see Figure 83).....	93
Table 22: The first three flow rates of AMOVI experiment for the single cylinder configuration. Characteristic velocities and Reynolds numbers.	93
Table 23: The meshes used for the single cylinder configuration depending on the flowrate Q. ref. is the reference mesh (the finest in the single cylinder configuration), coarse (a) and coarse (b) are two levels of coarse refinements, (b) being coarser than (a)	95

Table 24: The number of discretization points in the spanwise and azimuthal direction around the single cylinder, the number of cells for the precursor simulation and the number of cells in the main computational domain (without the precursor domain)	96
Table 25: The gap velocity U_g , the upstream velocity U_∞ , the gap Reynolds number Re_g , the upstream Reynolds number Re_∞ and the hydraulic Reynolds number of the square duct at the inlet Re_H	99
Table 26: Representative lengths of the refinements shown in Figure 82	99
Table 27: The number of discretization points in the spanwise and azimuthal direction around cylinders, the number of cells for the precursor simulation and the number of cells in the main computational domain (without the precursor domain).....	99
Table 28: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes – single rigid cylinder configuration.....	102
Table 29: The computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length and the main frequency of the lift force - results of the reference mesh for $Q = 0.5$ l/s and $Q = 1.7$ l/s – single rigid cylinder configuration - comparison with experimental data.....	107
Table 30: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes and the coarse ones at $Q = 1.7$ l/s and $Q = 2.9$ l/s – single rigid cylinder configuration.....	109
Table 31: The computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length and the main frequency of the lift force - sensitivity to mesh refinement at $Q = 1.7$ l/s and $Q = 2.9$ l/s for the single cylinder configuration	111
Table 32: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes – single flexible cylinder configuration.....	112
Table 33: The computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length, the mean displacement of the cylinder and its standard deviation at $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes – single flexible cylinder configuration	112
Table 34: The main frequencies for the displacement, the drag and lift coefficient for $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes – comparison with the experimental data.....	113
Table 35: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes and the coarse ones at $Q = 1.7$ l/s and $Q = 2.9$ l/s – single flexible cylinder configuration.....	114
Table 36: The computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length, the mean displacement of the cylinder and its standard deviation at $Q = 1.7$ l/s and $Q = 2.9$ l/s – single flexible cylinder configuration	115
Table 37: The main frequencies for the displacement, the drag and lift coefficient for $Q = 1.7$ l/s and $Q = 2.9$ l/s with the reference meshes – comparison with the experimental data	115

Table 38: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes at $Q = 0.5$ l/s, $Q = 1.7$ l/s and $Q = 2.9$ l/s – rigid tube bundle configuration	117
Table 39: The computational/experimental time, the mean drag and lift forces per unit of length, and the standard deviation of the drag and lift forces per unit of length and the main frequency of the lift force - results of the reference mesh for $Q = 0.5$ l/s and $Q = 1.7$ l/s – rigid tube bundle configuration - comparison with experimental data	122
Table 40: The time-step, the computational time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length and the main frequency of the lift force - results of the reference mesh for several flow rates between $Q = 0.5$ l/s and $Q = 2.9$ l/s – rigid tube bundle configuration	123
Table 41: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes at $Q = 0.5$ l/s and $Q = 1.6$ l/s – flexible cylinder in a rigid tube bundle configuration	125
Table 42: The computational/experimental physical time, the mean displacement of the cylinder and its standard deviation and the dominant frequency at $Q = 0.5$ l/s and $Q = 1.6$ l/s with the reference meshes – single flexible cylinder in a rigid tube bundle configuration	127
Table 43: The mesh type, the physical time, the time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for $Q = 1.95$ l/s – flexible tube in a rigid tube bundle configuration.....	129
Table 44: The mesh type, the physical time, the mean drag and lift forces per unit of length, and the standard deviation of the drag and lift forces per unit of length – $Q = 1.95$ l/s with the reference and the coarsened mesh – rigid tube bundle configuration.....	130
Table 45: The mean displacement of the cylinder, its standard deviation and its main frequency for different flowrates – the meshes for $Q = 1.85$ and $Q = 1.95$ l/s are coarsened compared to the reference one – flexible cylinder in a rigid tube bundle configuration	131
Table 46: The mean drag and lift forces per unit of length and their standard deviation for different flowrates – the meshes for $Q = 1.85$ and $Q = 1.95$ l/s are coarsened compared to the reference one – flexible cylinder in a rigid tube bundle configuration	132
Table 47: Performed cases.....	135
Table 48: SRT - Key information of the submissions	136
Table 49: SRT - Calculation Domain Details.....	136
Table 50: SRT - Fluid Boundary Conditions.....	137
Table 51: SRT - Numerical Settings.....	137
Table 52: Comparison of the resulting RMS values of LIFT and DRAG forces obtained by participants for the case SRT-290.....	140
Table 53: SFT - Key information of the submissions	140

Table 54: SFT - Calculation Domain Details	141
Table 55: SFT - Fluid Boundary Conditions	141
Table 56: SFT - Numerical Settings	142
Table 57: Comparison of the resulting RMS values of tube displacement and appropriate APSD peaks frequencies obtained by participants for the case SFT-170.....	143
Table 58: Comparison of the resulting RMS values of tube displacement and appropriate APSD peaks frequencies obtained by participants for the case SFT-290	145
Table 59: RTB - Key information of the submissions	145
Table 60: RTB - Calculation Domain Details	146
Table 61: RTB - Fluid Boundary Conditions	146
Table 62: RTB - Numerical Settings.....	147
Table 63: Comparison of the resulting RMS values of LIFT and DRAG forces on the central rod obtained by participants for the case RTB-170	149
Table 64: Comparison of the resulting RMS values of LIFT and DRAG forces on the central rod obtained by participants for the case RTB-290	154
Table 65: FTRTB - Key information of the submissions	155
Table 66: FTRTB - Calculation Domain Details	155
Table 67: FTRTB - Fluid Boundary Conditions	156
Table 68: FTRTB - Numerical Settings	156
Table 69: Comparison of the resulting RMS values of central rod displacement and appropriate APSD peaks frequencies obtained by participants for the case FTRTB-160.....	159
Table 70: Comparison of the resulting RMS values of central rod displacement and appropriate APSD peaks frequencies obtained by participants for the case FTRTB-195.....	160
Table 71: Computational Resources used for calculations	162

Abbreviations and Acronyms

Acronym	description
ALE	Arbitrary Lagrangian Eulerian
BC	Boundary conditions
BWR	Boiling water reactors
CAD	computer-aided design
CFD	Computational fluid dynamics
CFL	Courant-Friedrichs-Lewy
CSM	Computational structural mechanics
DNS	Direct numerical simulation
EBNF	Euler-Bernoulli Normal Form
FA	Fuel assemblies
FDMK	Finite Difference Method Kinematic
FEM	Finite element method
FFT	Fast Fourier Transform
FIV	Flow-induced vibrations
FSI	fluid-structure Interaction
FTRTB	Flexible Tube in Rigid Tube Bundle
GO-VIKING	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
GTRFW	Grid-to-rod-fretting wear

LDV	Laser Doppler Velocimetry
LES	Large eddy simulation
LTO	Long-term operation
MOR	Model Order Reduction
NPP	Nuclear power plant
PIV	Particle Image Velocimetry
PSD	Auto-spectra or power spectral density
PWR	Pressurized water reactor
RANS	Reynolds-Averaged Navier-Stokes
RMS	Root Mean Square
RSM	Reynolds Stress Model (RSM)
RTB	Rigid Tube Bundle
SAS	Scale adaptive simulation
SFT	Single Flexible Tube
SRT	Single Rigid Tube
SG	steam generator
URANS	Unsteady Reynolds-Averaged Navier-Stokes
WP	Work package
WR-LES	Wall-resolved Large Eddy Simulation
ZLES	Zonal Large Eddy Simulation

Summary

This work package 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. The GO-VIKING deliverable D3.2 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.

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 in the corresponding Chapters. 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 section 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.

Keywords

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



Funded by
the European Union

1 Introduction

The integrity of the nuclear steam supply system (NSSS) of a nuclear power plant (NPP) is essential for its safety and operability. Key NSSS components such as fuel assemblies (FA) or steam generator (SG) tubes are subject to several ageing challenges like corrosion mechanisms, neutron flux embrittlement, and vibratory loads. Vibrations induced by coolant flow are particularly important, as they lead to increased wear and tear and/or material fatigue. Flow-induced vibrations (FIV) remain important contributors to key components' failures that may degrade NPP safety features and breach confinement barriers, thus generating operational and safety issues.

Over 70% of all fuel rod leakages in US pressurized water reactors (PWR) are due to FIV, according to EPRI [1]. According to IAEA, grid-to-rod-fretting wear (GTRFW), resulting from such vibrations, is the cause of 58% of fuel failures in PWRs worldwide [2]. The highly turbulent coolant flow in FAs during normal operation results in asymmetric and fluctuating pressure on the fuel rods' surfaces. The pressure fluctuations are random and oscillate over a broad band of frequencies and amplitudes over the entire rod surface. This leads to dynamic spacer grid-to-rod contacts and frictions, resulting in material wear in the contact region between the rod and its support (springs and dimples).

FIV cause damage also in SGs. The primary side of a PWR SG consists of tube bundles, cooled by the secondary side water. Depending on fluid velocity, tube geometry, and tube fixation, amongst others, secondary water vortex shedding may excite tube vibrations. As a consequence, repeating tube bending may lead to increased material fatigue and eventually damage [3]-[5]. In 2012, a SG tube rupture (SGTR) accident occurred in the US NPP San Onofre. Extensive wear of more than 3000 SG tubes was found in its units 2 and 3, leading to primary-to-secondary circuit coolant leaks. The SGs in these units were put into operation one year earlier. The principal reason for the extensive wear and the tube damage was found to be related to FIV [6]. Since then, these units are in a shut-down condition, pending decommissioning. SGTR as a result of FIV has also been observed in many other NPPs in the past [7]. The enhancement of the heat transfer in the heat exchangers through increased flow velocities and thinner tube walls increases the susceptibility of the tubes to FIV. This is further increased by ageing mechanisms that have a negative impact on material wear and the mechanical contacts. Therefore, it is important to properly consider FIV loads in the component design and the operational surveillance and maintenance programs. This also holds for power uprates and plant long-term operation (LTO) programs.

To assess and further advance the current state-of-the-art of studying FIV in NPPs, the joint industry VIKING (Vibration ImpaKt In Nuclear power Generation) project was launched at the beginning of 2020. In this project, nine organizations collaborated for almost four years on FIV of configurations representative of steam generators and fuel rods and assemblies. This was done by performing numerical benchmark studies on five different experimental facilities [13]. To further advance the knowledge of solving FSI problems with the help of numerical tools, the current Horizon Europe GO-VIKING project (<https://go-viking.eu/>) was granted by the European Commission in 2022 [14]. It gathers expertise from academia and research organizations, industry, and technical safety organizations (TSO) to synthesize and improve good practices, as well as to develop accurate numerical simulation methods for FIV evaluation. These will support the prediction and FIV evaluation by EU stakeholders, as well as their decision making on FIV countermeasures in the NPPs. In this deliverable, the numerical efforts to simulate the AMOVI experiment are described. AMOVI deals with representative geometries for a SG tubes exposed to a single phase cross-flow.

2 AMOVI benchmark

This chapter presents a short description of the AMOVI experiment and provides information on the benchmark cases selected by each participant for performing the simulations and appropriate comparisons.

2.1 Experiment description

AMOVI [15] is a small hydraulic loop operated by CEA Saclay (France) to study the fluid-structure interaction between a square tube bundle, arranged in different configurations, and a water flow characterized by a low Reynolds number. The hydraulic loop consists of a pump with a maximum flow rate of $Q = 4$ l/s; a water tank of about 200 l, and a test section with dimensions of 70 x 100 x 600 mm (width x depth x height). The square bundle consists of 3x5 tubes plus two columns of 5 semi-tubes. The diameter of the tubes is 12.15 mm, the length is 99 mm and the bundle pitch is 17.5 mm, hence $P/D = 1.44$, which is representative for SG tube bundles. The central stainless-steel tube of the bundle can be mounted in a rigid or flexible setup to get force or displacement measurements, respectively. The other rigid plexiglass tubes can be plugged in and out in order to study different basic configurations. Two basic arrangements are tested:

- a single tube in a reduced flow channel (Figure 1, left)
- a full tube bundle with central flexible tube in the flow channel (Figure 1, right).

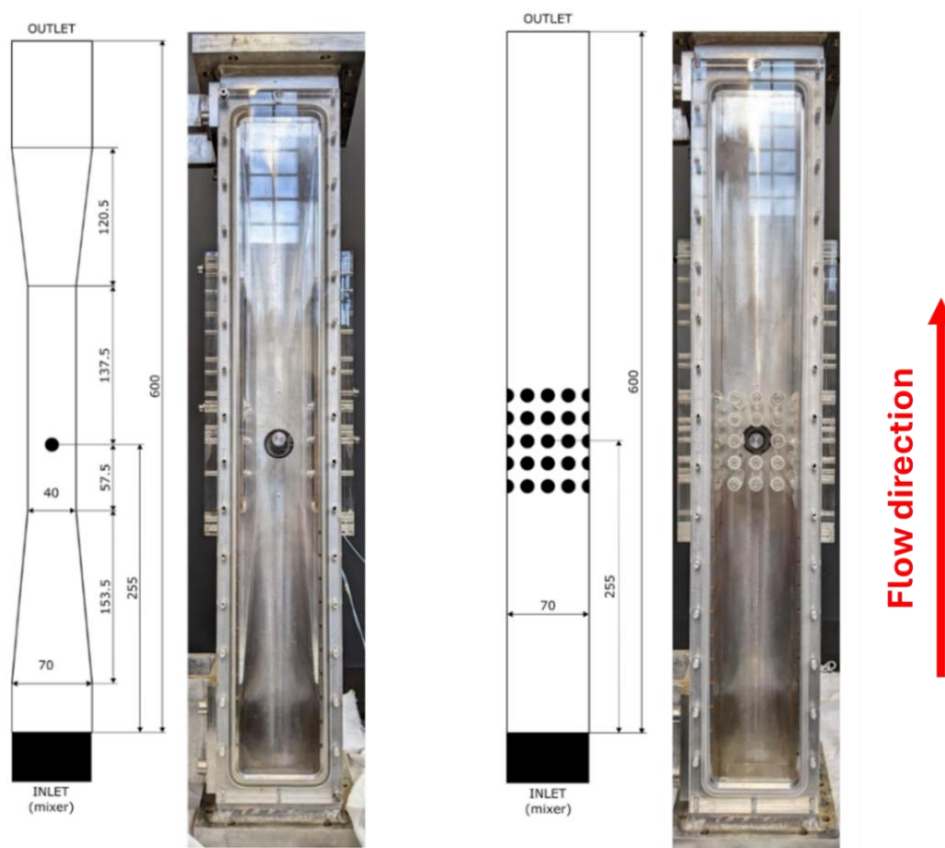


Figure 1: AMOVI test section with two different configurations: single tube (left) and tube bundle (right) in a flow channel

Each of these two configurations is then tested with the central tube being either rigid or flexible in the lift direction. This results in four test section configurations in total.

In the flexible arrangement, the central tube is supported by a flexible blade (Figure 2), allowing it to vibrate primarily in lift (perpendicular to the main flow) direction, with a frequency in air $f_n = 27.5$ Hz and a damping of $\xi_n = 0.064$ %. The displacement at the tip of the tube is deduced from a strain gauge instrumenting the root of the flexible blade (Figure 2). In the rigid arrangement, a tri-axial force sensor measures the forces acting on the central tube (Figure 3). Fluid flow parameters are obtained only for selected runs of interest using Particle Image Velocimetry (PIV) and Laser Doppler Velocimetry (LDV) setups. In the PIV setup, a high-speed camera is utilized to capture the flow around the tubes in a 2D central plane within the test section. To visualize the flow, spherical fluorescent particles are added to the water as seeding particles. In the LDV setup, the axial and horizontal time-average velocities and the turbulence fluctuations were probed upstream the test section thanks to two crossed lasers. Flow temperature, relative pressure and flow rate are monitored respectively with a thermometer placed on the tank, a pressure sensor positioned upstream of the mixer and a flow meter placed downstream of the pump.

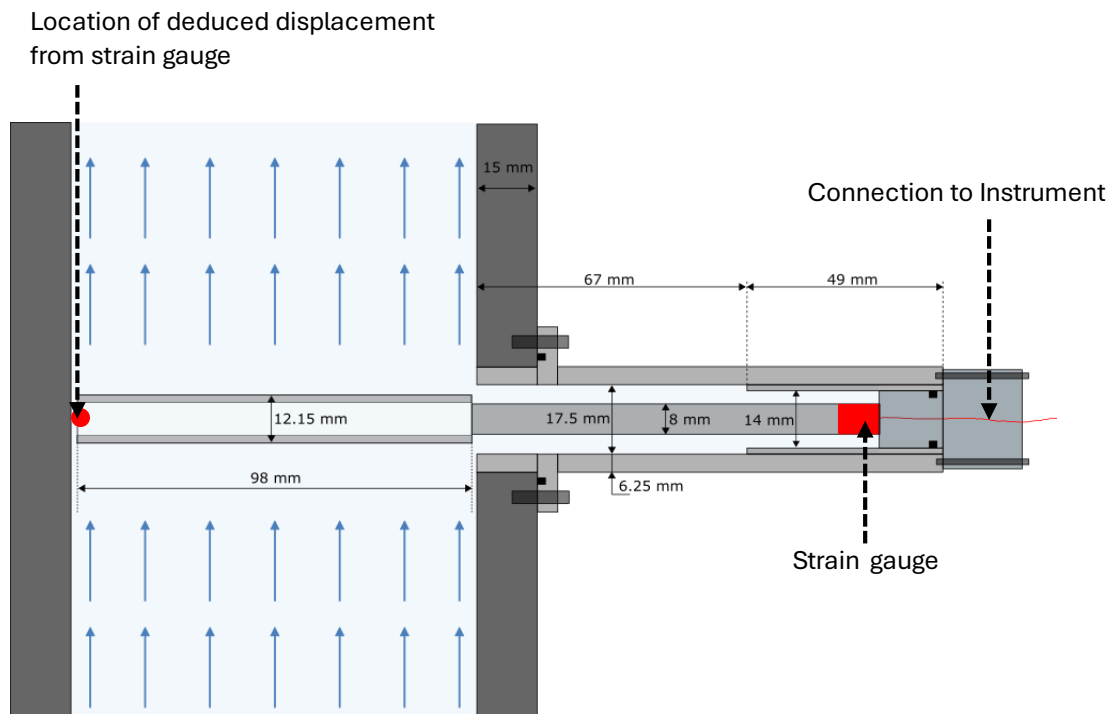


Figure 2: AMOVI cross-sectional view parallel to flow direction: Flexible tube configuration for displacement measurements through a strain gauge

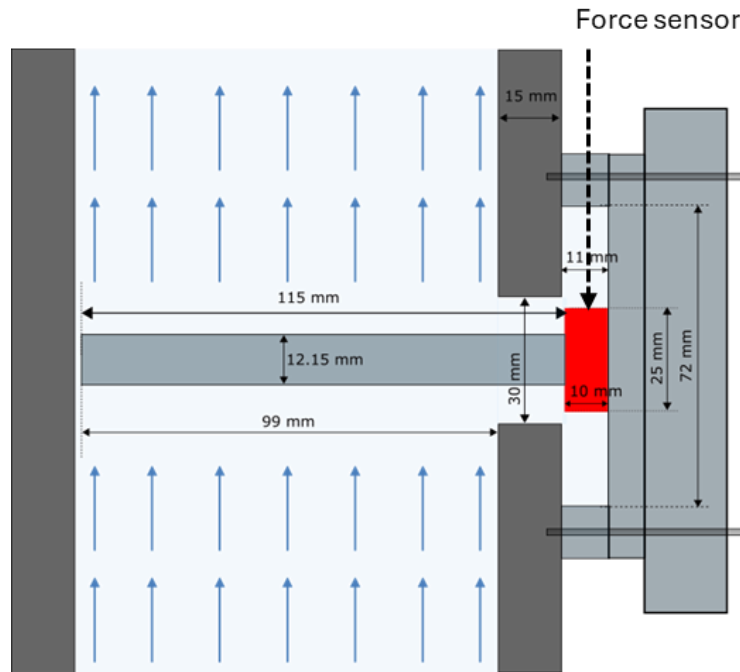


Figure 3: AMOVI cross-sectional view parallel to flow direction: Rigid configuration for force measurements through a tri-axial force sensor.

2.2 Selected benchmark cases

Sixteen different test cases are made available by CEA to the GO-VIKING partners. Tests are performed for four water flow rates for each configuration and the vibrations of the central tube or the forces acting on its surface are measured. For some cases of interest, LDV and PIV measurements are also performed to characterize the flow pattern.

Table 1 summarizes the main parameters of AMOVI test cases provided and indicates which of them were selected by current benchmark participants for numerical calculations.

In experiment, for all cases, the auto-spectra or power spectral density (PSD) were generated. At high frequencies, spectra are biased by noise: high unphysical peaks appear in the spectra, distorting the content of interest, which should instead tend toward zero. For this reason, in the PSD, frequencies above 200 Hz are not considered. For all the considered cases, the PSD of force and displacement signals are estimated via the Welch's method: recorded signals are divided into 100 segments of 4096 points with 50% overlap; each segment is windowed with a Hamming window.

The details on the prepared models and simulations performed by each participant are presented in the corresponding chapters below (see Chapters 3, 4 and 5). This is followed by Chapter 6, which is devoted to a general comparative analysis of the results obtained by all participants with experimental data for selected cases, where the approaches and modelling tools used, as well as the computational costs, are directly indicated and compared.

Table 1: Overview of available test cases with flow conditions and an indication of those selected for simulations

	Flow Rate (L/s)	Pressure (bar)	Temperature (°C)	PIV/LDV data available	Simulations performed		
					GRS	IPP	EDF
Single rigid tube	0.5	1.33	18	-	-	-	+
	1.7	3.80	25	-	-	-	+
	2.9	2.88	26	LDV	+	+	+
	4.0	3.5	26	-	-	-	-
Rigid 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 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	-	-	+	+

3 Simulations performed by GRS

In this work, GRS is responsible for simulating the AMOVI experiment. GRS has calculated the time dependent vibration amplitudes with ANSYS CFX Rigid Body Approach R21.2 and force measurement with ANSYS CFX R21.2 for selected test cases. Here the selected cases were investigated with RANS (Reynolds-Averaged Navier–Stokes) and hybrid models. Further details of AMOVI Modelling by GRS are provided in the following sections.

3.1 Modelling of AMOVI Experiment

This chapter introduces the CAD (computer-aided design) Modelling of AMOVI Experiment and provides the numerical mesh development methodology, including a mesh independence study for the numerical calculations.

3.1.1 CAD Modelling of AMOVI Experiment

The 2D drawing of the complete test section with the flow mixer at inlet was provided by CEA. Based on available drawing, the 3D model was prepared in the ANSYS DesignModeler. The geometry (CAD model) did not include the mixing section in the inlet chamber. The rigid tubes are 100 mm long with a diameter of $D=12.5$ mm and pitch of $P=17.5$ mm. First, the CAD model for the simulations for force calculation cases was prepared. As shown in Figure 3, the central tube has total length of 115 mm, however only 99 mm of its length is exposed to the flowing water with the 1 mm of tolerance. Consequently, the dead zone, where the flow has negligible impact (see Figure 3), is not modelled. Further, the original developed CAD model for the single rigid tube and single flexible tube configurations has a diverging part, which starts 137.5 mm downstream the central tube in the narrowed downstream section. After performing the initial CFD simulations, it was noticed that putting the OPENING boundary condition at this location (137.5 mm downstream from central tube) is reasonable, due to the absence of a re-circulation zone in this region. This simplification also helps to decrease the total number of mesh elements. The final models have dimension of 100 x 70 x 392.5 mm. Figure 4 represents the CAD models for the force calculation cases with single rigid tube (left) and rigid tube bundle (right).

Similarly, the CAD models for the flexible tube cases were generated (Figure 5). In these cases, the central flexible tube (tube + flexible blade) was modelled. The modelling of the outer cylinder part was important due to altered flow development towards the flexible tube mounting. Despite the test section symmetry, a full Rigid body (FSI) model of the AMOVI experiment was generated, due to the complexity of the tube movement, found in the experimental data.

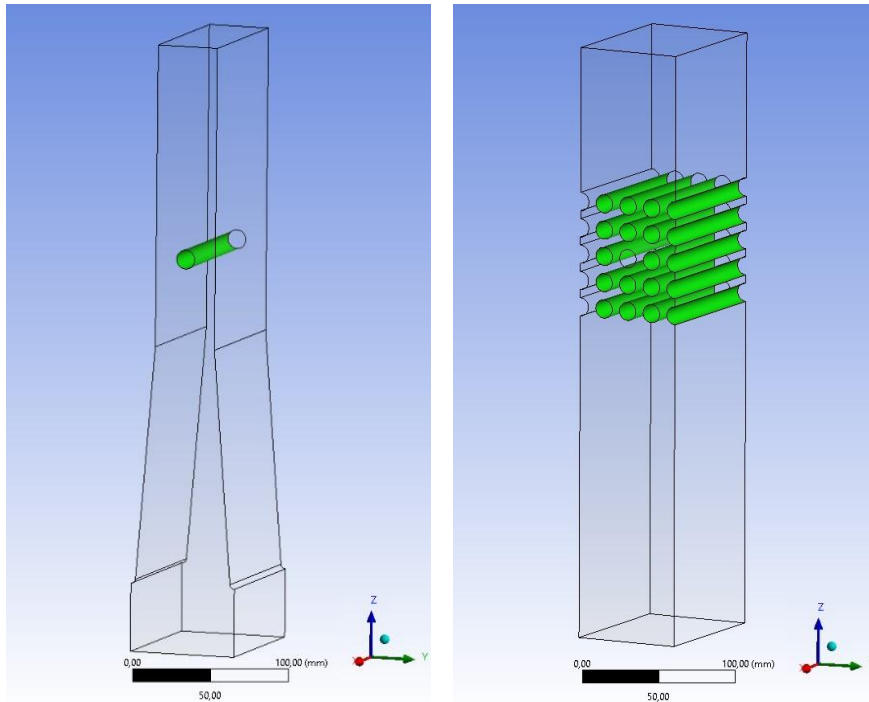


Figure 4: CAD models: Single rigid tube (left) and Rigid tube bundle (right).

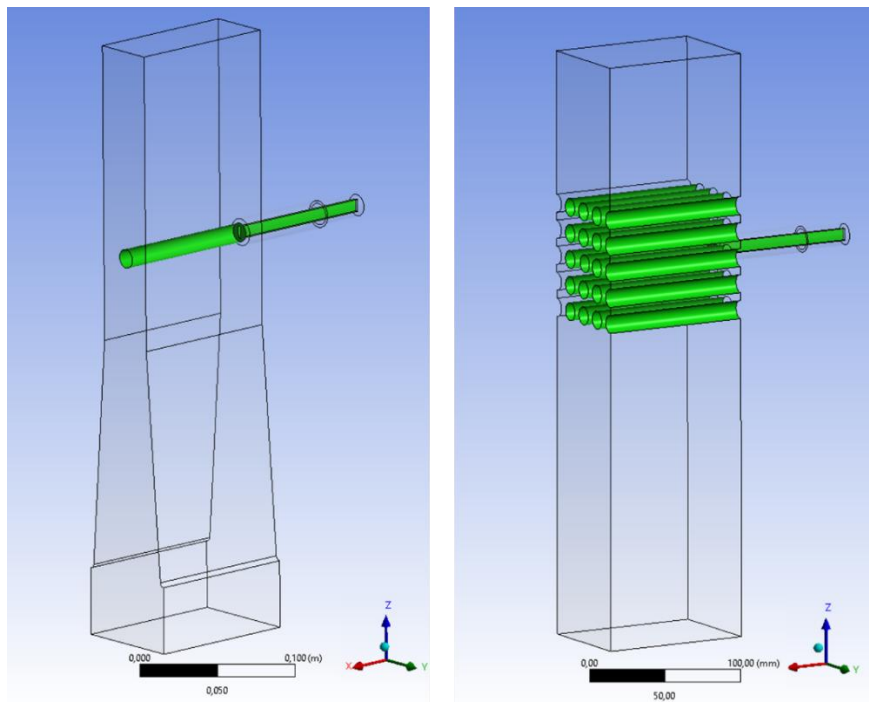


Figure 5: CAD models: Single flexible tube (left) and Flexible tube in a rigid tube bundle (right)

3.1.2 Numerical Mesh Generation

The RANS mesh is generally less demanding than LES or DNS meshes, since RANS models account for all turbulence scales through modeling rather than direct resolution. RANS models that predict large zones of flow separation will be more likely to develop unsteady solutions than models that predict small or no separation zones. Often, the models that predict larger separation zones are closer to the experimental data. Nevertheless, adequate mesh resolution remains crucial to

accurately capture key flow features. This is particularly important in the near-wall region, where turbulence models such as $k-\omega$ SST require fine resolution to correctly predict boundary layer behavior.

3.1.2.1 Force measurement cases

Hexahedral meshes are particularly advantageous for flows with thin shear layers or narrow gaps, as they provide higher numerical efficiency and accuracy than tetrahedral meshes. However, generating such meshes with ANSYS ICEM CFD is more complex and typically requires considerable manual effort. To keep the computational effort within practical limits, a hexahedral mesh was generated for the AMOVI test-section fluid domain in the force-measurement cases using ANSYS ICEM CFD. Mesh independence of the results is essential for the CFD analyses, as recommended in [16]. For the single rigid tube case, several meshes with 0.38 million (Coarse), 2.21 million (Medium), 5.15 million (Medium-Fine) and 8.52 million (Fine) elements were generated. Figure 6 shows the generated Medium-Fine mesh. For the mesh independence study, the velocity profile was checked on several locations in the domain. Figure 7 shows the transient velocity for a certain location. It was found that the results with the Medium-Fine mesh deviates by just 2% from the ones generated with the Fine mesh. Therefore, the Medium-Fine mesh was selected for the URANS and Hybrid-URANS simulations. The overall mesh quality is good: expansion factor of 7, maximum aspect ratio of 344 and minimum orthogonality angle of 43° .

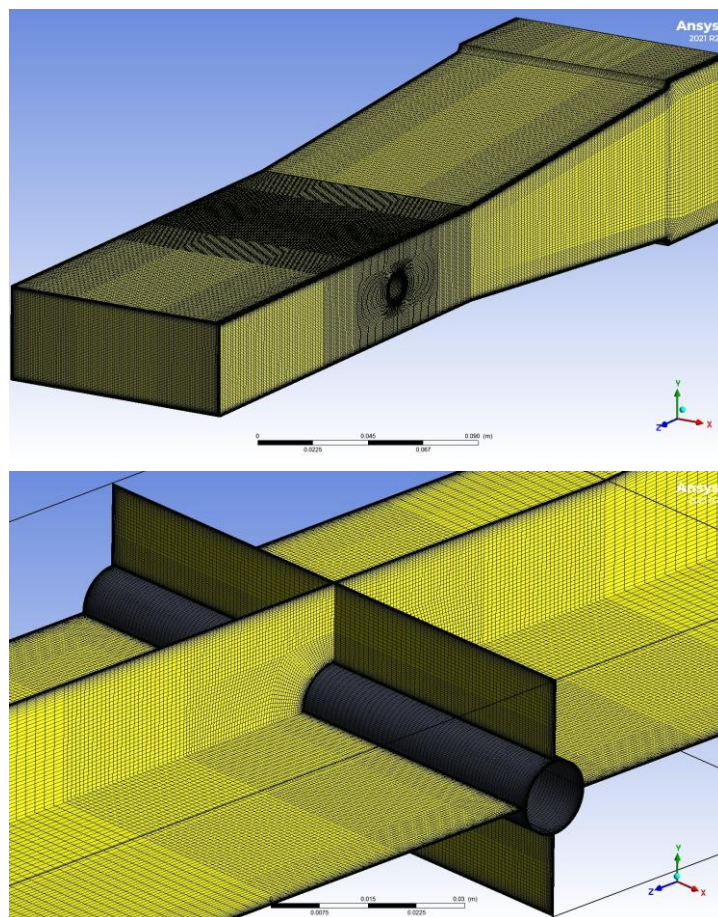


Figure 6: Single rigid tube mesh: Full domain (top) and a sectional view (bottom).

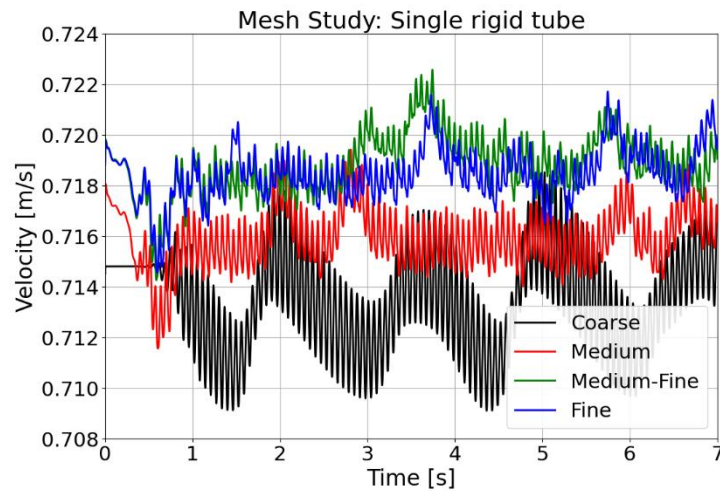


Figure 7: Mesh study for the single rigid tube case

Similarly, the rigid tube bundle geometry was also meshed with hexahedral elements, see Figure 8. During mesh generation, special care was taken for the half rigid tubes on the left and right sides of the test section channel to allow proper resolution of the wall boundary layers. Figure 9 shows the sectional view of the mesh with central rigid tube. Additionally, refinement was applied up- and downstream the tube bundle. Several meshes with 2.6 million (Coarse), 16 million (Medium) and 19 million (Fine) were generated. Both forces (Lift and Drag) calculated with coarse mesh showed a factor of 1.8~2 larger than experiment and were oscillating with larger amplitude. As we used refined meshes (mid and fine) the RMS value did not vary more than 0.02 N/m. Thus, mesh independency for forces was also achieved. Similar to the single rigid tube case, several velocity monitoring locations were defined in the CFD domain. Figure 10 shows the transient velocity progression shortly behind the rigid tube bundle. The results show the higher velocity fluctuations when using coarse mesh. Thus, the mesh with 16 million (Medium) mesh was selected for the URANS simulations. However, for the Hybrid-URANS simulations, the mesh with 19 million (Fine) was selected due to the better representation of the small turbulence structures. The quality of both meshes is good: the min. orthogonality angle is (49°), the maximum expansion factor is 4, and the aspect ratio is 85. The mean cell size in spanwise direction is 1.3 mm, while in azimuthal direction 0.225 mm.

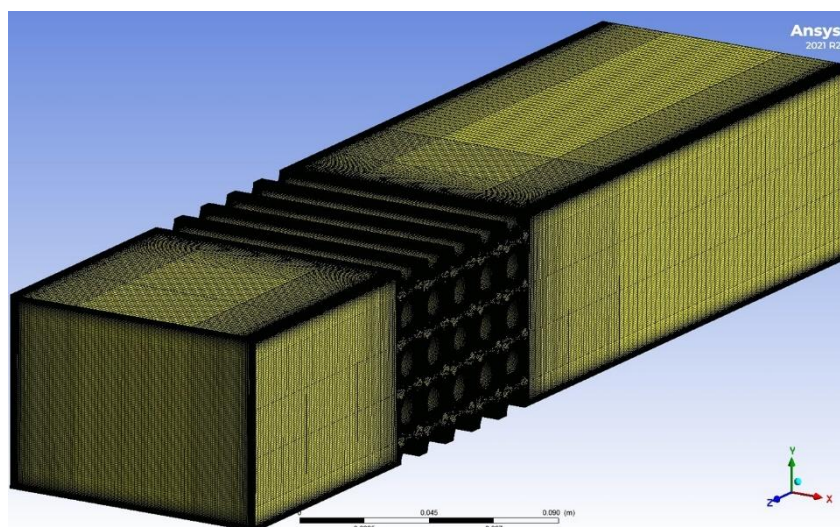


Figure 8: Rigid tube bundle mesh.

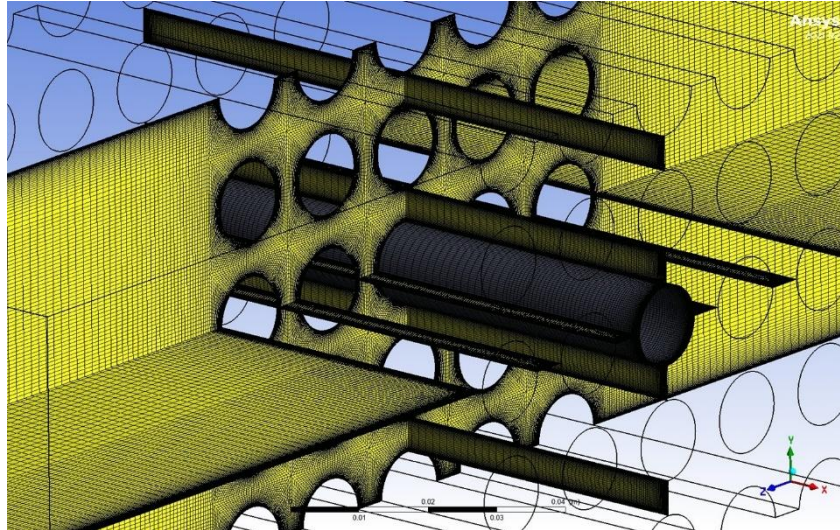


Figure 9: Detailed sectional mesh view of the rigid tube bundle with central rigid tube

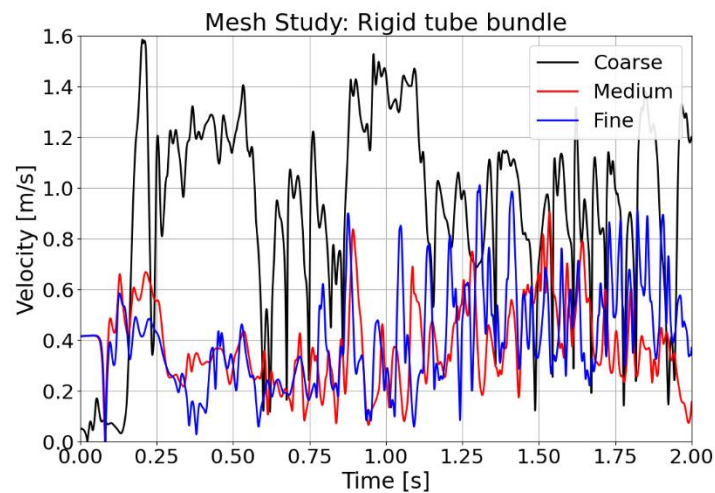


Figure 10: Mesh study for the rigid tube bundle case

3.1.2.2 Displacement measurement cases

For cases where unsteadiness is imposed from outside, like flows with moving solids, FSI, or unsteady boundary conditions, good-quality meshing is essential for obtaining accurate results. During FSI calculations, mesh deformation can occur (as in the present analysis), which directly affects mesh quality by altering key parameters such as minimum orthogonality angle, expansion factor, and aspect ratio. These changes influence the convergence behaviour of the CFD solver and, in some cases, may lead to numerical instabilities or other computational issues. With the help of ANSYS Meshing, the fluid domain was sliced in multiple blocks, and then hexahedral meshes were generated using the multizone meshing method, see in Figure 11 and Figure 12. For the single flexible tube configuration, several meshes with 0.47 million (Coarse), 3.95 million (Medium), and 9.7 million (Fine) elements were generated.

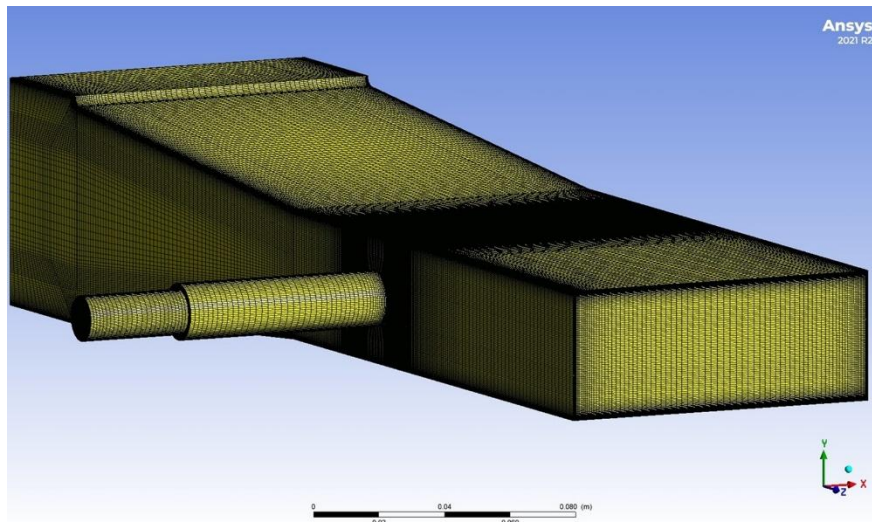


Figure 11: Isometric mesh view of single flexible tube case

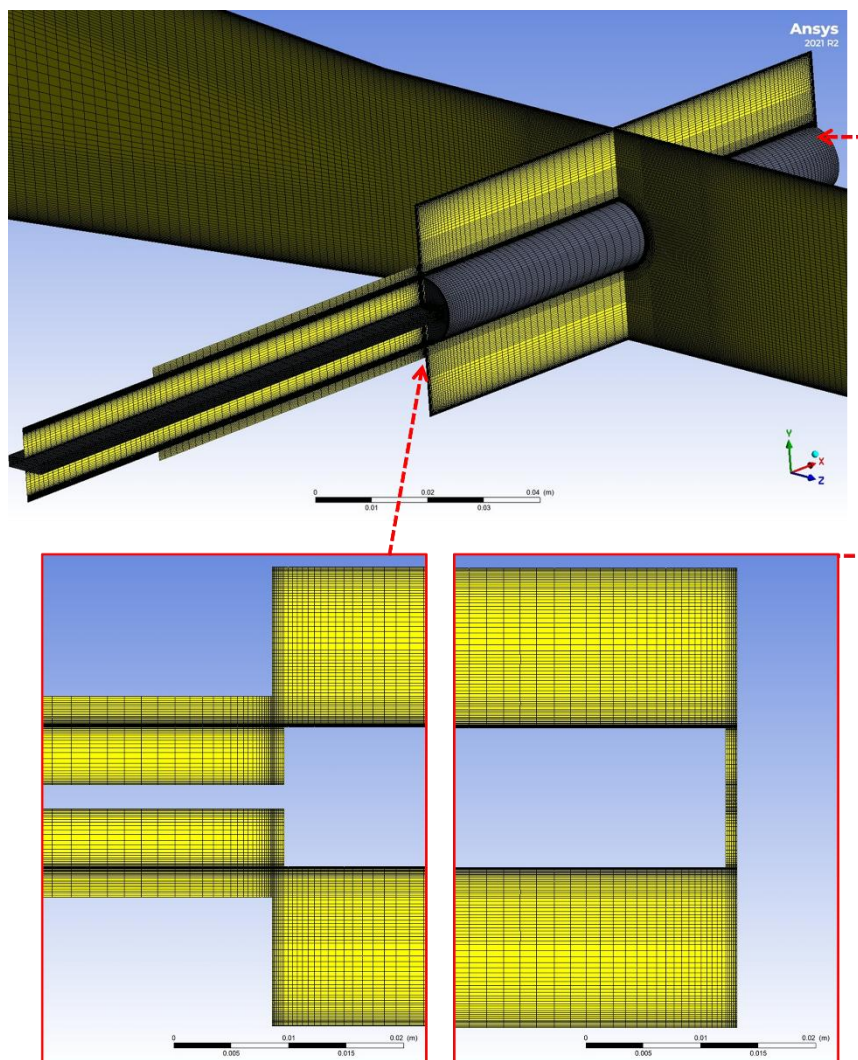


Figure 12: Mesh sectional view of single flexible tube including the fluid gap in the AMOVI domain

Further, a mesh study with the developed meshes was performed. The displacement measurement location (see Figure 2) was defined at the bottom of the flexible tube, where the displacement reaches its maximum value. Figure 13 shows the displacement of the flexible tube in lateral direction. The results show that selection the Medium mesh will save computational time and provide very similar results to the ones with the Fine mesh. The overall mesh quality is good: min orthogonal angle of 38° , element quality > 0.32 and max. aspect ratio – 151.

Also for the tube bundles with moving tube, the ANSYS Meshing tool was used to generate the numerical grids. Since a mesh study was, unfortunately, very expensive for this case, only a single mesh was generated. Its quality is good, although for some elements the aspect ratio reaches 135, which is beyond the best practice suggestions [16]. The min. orthogonality angle was 13° and the expansion factor - 88. These are outside the limits suggested in [16] for less than 1% of the elements. Furthermore, the CFD code converged within just a few iterations proving the overall high mesh quality.

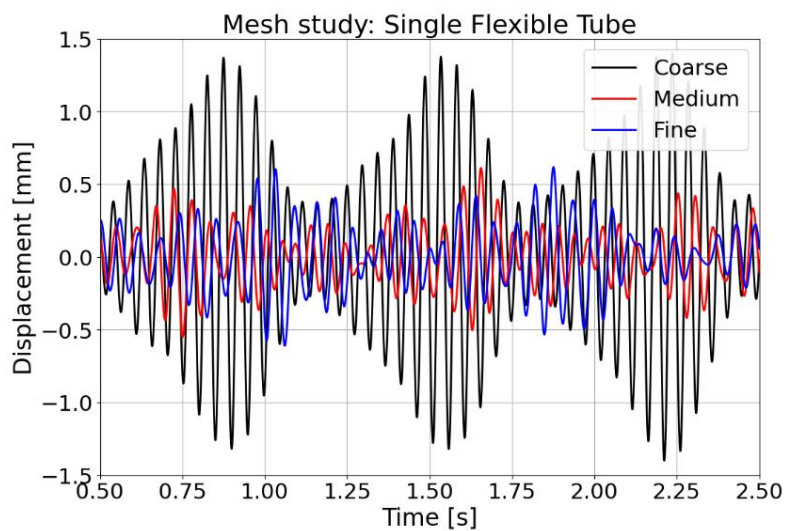


Figure 13: Mesh independence study for single flexible tube case

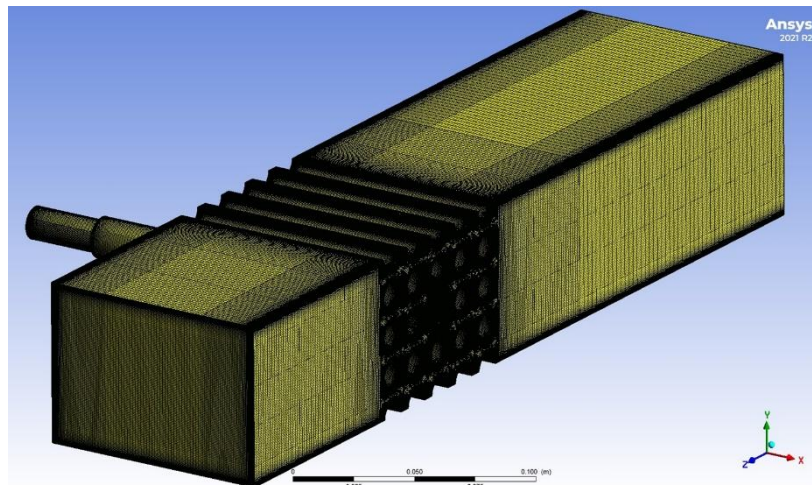


Figure 14: Isometric mesh view of flexible tube in rigid tube bundle.

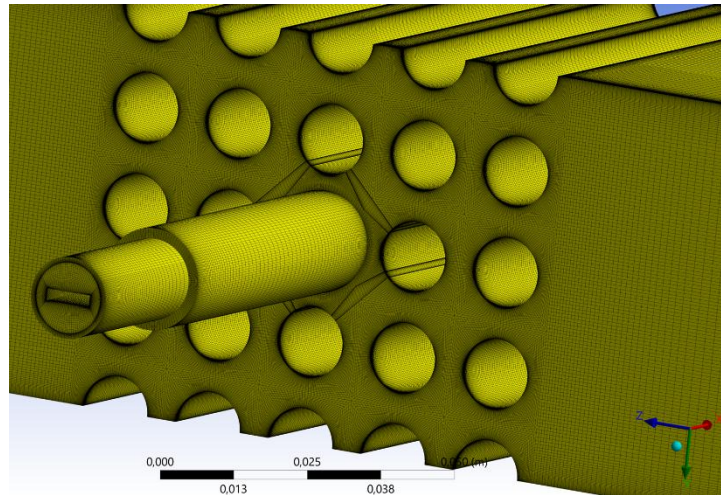


Figure 15: Isometric mesh view of the outer cylinder of flexible tube in the rigid tube bundle.

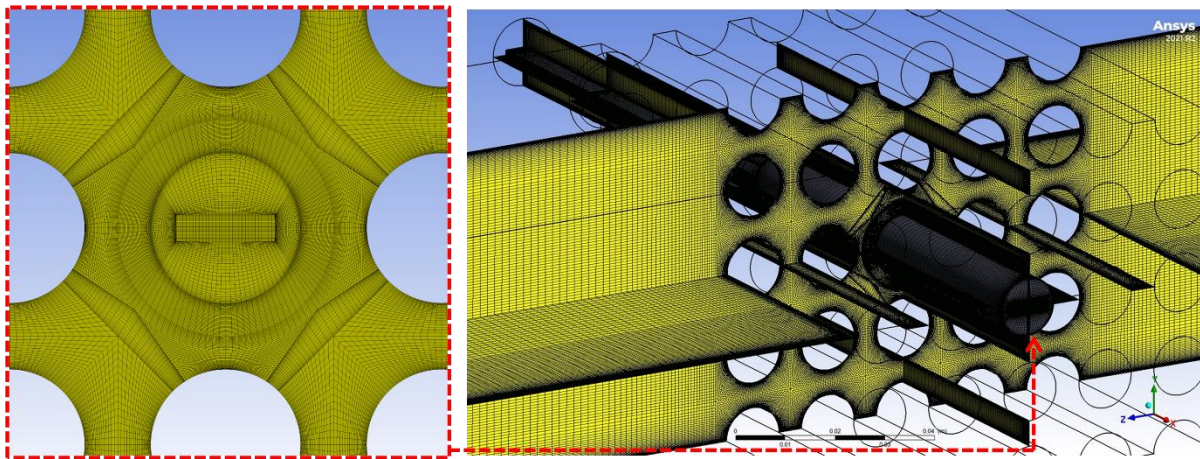


Figure 16: Sectional mesh view of the flexible tube.

3.2 Generation of an inlet velocity profile

Appropriate inlet boundary conditions are a prerequisite for accurate simulation results, especially in cases with flow-induced vibrations. Thus, it was decided to generate a small geometry representative of the upstream section of the AMOVI test section, with dimensions of $0.1 \times 0.07 \times 0.01$ m, and use it to generate a fully developed flow profile. Using ANSYS ICEM CFD, a fine computational hexahedral mesh was generated for this small domain, see in Figure 17. In CFX-Pre, a translation periodic boundary condition is applied at its inlet and outlet. Temperature, pressure and flow rate were defined as given in Table 1. The turbulence intensity was set to 5% using the two equation turbulence model $k-\omega$ SST [17], and the material properties were determined according to the IAPWS-IF97 standard. Further, the steady state simulations for all listed states were executed.

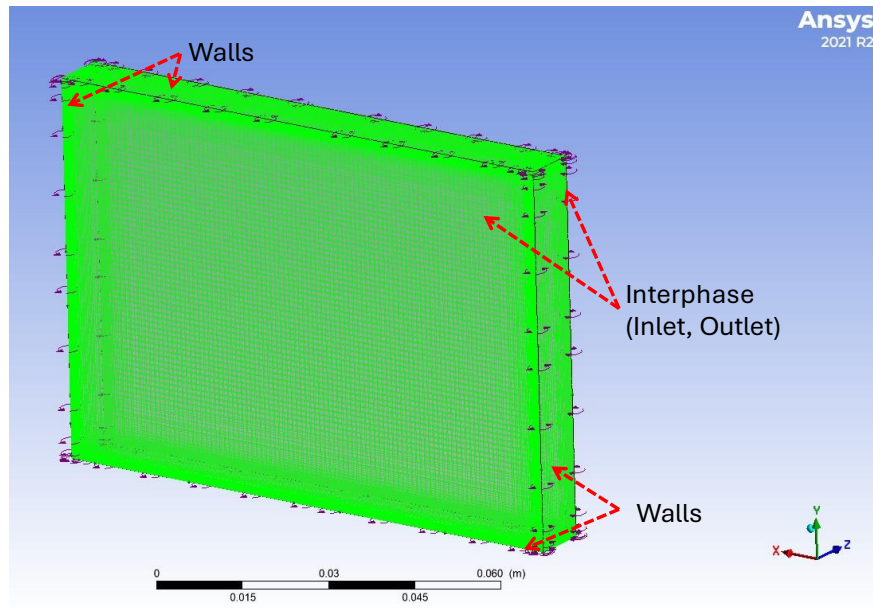


Figure 17: Computation domain of small sector for development of inlet velocity profile

The three velocity components, the turbulent kinetic energy, the turbulent eddy diffusivity, and the coordinates were exported in a .csv file, which is then imported in CFX-Pre. The velocity magnitude profile of the single flexible tube configuration is shown in Figure 18. As expected, the maximum velocity was found in the middle of the channel.

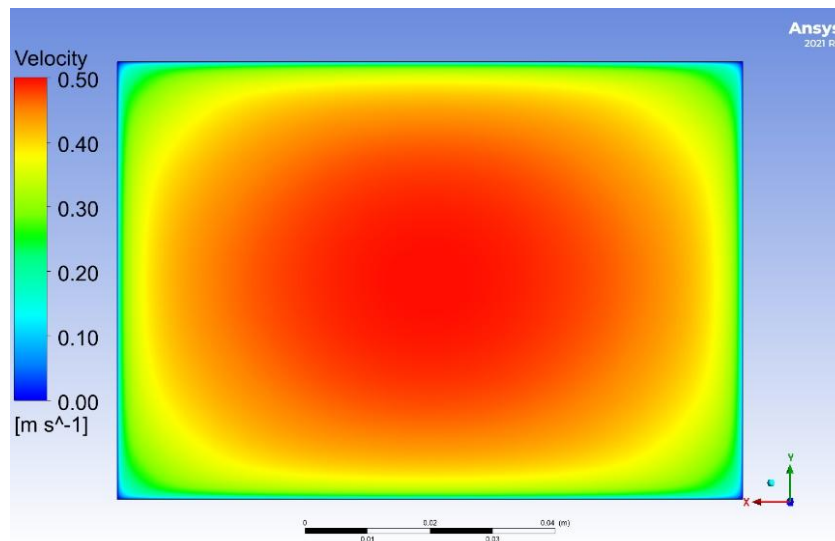


Figure 18: Fully developed velocity profile at the inlet for the AMOVI configuration with a single flexible tube.

3.3 Analysis of the Forces Acting on the AMOVI Tube

3.3.1 Simulation setup and boundary condition

As mentioned above, in all CFD simulations IAPWS IF97 water material data were used. The CFD simulations were isothermal, because all AMOVI experiments were performed at constant fluid temperature. Table 2 summarizes the most important fluid properties. For the present simulation cases, the generated fully developed velocity profiles were specified at the inlet of AMOVI test

section. The outlet boundary was modelled as an opening with a subsonic flow regime. At outlet, the flow direction was imposed normal to the outlet boundary, and the constant 0 Pa relative pressure was specified. The wall was assumed hydraulically smooth. In the CFD domain, the gravity force was applied in the negative Z direction. Near-wall treatment of turbulence was handled with automatic wall functions. In this study, different turbulence models (two equation ($k-\omega$ -SST), Reynolds Stress Model (RSM), SAS (Scale-Adaptive Simulation) and ZLES (Zonal Large Eddy Simulation) [18]) were investigated.

The Zonal LES (ZLES) turbulence model is a hybrid turbulence model that combines turbulence approaches based on LES and URANS. The advantage of the hybrid model over the pure LES is the reduced computational effort. The idea behind ZLES is to predefine zones with different turbulence treatments at the pre-processing stage. Between the different regions the turbulence model is switched from URANS (SST in the present analysis) to LES/WMLES (wall-modelled LES). ZLES is actually not a new model, but an infrastructure that combines existing elements of technology in a zonal fashion [18]. The recommendations for each zone are the same as those applicable to the individual models. The AMOVI domain was split into a URANS and LES zones before the simulation. In these particular cases, the bundle/the tube is simulated with LES, while the rest of the computational domain is treated with SST, see Figure 19.

For the numerical transport of the quantities (velocity, temperature, etc.) through the solution domain a High-Resolution advection scheme was used. Attention was paid also to the time discretization for which a Second Order Backward Euler scheme was chosen. Sensitivity studies on the time step size were performed. Previous analyses revealed strong dependence of the simulation results on the time step size. It was found that it has a significantly larger effect on the force calculation. Large time steps ($1\text{E-}3$ s) led to underpredict the local turbulence force on central tube. The studies showed that reducing the time step size below $2\text{E-}4$ s does not have noticeable impact on the numerical results. The time step size studies were performed for the final chosen mesh for the computations. Reducing to $1\text{E-}4$ s only negligibly increased the displacements. In the presented simulations the time step was reduced to $2\text{E-}4$ s in order to keep the Courant-Friedrichs-Lewy (CFL) number low. The mean CFL number observed in the RANS calculation was less than 0.5. The convergence of the simulation was assured in every time step by convergence criteria of $1\text{E-}5$.

Table 2: Fluid properties of force calculation cases

	Flow rate (L/s)	Fluid density (kg/m ³)	Fluid viscosity (Pa.s)	Thermal conductivity (W/m.K)	Pressure (bar)	Temperature (°C)
Single rigid tube	2.9	996.87	8.7014E-4	0.6089	2.88	26
Rigid tube bundle	1.7	998.59	1.052E-3	0.5948	0.90	18
	2.9	991.93	6.4092E-4	0.6321	3.19	41

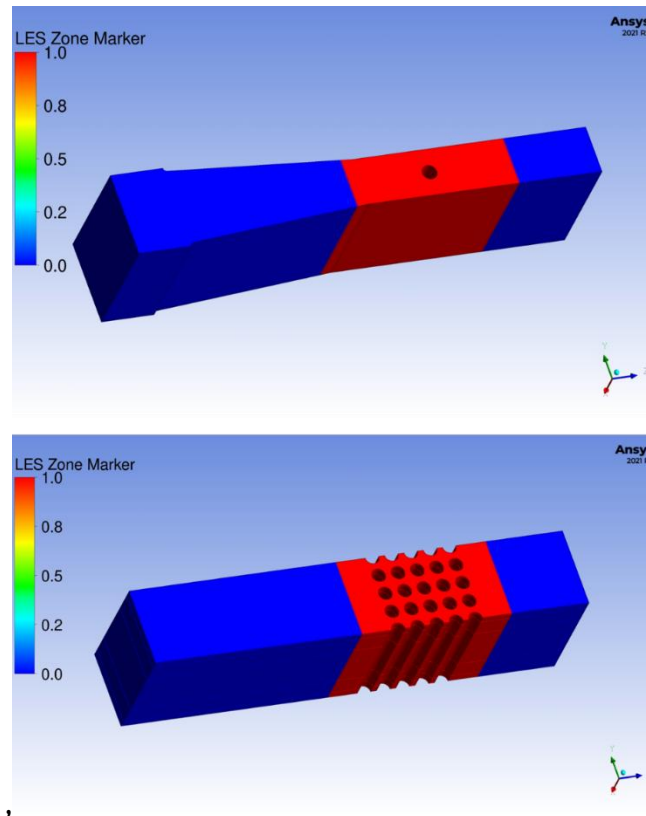


Figure 19: LES (red) and URANS (blue) Zones: Single rigid tube (top) and rigid tube in rigid tube bundle (bottom)

3.3.2 Single rigid tube

At the beginning, a steady state RANS calculation with a single rigid tube case was performed with the SST turbulence model [17]. The results were then used to initialize a URANS/Hybrid-URANS simulation. All simulations were performed for 10 s of problem time. This was sufficient to have several flow-through times.

The APSD (Auto Power Spectral Density) was calculated from forces (lift and drag forces) using Python scripts (Figure 20 and Figure 21). The numerical function matplotlib.mlab with Welch's average periodogram was used. The sampling frequency (number of samples per second) was 5000 Hz, while in the experimental data 1000 Hz was used. In the FFT analysis, a Hanning windowing was applied and PSD extracted up to 200 Hz. In Figure 20, a single prominent peak is present in the experimental curve corresponding to the vortex shedding frequency (15.2 Hz), which is associated with the periodic formation and detachment of vortices from the tube surface. All models capture the dominant vortex shedding frequency with reasonable accuracy (below 1 Hz deviation from experiment). The sensitivity of turbulence models is low, which represents good quality of mesh resolution for the accuracy of the simulations. At higher frequencies, all models show a decay in spectral energy, indicating reduced lift force fluctuations. SST underpredicts lift force levels before and after the main peak, while Omega-RSM model shows better prediction of lift force compared to SST. However, the Hybrid-URANS (SAS and ZLES models) showed the closest agreement with the experimental results in both peak amplitude and frequency location.

In Figure 20, a second prominent frequency peak is observed in all simulations that corresponds to the third harmonic of the vortex shedding frequency (~47 Hz), indicating complex interactions between the flow and the tube surface. This suggests that higher harmonics can contribute to

material stress and fatigue damage in the structure. In the experiment, however, this second prominent peak is not observed. Instead, a small peak at ~ 50 Hz appears in the experimental curve, originating from electrical current interference. This artifact should therefore be disregarded when comparing experimental results with numerical results.

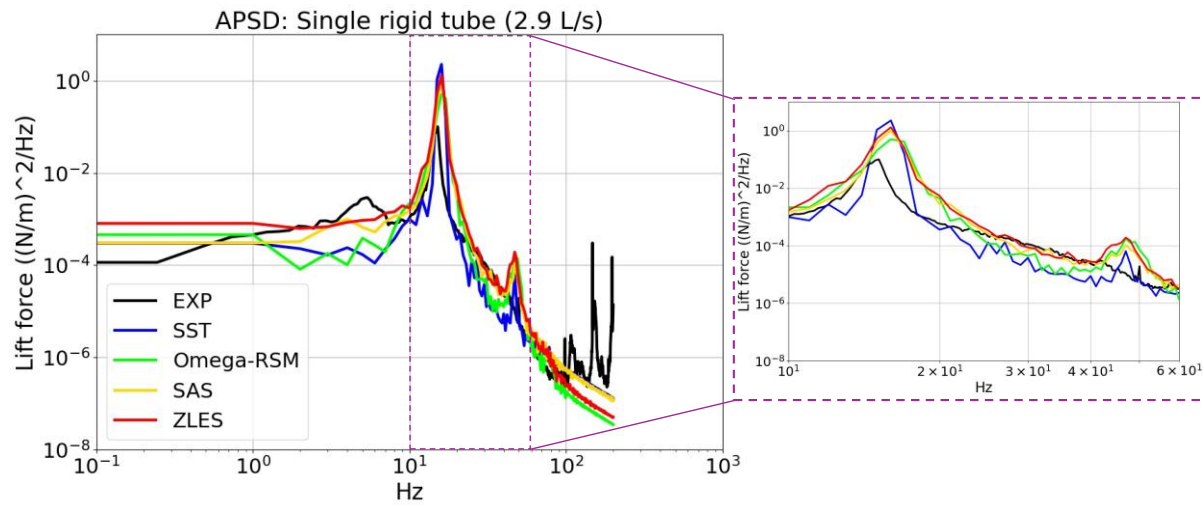


Figure 20: APSD of lift force for single rigid tube

Figure 21 presents the APSD of the drag force. In the experimental curve, a peak appears at ~ 50 Hz (in every interval) originating from the electrical current. In all simulations, a dominant frequency corresponding to twice the vortex shedding frequency ($2f$, ~ 32 Hz) is observed, indicating the periodic nature of the drag force induced by alternate vortex shedding from both sides of the tube. The numerical results reproduce this primary frequency peak with good agreement among RANS and Hybrid models. Such frequency peak (twice the peak of the lift force) in the drag force spectrum is expected, but not present in the experimental results, which is currently being discussed with CEA. In general, the trends of both simulation and experiment are similar, but with a shift of ~ 20 Hz. Similar to the lift force, Omega-RSM and SST models underpredict the drag force in comparison with Hybrid-URANS models.

Table 3 presents the RMS lift and drag forces per unit length obtained from experimental measurements and from numerical simulations using various turbulence models. As discussed above, Omega-RSM and SST models underestimate the measured value. However, the Hybrid models demonstrate improved performance due to their enhanced ability to capture large-scale unsteady structures and showed good agreement for both spectra with the experiment. Among them, particularly in this case, the SAS model showed even improved agreement with the experimental data, by capturing the lower and higher frequency profiles. The SAS model provides results that are agree best with the measurements, compared to the other models. Still, it overestimates by a factor of three the lift force and a factor of 1.3 the drag force.

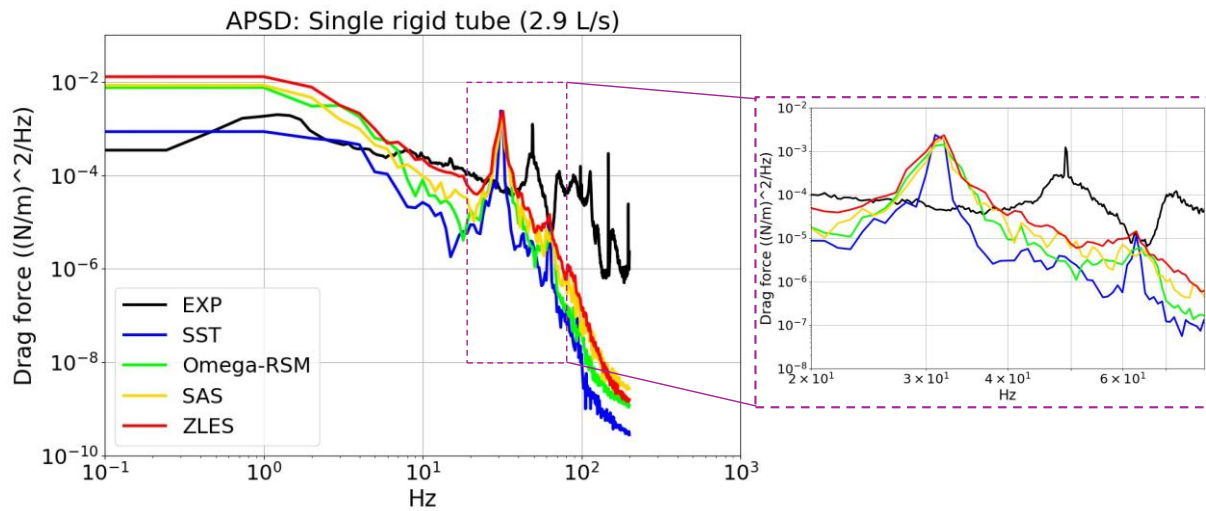


Figure 21: APSD of drag force for single rigid tube

Table 3: Comparison of calculated and measured RMS of forces

	Experiment	SST	Omega-RSM	SAS	ZLES
Lift force (N/m)	0.411	1.87	1.11	1.31	1.50
Drag force (N/m)	0.120	0.088	0.153	0.156	0.197

The mean free stream velocity in the reduced channel is ≈ 0.7 m/s. As expected, the flow behind the cylinder is decelerated, while the water is accelerated around the tube. This is well seen in Figure 22 (flow from left to the right), which also depicts the flow re-circulation zones behind the tube. The maximum velocity observed is 1.7 m/s. Behind the tube, a von Kármán vortex street developed. The vortices are generated at the back of the tube and periodically detach from either side of the tube.

Figure 23 and Figure 24 show resolved structures for the tube in cross flow based on Q-Criterion iso-surfaces (1) for $Re(D_h) \approx 1.0 \times 10^4$ flow, calculated using the SAS model (Hybrid) and the SST model (URANS), respectively. These iso-surfaces visualize turbulence structures coloured with instantaneous velocity at $t=10$ s. The 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.

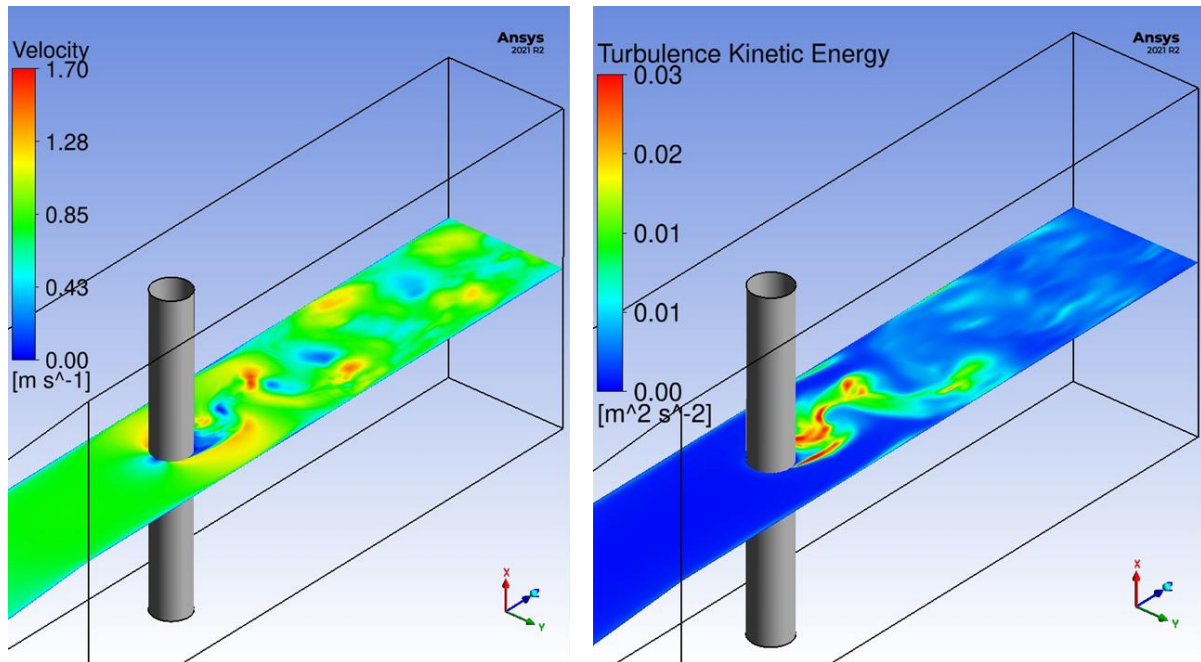


Figure 22: Velocity contour in a longitudinal plane showing flow acceleration and deceleration around the tube as well as re-circulation zones (left) and formation of a von Kármán vortex street behind the rigid tube (right), visualized with the turbulence kinetic energy

$$S^2 - \Omega^2 = 200 [s^{-2}] \quad (1)$$

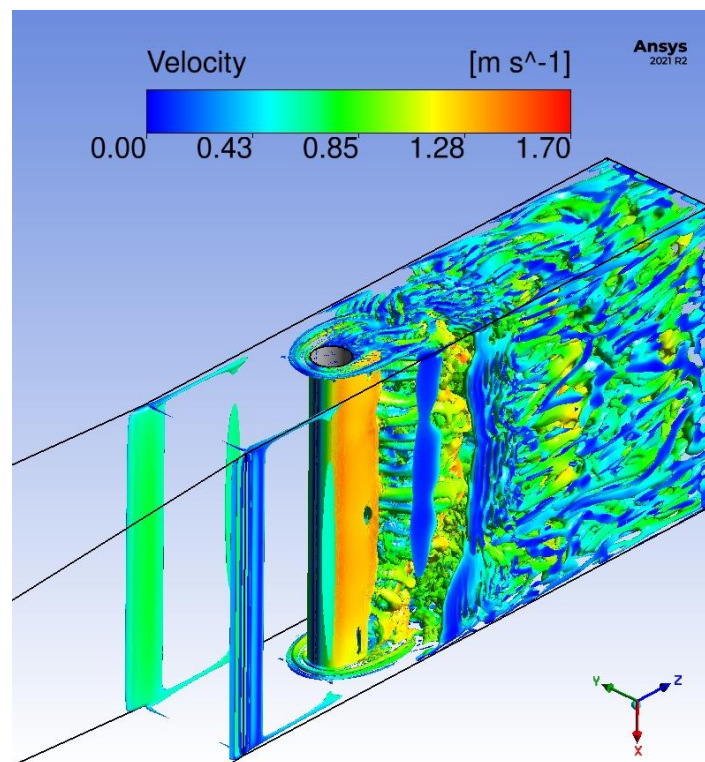


Figure 23: Resolved structures for single rigid tube in cross flow (SAS model)

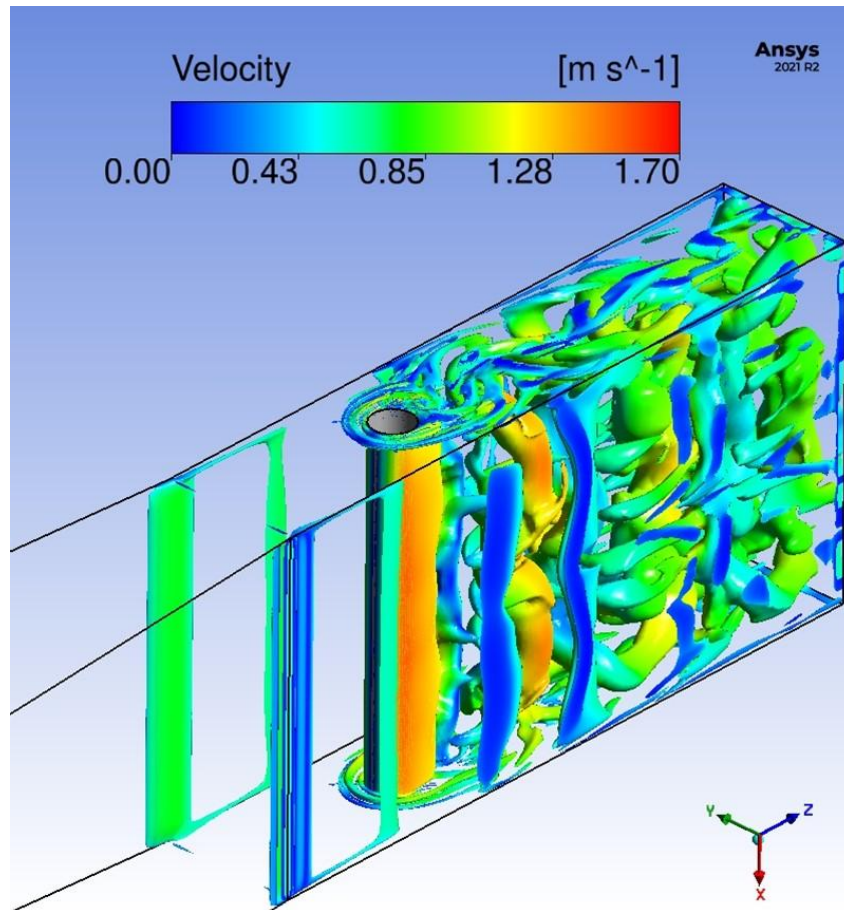


Figure 24: Resolved structures for single rigid tube in cross flow (SST model)

3.3.3 Rigid tube bundle

At the beginning, a steady state RANS calculation with a rigid tube bundle case was performed with the SST turbulence model. The results were then used to initialize a URANS/Hybrid simulation. In this study, two cases based on the following flow rates are investigated:

- 2.9 L/s case
- 1.7 L/s case

3.3.3.1 Analysis of force calculation for flow with 2.9 L/s

For the rigid tube bundle case with 2.9 L/s, different turbulence models (i.e. SST, Omega-RSM, SAS and ZLES) were investigated. The SST turbulence model was calculated up to $t=3$ s of problem time due to some technical reasons. However, simulation with the Omega-RSM model was performed for $t=5$ s of problem time, while Hybrid models (SAS and ZLES) were performed for $t=10$ s and $t=16$ s of problem time, respectively. The first 2 s of the problem time allowed the flow to statistically develop. This was sufficient to have several flow-through times, since the mean velocity in the test section with the tube is around ~ 0.54 m/s and its length is around 0.39 m.

Figure 25 shows the calculated instantaneous velocity with ANSYS ZLES at $t=10$ s in ZX (left) and YZ planes (right). The YZ plane is positioned at half cylinder height. The water flows from left to the

right with free stream velocity in the channel of 0.54 m/s. The more or less uniform velocity profile is destroyed after the impingement of the water on the first row of the tube bundle. On the front face of first row of the tube bundle, stagnation points are visible. The reduced hydraulic diameter of the flow channel leads to a significant increase in the velocity magnitude in the streamwise gap. Four higher-velocity jets with local maxima up to 1.9 m/s develop in the AMOVI bundle. These jets are unstable in time and oscillate in flow transverse direction. After passing the last bundle row, the jets enter the tube-free channel and then decay a few tube diameters downstream the last tube row. The high velocity jets downstream the last tube row are unstable in transverse direction. Recirculation zones develop behind the cylinders and in the corners formed by the half-cylinders and the side walls (Figure 26, calculated by ANSYS ZLES model). It is interesting to see that the flow circulates in clockwise direction behind the first row, while behind rows 2-4 it circulates in counter-clock direction. Figure 27 shows resolved structures for the rigid tube bundle in cross flow based on iso-surfaces of Q-Criterion 200 (1) for $Re(D_h) \approx 5.2 \times 10^4$ flow, calculated using the ZLES model.

In order to compare measured and calculated velocities with the same color map, the experimental data and ANSYS CFX results (ZLES) were imported in the ParaView software (see Figure 28). Jet formations are in qualitative agreement between tube gaps with the experiment. In the simulation, the four high-velocity jets coalesce some 3 or 4 tube diameters behind the bundle, while in the experiment these tend to attach to the side walls of the test section and broader wake regions in the center at end of the tube bundle. Possible reason for this deviation might be the too short computational time. The numerical results were averaged over 8 s, while the experiment over 400 s. Still, a very good agreement between PIV and simulation was observed for the $Q=1.6$ l/s case [19].

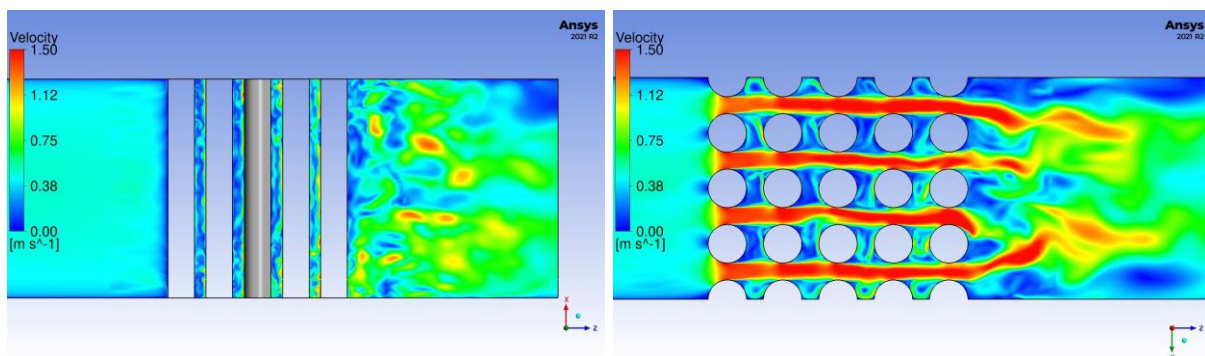


Figure 25: Calculated instantaneous velocity for 2.9 L/s at $t = 10$ s on middle ZX (left) and YZ Planes (right).

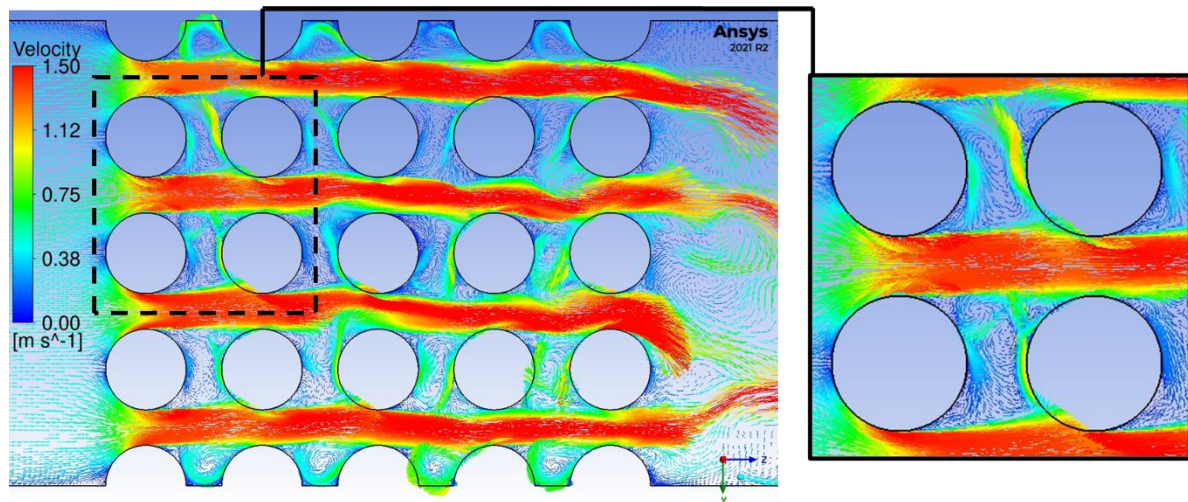


Figure 26: Recirculation zones visualized with velocity vectors on middle ZX plane for 2.9 L/s case.

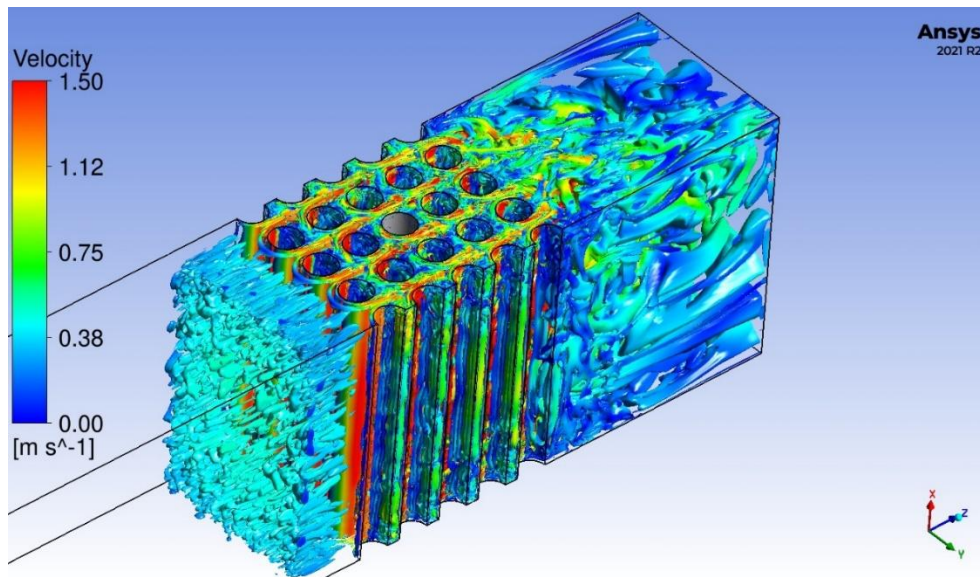


Figure 27: Resolved structures for rigid tube bundle in cross flow (ZLES)

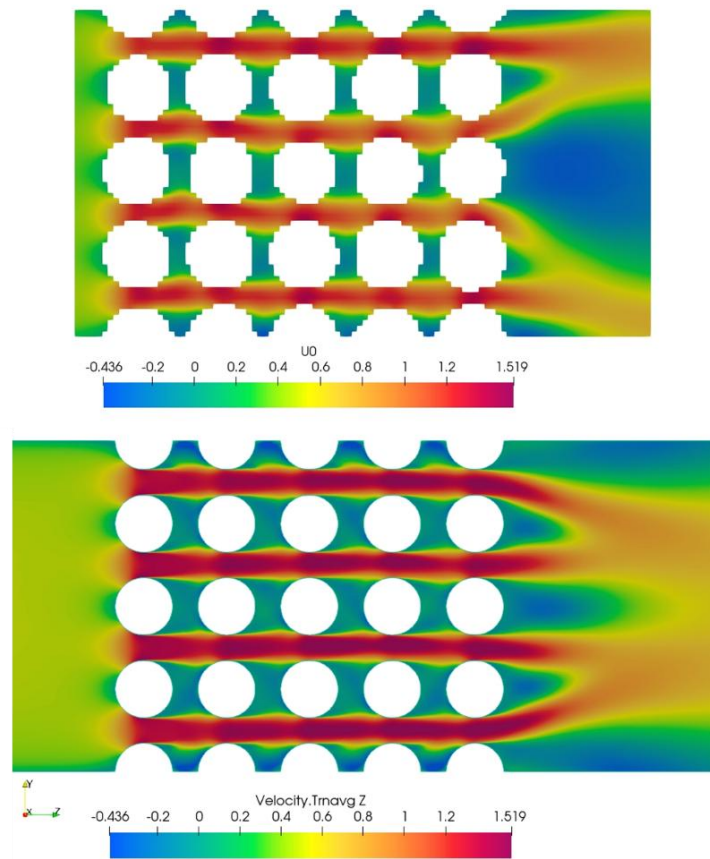


Figure 28: Measured (top) and calculated (bottom) time-averaged streamwise velocity in the tube bundle for 2.9 L/s case.

The APSD was calculated from forces (lift force and drag force) using Python scripts (Figure 29 and Figure 30). Here, the forces measured on the central tube within the rigid tube bundle subjected to a cross-flow rate of 2.9 L/s. The numerical function `matplotlib.mlab` with Welch's average periodogram was used. The sampling frequency (number of samples per second) was 2000 Hz (SST), 5000 Hz (Omega-RSM and SAS) and 6000 Hz (ZLES), while in the experimental data 1000 Hz was used. In the FFT analysis, a Hanning windowing was applied and PSD extracted up to 200 Hz. The frequency spectra illustrate the distribution of fluctuating lift and drag force energy across different frequencies respectively.

In Figure 29 and Figure 30, the APSD illustrates how the lift and drag force energy is distributed over a wide range of frequencies, reflecting both steady and unsteady flow contributions. The experimental spectrum indicates a relatively broad distribution of energy without a sharp dominant peak, suggesting that the lift and drag force fluctuations are mainly associated with turbulent wake interactions and vortex shedding suppression within the tube array. Among the CFD models, the ZLES approach most accurately reproduces the experimental spectral trends, showing strong agreement in both the shape and magnitude of the spectra for both forces. ZLES predictions align particularly well in the mid- to high-frequency range (10–100 Hz), effectively capturing the gradual decay of lift and drag force energy and reflecting their superior ability to resolve unsteady flow dynamics within the tube bundle. Similarly, SAS model comes very close to the ZLES model trend. However, SAS slightly underpredicts the spectrum in the interesting frequency range (10-100 Hz). The Omega-RSM model also follows the general trend, but moderately underestimates the energy levels in the low- to mid-frequency range for both forces. In contrast, the SST model significantly underestimates the spectral energy beyond the primary frequency range, indicating its limited

ability to resolve unsteady flow structures and inter-tube vortex interactions. Overall, the results demonstrate that Hybrid-URANS turbulence models have ability to provide a more realistic prediction of forces. Among them, ZLES showed best agreement by reproducing both the shape and magnitude of the experiment for both forces.

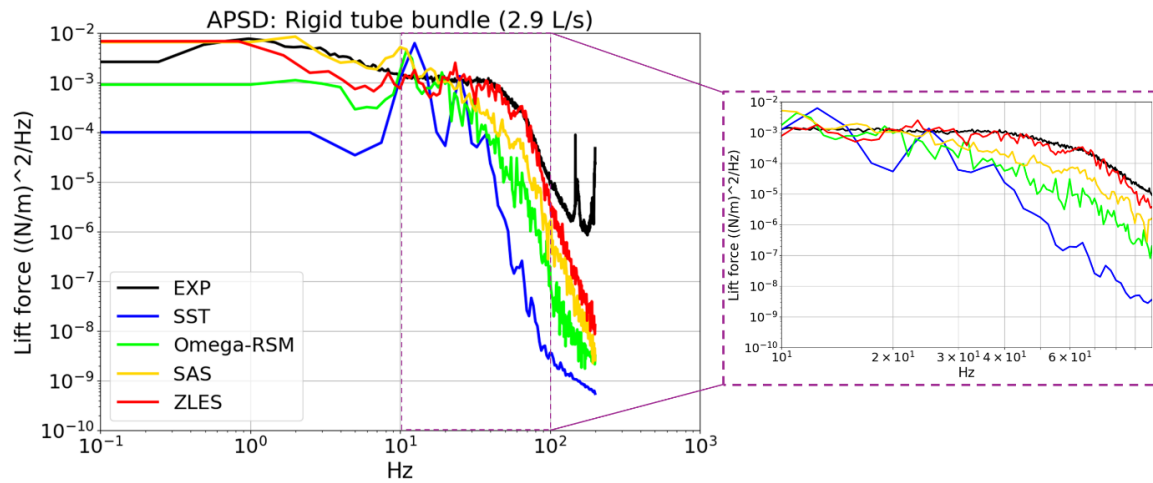


Figure 29: APSD of lift force for rigid tube bundle for 2.9 L/s

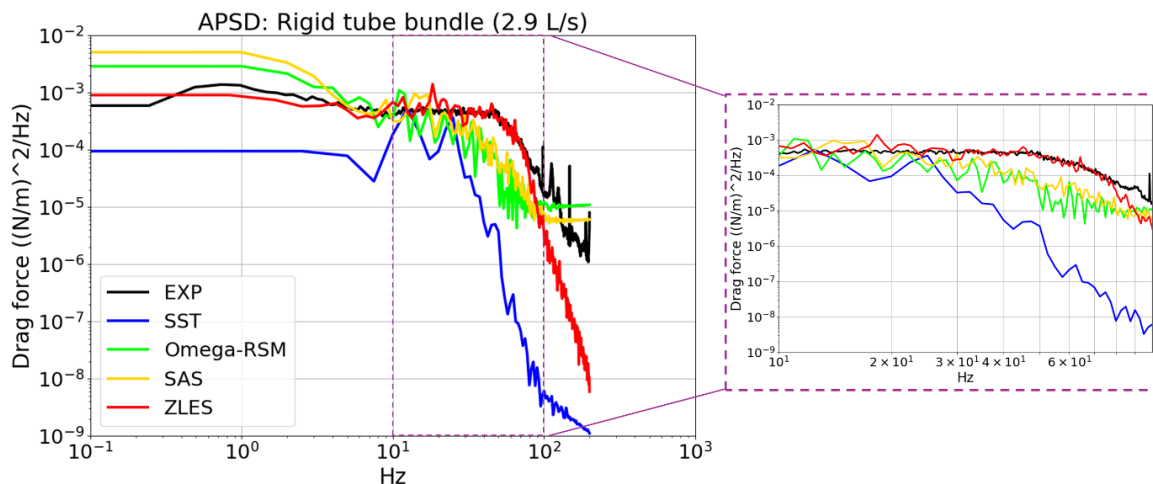


Figure 30: APSD of drag force for rigid tube bundle for 2.9 L/s

Table 4 summarizes the RMS lift and drag forces per unit length obtained from experimental measurements and from numerical simulations using various turbulence models. The ZLES model demonstrates the best agreement in terms of RMS lift force (0.251 N/m vs 0.291 N/m) and drag force (0.178 N/m vs 0.179 N/m).

Table 4: Comparison of calculated and measured RMS of forces for rigid tube bundle 2.9 L/s case

	Experiment	SST	Omega-RSM	SAS	ZLES
Lift force (N/m)	0.291	0.173	0.170	0.266	0.251
Drag force (N/m)	0.179	0.0728	0.146	0.167	0.178

3.3.3.2 Analysis of force calculation for flow with 1.7 L/s

For rigid tube bundle case with 1.7 L/s, only the ZLES model was investigated over 10 s of problem time. Figure 31 shows the calculated instantaneous velocity fields with ANSYS ZLES at $t = 10$ s in ZX (left) and YZ planes (right). The YZ plane is positioned at half cylinder height. The water enters the domain with 0.2428 m/s. The more or less uniform velocity profile is destroyed after the impingement of the water on the first row of the tube bundle. The reduced hydraulic diameter of the flow channel leads to a significant increase in the velocity magnitude in the streamwise gap. Four higher-velocity jets with local maxima up to 1.14 m/s develop in the AMOVI bundle. The jets and flow behaviour in tube bundle and further downstream after tube row is similar as described in 3.3.3.1 (for case 2.9 L/s) with some changes in local scalar magnitudes.

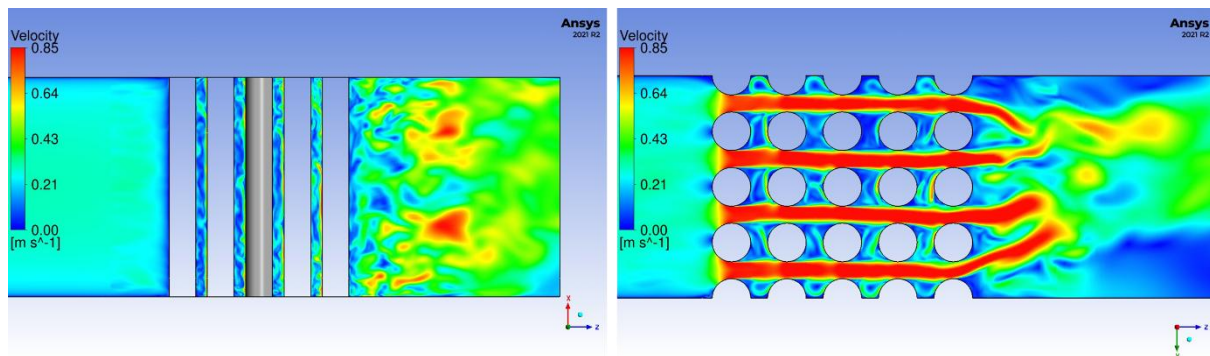


Figure 31: Calculated instantaneous velocity for 1.7 L/s at $t = 10$ s on middle ZX (left) and YZ planes (right).

In order to compare measured and calculated velocities with the same color map, the experimental data and ZLES results were imported in the ParaView software (see Figure 32). The qualitative agreement between experiment and simulation is good, although the averaging was performed over 8 s of simulation time and over 410 s of experimental data. The similarity is visible: the upper two jets merge shortly downstream the tube bundle, as this is the case for the lower two jets. Still, ANSYS CFX slightly overpredicts the maximal jet velocity and suggests stronger recirculation zones behind the last tube bundle row compared to the experiment.

The APSD was calculated from forces (lift and drag forces) using Python scripts (Figure 33 and Figure 34). Here, the forces were measured on the central tube within the rigid tube bundle subjected to a cross-flow rate of 1.7 L/s. The numerical function matplotlib.mlab with Welch's average periodogram was used. The sampling frequency was 8000 Hz, while in the experimental data 1000 Hz was used. In the FFT analysis, a Hanning windowing was applied and the PSD was extracted up to 200 Hz. The experimental spectrum indicates a relatively broad distribution of energy without a sharp dominant peak, suggesting that the lift and drag force fluctuations are mainly associated with turbulent wake interactions and vortex shedding suppression within the tube array. ZLES approach accurately reproduces the experimental trend, showing good agreement for both, the shape and magnitude of the spectra, in the important mid-frequency range (10–50 Hz), effectively capturing the gradual decay of lift and drag force energy and reflecting its superior ability to resolve unsteady flow dynamics within the tube bundle. The ZLES model demonstrated good agreement of predicted RMS lift force (0.06634 N/m) with experiment (0.066 N/m), as well as for the RMS drag force (0.045 N/m vs 0.048 N/m).

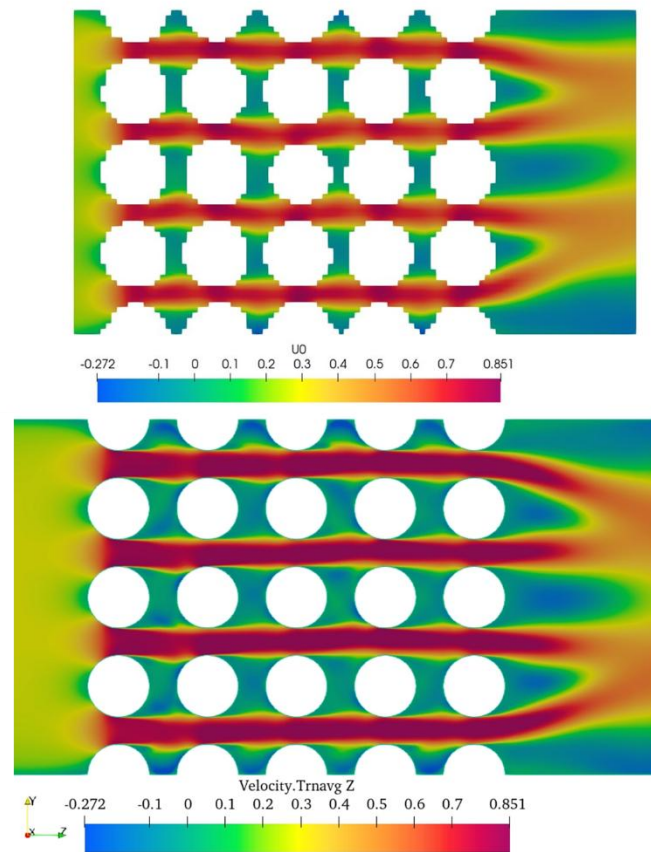


Figure 32: Measured (top) and calculated (bottom) time-averaged streamwise velocity in the tube bundle for 1.7 L/s case.

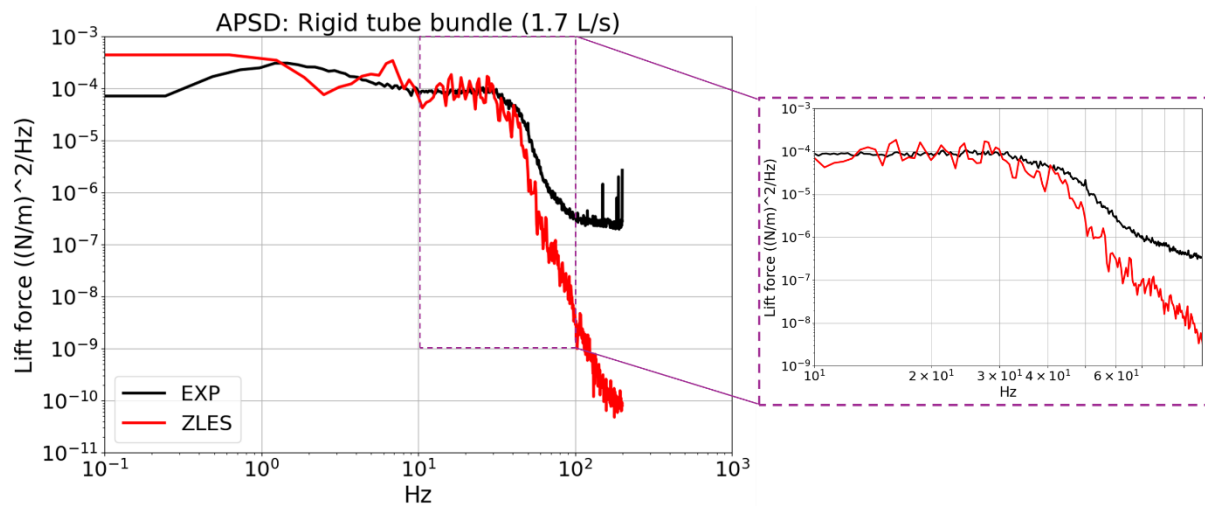


Figure 33: APSD of lift force for rigid tube bundle for 1.7 L/s

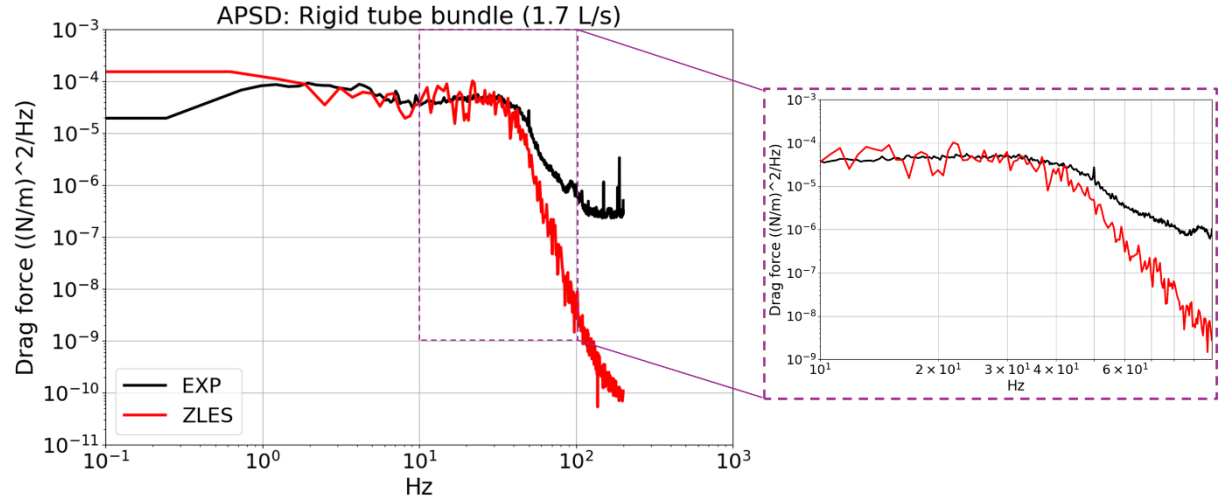


Figure 34: APSD of drag force for rigid tube bundle for 1.7 L/s

3.4 The Rigid Body Motion Approach

According to WP3 description, the objective is to simulate the flow induced vibration with two-way coupling of fluid and solid solvers, which is possible with ANSYS Workbench FSI. To reduce the effort of solving the full mechanical model, the response of the mechanical body motion could be solved using a rigid body solution method instead. The objective here is to prove the adequacy of this simplified method that accounts for the structural deformation in an FSI run.

A rigid body is a solid object that moves through a fluid without deforming. Its motion is dictated by the fluid forces and torques acting upon it, plus any external forces (such as gravity) and external torques. In this work, the rigid body (vibrating tube) is defined by a collection of 2D regions that form its faces. Hence, the AMOVI tube itself does not need to be meshed. Mesh motion is used to move the mesh on the rigid body faces in accordance with the solution of the rigid body equations of motion [20]. The rigid body solution algorithm uses a linear momentum solver for the translation (P) of the rigid body and a separate angular momentum solver for the rotation (Π) of the rigid body, which is equal to the applied force (F) and torque/-momentum (M) respectively, as the following equation states:

$$\frac{d(P|\Pi)}{dt} = (F|M) \quad (2)$$

For the selection of a suitable algorithm, initial analyses were performed and based on that the angular momentum algorithm was selected [20].

3.4.1 Angular Momentum Algorithm

The equation of motion for a rotating rigid body can be written in body-fixed coordinates (about the center of mass) as [20]:

$$d \frac{(\bar{I}\omega_B)}{dt} = \omega_B \times \bar{I}\omega_B + \bar{I}\dot{\omega}_B \quad (3)$$

Where $\bar{I}$ is moment of inertia tensor, which defines how the body's mass is distributed relative to its rotation axes. ω_B and $\dot{\omega}_B$ are angular velocity vector and angular acceleration with body coordinate system, respectively. The total moment acting on the body is calculated as follows [20]:

$$M_B = M_{Aero} - \sum [k_{Spring}(\theta - \theta_{so})] + \sum M_{Ext} \quad (4)$$

where M_{Aero} is the aerodynamic torque, k_{Spring} is the rotational spring constant and M_{Ext} is all other external torques acting on the body. Using equation (3) and (4), the angular momentum equation can be solved by choice of algorithm accuracy, as follows [20]:

- Simo Wong algorithm, which is second order accurate (a modified Newmark integration scheme)
- Backward Euler, which is first order accurate

To solve the angular momentum equation in the solver, external forces and torques need to be defined as well as the degrees of freedom. One of the most important parameters in the rigid body model is the spring constant in x, y, z directions that brings the vibrating structure back to its original equilibrium position.

3.4.2 Derivation of Numerical Parameter for the Rigid Body Motion Approach

The Rigid Body Motion approach allows the modelling of the rotation of the AMOVI flexible tube about an axis due to an external torque. In AMOVI, the rotational motion is constrained to occur about the z-axis only, which represents one degree of freedom. Based on the boundary conditions and geometrical configurations, the angular momentum solver calculates the torque, resulting from the fluid forces acting on the rigid body. Thereafter, the solver calculates the mesh motion. To do this, three important parameters have to be input by the user: a mass, a mass moment of inertia (kg.m^2) and a torsional spring constant (N.m/radian).

3.4.2.1 Numerical modelling and structural analysis of the flexible tube

The vibrating tube was produced of stainless steel (density of 8300 kg/m^3 , 200 GPa elasticity). First, a CAD model from the CEA's drawings was generated for the flexible AMOVI tube and the attached blade. In total, 1.04 million of elements were used in ANSYS Mechanical to generate an unstructured mesh for the 0.2 m long tube and its blade (see Figure 35). Further refinement had no noticeable effect on the results. The mass of the flexible tube is 0.045 kg. The mass moment of inertia tensor describes the body resistance to changes in its rotation rate. It is a symmetric tensor ($I_{xy} = I_{yx}$) with six components that have to be input by the user. In AMOVI, the rigid body rotation occurs only about the z-axis, hence the tensor is simplified to I_{zz} and its other components are zero. ANSYS derives the mass moment of inertia at the center of mass ($I_{zz} = 0.00013342 \text{ kg.m}^2$), which is located at x, y and z (0.017646 m, 0.035 m, 0.255 m). However, the final mass moment of inertia ($I_{zz} = 7.6\text{e-}4 \text{ kg.m}^2$) was derived based on the rigid body coordinate system using the parallel axis theorem.

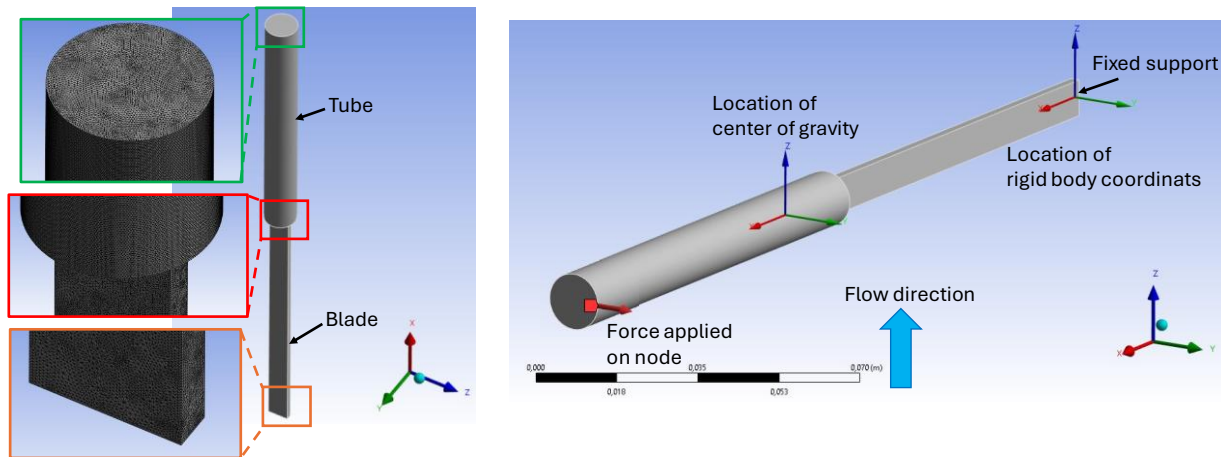


Figure 35: Flexible tube model: Numerical mesh of structural model (left); Boundary condition and coordinates system (right)

To bring the rigid body back to its original position after external load, the torsional spring constant has to be derived from structural analysis with Mechanical. The right end of the structure was fixed in x, y, z directions, while the left end of the tube was free to vibrate. A force (1 N, 2 N and 3 N, respectively) was applied on the single node in lift direction (transverse flow direction). The corresponding displacements (Figure 36) and the resulting moments of force on the fixed support (blade's right end in Figure 35) were calculated. Based on these results, the torsional stiffness of the whole rigid body is calculated to be 21.13 N.m/rad.

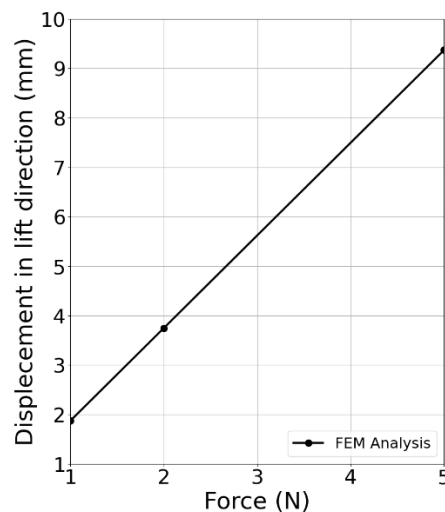


Figure 36: Measured displacement under application of external force

3.5 Analysis of AMOVI cases with mesh displacement

In this chapter, the results of the simulation for AMOVI vibrating experiment cases will be presented. The experimental data will be compared with simulation data.

3.5.1 Simulation setup and boundary conditions

For the vibrating tube cases, most of the boundary conditions are the same as described in 3.3.1. For the modelling of the rigid body approach, the flexible tube is selected as rigid body and the wall velocity is specified relative to mesh motion. Only one rotational degree of freedom on the Z-axis

is specified (Figure 37). The mass moment of inertia of ($I_{zz} = 7.6e-4 \text{ kg.m}^2$), the mass (0.045 kg), and the torsional spring constant (21.13 N.m/radian) are input parameters of the Rigid Body Motion approach. The end of the flexible tube is considered as surface of revolution about Z-axis. Several analyses were performed for a stable mesh motion boundary condition for the front wall. Based on these, the mesh motion was selected to be parallel to boundary. This improves the mesh quality parameters during the transient calculation. The rest of the walls were considered to be stationary and wall velocity relative to the boundary frame was selected. All walls were specified to be smooth. Table 5 summarizes some fluid properties.

In the CFD domain, the gravity force was applied in the negative Z direction. Along with that, the displacement diffusion model for mesh deformation was specified in CFD domain. The standard mesh stiffness option (Increase near small volumes with constant 2.0) is selected. Near-wall treatment of turbulence was handled with automatic wall functions. In this study, different turbulence models (two equation ($k-\omega$ -SST), Reynolds Stress Model (RSM), and ZLES (Zonal Large Eddy Simulation)) were investigated. The AMOVI domain was split into a URANS and LES zones for the ZLES simulation. In these particular cases, the bundle/the tube is simulated with LES, while the rest of the computational domain is treated with SST, see Figure 38.

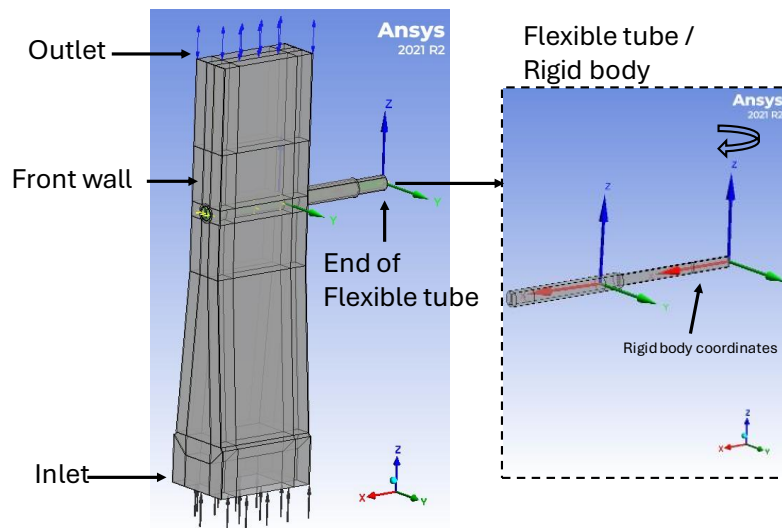


Figure 37: Modelling of rigid body for vibrating cases

In the present study, the numerical transport of the quantities (velocity, temperature, etc.) through the solution domain was done with a High-Resolution advection scheme. Attention was paid also to the time discretization, for which a Second Order Backward Euler scheme was chosen.

For the single flexible tube case, sensitivity studies on the time step size were performed. Previous analyses revealed strong dependence of the simulation results on the time step size. It was found that it has a significantly larger effect on the vibration calculation. Large time steps ($1E-3 \text{ s}$) led to underprediction of the local turbulence force and vibration amplitude. The studies showed that reducing the time step size below $2E-4 \text{ s}$ does not have noticeable impact on the numerical results. In the presented simulations, the time step was reduced to $2E-4 \text{ s}$ in order to keep the Courant-Friedrichs-Lewy (CFL) number low. The mean CFL number observed in the RANS calculation was less than 0.6. The convergence of the simulation was assured in every time step by convergence criteria of $1E-5$. Further, for this approach, the Simo Wong algorithm for the solution of the angular momentum equation was used. Further, time step size studies were also performed with the different turbulence models, as these have influence on the time step used and the results. The convergence of all simulations was assured in every time step with convergence criteria set to $1E-5$.

Table 5: Fluid properties of displacement calculation cases

	Flow rate (L/s)	Fluid density (kg/m ³)	Fluid viscosity (Pa.s)	Thermal conductivity (W/m.K)	Pressure (bar)	Temperature (°C)
Single flexible tube	2.9	997.14	8.9003E-4	0.6069	3.2	25
Flexible tube in a rigid tube bundle	1.6	998.77	1.079E-3	0.5948	1.07	17

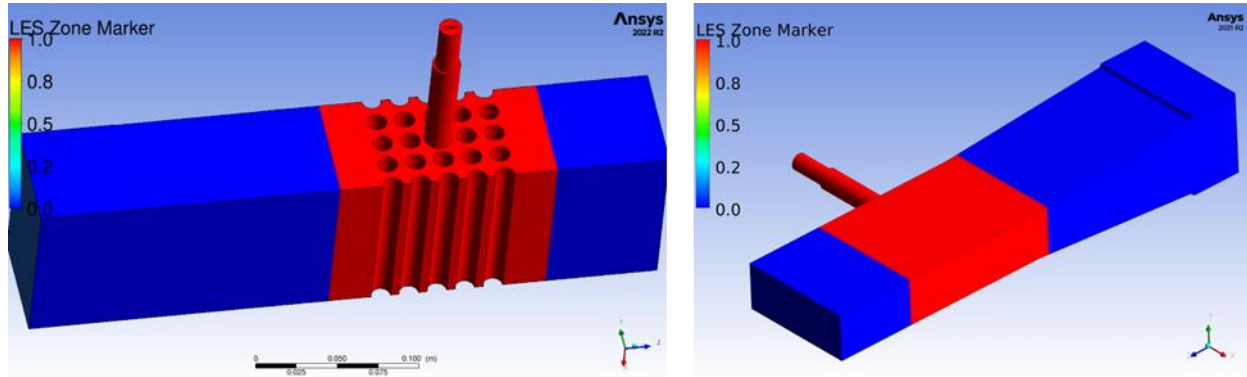


Figure 38: LES (red) and URANS (blue) zones: flexible tube in rigid tube bundle (left) and Single flexible tube (right)

3.5.2 Single flexible tube

At the beginning, a steady state RANS calculation was performed using the SST turbulence model. These results were then used to initialize the URANS/Hybrid simulations. In this study, experiments with a higher flow rate of 2.9 L/s and a lower flow rate of 1.7 L/s were investigated.

3.5.2.1 Analysis of FIV with 2.9 L/s flow rate

The single flexible tube case was investigated with SST, Omega-RSM, and ZLES turbulence models. These simulations are performed for more than ~6 sec of problem time.

Figure 39 shows the calculated instantaneous velocity with Omega-RSM at $t = 6.8$ s. The mean free stream velocity in the channel is 0.63 m/s (water flows bottom-top). As expected, the flow behind the cylinder is decelerated, while the water is accelerated around the tube. This is well seen in Figure 40 (top), which also depicts the flow re-circulation zones behind the tube. The maximum velocity of 1.3 m/s is observed in the 1 mm gap between the left tube end and the channel wall (see Figure 39). The flow in the right part of the blade channel, close to the blade fixation point is stagnant. Figure 39 reveals that there is mass exchange between both channels: some water leaves the main channel and then enters through a small gap into the blade channel. In the gap, the water is accelerated up to 1.3 m/s. At the same time, some water leaves the blade channel and mixes with the water in the main channel behind the cylinder. Figure 40 (bottom) shows the formation of a von Kármán vortex street behind the vibrating tube, visualized with the help of the turbulence kinetic energy. The vortices are generated at the back of the tube and periodically detach from either side of the tube, generating forces that move the structure.

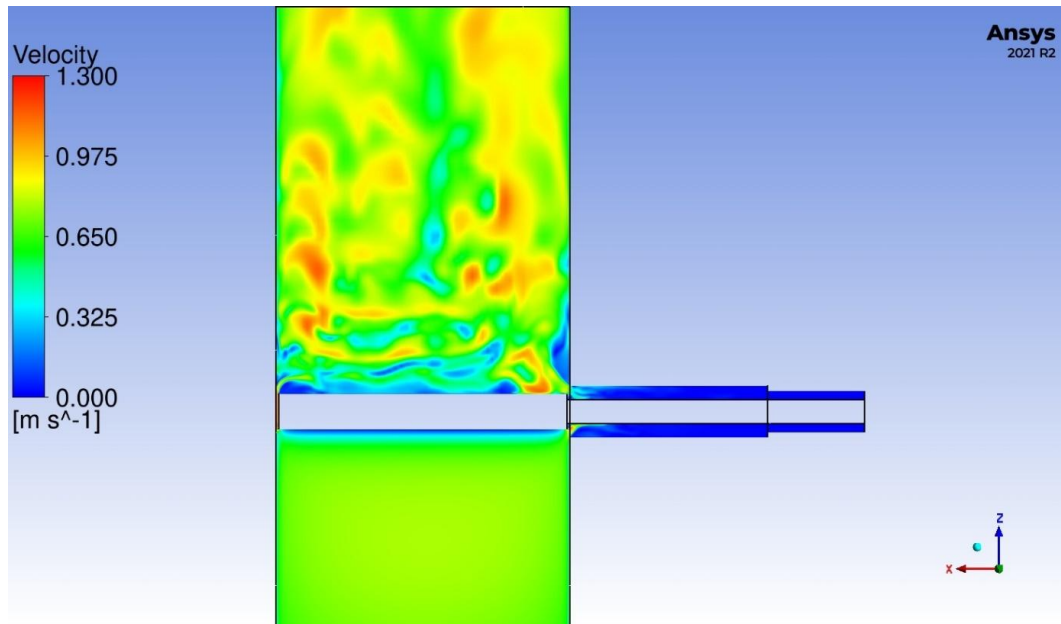


Figure 39: Velocity magnitude distribution in a longitudinal plane (water flows bottom-top) through the AMOVI test section.

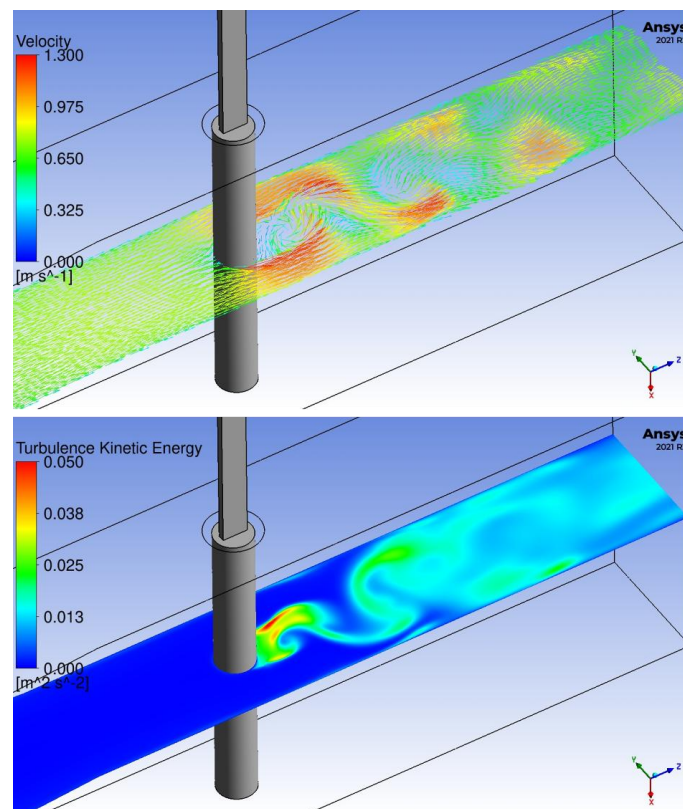


Figure 40: Velocity colored vectors in a longitudinal plane showing flow acceleration and deceleration around the tube as well as re-circulation zones (top) and formation of a von Kármán vortex street behind the vibrating tube (bottom), visualized with the turbulence kinetic energy.

The time-dependent vibration amplitudes are compared with data and the simulation results are further analysed with the help of a Fast Fourier Transform (FFT) analysis. The FFT is performed on

6 s of simulation time. The first $t=0.5$ s of simulation time was omitted in all cases. These data acquisition times should provide sufficient resolution for the FFT analysis. Experimental data of 30 s was also used for the comparison. The generated curves were plotted in three separate plots for better visibility.

Figure 41 presents the calculated and the experimental tube displacements. It can be seen that the vibration displacements in the Omega-RSM and ZLES simulations are very similar. SST shows underprediction of vibration amplitude compared to other turbulence models. The RMS value of the time-dependent displacement was calculated for the available simulation results and the experiment. SST predicts an RMS value of 0.085 mm, while Omega-RSM and ZLES give 0.179 mm and 0.155 mm. These models calculate 55%, 5.7% and 18.4% smaller displacements than the measured one (0.19 mm).

For the design of heat exchangers and other equipment in the energy sector the maximum vibration amplitude is of great importance. The maximum displacement of the tube in lift direction from its normal position is calculated to be 0.184 mm (SST), 0.61 mm (Omega-RSM), and 0.494 mm (ZLES), while the measured value is 0.6 mm. The displacement prediction is considered acceptable as long as the calculated value is within the same order of magnitude of the experimental data.

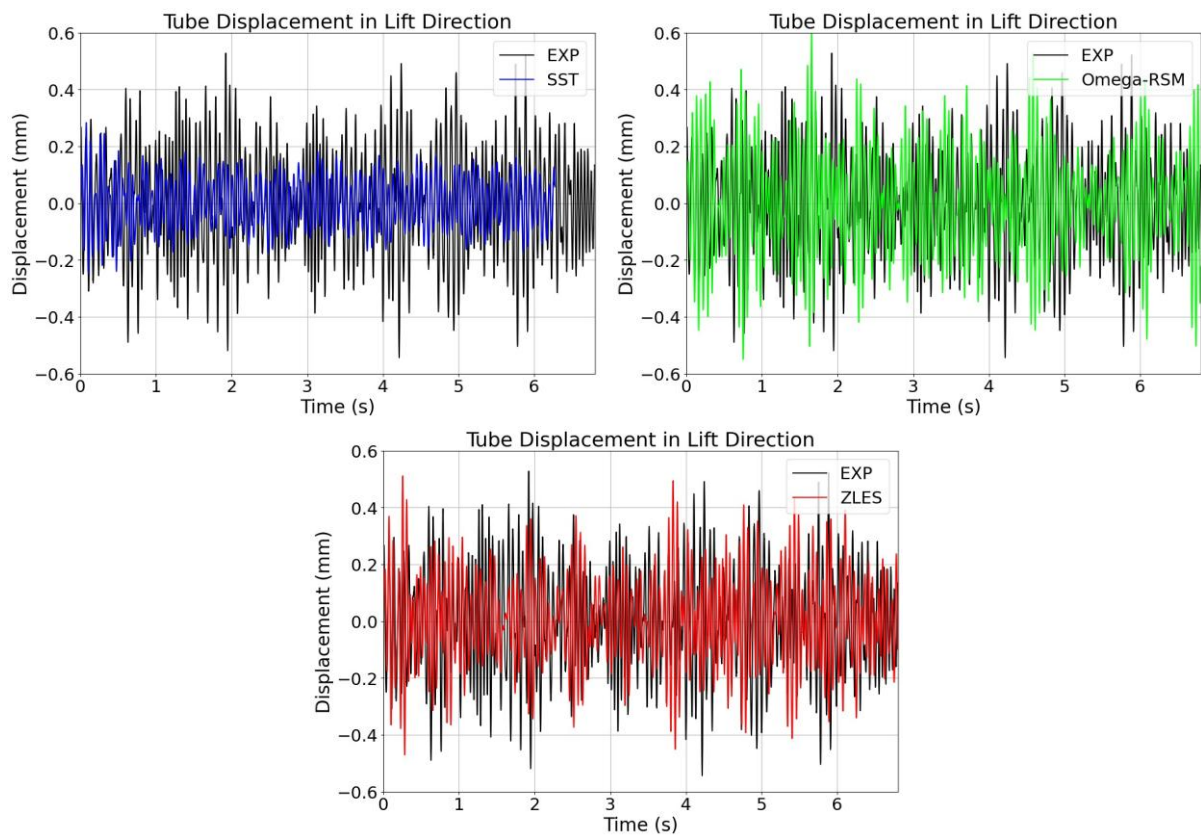


Figure 41: Comparison of tube displacements based on turbulence models

The APSD was calculated from the displacements using Python scripts (Figure 42). The numerical function matplotlib.mlab with Welch's average periodogram was used. The sampling frequency was 2500 Hz (SST), 5000 Hz (Omega-RSM) and 1000 Hz (ZLES). The sampling frequency of the experimental data was 1000 Hz. In the FFT analysis, a Hanning windowing was applied. Two prominent peaks are visible in the experimental curve: the first results from the vortex-shedding from the tube surface (15.2 Hz), while the second one represents the reduced first eigenfrequency of the vibrating tube (22.1 Hz).

A good agreement between simulation and experiment is observed: SST and ZLES predict 16.1 Hz and 16.15 Hz for the vortex-shedding frequency, while Omega-RSM comes to 16.8 Hz, which is almost the same as the measured value. For the second frequency peak at 22.1 Hz, SST predicts 21.3 Hz, while the Omega-RSM and ZLES calculates the same value of 21.5 Hz. Further, the experimental PSD curve shows a low frequency peak at 4.8 Hz. No physical explanation for this low peak was found so far. Nevertheless, it is also visible in the PSD in the simulations: in the ANSYS simulations it is reproduced, but at a frequency almost lower by ~ 1 Hz. One sees a 50 Hz peak in the experimental PSD, but according to CEA, this and the one at ~ 90 Hz result from the power network (electrical noise). Hence, these cannot be reproduced by the simulations.

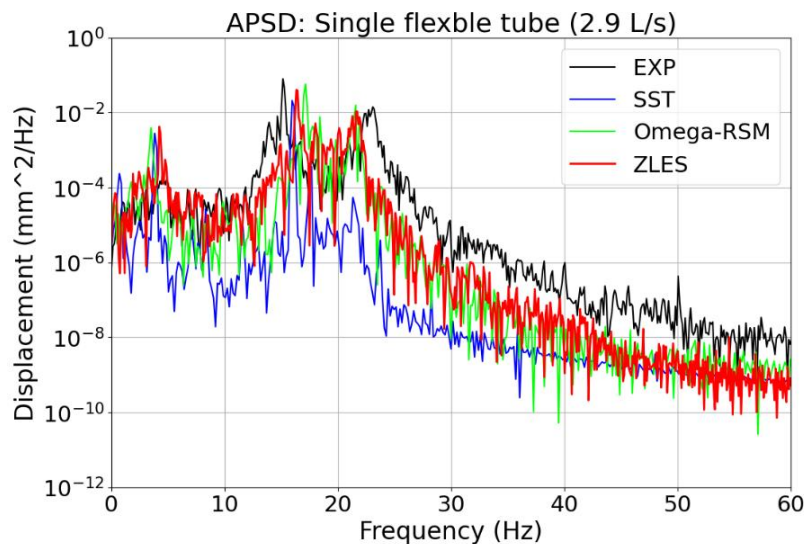


Figure 42: APSD of displacement for single flexible tube (2.9 L/s)

The comparison of the most important vibration parameters is summarized in Table 6. The vortex-shedding and the reduced eigenfrequency peaks are captured by all simulations. These are the vibration frequencies, which are the interesting ones from technical point of view. Although the accurate calculation of the tube RMS displacement is still a bit of a challenge to some extent, the calculated tube displacement is still in the same order of magnitude as the experimental one. Omega-RSM model seems to be the most suitable turbulence model for this particular case, and the presented Rigid Body Motion method can be used for the prediction of FIV of such tube-blade configuration.

Table 6: Comparison of calculated and measured tube displacements and vibration frequencies for single flexible tube case

	Experiment	SST	Omega-RSM	ZLES
Displacement RMS, mm	0.19	0.085	0.179	0.155
Max. displacement, mm	0.6	0.184	0.61	0.49
Vortex-shedding frequency, Hz	15.2	16.1	16.7	16.15
Reduced eigenfrequency, Hz	22.1	21.3	21.5	21.5
Low-frequency peak, Hz	4.8	3.7	3.5	4.2

3.5.2.2 Analysis of FIV using 1.7 L/s flow rate

The simulation with a lower flow rate of 1.7 L/s is performed only with Omega-RSM and ZLES models. These simulations are performed for 10 sec of problem time.

Figure 43 shows the calculated instantaneous velocity with Omega-RSM at $t = 10$ s. The mean free stream velocity in the channel is 0.42 m/s (water flows bottom-top). The maximum velocity of 0.76 m/s is observed in the 1 mm gap between the left tube end and the channel wall (see Figure 43). The flow in the right part of the blade channel, close to the blade fixation point, is stagnant. Like the higher flow rates, flow behavior in reduced flow rate is identical; only the velocity magnitude is reduced with reduction of turbulent kinetic energy.

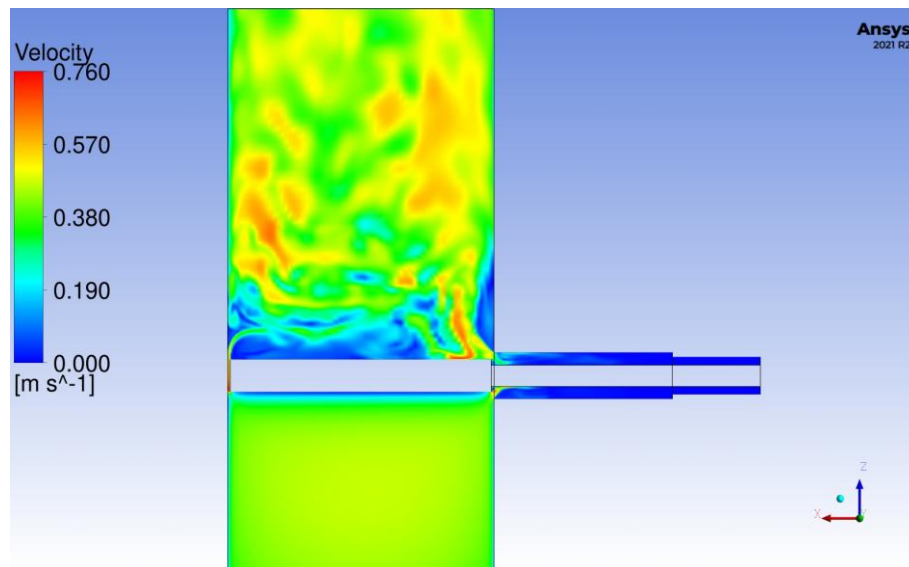


Figure 43: Velocity magnitude distribution in a longitudinal plane (water flows bottom-top) through the AMOVI test section.

The time-dependent vibration amplitudes are compared with data (Figure 44) and the simulation results are further analysed with the help of a Fast Fourier Transform (FFT) analysis (Figure 45). The FFT is performed for 7 s of the simulation time. The first 1 s of simulation time was omitted in both cases. These data acquisition times should provide sufficient resolution for the FFT analysis. Experimental data of 20 s was also used for the comparison. The generated curves were plotted in two separate plots for better visibility.

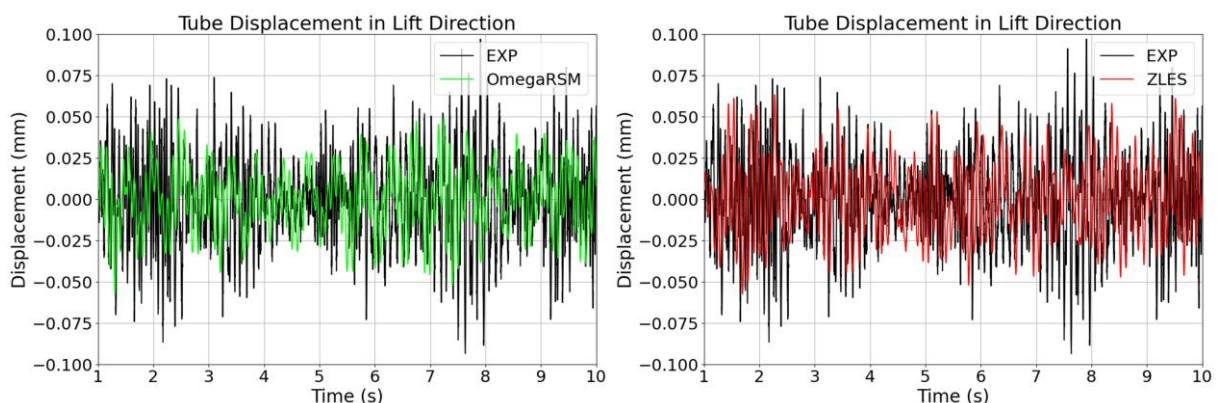


Figure 44: Displacement over time for single flexible tube (1.7 L/s)

Figure 44 shows the similarity of the vibration displacements between both turbulence model. The RMS value of the time-dependent displacement was calculated for the simulation and the experiment. Omega-RSM predicts an RMS value of 0.019 mm, while ZLES give 0.023 mm. Omega-

RSM and ZLE models calculate 26% and 14% smaller displacements than the measured one (0.026 mm).

The APSD was calculated from the displacements using Python scripts (Figure 45). The numerical function matplotlib.mlab with Welch's average periodogram was used. The sampling frequency was 5000 Hz for both models, while experiment was 1000 Hz. In the FFT analysis, a Hanning windowing was applied. Two prominent peaks are visible in the experimental curve: the first results from the vortex-shedding from the tube surface (9.2 Hz), while the second one represents the reduced first eigenfrequency of the vibrating tube (23.5 Hz). At lowest frequency of 2.24 Hz, experiment showed small energy spike, while both simulations calculated higher energy spectrum. A good agreement between simulation and experiment is observed: Omega-RSM and ZLES predict 10 Hz and 9.5 Hz for the vortex-shedding frequency, respectively. For the second frequency peak at 23.5 Hz, both simulation deviates <1 Hz (22.5 Hz) from experiment. Both models showed good agreement with the experiment. The comparison of the most important vibration parameters is summarized in Table 7.

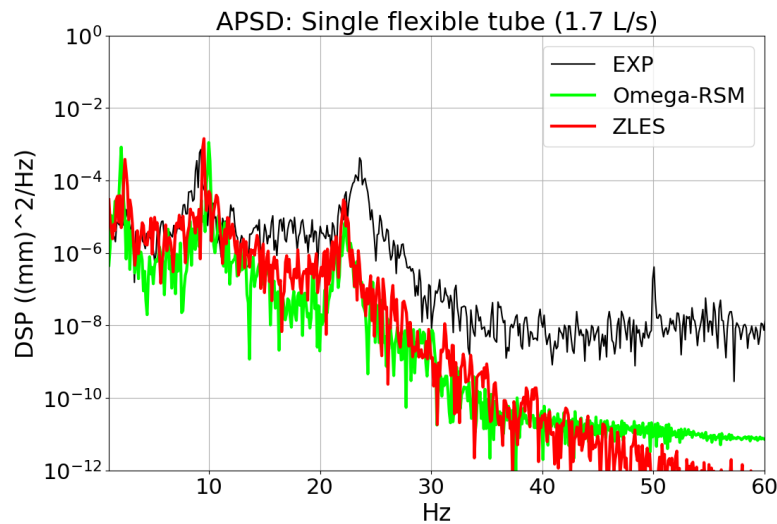


Figure 45: APSD of displacement for single flexible tube (1.7 L/s)

Table 7: Comparison of calculated and measured tube displacements and vibration frequencies for single flexible tube case (1.7 L/s)

	Experiment	Omega-RSM	ZLES
Displacement RMS, mm	0.026	0.019	0.022
Max. displacement, mm	0.096	0.063	0.049
Vortex-shedding frequency, Hz	9.2	10	9.5
Reduced eigenfrequency, Hz	23.5	22.5	22.5
Low-frequency peak, Hz	2.24	2.2	2.4

3.5.3 Flexible tube in a rigid tube bundle ($\dot{Q}=1.6$ l/s)

In the flexible tube in rigid tube bundle case, different analysis based on turbulence model, time steps and rigid body algorithm were performed.

Figure 46 shows the calculated instantaneous velocity with ANSYS Omega-RSM model at $t = 5.5$ s in ZX (left) and YZ planes (right). The YZ plane is positioned at half cylinder height. The water flows from left to the right with mean free stream velocity in the channel of 0.29 m/s. As in the previous bundle cases, the uniform velocity profile is destroyed after the impingement of the water on the first bundle row. The reduced hydraulic diameter of the flow channel leads to a significant increase in the velocity magnitude in the streamwise gap. Four higher-velocity jets with local maxima up to 1 m/s develop in the AMOVI bundle. These jets are unstable in time and oscillate in flow transverse direction, which drives the vibration of the central, flexible tube primarily in its lift direction. After passing the last bundle row, the jets enter the tube-free channel and then decay a few tube diameters downstream the last tube row. Recirculation zones develop behind the cylinders and in the corners formed by the half-cylinders and the side walls (Figure 47).

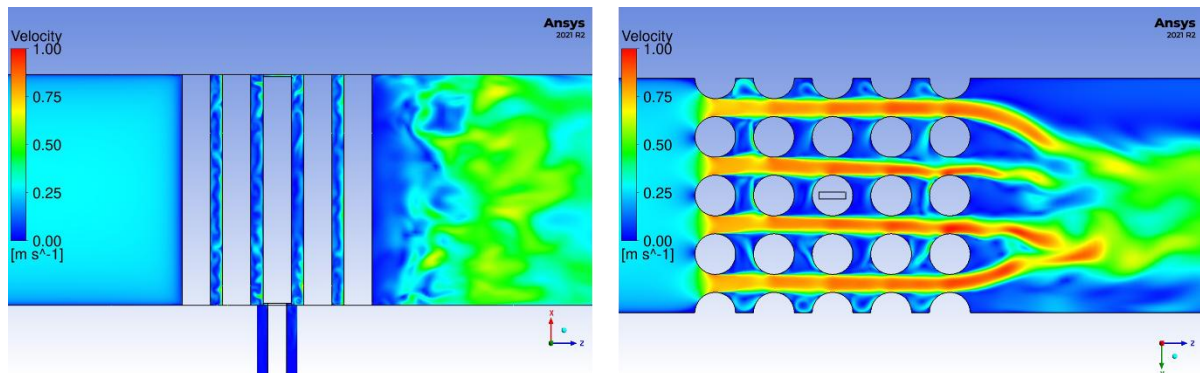


Figure 46: Calculated instantaneous velocity at $t = 5.5$ s in ZX (left) and YZ planes (right).

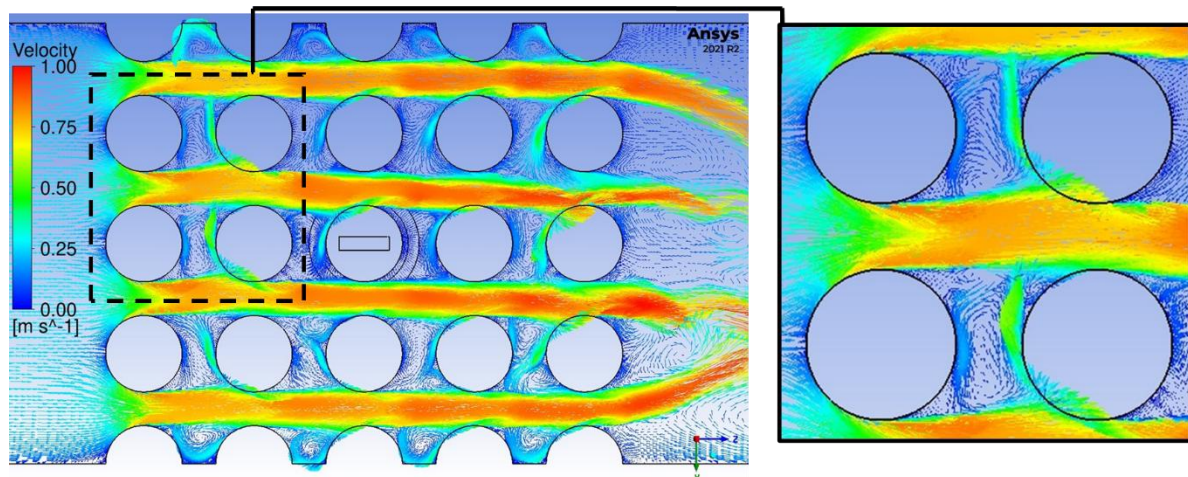


Figure 47: Recirculation zones visualized with velocity vectors.

In order to compare measured and calculated velocities with the same color map, the experimental data and ANSYS CFX-Omega-RSM results were imported in the ParaView software (Figure 48). The qualitative agreement between experiment and simulation is good, although the averaging was performed over 3.5 s of simulation time and over 410 s of experimental data. The similarity is visible: the upper two jets merge shortly downstream the tube bundle, as this is the case for the lower two jets. Still, ANSYS CFX-Omega-RSM slightly overpredicts the maximal jet velocity and suggests stronger recirculation zones behind the last tube bundle row compared to the experiment. The wiggling in the numerical solution does not result from too short simulation time (~ 4 s), but rather due to low frequency data output for the averaging procedure. Unfortunately, in this simulation, the preferred “Transient Statistics” option that averages the solution over all time steps was not

activated. Therefore, the comparison below might have looked differently with this option activated.

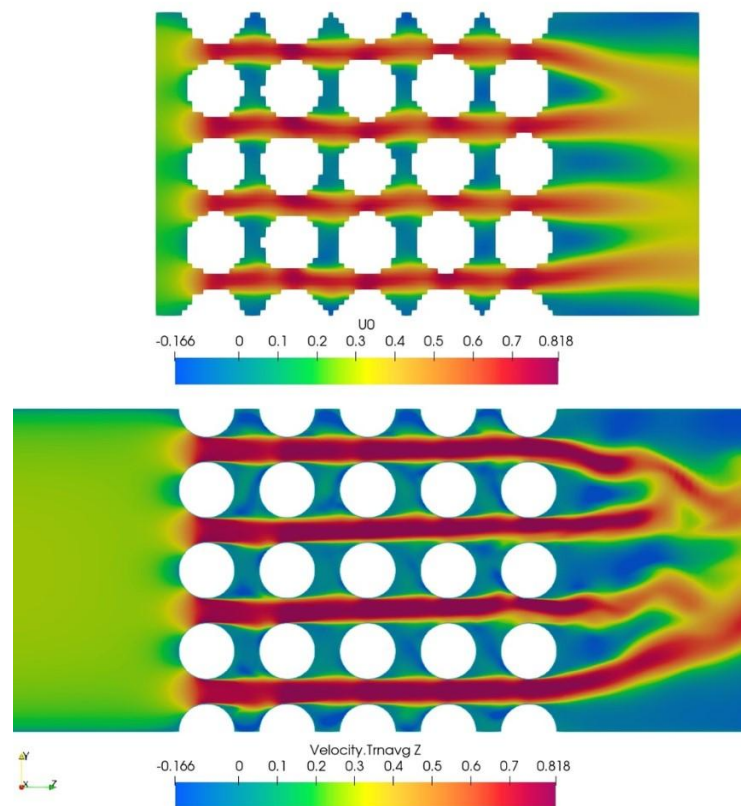


Figure 48: Measured (top) and calculated (bottom) time-averaged streamwise velocity in the tube bundle.

Further, the time-dependent vibration amplitudes of the tube bottom are compared with measurement data and the simulation results are further analysed with the help of a Fast Fourier Transform (FFT) analysis. Two different discretization methods of the angular momentum algorithm were performed:

- Simo Wong
- First Order Backward Euler

Figure 49 and Figure 50 present the calculated and the experimental tube displacements over time. In this first approach (Figure 49), the angular momentum algorithm was set to Simo Wong, which is second order accurate, and time step size was set to 0.2 ms (as default value in the vibration case). Then simulations with SST, Omega-RSM and ZLES turbulence approaches were carried out for 1.2 s, 2.7 s, 3.3 s of time. All data vary with time, therefore, the RMS value of the time-dependent displacement was calculated for the available simulation results and the experiment. The SST simulation showed the lowest vibration amplitude and predicted RMS displacement of 0.014 mm. The Omega-RSM and ZLES model showed increasing amplitudes with the time. This looks like a prediction of a lock-in by the code, which does not really exist in the experimental data. This was the reason for the decision to stop these simulations after a relatively short problem time.

In the second approach (Figure 50), the angular momentum algorithm was set to First Order Backward Euler method. Omega-RSM model was investigated with time step of 0.5 ms, while several simulations with different time steps were performed with ZLES. It can be seen that the ZLES solution depends on the time step size. Reducing the time step size down to 0.125 ms leads to an excessive increase of the tube displacement amplitudes. As in the Simo Wong simulation, a lock-in is predicted here as well, which is not present in the real experiment.

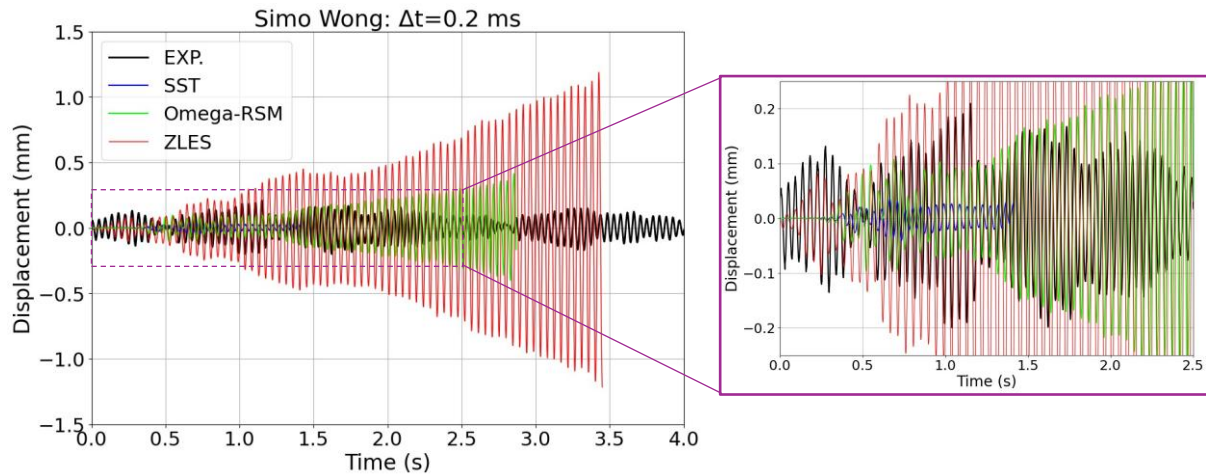


Figure 49: Comparison of tube displacements based on Simo Wong (Rigid Body Motion algorithm accuracy) with experimental data

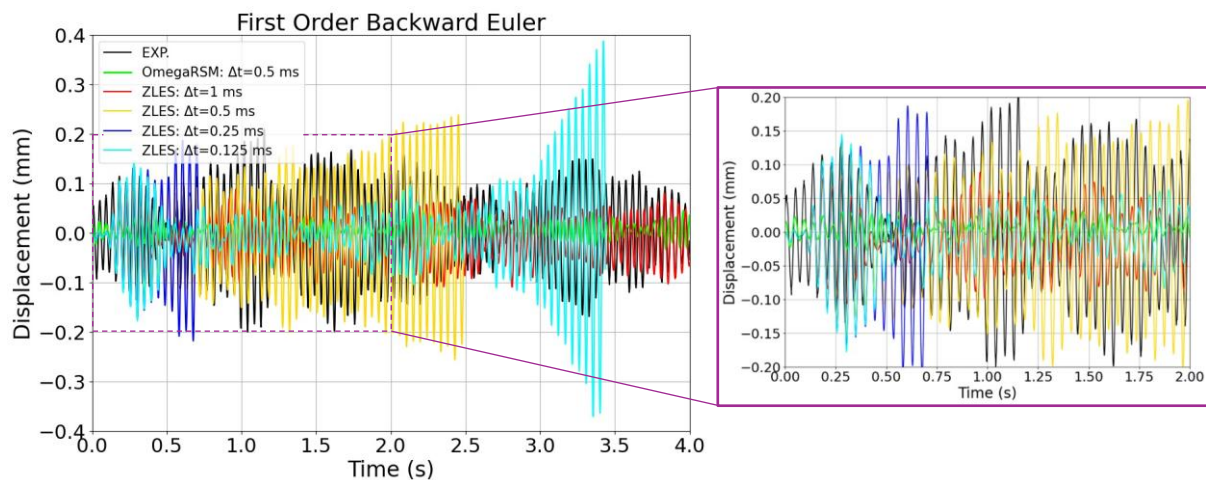


Figure 50: Comparison of tube displacements based on First Order Backward Euler (Rigid Body Motion algorithm accuracy) with experimental data

The APSD was calculated from the displacements using Python scripts (Figure 51). The numerical function matplotlib.mlab with Welch's average periodogram was used. The sampling frequency was 5000 Hz (SST, Omega-RSM and ZLES). The sampling frequency of the experimental data was 1000 Hz. In the FFT analysis, a Hanning windowing was applied. The FFT is performed over 3.5 s of simulation time and 400 s of AMOVI measured data. The prominent peak displays the frequency of the vortex-shedding from the surface of the central tube. A very good agreement between simulation and experiment is observed: ZLES predicts $f = 22.4$ Hz for the reduced eigenfrequency, while the experimental value reads 23.4 Hz. SST and Omega-RSM model show 21.5 Hz. The exact numbers are summarized in Table 8.

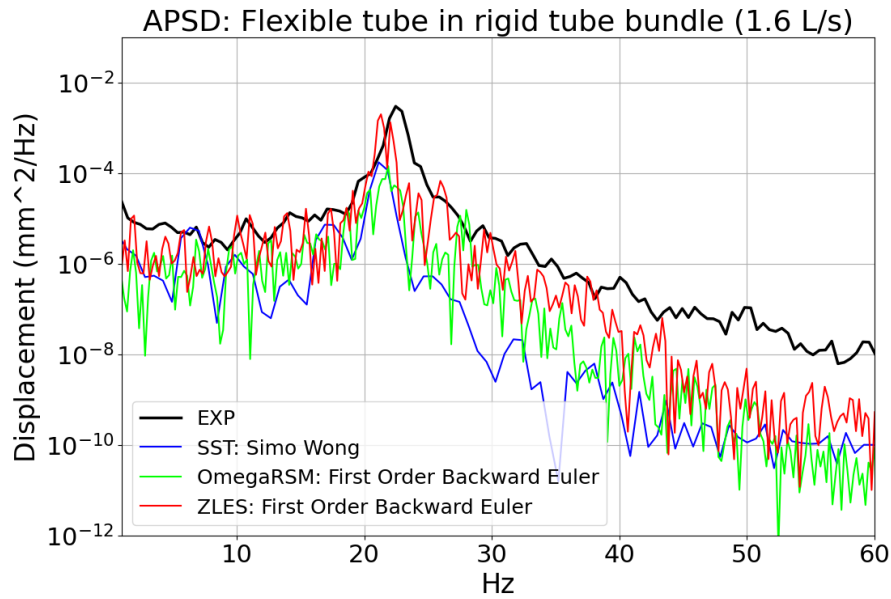


Figure 51: APSD of flexible tube in rigid tube bundle case with 1.6 L/s

Table 8: Comparison of calculated and measured tube displacements and vibration frequencies for flexible tube in rigid tube bundle case (Q=1.6 l/s)

	Experiment	SST	Omega-RSM	ZLES
Displacement RMS, mm	0.073	0.014	0.0156	0.045
Max. displacement, mm	0.30	0.035	0.05	0.088
Vortex-shedding frequency, Hz	23.4	21.5	21.5	22.4

From the current study, it was found that the complex configuration simulated with current rigid body has difficulties to match the experimental displacements, while the frequencies are well predicted. The time step size dependence was found to be significant, especially concerning the tube displacement amplitudes. The different turbulence models also predicted different tube amplitudes, while the variation in the predicted vortex-shedding frequency was small. It was also shown that the Rigid Body Motion discretization algorithm has an effect on the numerical results. Thus, it is important to compare these results with more advanced structural models like the MOR (Model Order Reduction) from ANSYS and the full order model ANSYS CFX-Mechanical. The MOR results of the presented cases can be found in [14] and [19]. The comparison of MOR with the full-order model ANSYS CFX-Mechanical will be described in GO-VIKING D5.2 “Validation of fast-running FSI methods” to be published soon under <https://go-viking.eu/results>.

3.6 Conclusions

To analyse, predict and eventually avoid potential vibration problems, validated 3D advanced numerical tools, based on CFD methods are required. Within the European GO-VIKING project’s WP3, GRS performed analyses of the AMOVI experiment using different simulation approaches. AMOVI is dedicated to FIV in SGs and its test section was equipped with a single tube exposed to a cross-flow in a test channel. In another configuration, the tube was replaced by a tube bundle. The

experiments were performed with rigid and flexible tubes exposed to different inlet volumetric flow rates.

Different turbulence models (i.e. SST, Omega-RSM, SAS and ZLES) were investigated for the calculation of the drag and lift fluid-induced forces on the tube surface for two different configurations in the AMOVI test channel – single tube and flexible tube in a tube bundle. The SAS hybrid model provided best agreement with experimental data for the single tube case. This model was able to capture the first and third vortex shedding harmonic frequency. For the rigid tube bundle case, the ZLES model was found to perform best: 13% deviation from experiment for the case with 2.9 l/s at the test section inlet 6% deviation for the $Q=1.7$ l/s case. The comparison between the PIV measurements and CFD simulation results for both cases showed relatively good agreement, with similar local minima and maxima in the velocity field, even when using a shorter averaging period in the computational data. The numerical results were averaged over 8 seconds, whereas the experimental data were averaged over 400 seconds.

The FSI analyses with moving tubes were performed for a single flexible tube and flexible tube in rigid tube bundle in the test section channel. Also here, different turbulence models were investigated (i.e. SST, Omega-RSM and ZLES). In the single flexible tube case, Omega-RSM showed best results by not only capturing first vortex shedding frequency and the reduced natural frequency of vibration, but also the RMS displacement. Among all other turbulence models, the ZLES model showed best agreement with experiment for the bundle case. Vibration frequency and tube displacement were predicted well by the code using the Rigid Body Motion approach. Time step size studies showed the sensitivity of the results on the time step size. Small time step sizes led to a large increase in the amplitudes. Its origin still has to be further investigated, as in the simulated cases no lock-in is present.

4 Simulations performed by IPP

The cross-flow around a rod is a highly non-stationary problem in which the interaction of the boundary layer, flow separation and vortex path formation [21]-[22] leads to the generation of periodic lift forces. These forces result from the asymmetric separation of vortices and cause vibrations of the structure in the transverse direction [23].

From a numerical modeling perspective, the problem is significantly complicated by the need to account for a strong two-way coupling between the flow and the deformable structure - Fluid Structure Interaction. In such problems, changes in the shape or position of the rod affect the structure of the vortex path, so the pressure redistribution and vortex formation, in turn, determine the oscillation dynamics.

Particular attention in modern research is paid to coupled modelling methods (one/two way FSI) [10]-[14] [24], the influence of mesh and boundary layer resolution, the correct reproduction of spectral characteristics (PSD lift/response), as well as the modelling of turbulent effects (LES/DES vs. URANS).

The practical significance of such problems is extremely high: flow-induced vibrations are one of the main causes of fatigue failure in structures [1]-[4], [25], including heat-exchange tube bundles and fuel rods in nuclear reactors, where cyclic vibrations can lead to wear and damage to components.

The question of the feasibility of implementing so-called engineering approaches and less resource-intensive models, with a demonstration of their applicability for predicting the rod's lift forces and displacements (vibrations) characteristic of the FIV phenomenon, remains particularly relevant. The development of such computationally lightweight methods and models is of particular interest and holds promise for subsequent application to operating equipment, such as steam generators.

The following sections present the development by IPP, where, within the framework of the AMOVI benchmark modeling, emphasis is placed on using “lightweight” models with lower computational cost to provide sufficiently reliable simulations.

4.1 Physics and numerical modelling

The ANSYS CFX code is applied for CFD calculations. For both Single Rod and Rod Bundle configurations, a 2D (two-dimensional) model approach is used to test its applicability for modeling FIV and for restoring the development of forces over time, their APSD, and appropriate rod displacement characteristics in cases with Flexible Rod. For organizing FSI analysis of the AMOVI cases with a flexible rod, two options for the mechanical modeling are considered: the Rigid Body Solution or a user FORTRAN subroutine with the Beam equation solver. Further sections provide the details on model development and calculations performed by IPP in the framework of the AMOVI benchmark.

4.1.1 Fluid modelling

4.1.1.1 CFD models for AMOVI Experiment

Based on the drawings of the AMOVI experimental setup (see section 2.1), two-dimensional models of test cross-sections were prepared for both geometric configurations – flow around a single rod and a bundle of rods. Each model is represented as a central cross-section of a fluid domain

enclosed within a flow channel with the fluid flowing around a single rod (Figure 52, left) or rod bundle (Figure 52, right). Application of both the Rigid Tube Bundle as well as the Beam Subroutine approaches for accounting for Rod movement in Flexible Rod configurations allow us to exclude the Solid part from the modelling.

Since each AMOVI experimental case was performed at a constant temperature (see Table 1, Section 2.2), an isothermal formulation is used for all simulated cases. The thermophysical properties of the fluid for each case are determined according to the IAPWS Library.

As for the modeling boundaries, neither model accounts for the mixing section in the inlet chamber; both include only the straight sections of the test section, with lengths of 195 mm and 600 mm in the Single Rod and Rod Bundle models, respectively. To ensure correct accounting of the longitudinal component of the inlet flow velocity profile, it is specified at the inlet boundary in the form of a user-defined function depending on the value of the volumetric flow rate and the geometric parameters of the channel:

$$W_{in} = \frac{Q}{S_{ref}} \cdot \left(1 + \frac{1}{m}\right) \cdot (1 - |2 \cdot x/H|)^{1/m} \quad (5)$$

$$m = f(Re), \quad S_{ref} = 0.1 \cdot H[m^2]$$

Here H is the channel width, which is equal to 0.04 [m] in the Single Rod model and 0.07 [m] in the Rod Bundle model.

A soft boundary condition of the “opening” type is applied at the outlet, while the walls of the test section and fixed rods walls in the assembly are assumed to be no slip and smooth. In calculation cases with Flexible Rod, the movement of the rod mesh nodes is determined by the Rigid Body Solution or by the Beam solution implemented as a User library in CFX (for a more detailed description, see section 4.1.2). The general view of the calculation models and the specification of boundary conditions are shown in Figure 52.

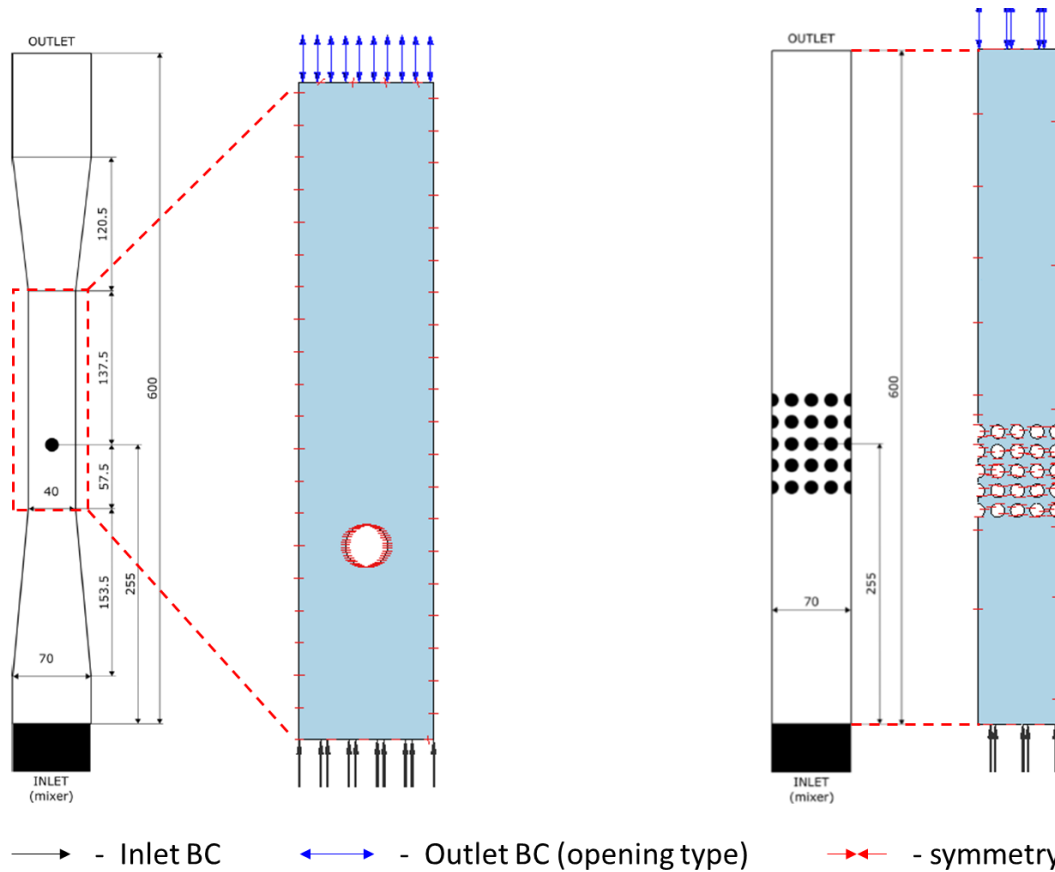


Figure 52: General view and schematic representation of boundary conditions applied in CFD models for single rod (left) and rod bundle (right) cases

Hexahedral meshes were generated for both the Single Rod and Rod Bundle CFD models. For this purpose, the ANSYS Meshing as well as ICEM CFD tools were applied. The main focus was on the near-wall mesh resolution and on maintaining low skewness and smooth transitions, which ensure numerical stability.

During the mesh study for the first Single Rod CFD model, four different meshes were developed. The general view and the corresponding information on the number of elements in the developed meshes is given in Figure 53. The first mesh named “LowRe” was prepared in Ansys Mesher for the first test calculations. Then, to investigate the influence of whole-domain mesh refinement, near-wall mesh refinement, and their combined improvement, three different meshes were generated in ICEM CFD.

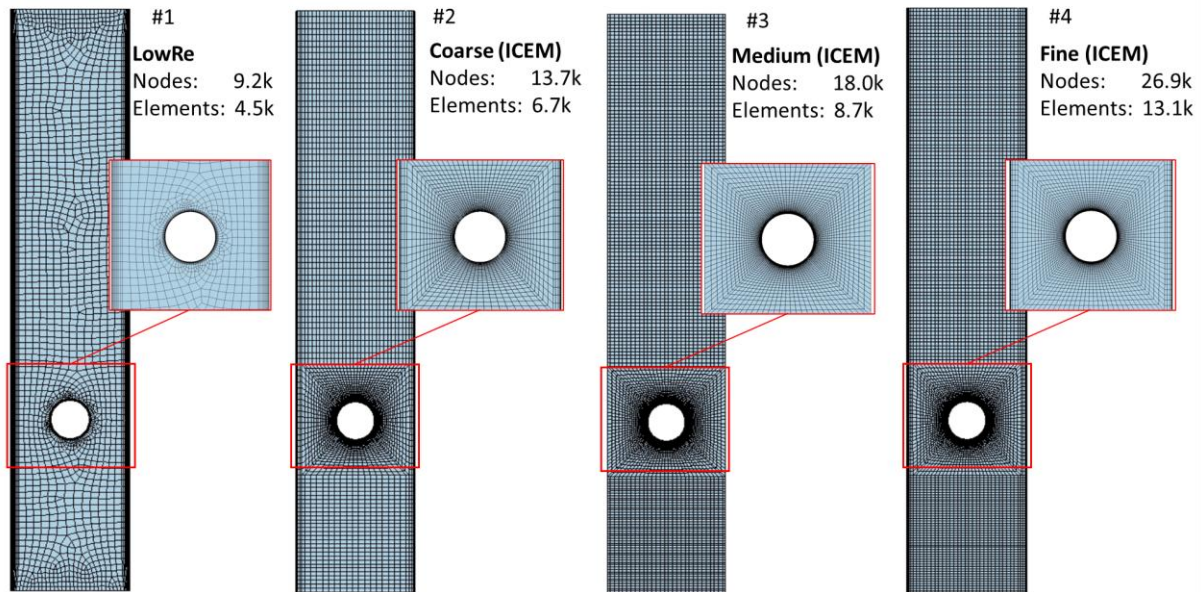


Figure 53: Calculation meshes applied for single rod CFD model

A separate mesh-refinement study was carried out for the case with an absolutely rigid rod, followed by the flexible rod, to assess the possible influence of mesh variation on subsequent calculation results.

It is worth noting that when applying all four meshes for the Rigid Tube configuration, a characteristic pattern of Karman vortices is observed. The frequency of vortex shedding in a fluid flow can be characterized by the Strouhal Number (St) - a dimensionless quantity that relates the vortex shedding frequency, the flow velocity, and the characteristic size of the body. In the range $200 < Re < 200000$ the approximate empirical law of constancy of the Strouhal number applies: $Sh \approx 0.2$, based on which the theoretical value of the vortex separation frequency in the Single Rod model is equal ~ 11.37 Hz. The corresponding values of the vortex separation frequencies obtained from calculations on different meshes are as follows: Mesh#1 – 14.28Hz, Mesh#2 – 16.66Hz, Mesh#3 – 15.66Hz, Mesh#4 – 15.62Hz. The calculated values of the vortex shedding frequency are approximately 25% higher than the theoretical value, which is explained by the confined flow in the channel, which gives a corresponding increase in the Strouhal number.

Figure 54 shows the transient axial velocity at two locations: the first one – just behind the rod, a little downstream in the place after the vortex separation, and the second – in the center of the channel downstream. The results demonstrate the higher velocity fluctuations when applying coarse meshes #1 and #2. For the medium and fine meshes, these fluctuations decrease and approach the same level.

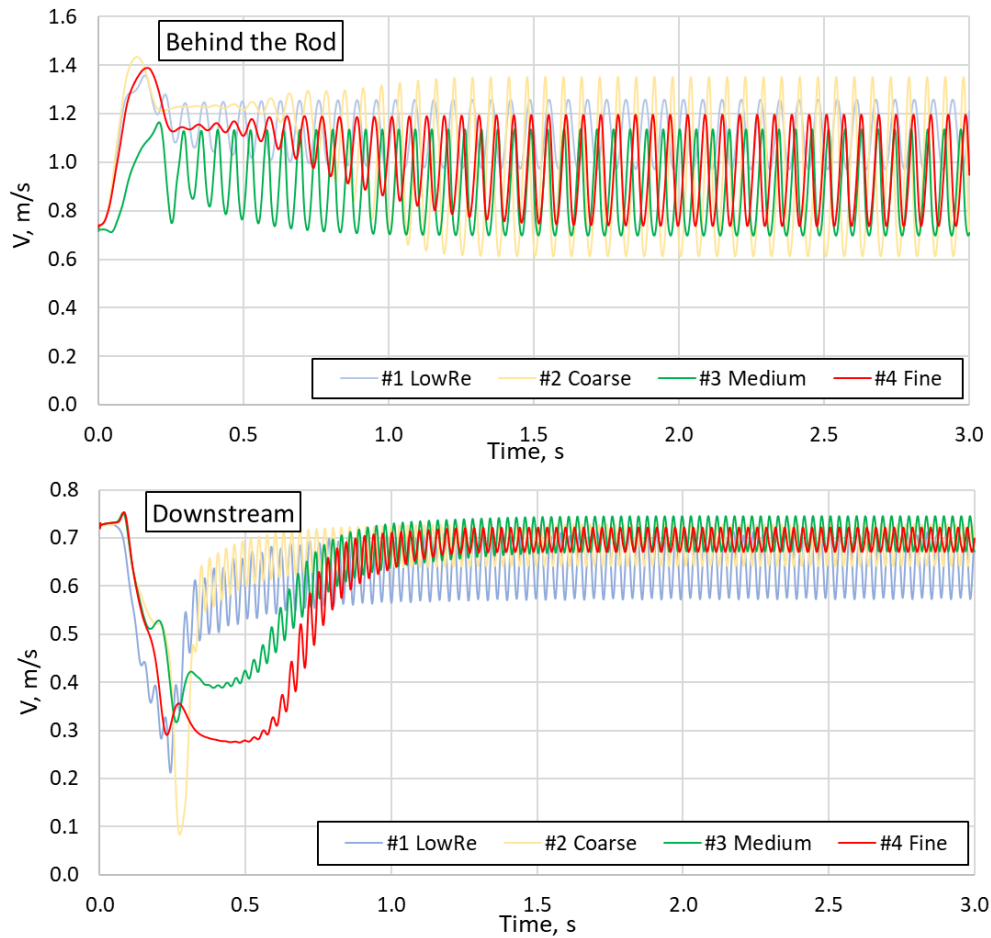


Figure 54: Mesh independence study for the single rigid tube case

Figure 55 shows the results of applying the developed meshes for the Single Flexible Rod case. It can be seen that the obtained rod displacement amplitudes exhibit higher fluctuations when using a coarse mesh, but when using medium and fine meshes, these fluctuations decrease and the amplitudes are similar. Thus, mesh #3 (Medium) was selected for further calculations for both the Single Rigid Tube and Single Flexible Tube cases.

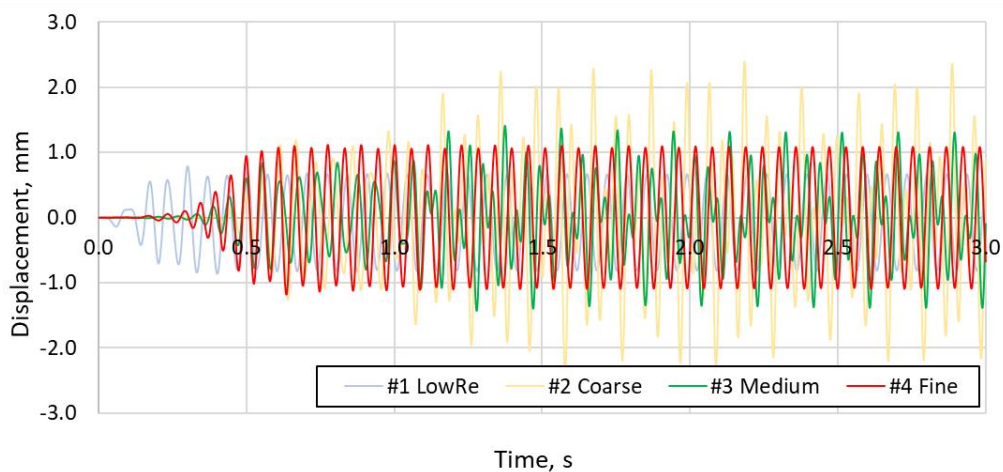


Figure 55: Mesh independence study for the single flexible tube case

Then, the mesh study for the second CFD model with the Rod Bundle was performed. To investigate the influence of whole-domain mesh refinement, near-wall mesh refinement, and their combined improvement, four meshes were generated in ICEM CFD. The general view and the corresponding information on the number of elements in prepared meshes is given in Figure 56.

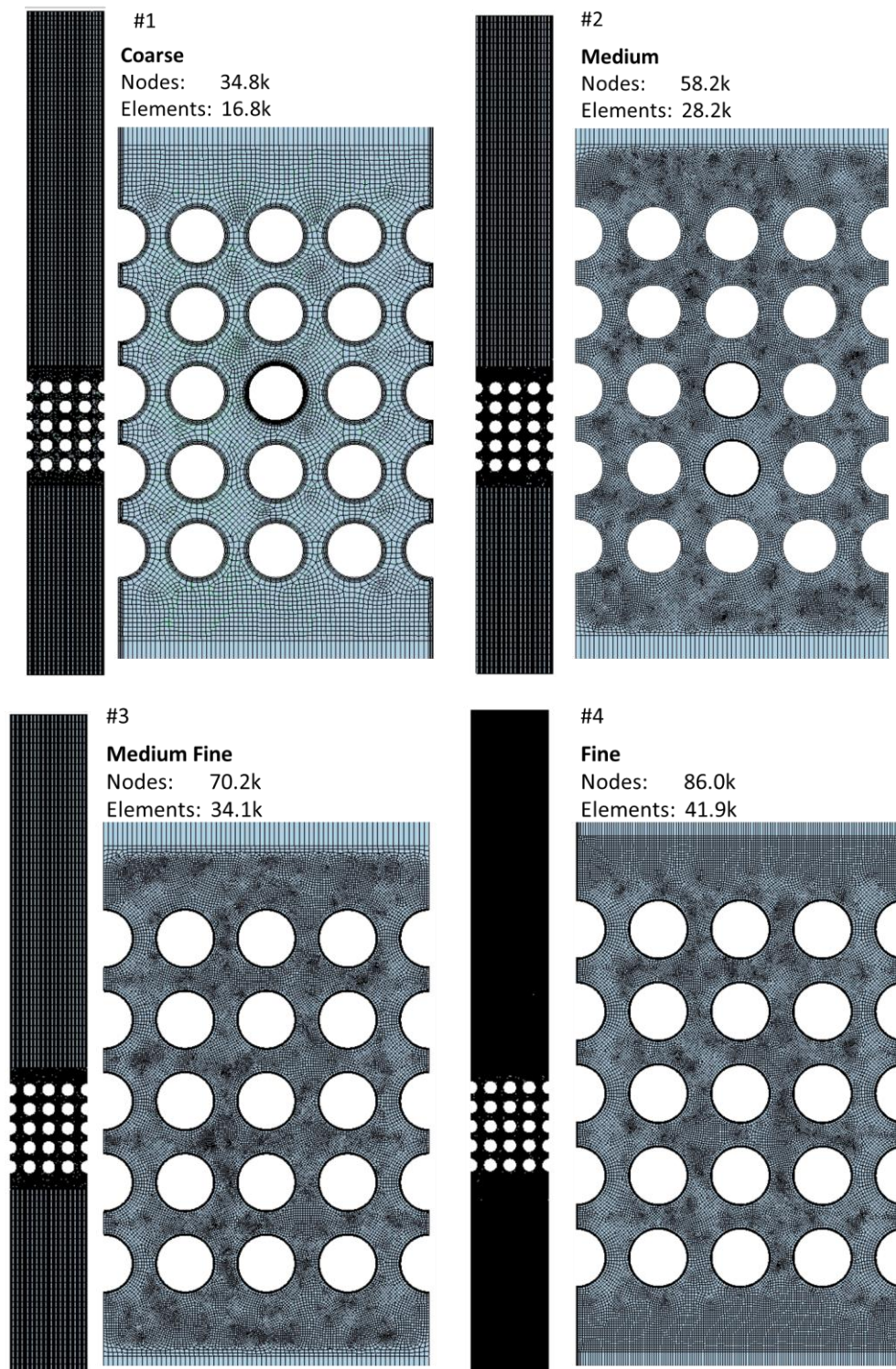


Figure 56: Calculation meshes applied for rod bundle CFD model

To investigate the influence of mesh refinement on calculation results, mesh studies were performed for both the Rigid Tube Bundle and Flexible Tube in Rigid Tube Bundle cases. Figure 57 shows the transient axial velocity at point behind the rod bundle (a little downstream). It can be seen that for the first two meshes (Coarse and Medium) the velocity fluctuations were higher, but for the third (Medium Fine) and fourth (Fine) meshes the fluctuations were lower and maintained a similar amplitude. The same situation occurred when applying these meshes for the Flexible Tube in Rigid Tube Bundle case (see Figure 58). Thus, mesh #3 (Medium Fine) was selected for further calculations for both the Single Rigid Tube and Single Flexible Tube cases.

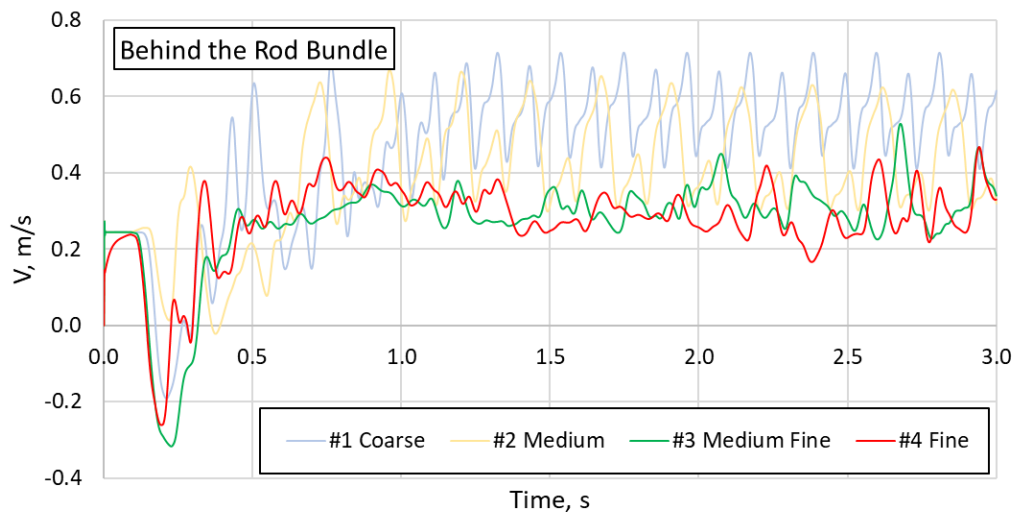


Figure 57: Mesh independence study for the rigid tube bundle case

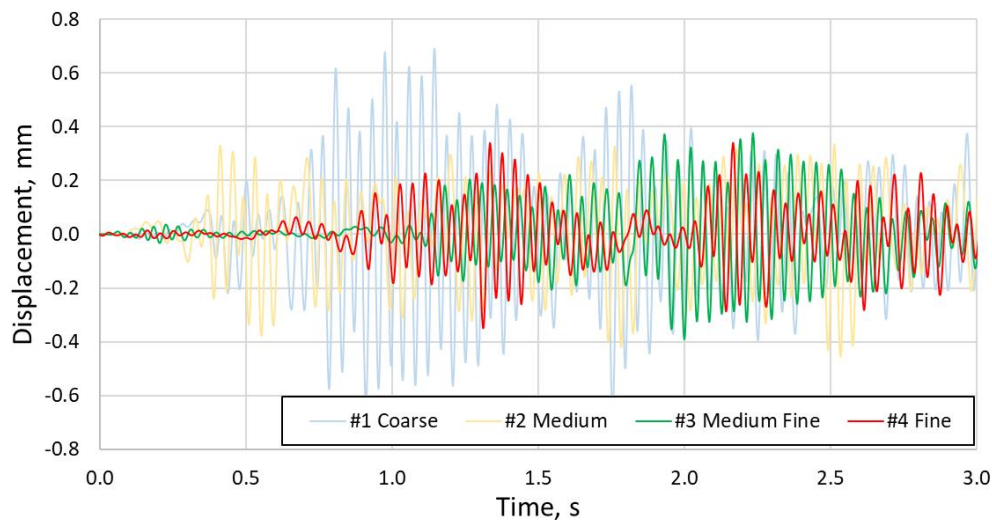


Figure 58: Mesh independence study for the flexible tube in a rigid tube bundle case

4.1.2 Mechanical modelling

From a mechanical point of view the AMOVI Rod movement was modeled in two different ways. The first option was to apply the Rigid Body Solution. In that case, the flexible rod wall is treated as a rigid body, and the mesh motion around the rod is simulated based on the solution of the rigid-body equations of motion.

The parameters that define the rigid body are:

- The mass of the modeled section of the rod – 0.906 g (according to the specifics of the ANSYS CFX, a quasi 2D model of the rod cross section located at the central position in the channel is represented by 2mm thick section, which is 1/50 part of the whole active part of the flexible rod).
- The mass moment of inertia in the YY (vertical) direction – 0.00075987 kg/m².
- The linear spring constant is set in the direction of the Rod movement (X) and equal to – 30.8676 N/m.
- The equivalence of the hydrodynamic force (effective length of the control volume in the transverse direction) is taken into account in a quasi-two-dimensional approximation (a feature of using ANSYS CFX).
- Gravity is accounted.

The second approach was to implement the User Library for FIV analysis in FORTRAN. This Library includes different options of Beam solution for the modelled AMOVI flexible rod:

- Analytical solution of the Euler-Bernoulli Beam
- Analytical solution of the Timoshenko Beam
- FDM implementation of the Euler-Bernoulli Beam (no preliminary determined frequencies in this case)

Some additional details of the Analytical solution for the Beam are described in section 4.1.2.1. The FDM implementation description could be found in [26].

4.1.2.1 Beam solution

The governing equations applied for the Analytical solution of the Beam are summarized in D2.2 [27], so here we will focus on the two most interesting models: two consecutively connected beams (Two Beams) and a beam with a rigid body at the end (Beam+RB).

Let's start with a more simplified model of the beam with the rigid body, in this case the equations of motion for the beam are the same as described in [27], but the boundary conditions where the rigid body is connected, should be modified:

$$\begin{cases} \frac{\partial^2 y_0}{\partial t^2} = -Q(l, t) \\ J_0 \frac{\partial^2 \varphi_0}{\partial t^2} = Q(l, t)d - EJ \frac{\partial^2 y}{\partial x^2} \end{cases} \quad (6)$$

Here the following notations are used:

$$y_0 = y(l, t) + d \frac{\partial y}{\partial x_{x=l}}; \quad \varphi_0 = \frac{\partial y}{\partial x_{x=l}}; \quad (7)$$

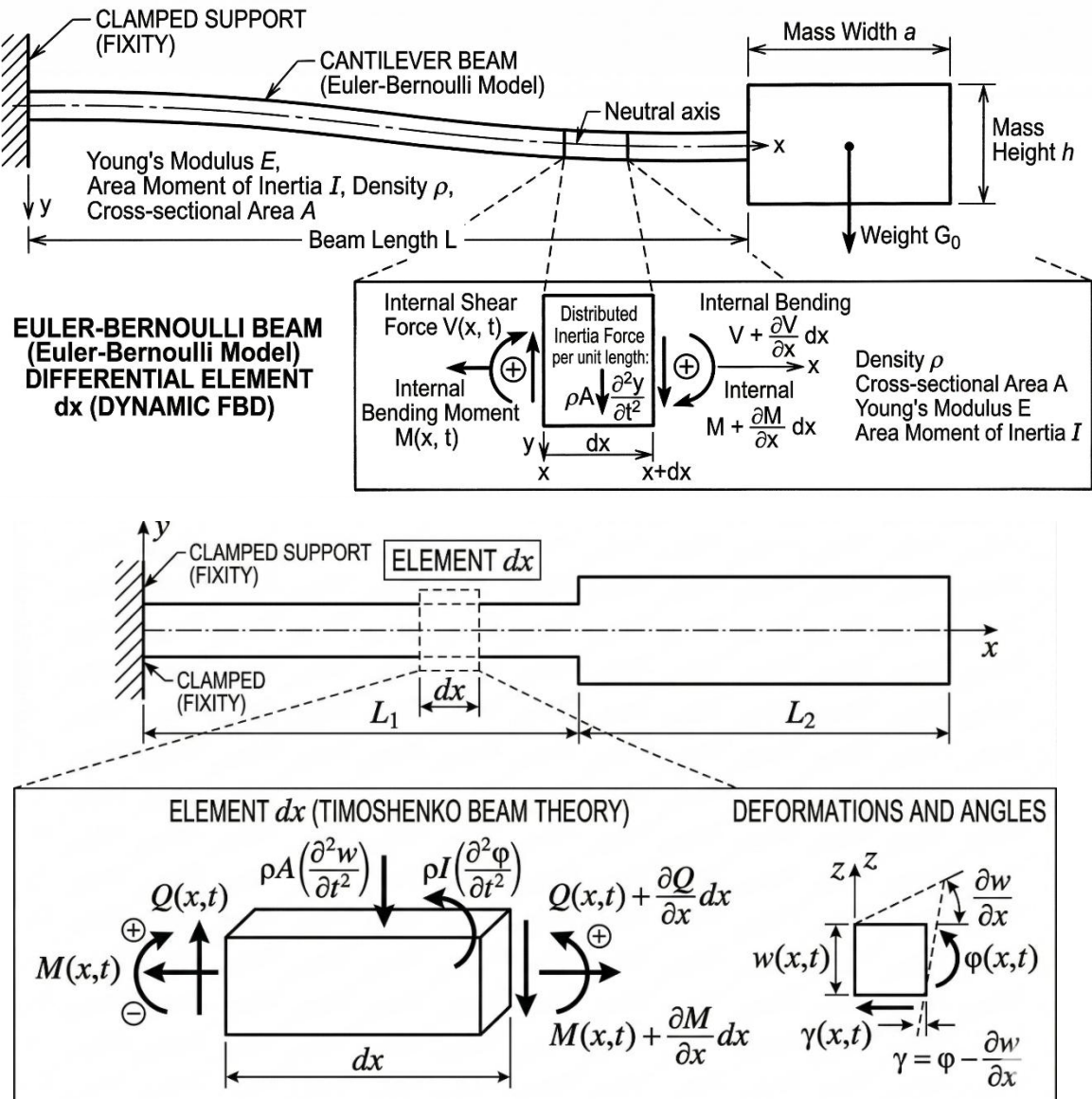


Figure 59: Beam with Rigid body (up), Two Beam model (bottom)

Thus, the equations of motion for the beam with a rigid body:

$$\begin{cases} -EJ \frac{\partial^3 y}{\partial x^3} + M_0 \frac{\partial^2 y}{\partial x^2} + M_0 d \frac{\partial^3 y}{\partial t^2 \partial x} = 0 \\ EJ d \frac{\partial^3 y}{\partial x^3} + EJ \frac{\partial^2 y}{\partial x^2} + J_0 \frac{\partial^3 y}{\partial t^2 \partial x} = 0 \end{cases} \quad (8)$$

Using the solution, which satisfies the:

$$Y(x) = C_3 Kr_3(\beta x) + C_4 Kr_4(\beta x) \quad (9)$$

Here Kr_3, Kr_4 – are the Krylov or beam functions described in [27], $\beta = \sqrt[4]{\frac{\rho A \omega^2}{EI}}$. If we take into account the boundary conditions at the clamped end $Y(0) = 0$; $Y'(0) = 0$ Frequency equation for the Euler-Bernoulli beam with rigid body:

$$\frac{1}{\alpha}(1 + ch\beta\cos\beta) - \beta(\sin\beta ch\beta - \cos\beta sh\beta) - 2\varepsilon\beta^2\sin\beta sh\beta - (\delta + \varepsilon^2)(\sin\beta ch\beta + \cos\beta sh\beta)\beta^3 + \alpha\delta\beta^4(1 - \cos\beta ch\beta) = 0 \quad (10)$$

Here the following notations are used:

$$\alpha = \frac{M_0}{\rho Fl}; \delta = \frac{J_0}{M_0 l^2}; \varepsilon = \frac{d}{l} \quad (11)$$

In the case of the two beam model the characteristic equation is the determinant of the following matrix:

$$\begin{vmatrix} Kr_{12} & Kr_{22} & Kr_{31} & Kr_{41} \\ \beta_2 Kr_{42} & \beta_2 Kr_{12} & \beta_1 Kr_{21} & \beta_1 Kr_{31} \\ -J_2\beta_2^2 Kr_{32} & -J_2\beta_2^2 Kr_{42} & J_1\beta_1^2 Kr_{11} & J_1\beta_1^2 Kr_{21} \\ J_2\beta_2^3 Kr_{22} & J_2\beta_2^3 Kr_{32} & J_1\beta_1^3 Kr_{41} & J_1\beta_1^3 Kr_{11} \end{vmatrix} \quad (12)$$

Here β_1 corresponds to the first and β_2 to the second beam.

The results of the natural frequencies are summarized in Table 9, also the solution of the Timoshenko beam(s) is calculated. We can see that the error between the Beam+RB vs Two Beam solution for frequencies 1 to 5 is less than 10%. Moreover, the differences between the beam models, i.e. Euler-Bernoulli vs Timoshenko are also less than 10% for 1-10 natural frequencies.

Table 9: Comparison of natural frequencies (in Air)

	Beam+RB		Two Beams	
	Euler-Bernoulli	Timoshenko	Euler-Bernoulli	Timoshenko
1	29.56	29.56	31.04	31.02
2	284.06	283.88	300.87	299.26
3	1245.69	1242.16	1256.07	1249.95
4	3110.20	3089.60	3044.36	3018.68
5	5922.20	5852.80	5450.47	5333.86
6	9675.00	9500.06	7208.61	6929.79
7	14368.09	13999.99	10000.55	9739.37
8	20000.94	19316.26	14272.42	13733.26
9	26573.31	25409.08	18427.07	16694.36
10	34085.04	32236.13	21376.68	19952.09

From this point of view in the next analysis we will consider two models, beam+RB and two beams both Euler-Bernoulli type.

4.1.2.2 Added mass effect

For the analysis of rod vibration in the fluid we should include added mass. In a very simple case references from Blevins [28] could be used.

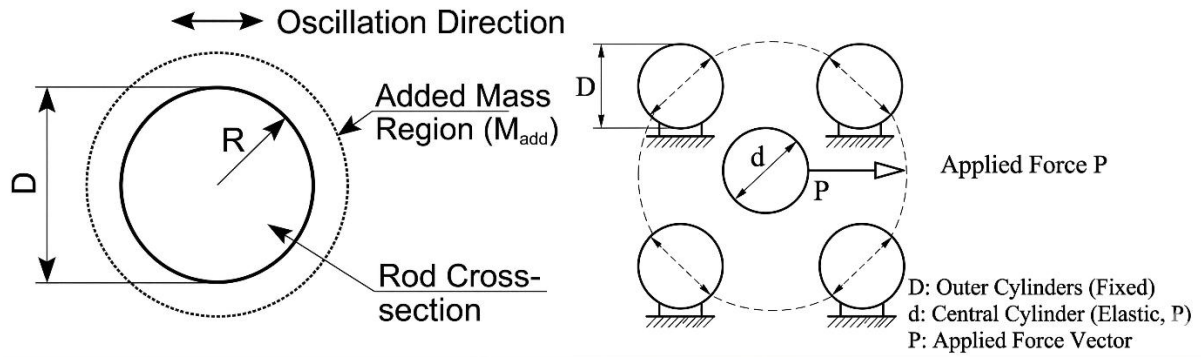


Figure 60: Added mass: a) flexible cylinder oscillating in infinite fluid, b) flexible cylinder oscillating in rigid bundle

Added mass for the single cylinder per unit length:

$$m_{add} = \frac{\rho \pi D^2}{4} = \frac{998 \pi 0.01215^2}{4} = 0.1157 \text{ kg/m}; \quad (13)$$

Added mass for the flexible cylinder oscillating in the rigid bundle:

$$m_{add} = \frac{\rho \pi D^2}{4} \left[\frac{(D_e/D)^2 + 1}{(D_e/D)^2 - 1} \right]; \quad \frac{D_e}{D} = \left(1 + \frac{1}{2} \frac{P}{D} \right) \frac{P}{D} \quad (14)$$

$$m_{add} = \frac{998 \pi 12.15^2}{4} \left[\frac{(0.0301/0.01215)^2 + 1}{(0.0301/0.01215)^2 - 1} \right] = 0.1607 \text{ kg/m}$$

In Figure 61 we can see the effect of added mass on the oscillating Beam+RB and Two Beam models, in this case, fluid added mass effects just lower the first frequency by approximately 5%.

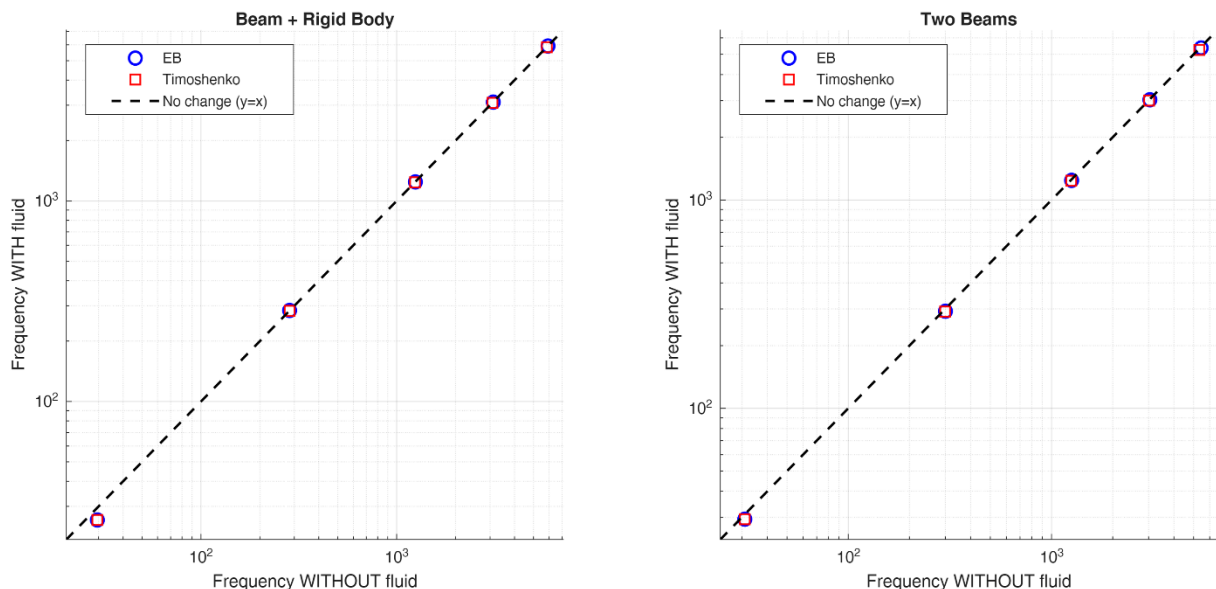


Figure 61: Added mass: effect for the flexible cylinder oscillating in infinite fluid

4.2 Numerical calculations results

The results of the simulations for AMOVI Fixed Rod and Flexible Rod cases performed by IPP are presented in sections below.

- Sections 4.2.1.1 and 4.2.1.2 represent the simulation results for the Single Rigid Tube case with $Q=2.9$ l/s and the Rigid Tube Bundle cases with $Q=1.7$ l/s and with $Q=2.9$ l/s respectively.
- Sections 4.2.2.1– 4.2.2.4 represent the simulation results for Single Flexible Tube cases with $Q=0.5$ l/s, $Q=1.7$ l/s, $Q=2.9$ l/s and $Q=4.0$ l/s.
- Section 4.2.2.5 – 4.2.2.8 represent the simulation results for Flexible Tube in a Rigid Tube Bundle cases with $Q=0.5$ l/s, $Q=1.1$ l/s, $Q=1.6$ l/s and $Q=1.95$ l/s.

For each simulated case the results are compared with the experimental data.

4.2.1 Analysis of AMOVI cases with fixed rod (Rigid tube cases)

At the initial stages of simulating AMOVI test cases with Single Rigid tube and Rigid tube bundle, the primary analysis was performed using the SST turbulence model with varying time steps. Then, after a series of calculations for the Flexible Tube in a Rigid Tube Bundle, the application of SAS and Omega-RSM turbulence models to the Rigid Tube Bundle cases was also performed and analyzed. Appropriate results are demonstrated in subsequent sections.

4.2.1.1 Single rigid tube - SRT-290 ($Q=2.9$ l/s)

During the analysis of the Single rigid tube configuration, the case with 2.9 l/s was simulated. For modelling, the SST turbulence model was applied with the calculation time step varying from 1.0ms to 0.1ms. While performing the simulation with a 1.0ms time step – both the first developed mesh (LowRe, see Figure 53) and subsequently chosen ICEM (Medium, see Figure 53) mesh were applied. Total simulation time for all three calculations was 10.0 seconds. The first one second of the simulated transients was discarded during post-processing. As can be seen from the obtained results summarized in Table 10– in all simulations both Lift force and Drag force RMS values are underestimated, while the position of the PSD peak is captured with sufficient accuracy for engineering assessment (see Figure 62).

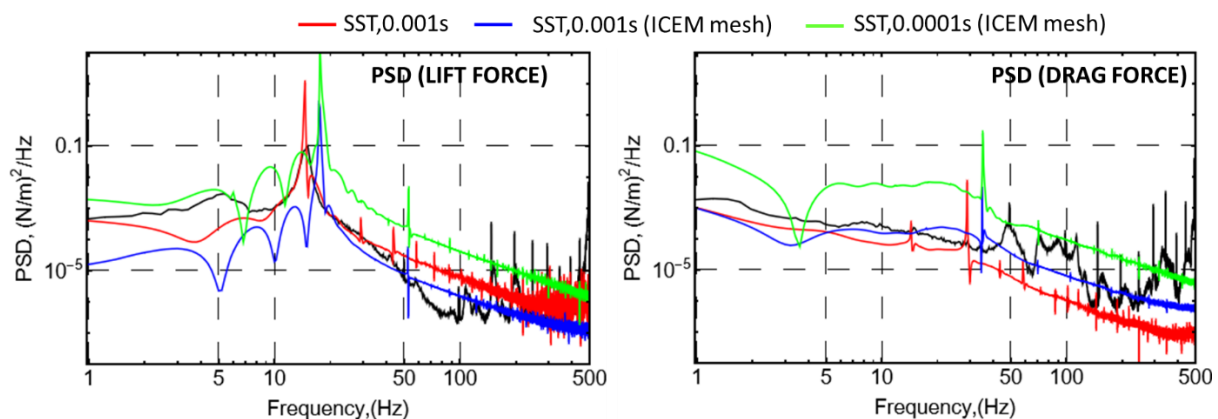


Figure 62: Comparison of resulting APSD of lift force (left) and drag force (right) with experimental APSDs for the single rigid tube with $Q=2.9$ l/s

Obtained from simulations the longitudinal velocity and vorticity fields that demonstrate the von Karman vortex pattern, are shown in Figure 63.

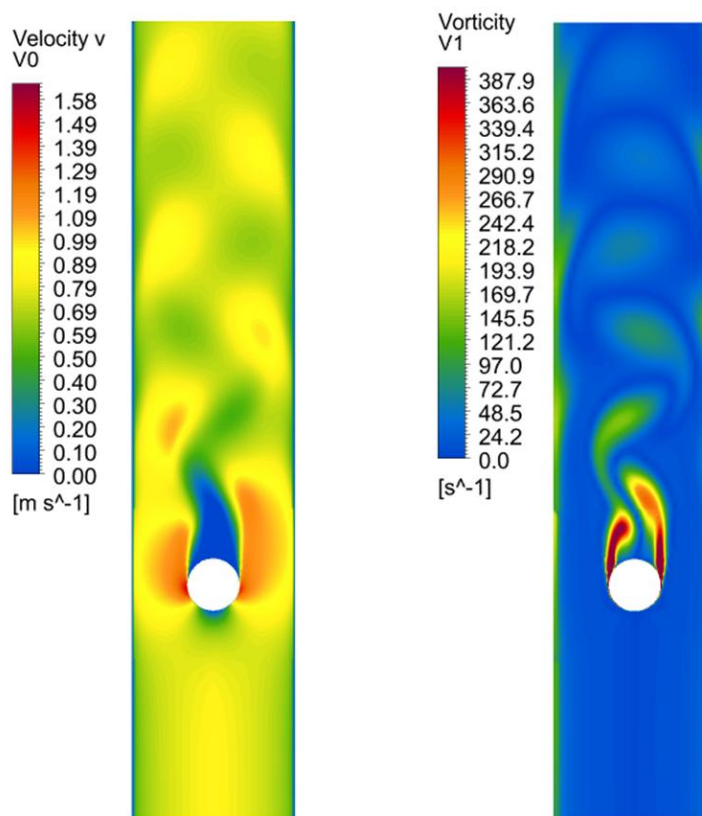


Figure 63: Simulated fields of the longitudinal velocity component and vorticity picture obtained for the single rigid tube with $Q=2.9$ l/s

Table 10: Comparison of RMS of forces obtained in calculations for a single rigid tube ($Q=2.9$ l/s) with the measured values

	Experiment	SST, 1.0ms	SST, 1.0ms (ICEM mesh)	SST, 0.1ms (ICEM mesh)
RMS Lift Force, [N/m]	0.411	0.197	0.117	0.248
RMS Drag Force, [N/m]	0.120	0.01	0.007	0.023

4.2.1.2 Rigid tube bundle - RTB-170 ($Q=1.7$ l/s) and RTB-290 ($Q=2.9$ l/s)

Two AMOVI cases with Rigid Tube bundle configuration were simulated – RTB-170 and RTB-290. For both cases, the SST model with 1.0ms and 0.1ms time steps, as well as the SAS and Omega-RSM turbulence models with 0.1ms time steps, were applied. The total simulation time in both cases was 10.0 seconds. The first one second of the simulated transients was discarded during post-processing. When analyzing the results, it was observed that a good ability to simulate RMS Lift forces was achieved for almost all calculation configurations, except when applying the SAS turbulence model with a 0.1 ms timestep, where the deviation of the resulting RMS from the experimental value was more than 10-15%. Moreover, when using the SAS for RTB at 1.7 l/s, the RMS

lift force was underestimated, whereas for RTB at 2.9 l/s, on the contrary, it was overestimated. As for the drag force RMS, in both analyzed cases it was overestimated regardless of the turbulence model used or the calculation step applied (see Table 11 and Table 12).

Figure 64 and Figure 66 show the time evolution of Lift and Drag forces for the simulated RTB cases compared with the experimental data. The corresponding APSD for Lift and Drag forces are presented in the Figure 65 and Figure 67.

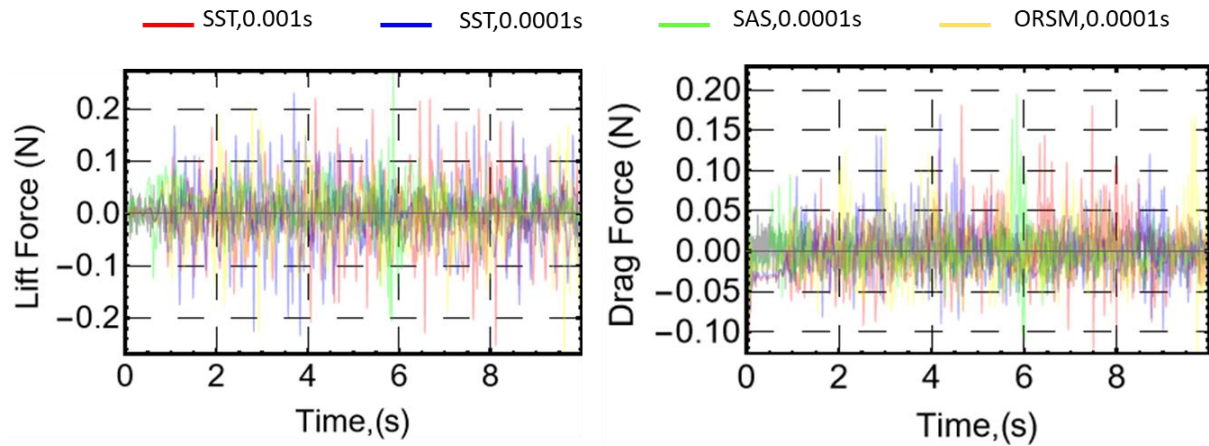


Figure 64: Comparison of the simulated lift force (left) and drag force (right) in time with experimental forces measured for the rigid tube bundle with $Q=1.7$ l/s

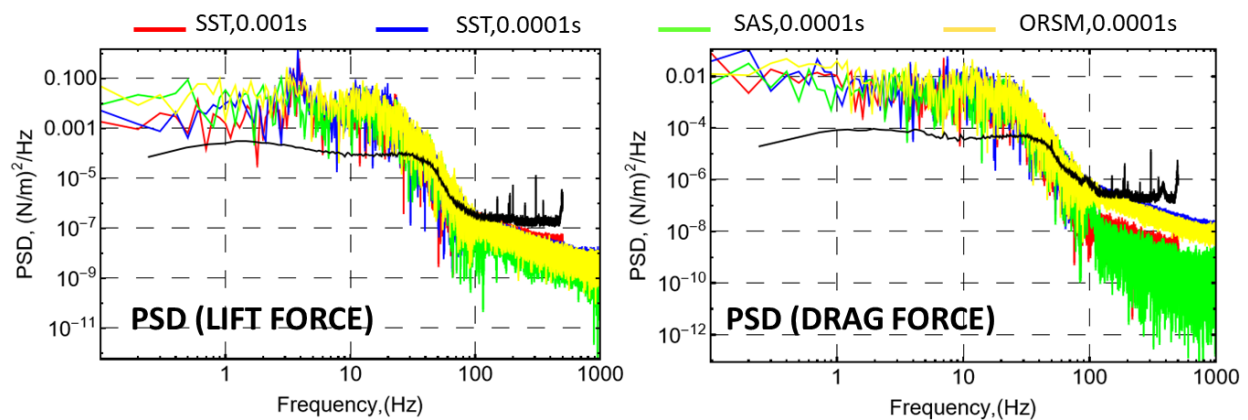


Figure 65: Comparison of resulting APSD of lift force (left) and drag force (right) with experimental APSDs for the rigid tube bundle with $Q=1.7$ l/s

Table 11: Comparison of RMS of forces obtained in calculations for rigid tube bundle ($Q=1.7$ l/s) with the measured values

	Experiment	SST, 1.0ms	SST, 0.1ms	SAS, 0.1ms	ORSM, 0.1ms
RMS Lift Force, [N/m]	0.066	0.067	0.061	0.043	0.057
RMS Drag Force, [N/m]	0.048	0.036	0.133	0.131	0.134

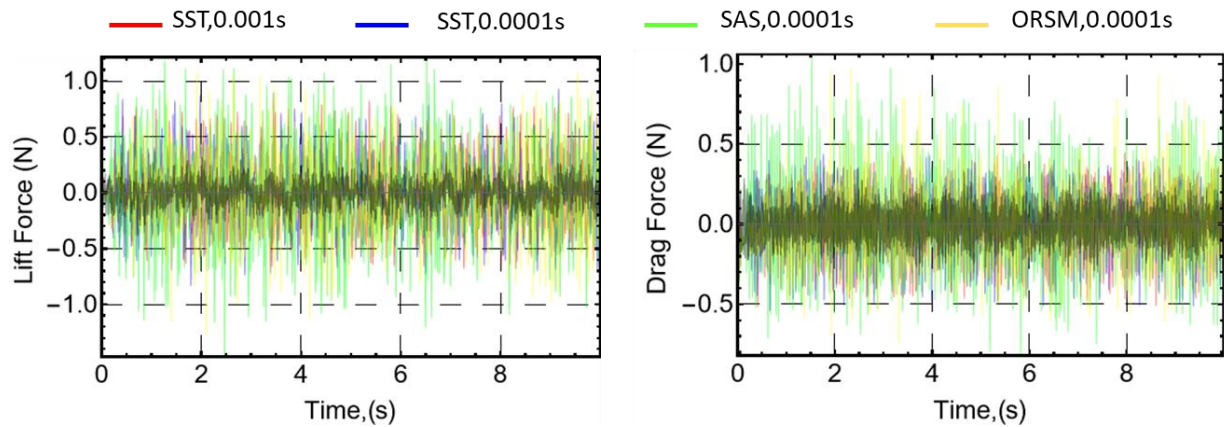


Figure 66: Comparison of the simulated lift force (left) and drag force (right) in time with experimental forces measured for the rigid tube bundle with $Q=2.9$ l/s

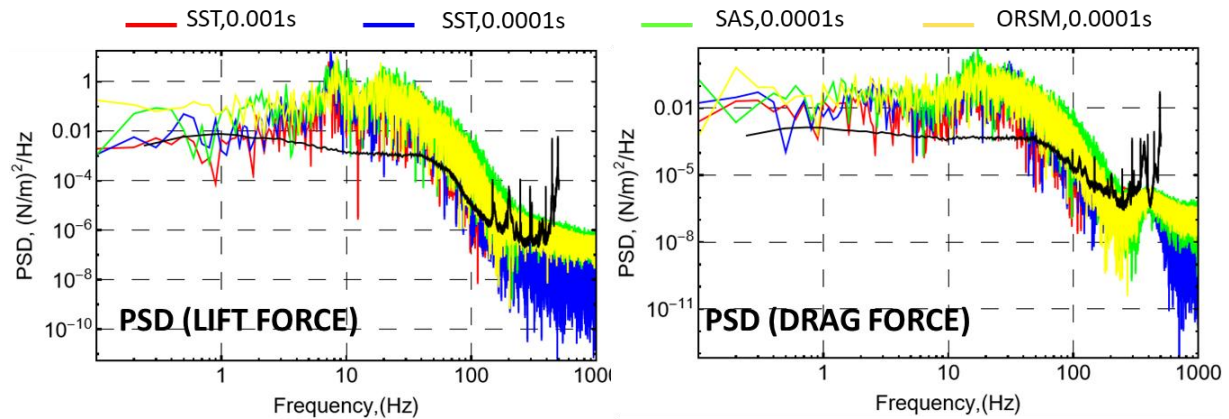


Figure 67: Comparison of resulting APSD of lift force (left) and drag force (right) with experimental APSDs for the rigid tube bundle with $Q=2.9$ l/s

Table 12: Comparison of RMS of forces obtained in calculations for rigid tube bundle ($Q=2.9$ l/s) with the measured values

	Experiment	SST, 1.0ms	SST, 0.1ms	SAS, 0.1ms	ORSM, 0.1ms
RMS Lift Force, [N/m]	0.291	0.283	0.282	0.405	0.328
RMS Drag Force, [N/m]	0.179	0.239	0.276	0.343	0.296

Figure 68 compares the calculated and measured streamwise velocity pictures obtained in both analyzed Rigid Tube Bundle cases. For this purpose, the calculated results were illustrated with the same legend bounds and color map as it was visualized for the measured picture. It can be seen that very good agreement in the formation of high-velocity jets behind the bundle is observed in the 1.7 l/s case, whereas in the 2.9 l/s case the development of these jets differs slightly from the

measured. The reason for this situation may be the significantly reduced simulation time - 10.0 seconds compared to 400 seconds of measurements.

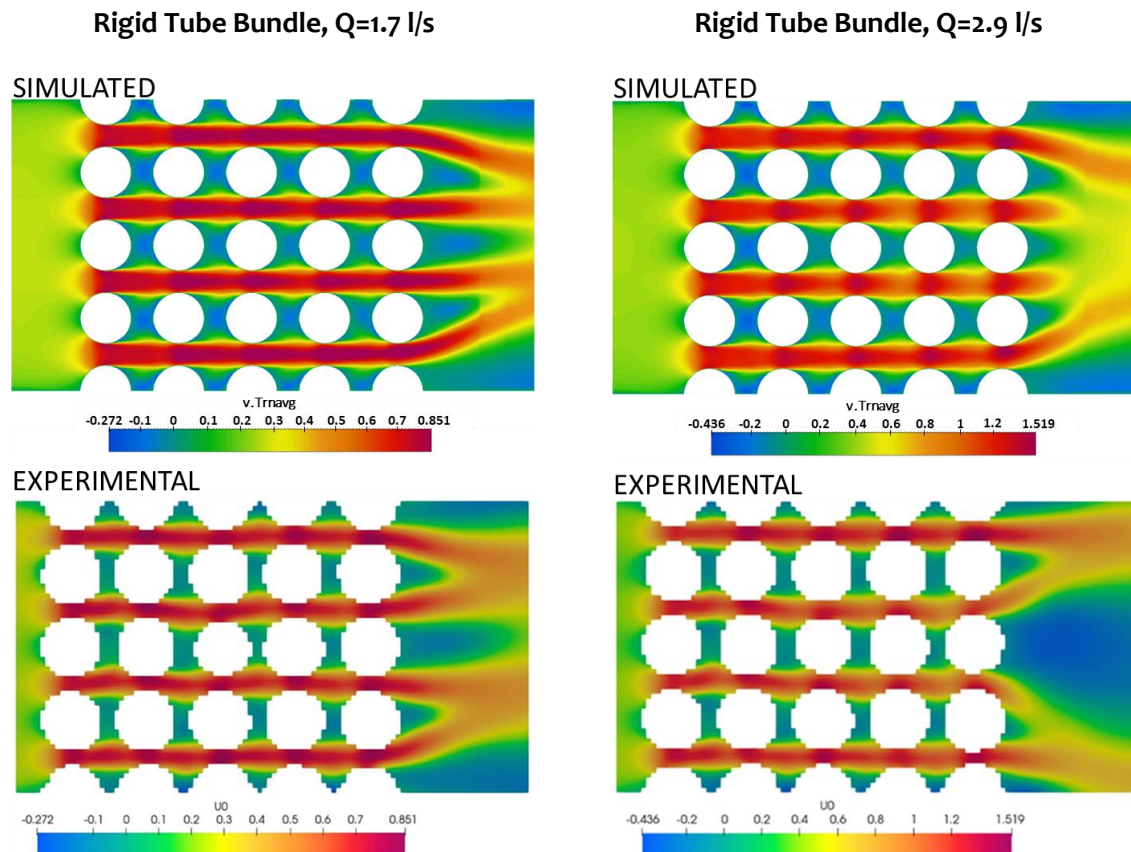


Figure 68: Comparison of resulting time-averaged streamwise velocity for RTB-170 (left) and RTB-290 (right) with the measured in experiment (bottom left/right)

4.2.2 Analysis of AMOVI cases with flexible Rod

This section demonstrates the results of simulations performed for configurations with the Single Flexible Tube and Flexible Tube in a Rigid tube bundle. For each of these configurations, four cases from the AMOVI benchmark were considered, with flow rates ranging from 0.5 to 4.0 l/s for SFT and from 0.5 to 1.95 l/s for FTRTB. Vibration analysis was performed using either the custom beam solver functions implemented in CFX or the built-in Rigid Body Solution. In some cases, both approaches were used. In custom Beam Solver Libraries two different settings were applied – the first named EBNF is based on the analytical solutions described in section 4.1.2, and the second – FDMK, is based on the Finite Difference Method application for Beam solution and described in section 3.2 of D5.2 [26]. While performing simulations using User Beam Solvers, the SST turbulence model with 1.0ms timestep was applied. In the case of Rigid Body Solution application, different turbulence models were used, starting from SST and SAS and then including Omega-RSM for FTRTB cases with the 1.0 ms/0.1-0.2 ms timesteps. For each analyzed case, the displacements over time, as well as the corresponding APSD, were compared with the experimental data.

4.2.2.1 Single flexible tube - SFT-50 (Q=0.5 l/s)

The Single flexible tube case with a flow rate of Q=0.5 l/s was analyzed using two options for the user beam subroutine implemented in ANSYS CFX: EBNF and FDMK. Rigid body solution application

was tested as well, but the resulting displacements were overestimated by more than 100 times compared to experimental values. Therefore, the Rigid Body Solution was unsuccessful in this case.

Figure 69 (left) demonstrates the simulated and the experimental tube displacements over 6s. It can be seen that the amplitude of displacements is also overestimated during simulations with the User Beam Subroutine compared to the experimental data, but not as much as with the Rigid Body Solution. The root mean square (RMS) was calculated for the available 6 seconds of simulation results and compared with the experimental value. Simulation with the EBNF option predicts an RMS value of 0.014 mm and FDMK of 0.012 mm (a little closer to the EXP value), while the experimental RMS value is 0.002 mm. The experimental RMS was determined over the entire 400 s time range. The APSD was calculated based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 69 (right) compares the calculated APSD with the experimental one. From the presented comparison, it could be noted that the first peak present in the experimental spectrum (3.17 Hz) was captured during the simulations with sufficiently good accuracy – 2.86 Hz, but with a higher amplitude.

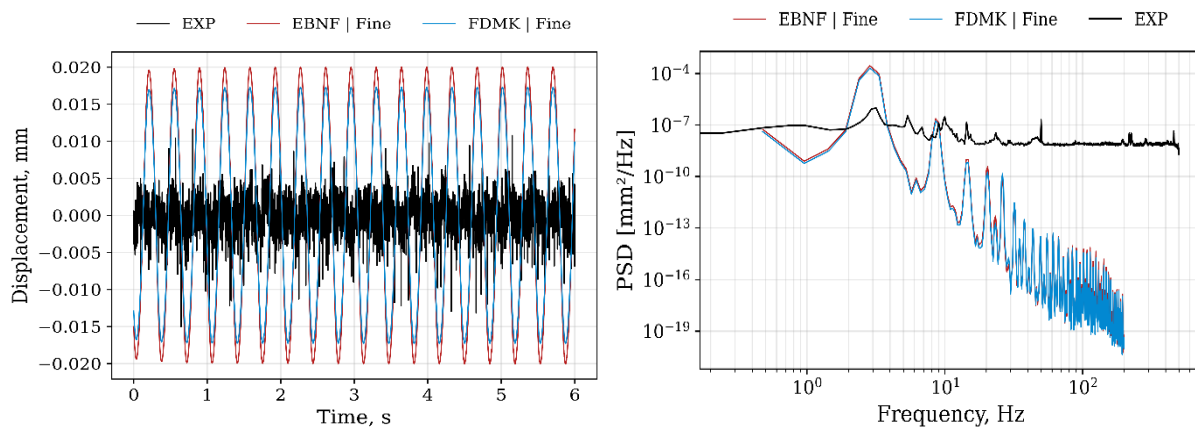


Figure 69: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=0.5$ l/s

Table 13: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube ($Q=0.5$ l/s)

	Experiment	EBNF	FDMK
Displacement RMS, mm	0.002	0.014	0.012
Max. displacement, mm	0.012	0.02	0.017
Peak#1, Hz	3.17	2.86	2.86

4.2.2.2 Single flexible tube - SFT-170 ($Q=1.7$ l/s)

For simulations of the Single flexible tube case with a flow rate of $Q=1.7$ l/s, both the Rigid Body Solution and the User Beam Library were applied. Two options of the user beam subroutine, implemented in ANSYS CFX - EBNF and FDMK were tested.

Figure 70 (left) demonstrates comparison of the simulated with Rigid Body Solution and the experimental tube displacements over 4s. The simulations with a refined mesh overestimate the

displacement amplitude relative to the experimental data, whereas a coarser mesh captures it well. The root mean square (RMS) was calculated for the available 4 seconds of simulation results and compared with the experimental value. Simulation with a refined mesh predicts an RMS value of 0.179 mm, and with a coarser mesh – 0.019 mm, while the experimental RMS is 0.028 mm. The experimental RMS was determined over the entire 400 s time range. The APSD was calculated based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 70 (right) compares the APSD calculated based on RB application with the experimental one. From the presented comparison, it can be noted that the first peak in the experimental spectrum (9.03 Hz) was captured only during simulations with a refined mesh, with sufficiently good accuracy (10.05 Hz), but with a higher amplitude. The second peak captured in the simulation with the refined mesh is 20.01 Hz, while the experimental one is 22.4 Hz. In the case of a coarser mesh, the APSD prediction is poor – captured peaks (especially the first peak) are shifted significantly to higher frequencies, while their amplitudes are significantly lower. The second peak predicted with the good agreement in times of frequency – 24.1 Hz, but the amplitude is significantly lower as well.

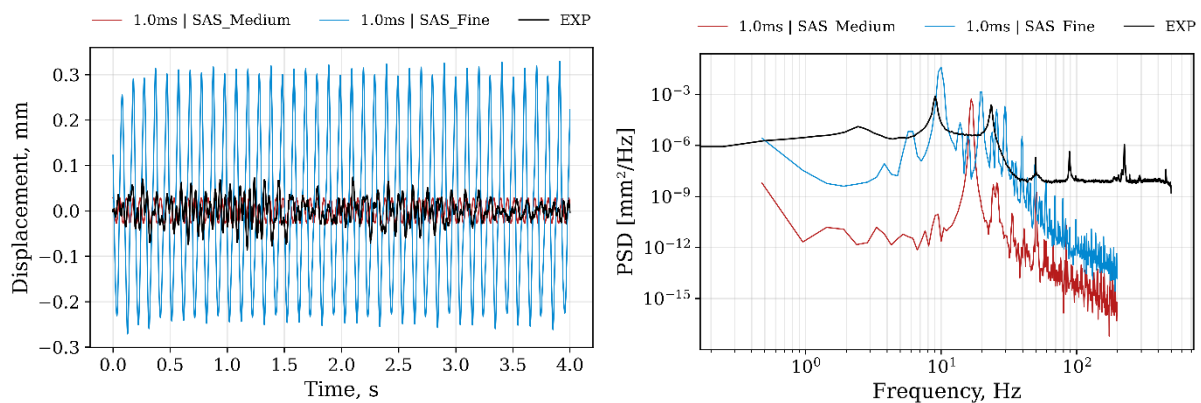


Figure 70: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=1.7$ l/s (Rigid Body approach)

Application of the User Beam Library with the EBNF and FDMK Beam options yielded results similar to those obtained with the Rigid Body Solution and a refined mesh. The amplitudes of the simulated displacements in both cases reach ~0.3mm.

Figure 71 (left) demonstrates the simulated (applying the User Beam Library) and the experimental tube displacements over 7s. It can be seen that the amplitude of the displacements is also overestimated during simulations with the User Beam Subroutine compared to the experimental data. The root mean square (RMS) was calculated for the available 7 seconds of simulation results and compared with the experimental value. Simulation with the EBNF option predicts an RMS value of 0.21 mm and FDMK – 0.18 mm (a little closer to the EXP value), while the experimental value of RMS is 0.028 mm. The experimental RMS was determined over the entire 400 s time range. The APSD was calculated based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 71 (right) compares the calculated APSD with the experimental one. From the comparison, it can be seen that the first peak in the experimental spectrum (9.03 Hz) is captured in the simulations and is equal to 10.07 Hz for both EBNF and FDMK options. The amplitude of the first peak is higher than the experimental one. The second peak captured in simulation with both beam options is 28.02 Hz while the experimental one is – 22.4 Hz, indicating that the captured second peak is shifted to higher frequencies, while the amplitudes are predicted well.

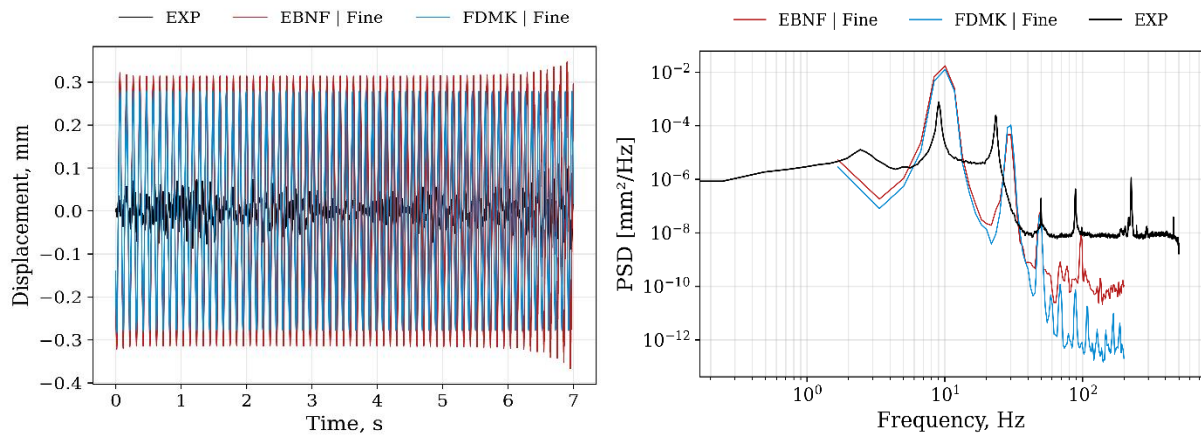


Figure 71: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=1.7$ l/s (User Beam lib)

Table 14: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube ($Q=1.7$ l/s)

	Experiment	RB-SAS refined	RB-SAS coarse	EBNF	FDMK
Displacement RMS, mm	0.028	0.179	0.019	0.21	0.18
Max. displacement, mm	0.08	0.31	0.05	0.34	0.28
Peak#1, Hz	9.03	10.05	16.69	10.07	10.07
Peak#2, Hz	22.4	20.01	24.1	28.02	28.02

4.2.2.3 Single flexible tube - SFT-290 ($Q=2.9$ l/s)

For simulations of the Single flexible tube case with a flow rate of 2.9 l/s, both the Rigid Body Solution and the User Beam Library were applied. Two options of the user beam subroutine, implemented in ANSYS CFX - EBNF and FDMK were tested.

Figure 72 (left) and Figure 73 (left) demonstrate the comparison of the simulated with Rigid Body Solution and the experimental tube displacements over 4s. First, simulations were performed to test the predictive capabilities of SST and SAS turbulence models. For simulations with the SAS turbulence model, both 1.0ms and 0.1 ms timesteps were used. For these test calculations, the first LowRe mesh was used. Results showed a good to moderate prediction of displacement amplitudes when applying the SST and SAS models with a 1.0ms timestep (displacement amplitudes were slightly overestimated), but the calculated APSD showed poor prediction and captured only the first peak (15.13 Hz). The 0.1ms timestep, applied with the SAS turbulence model, gave us almost exact prediction of the calculated APSD, while displacement amplitude increased (see Figure 72).

Then, further calculations on ICEM (refined) mesh were performed with SAS turbulence model and 0.1ms/1.0ms timesteps. It could be seen that the displacement amplitude was overestimated in both simulations as well. The root mean square (RMS) was calculated for the available 4 seconds of simulation results and compared with the experimental value. Simulation with a refined mesh on the SAS model with a 1.0ms timestep predicts an RMS value of 0.757 mm, and with a 0.1ms timestep – 0.794 mm, while the experimental RMS value is 0.205 mm. The experimental RMS was

determined over the entire 400 s time range. The APSD was calculated based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 73 (right) compares the calculated APSD for these simulations with the experimental one. From the comparison, it can be noted that both peaks are captured in simulations on the ICEM mesh – 14.74/14.76 Hz for the first peak and 25.74/26.25 Hz for the second peak, while the experimental peaks are 15.14 Hz and 22.7 Hz.

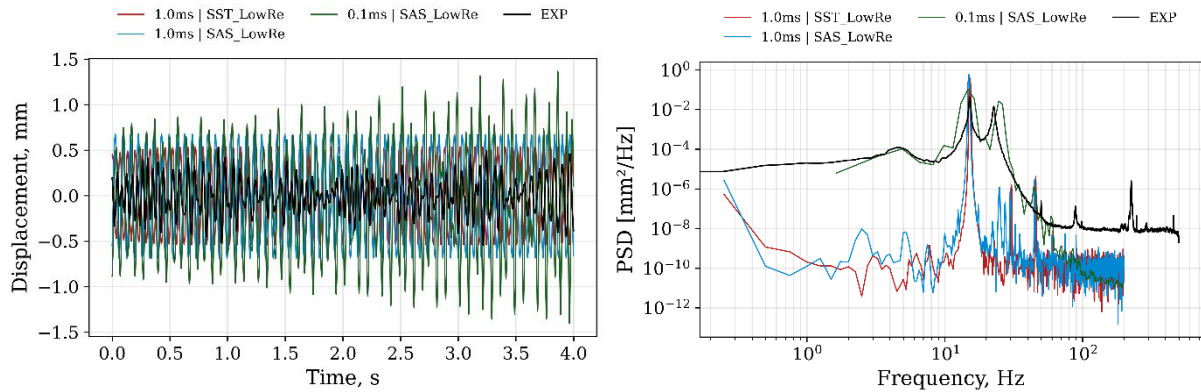


Figure 72: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=2.9$ l/s (Rigid Body approach; LowRe mesh)

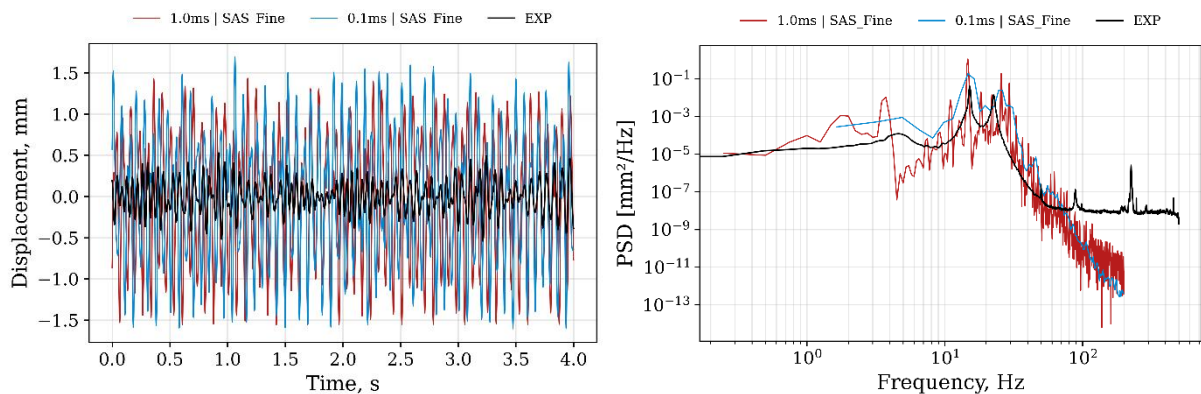


Figure 73: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=2.9$ l/s (Rigid Body approach; ICEM mesh)

The application of the User Beam Library with the EBNF and FDMK Beam options yielded results that are partly similar to those obtained with the Rigid Body Solution and a refined mesh. The amplitudes of the simulated displacements in both cases reach ~1.5mm.

Figure 74 (left) demonstrates the simulated (applying the User Beam Library) and the experimental tube displacements over 5s. It can be seen that the amplitude of the displacements is also overestimated during simulations with the User Beam Subroutine compared to the experimental data. The root mean square (RMS) was calculated for the available 5 seconds of simulation results and compared with the experimental value. Simulation with the EBNF option predicts an RMS value of 0.821 mm and FDMK of 0.658 mm (a little closer to the EXP value, but still overestimated by a lot), while the experimental RMS is 0.205 mm. The experimental RMS was determined over the entire 400 s time range. The APSD was calculated based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 74 (right) (right) compares the

calculated APSD with the experimental one. From the comparison presented, it can be seen that the prediction accuracy is even better than in the last Rigid Body applications. The first peak in the experimental spectrum (15.14 Hz) is captured during simulations and is equal to 15.1 and 16.78 Hz for the EBNF and FDMK options, respectively. The second peak captured in simulation is 21.8 Hz for the EBNF option and 21.4 Hz for the FDMK solution, while the experimental one is 22.7 Hz, indicating that the obtained APSD provides better predictions than the Rigid Body Solution results.

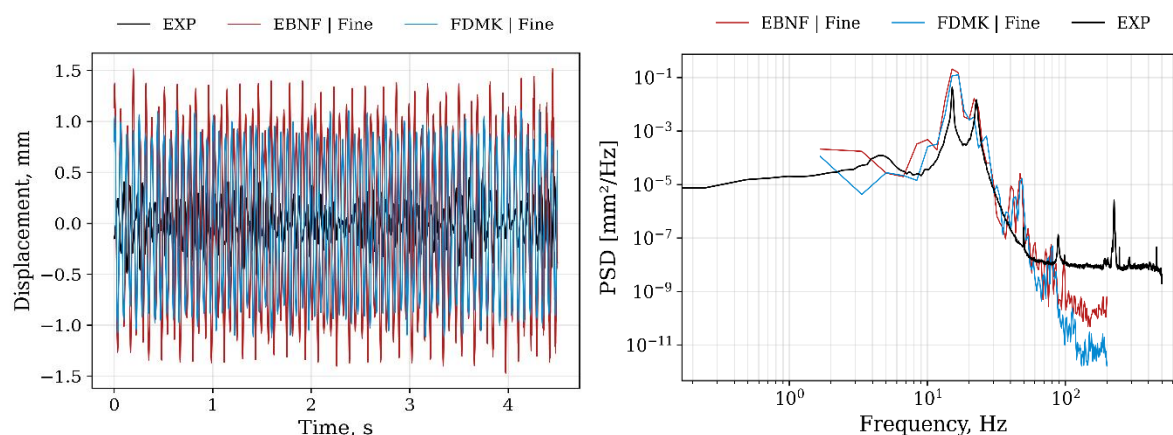


Figure 74: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=2.9$ l/s (User Beam lib; ICEM mesh)

Table 15: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube ($Q=2.9$ l/s)

	Experiment	RB-SAS, 1.0ms	RB-SAS, 0.1 ms	EBNF	FDMK
Displacement RMS, mm	0.205	0.757	0.794	0.821	0.658
Max. displacement, mm	0.510	1.480	1.720	1.501	1.151
Peak#1, Hz	15.14	14.74	14.76	15.1	16.78
Peak#2, Hz	22.7	25.74	26.25	21.8	21.4

4.2.2.4 Single flexible tube - SFT-400 ($Q=4.0$ l/s)

For simulations of the Single flexible tube case with a flow rate of 4.0 l/s, both the Rigid Body Solution and the User Beam Library were applied. Two options of user beam subroutine, implemented in ANSYS CFX - EBNF and FDMK were tested.

Figure 75 (left) and Figure 76 (left) demonstrate a comparison of the simulated with Rigid Body Solution and the experimental tube displacements over ~4-5s. First, simulations were performed to test the predictive capabilities of SST and SAS turbulence models. For both simulations, the 1.0ms timestep was applied. For these test calculations, the first LowRe mesh was used. Results showed a good to moderate prediction of displacement amplitudes when applying the SST and SAS models with a 1.0ms timestep (displacement amplitudes were even slightly underestimated). The root mean square (RMS) was calculated for the available 5 seconds of simulation results and compared with the experimental value. Simulation with the LowRe mesh on the SST model with a 1.0ms timestep predicts an RMS value of 1.91 mm, and on the SAS model with a 1.0ms timestep – 1.69 mm,

while the experimental RMS is 1.945 mm. The experimental RMS was determined for the entire time range of 400 s.

The calculated APSD also demonstrated a sufficiently good prediction, especially when applying the SAS model, which captured the first peak at 20.99 Hz, while the experimental one is 19.53 Hz. The 0.1ms timestep, applied with the SAS turbulence model on both the LowRe and refined ICEM meshes, resulted in a significant overestimation in displacements (see Figure 76). Calculated for those results, APSD also has the worst picture – it is shifted toward the higher-frequency region.

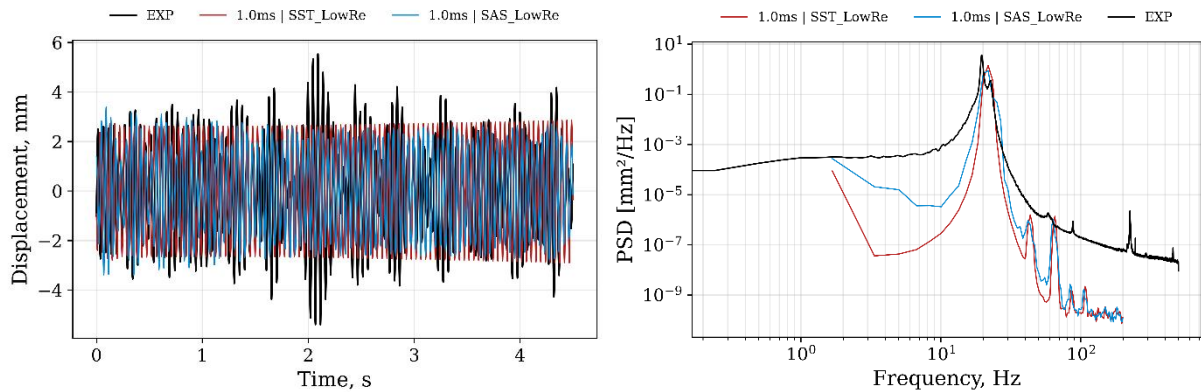


Figure 75: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=4.0$ l/s (Rigid Body approach; LowRe mesh)

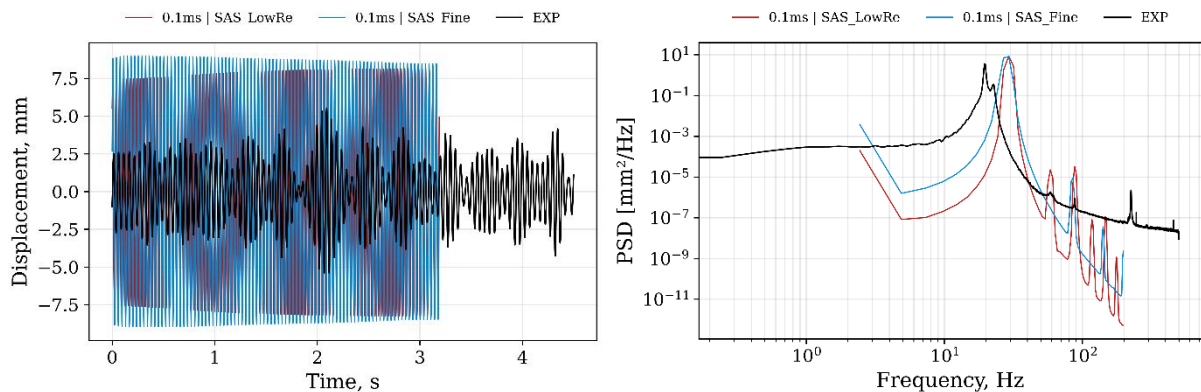


Figure 76: Comparison of tube displacements (left) and corresponding APSDs (right) for the single flexible tube with $Q=4.0$ l/s (Rigid Body approach; ICEM mesh)

Figure 76 (left) demonstrates the simulated (applying the User Beam Library) and the experimental tube displacements over 9s. It can be seen that the amplitude of displacements is significantly overestimated during simulations with the User Beam Subroutine and EBNF beam solution compared to the experimental data, which is very similar to the result of the Rigid Body Solution application with a 0.1ms timestep. It should be noted that this calculation was interrupted due to an error caused by high rod displacements. In the case of the FDMK option, the displacement amplitudes were predicted with very good agreement with the experimental data. The root mean square (RMS) was calculated for the available ~9 seconds of simulation results for FDMK and compared with the experimental value. The FDMK simulation predicts an RMS value of 2.056 mm, while the experimental RMS is 1.945 mm. The experimental RMS was determined over the entire 400 s time range. The APSD was calculated based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 77 (right) compares the calculated APSD

with the experimental one. From the presented comparison, it can be noted that the prediction accuracy for the calculation with the FDMK beam solver is almost the same as for the Rigid Body application with the SAS model and a 1.0ms timestep. The first peak in the experimental spectrum (19.53 Hz) is captured during simulations and is equal to 21.8 Hz for the FDMK option. The second peak captured in simulation is 30.2 Hz for the FDMK option, which is shifted to a higher frequency than the experimental one (22.7 Hz).

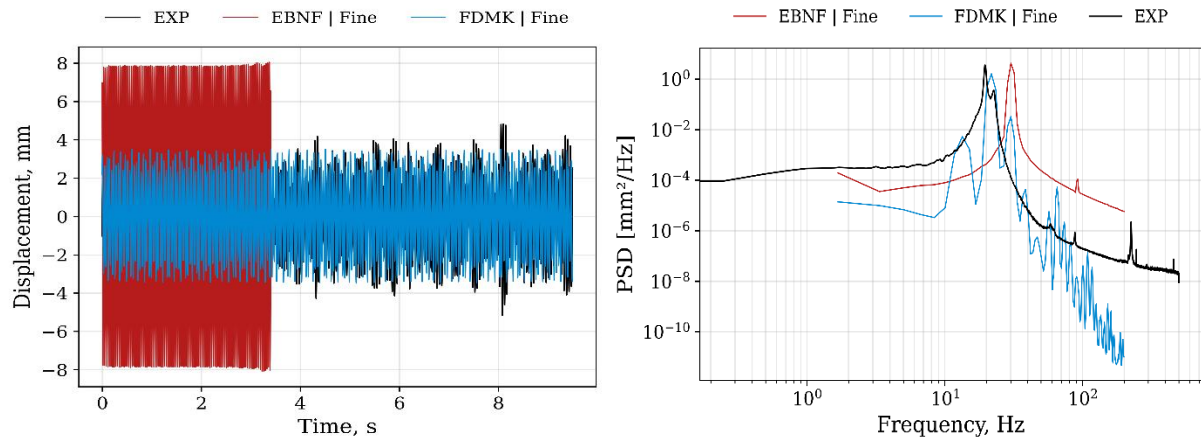


Figure 77: Comparison of tube displacements (left) and appropriate APSDs (right) for the single flexible tube with $Q=4.0$ l/s (User Beam lib; ICEM mesh)

Table 16: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube ($Q=4.0$ l/s)

	Experiment	RB-SST 1.0ms	RB-SAS 1.0ms	EBNF	FDMK
Displacement RMS, mm	1.945	1.91	1.69	3.335	2.056
Max. displacement, mm	5.2	2.7	3.4	8.0	3.85
Peak#1, Hz	19.53	21.72	20.99	30.2	21.8
Peak#2, Hz	22.7	-	-	-	30.2

4.2.2.5 Flexible tube in a rigid tube bundle - FTRTB-50 ($Q=0.5$ l/s)

The calculation case of a flexible tube in a rigid tube bundle with a flow rate of 0.5 l/s was analyzed using the Rigid body solution. This case proved difficult to simulate due to the low flow forces. Accordingly, rod vibrations in this case when applying SST and SAS turbulence models were not induced. Only the Omega-RSM turbulence model application with a 0.2ms timestep resulted in flexible rod vibrations.

Figure 78 (left) demonstrates the simulated and the experimental tube displacements over 7s. It can be seen that the amplitude of the displacements is also underestimated in simulations compared to the experimental data. The root mean square (RMS) was calculated for the available 7 seconds of simulation results and compared with the experimental value. Simulation predicts an RMS value of 0.00046 mm, while the experimental value of RMS is 0.003 mm. It should be noted that the experimental RMS was determined over the entire 400 s time range. Nevertheless, the amplitudes of displacement obtained in simulation are negligibly small. The APSD was calculated

based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 78 (right) compares the calculated APSD with the experimental one. From the presented comparison, it could be noted that the peak present in the experimental spectrum (~ 23.7 Hz) captured during the simulations with sufficiently good accuracy – 24.5 Hz, but with a lower amplitude.

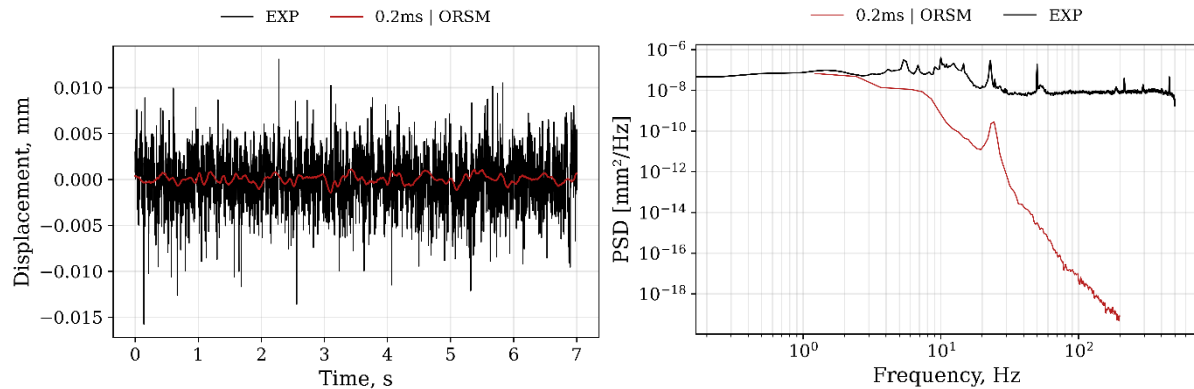


Figure 78: Comparison of tube displacements (left) and appropriate APSDs (right) for the flexible tube in a rigid tube bundle with $Q=0.5$ l/s (Rigid Body approach)

Table 17: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle ($Q=0.5$ l/s)

	Experiment	RB-ORSM
Displacement RMS, mm	0.003	0.000457
Max. displacement, mm	0.016	0.002
Peak#1, Hz	23.7	24.5

4.2.2.6 Flexible tube in a rigid tube bundle - FTRTB-110 ($Q=1.1$ l/s)

The calculation case of a flexible tube in a rigid tube bundle with a flow rate of $Q=1.1$ l/s was analyzed using the Rigid body solution. This case, as well as the previous one, proved difficult to simulate. The Omega-RSM turbulence model with the 0.2ms timestep was also applied to obtain the vibrations of the central rod.

Figure 79 (left) demonstrates the simulated and the experimental tube displacements over 8.5 s. It can be seen that the displacements are reproduced quite well during the first 6 seconds, after which the vibration amplitude became higher than the experimental one. The root mean square (RMS) was calculated for the first 6 seconds and then for the available 8.5 seconds of simulation results and compared with the experimental value. Simulation predicts an RMS value of 0.052 mm/0.108mm, while the experimental value of RMS is 0.038 mm. It should be noted that the experimental RMS was determined over the entire 400 s time range. The APSD was calculated based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 79 (right) compares the calculated APSD with the experimental one. From the presented comparison, it could be noted that the peak present in the experimental spectrum (~ 23.19 Hz) captured during the simulations very well – 23.189 Hz, with good amplitude prediction.

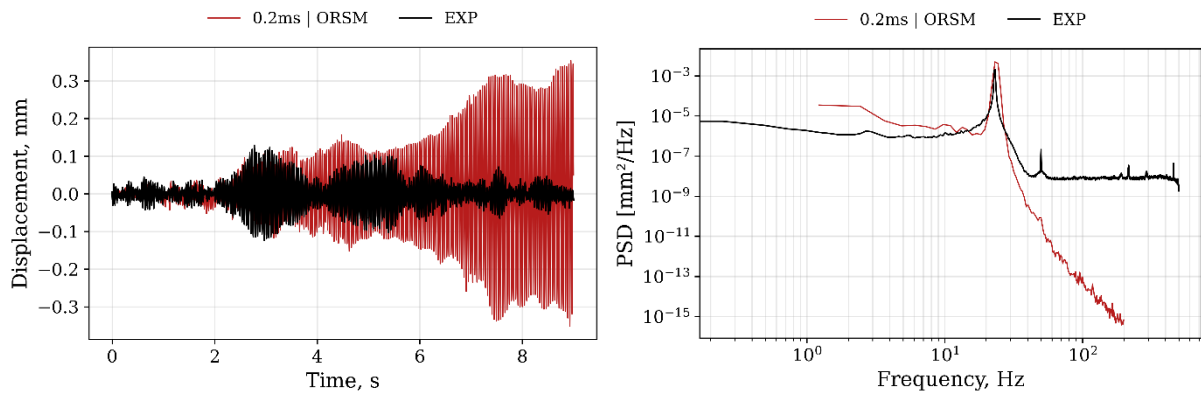


Figure 79: Comparison of tube displacements (left) and appropriate APSDs (right) for the flexible tube in a rigid tube bundle with $Q=1.1$ l/s (Rigid Body approach)

Table 18: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle ($Q=1.1$ l/s)

	Experiment	RB-ORSM
Displacement RMS, mm	0.038	0.052/0.108
Max. displacement, mm	0.125	0.155/0.35
Peak#1, Hz	23.19	23.189

4.2.2.7 Flexible tube in a rigid tube bundle - FTRTB-160 ($Q=1.6$ l/s)

The calculation case of a flexible tube in a rigid tube bundle with a flow rate of 1.6 l/s was analyzed using the Rigid body solution. The SST and Omega-RSM turbulence models with the 0.2ms timestep were applied to simulate the vibrations of the central rod.

Figure 80 (left) demonstrates the simulated and the experimental tube displacements over 4.0 s. It can be seen that the displacements are reproduced quite well, especially when using the Omega-RSM turbulence model. The root mean square (RMS) was calculated for the available 4.0 seconds of simulation results and compared with the experimental value. Simulation predicts an RMS value of 0.119 mm for SST and 0.069mm for Omega-RSM, while the experimental value of RMS is 0.073 mm. It should be noted that the experimental RMS was determined over the entire 400 s time range. The APSD was calculated based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 80 (right) compares the calculated APSD with the experimental one. From the presented comparison, it could be noted that the peak present in the experimental spectrum (~ 22.46 Hz) captured during the simulations very well – 22.74/22.59 Hz, with good amplitude prediction.

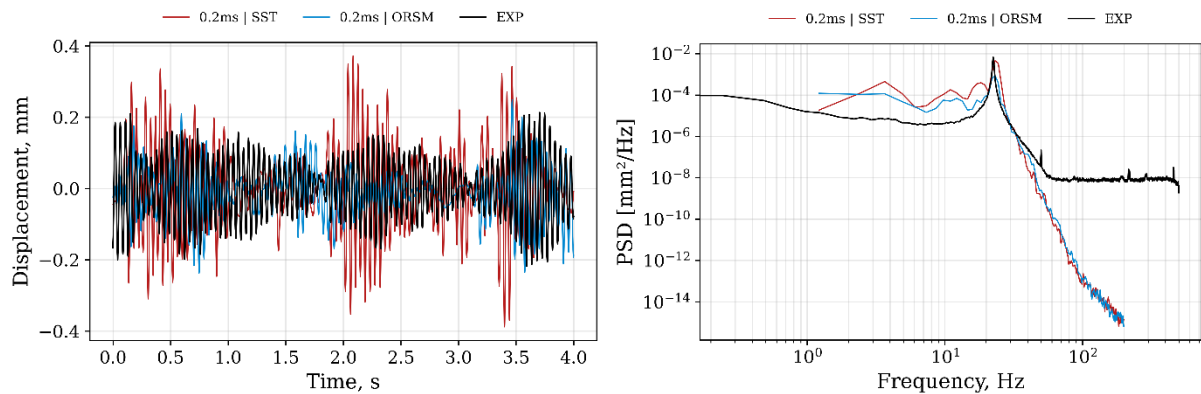


Figure 80: Comparison of tube displacements (left) and appropriate APSDs (right) for the flexible tube in a rigid tube bundle with $Q=1.6$ l/s (Rigid Body approach)

Table 19: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle ($Q=1.6$ l/s)

	Experiment	RB-SST	RB-ORSM
Displacement RMS, mm	0.073	0.119	0.069
Max. displacement, mm	0.22	0.397	0.22
Peak#1, Hz	22.46	22.74	22.59

4.2.2.8 Flexible tube in a rigid tube bundle - FTRTB-195 ($Q=1.95$ l/s)

The calculation case of a flexible tube in a rigid tube bundle with a flow rate of 1.95 l/s was analyzed using the Rigid body solution. The SAS and Omega-RSM turbulence models with the 0.5ms timestep were applied to simulate the vibrations of the central rod.

Figure 81 (left) demonstrates the simulated and the experimental tube displacements over 4.0 s. It can be seen that the displacements are underestimated when using both the SAS and Omega-RSM turbulence models. The root mean square (RMS) was calculated for the available 4.0 seconds of simulation results and compared with the experimental value. The simulation predicts an RMS value of 0.34 mm for SAS and 0.56mm for Omega-RSM, while the experimental RMS value is 3.223 mm. It should be noted that the experimental RMS was determined over the entire 400 s time range. The APSD was calculated based on simulated displacements using the matplotlib function with the Welch's average periodogram. Figure 81 (right) compares the calculated APSD with the experimental one. From the presented comparison, it could be noted that the peak present in the experimental spectrum (~ 23.68 Hz) captured during the simulations very well – 23.43/23.92 Hz.

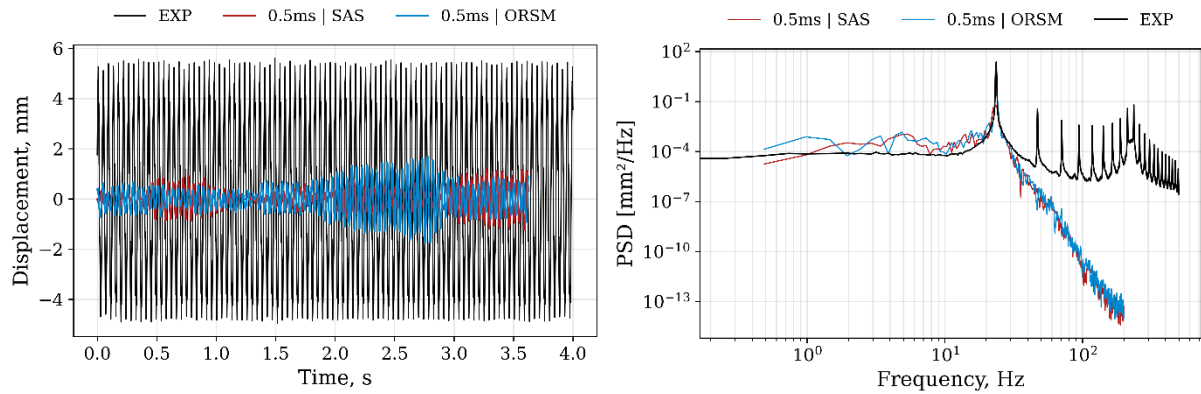


Figure 81: Comparison of tube displacements (left) and appropriate APSDs (right) for the flexible tube in a rigid tube bundle with $Q=1.95$ l/s (Rigid Body approach)

Table 20: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle ($Q=1.95$ l/s)

	Experiment	RB-SAS	RB-ORSM
Displacement RMS, mm	3.223	0.34	0.56
Max. displacement, mm	5.3	1.01	1.79
Peak#1, Hz	23.68	23.43	23.92

4.3 Conclusions

In the sections above, IPP described the models and approaches used to simulate flow induced vibrations of the rods. The main focus on developing relatively simple 2D models, mostly involving SST and SAS turbulence models. On the mechanical modelling side, both the Rigid Body Solution (available in ANSYS CFX, which was used for simulations) and a User library with implemented solvers for the Beam were applied. A quasi-two-dimensional formulation is considered within the URANS approach using a standard oscillator (rigid body), as well as a user library with a simple Euler-Bernoulli beam. The library includes several solvers: FDM (with a description of the time derivative in the form of kinematic quantities), as well as analytical solvers based on the normal mode method with a predetermined frequency spectrum (with and without the added effective mass of the fluid). The AMOVI experimental cases for the rigid tube configurations, as well as for the single flexible tube and the flexible tube in a rigid tube bundle, were simulated using the developed models.

The use of a 2D approximation has a number of advantages in terms of computational resources, especially when it comes to taking into account the vibration of each beam element in the cross-section. However, at least one additional parameter arises related to the selection of a reference point on the beam/rigid body for transferring displacements to the computational mesh. In terms of calculation settings, this adds flexibility (especially for engineering practice), but for modeling the physical processes of the generation, organization, and separation of a vortex structure (which determines the hydrodynamic forces), such an approximation may be insufficient. This is especially important when dealing with the transitional turbulent flow region around a single rod or rod bundle, which has a fundamentally three-dimensional structure. In this case, the oscillations are

determined by the complex wave structure in the boundary layer, the force impact of which on the moving wall is insignificant. In a sense, this problem can be "refined" by using more complex turbulence models such as SAS-SST or Omega-RSM, but this is the exception rather than the rule.

The use of the standard rigid body model is in some sense sufficient for predicting the average amplitude and frequency of oscillations, but has certain limitations in terms of the transition to the resonant region, since the natural oscillations are determined by only one frequency.

The use of user libraries for more complex oscillators (in this case, two-piece and equivalent one-piece beams) is generally advantageous, as it allows for better prediction of the fundamental characteristics of coupled oscillations, as well as the transition to resonance (Figure 82). In a sense, the use of solvers specifically for the differential formulation of oscillators is also preferable (if the solvers are non-dissipative) since it does not impose any conditions or restrictions on the oscillation spectrum.

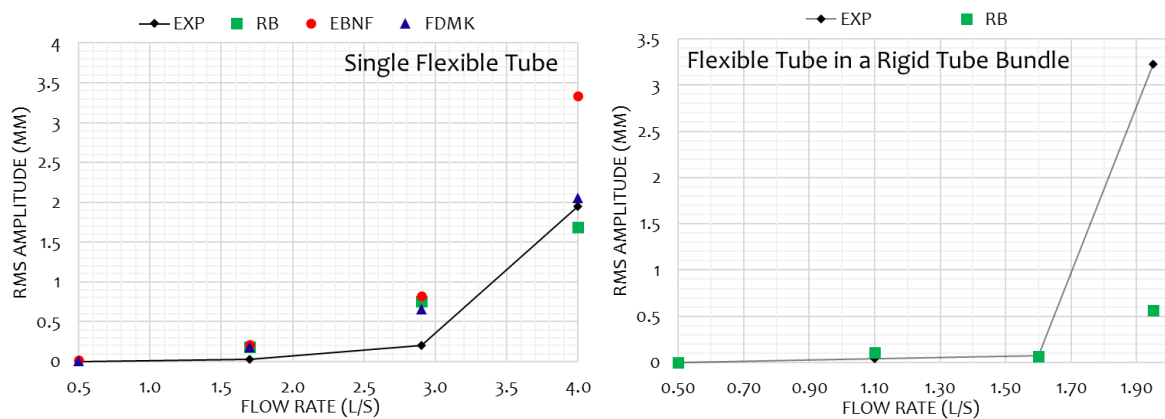


Figure 82: Comparison of simulated by IPP and experimental values of RMS value of the displacement as a function of the flow rate

5 Simulations performed by EDF

The open source CFD solver code `_saturne` v8.0 (www.code-saturne.org) developed by EDF has been utilized using massive parallel computing. Both supercomputers Cronos and Selena have been used. All the meshes are fully hexahedral and conforming and have been generated with Gmsh v4.7.1. All the walls are fully resolved and the inlet boundary conditions are fully developed, using a wall-resolved embedded LES, often called a precursor computation.

An equation of rigid body motion is implemented by default in the code to move the cylinder. It is combined with an Arbitrary Lagrangian Eulerian (ALE) approach to ensure mesh deformation for Fluid Structure Interaction. Both the configurations of the flow around a single cylinder and through an in-line tube bundle are simulated at different Reynolds numbers. The same meshes are used, when it is possible for the smaller flow rates, with and without ALE. Coarsening the meshes has been mandatory to allow large displacements while using ALE.

In the following, $\nu = 10^{-6} \text{ m}^2/\text{s}$ and $\rho = 1000 \text{ kg/m}^3$ are the kinematic viscosity and the density of water in normal conditions.

Four consecutive sections are dedicated to wall-resolved LES (high fidelity) simulations on four AMOVI experimental configurations:

1. The flow around a single rigid cylinder,
2. The flow around a single moving/flexible cylinder,
3. The flow through a rigid 5x3.5 tube bundle,
4. The flow through a 5x3.5 tube bundle where all the cylinders are rigid except the central one, located at (3,2) where the first argument stands for the column number and the second one for the line number.

5.1 Single cylinder configuration computational domain and mesh generation

Figure 83 shows the computational domain and the boundary conditions to represent the single cylinder configuration of AMOVI experiment [15], [29]. The different lengths are given in Table 21 and the boundary conditions are indicated in Figure 83. All the walls utilize a no-slip boundary condition and the outlet is a free outlet. For more details about the precursor computation which feeds the computations with a fully developed flow, the reader can go to section 5.3.

The hydraulic diameter of the rectangular channel at the inlet is equal to $D_H = 2HL_z / (L_z + H) = 0.082 \text{ m}$. $L_{\text{prec}} = 0.171 \text{ m}$ is equal to $2.1 D_H$. A value of $L_{\text{prec}} / D_H = 3$ would have been preferred, but this first estimate was done with a hydraulic diameter using H_s and not H as the convergent was not represented in the preliminary computations. The results should not change that much with $L_{\text{prec}} = 3D$ as the inlet is far enough from the cylinder ($L_{\text{in},1} + L_{\text{in},2} + L_{s,\text{up}} = 21D = 3,1 D_H$).

Table 21: The lengths of the computational domains for the single cylinder configuration of AMOVI (see Figure 83).

D (m)	H _s (m)	H (m)	L _{in,1} (m)	L _{in,2} (m)	L _{s,up} (m)	L _{s,do} (m)	L _{out,1} (m)	L _{out,2} (m)	L _z (m)
0.01215	0.04	0.07	0.044	0.1535	0.0575	0.1375	0.1205	0.087	0.1

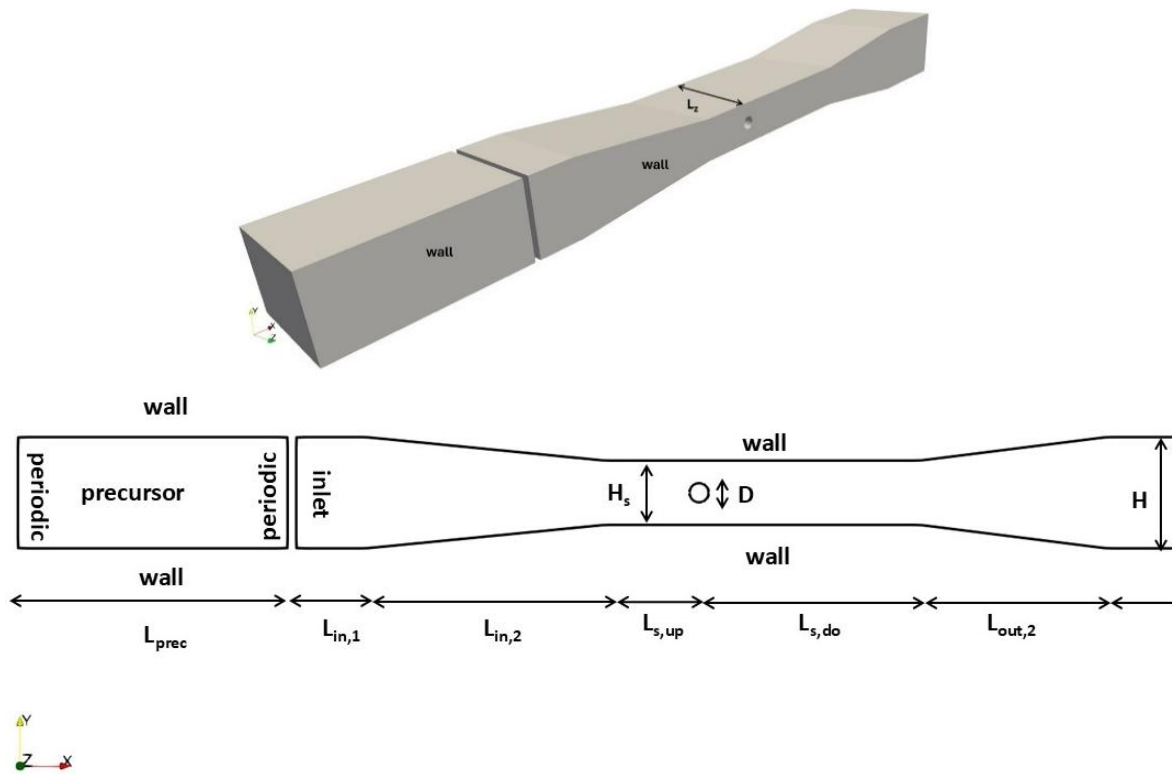


Figure 83: The computational domain and the boundary conditions for the single cylinder configuration of Amovi. The frame axes are the one used in this figure, the origine being at the center of the cylinder.

Three flowrates have been studied; the 4th one (4 l/s) leading to too fine meshes being unaffordable today. Table 22 gives the characteristic velocities and Reynolds numbers for the three flowrates. U_∞ and $U_{\infty 2}$ (resp. Re_H and Re_{H2}) are the bulk velocities (resp. hydraulic Reynolds numbers) in the narrow and large square ducts, respectively. Re_τ is the Reynolds number based on the friction velocity and the hydraulic radius $R_H = D_H/4$. One can see that for the lowest flow rate, the Reynolds number of the narrow channel is rather more characteristic of a transitional flow from the laminar to the turbulent regime than of a fully turbulent one.

Table 22: The first three flow rates of AMOVI experiment for the single cylinder configuration. Characteristic velocities and Reynolds numbers.

Q (l/s)	U_∞ (m/s)	Re_∞	Re_H	Re_τ	$U_{\infty 2}$ (m/s)	Re_{H2}
0.5	0.125	1519	7143	112	0.071	5882
1.7	0.425	5164	24286	336	0.243	20000
2.9	0.725	8809	41429	544	0.414	34118

Figure 84 shows the mesh around the cylinder and a portion of the end of the convergent section. The mesh is created using a Frontal-Delaunay approach and a Blossom algorithm to create 2D hexahedral elements (note that the Frontal-Delaunay for quads approach available in Gmsh has been used only for the tube bundles as the computations of this latter configuration came later).

The refinement details given herein are for so-called “reference meshes”; the finest mesh used for a given configuration in the single cylinder configurations. The first attempts to refine the configuration used the friction velocity of the narrow channel.

Δy_{cyl} and Δy_{side} are the refinements in the wall normal direction to the cylinders and toward the sidewall in Figure 84.

If U_∞ stands for the bulk velocity in the upstream channel (see Table 22) and $D_{HS}=2H_sL_z/(L_z+H_s)$ its hydraulic diameter, $Re_\infty = U_\infty D_{HS}/\nu$.

Δy_{in} is an *a priori* estimate of the required refinement at the inlet to obtain a non-dimensional normal width of 2 for the near-wall computational cells (note that this would give a wall computational cell center at a non-dimensional distance of 1 as this latter is located at the cell-center).

$$\Delta y_{in} = 2 \frac{\nu}{U_\infty} \sqrt{\frac{8}{C_f}} \text{ with } C_f = 0.184 Re_\infty^{-0.2}.$$

As there is an acceleration due to the cylinder, after several tests, the distances to the wall have been fixed to $\Delta y_{cyl} = \Delta y_{in}/8$ and $\Delta y_{side} = \Delta y_{in}/2$.

δ_{bl} and r_{bl} are the meshed boundary layer thickness and the progression ratio used for the refinement (see Figure 84). $r_{bl} = 1.05$ for the reference meshes. δ_{bl} is computed such that the refinement in the azimuthal direction of the boundary layer at its limit is equal to the larger cell size Δy_{out} in the wall normal direction after the progressive unrefinement (this is a condition to obtain homogeneous meshes using the Frontal-Delaunay and Blossom algorithms in Gmsh). Δy_{out} is actually the characteristic cell-size in 2D in the bulk region (see Figure 84).

In the z direction, the span-wise refinement is constant along a distance which allows a refinement towards the sidewalls with a progression ratio equal to 1.1 and a wall normal cell-size $\Delta y_{side,z} = \Delta y_{in}/5$. The azimuthal and the span-wise aspect ratios in the center-plane of the cylinder are equal to 20 and 15, respectively.

At the inlet and the outlet, the stream-wise refinement is constant up to the start or the end of the narrow rectangular channel with $\Delta x = 15\Delta y_{in}$ (see figure Figure 84).

Table 23 and Table 24 give the main parameter of the meshes used in the present report for the rigid and/or moving single cylinder configurations. The coarsest and finest meshes have around 50 million and 300 million computational cells, respectively.

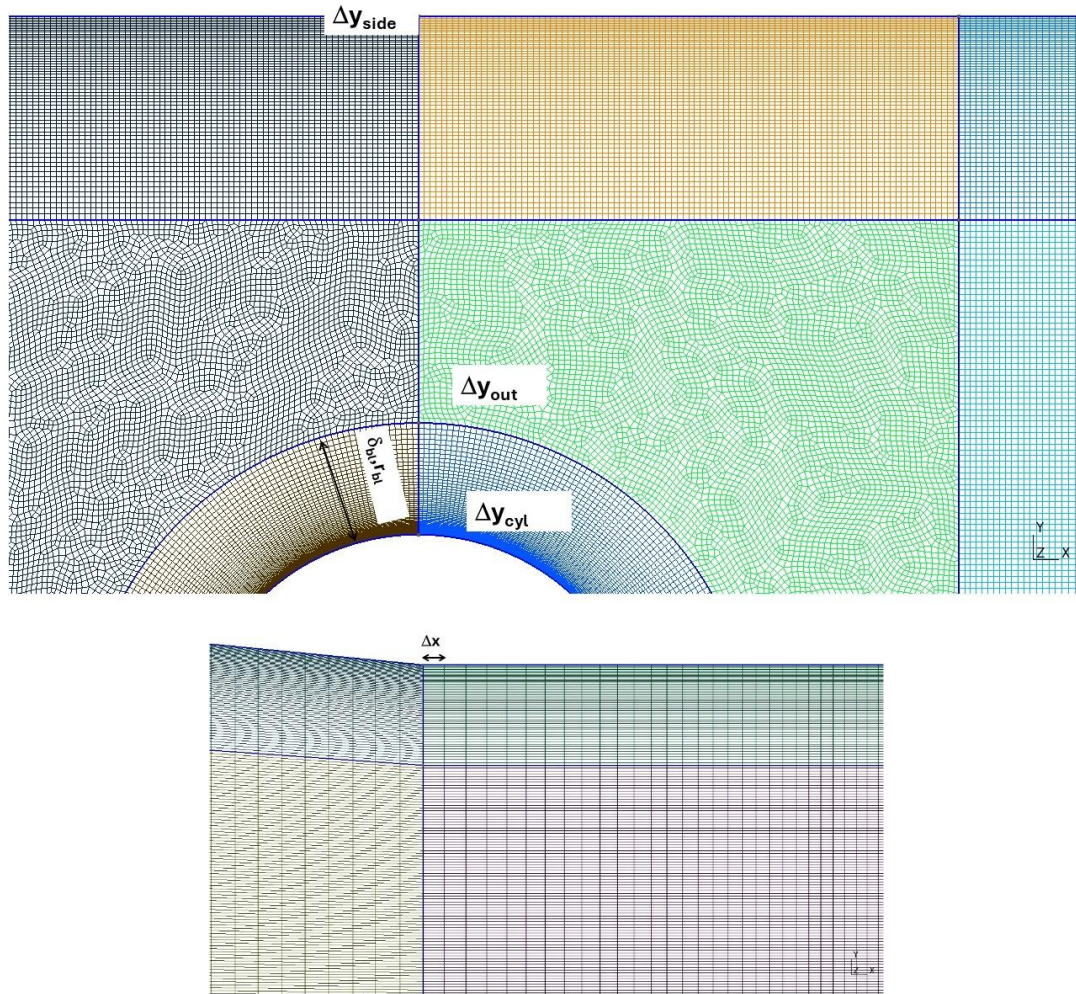


Figure 84: The main parameters for the mesh generation for the single cylinder configuration

Table 23: The meshes used for the single cylinder configuration depending on the flowrate Q . ref. is the reference mesh (the finest in the single cylinder configuration), coarse (a) and coarse (b) are two levels of coarse refinements, (b) being coarser than (a)

Q (l/s)	mesh	r_{bl}	δ_{bl} (m)	Δy_{cyl} (m)	Δy_{out} (m)	$\Delta x_{cyl} / \Delta y_{cyl}$
0.5	ref.	1.050	6.00E-03	3.200E-05	3.170E-04	10.0
1.7	ref.	1.050	3.00E-03	1.100E-05	1.560E-04	10.0
1.7	coarse (a)	1.050	6.10E-03	2.100E-05	3.270E-04	7.5
2.9	coarse (a)	1.050	2.70E-03	1.300E-05	1.000E-04	7.5
2.9	coarse (b)	1.025	1.10E-02	7.000E-06	2.740E-04	15.0

Table 24: The number of discretization points in the spanwise and azimuthal direction around the single cylinder, the number of cells for the precursor simulation and the number of cells in the main computational domain (without the precursor domain)

Q	mesh	n_z	n_θ	n_{prec} (Million)	n_{cells} (Million)
0.5	ref.	236	240	1.5	126.9
1.7	ref.	652	360	27.4	177.9
1.7	coarser (a)	352	240	7.2	39.9
2.9	coarser (a)	546	384	44.9	251.3
2.9	coarser (b)	546	384	19.6	117.5

5.2 Tube bundle configuration computational domain and mesh

Figure 85 shows the computational domain and the boundary conditions to represent the tube bundle configuration of AMOVI experiment [15], [29]. $D = 0.01215 \text{ m}$ and $P = 0.0175 \text{ m}$. The obtained pitch-to-diameter ratio is thus equal to $\frac{P}{D} = 1.44$. $H = 4P$ and $L_z = 0.1 \text{ m}$ are the width of the rectangular channel in the normal direction to the flow and the total length of the cylinders, respectively. The hydraulic diameter of the rectangular channel at the inlet is thus equal to $D_H = \frac{2HL_z}{(H+L_z)} = 0.082 \text{ m}$. L_{in} and L_{out} are equal to $10D$ and $22D$, respectively. L_z is equal to 0.1 m as it is the case in the experiment and the sidewalls are represented and finely refined. Note that the frame axes are the one used in Figure 85; x is the direction parallel to the main flow, z in parallel to the cylinders' axis (span-wise direction) and y in obtained by direct orientation perpendicular to x and z . “ y ” may be referred as the normal direction to the main flow.

Figure 86 shows the mesh around a central cylinder, a half cylinder (side cylinder) and a portion going towards the inlet. Unless specified, the mesh is created using a Frontal-Delaunay for quads approach and a Blossom algorithm to create 2D hexahedral elements. One can notice the very good quality of the mesh between the tubes and for the transition to the inlet or outlet. This has a non-negligible effect on the CPU cost of an iteration in time and on the quality of the wall-resolved LES. It has also been shown on previous configurations (VISCACHE, see [12]) that too important jumps on the cell size between the inlet channel and the first row of tubes may lead to spurious oscillation with LES. In the present computation, the cells at the interface of these two subdomains are the same. The mesh is also refined towards the walls normal to the tubes.

Figure 86 illustrates the main lengths and progression ratios used for mesh generation. Δy_{cyl} is the cell-size in the wall-normal direction to the cylinders and towards the wall intersecting the half-cylinders. Note that to obtain a non-dimensional distance of the first cell to the wall equal to 1, the non-dimensional distance of the cell-size should be equal to 2 in the wall-normal direction. For “so-called” reference meshes, the progression ratio in the wall normal direction to the cylinder and towards the upper and lower walls containing the half-cylinders are $r_{bl} = 1.1$ and $r_{sw,1} = 1.1$, respectively, “sw” standing for sidewall.

Two Reynolds numbers are used to create the mesh “a priori”:

$Re_g = \frac{2U_g(P-D)}{\nu}$ which is the Reynolds number based on the gap velocity $U_g = \frac{P}{P-D} U_\infty$ where U_∞ is the free-stream velocity and considering an infinite plane channel in the stream-wise and span-wise directions of half-width $(P - D)$ (the space between two adjacent cylinders).

$Re_H = \frac{U_\infty D_H}{\nu}$ which is the hydraulic Reynolds number of the inlet rectangular duct.

Table 25 gives these Reynolds numbers as well as the values of the gap and free-stream velocity three first cases of a rigid cylinder in a rigid tube bundle configuration studied in AMOVI.

C_f is an estimate of Darcy–Weisbach friction factor using the so-called McAdams correlation for a smooth pipe flow. For the cylinder we use

$$\Delta n = 2 \frac{v}{u_g} \sqrt{\frac{8}{C_f}} \text{ with } C_f = 0.184 Re_g^{-0.2}.$$

After a few computations, this estimate gave too high non-dimensional distances to the wall. Finally, the value $\Delta y_{cyl} = \Delta n/2$ has been taken and *a posteriori* analysis shows that this was enough to obtain adapted wall-refinement around the cylinders. The same refinement is used at the sidewalls containing the half cylinders.

Δx_{cyl} and Δz_{cyl} are the refinements in the azimuthal and in the span-wise directions around the cylinders. Δz_{cyl} has been fixed for all the meshes such that $\Delta z_{cyl}/\Delta y_{cyl}=15$. δ_{bl} is the meshed boundary layer thickness around the cylinder (see **Figure 86**). δ_{bl} is computed such that the refinement in the azimuthal direction of the boundary layer at its limit is equal to the larger cell size Δy_{out} in the wall normal direction after the progressive unrefinement, and to Δx_{cyl} . This is a condition to obtain homogeneous meshes using the Frontal-Delaunay and algorithms in Gmsh. Actually, Δy_{out} is the characteristic cell-size in 2D in the bulk region (see **Figure 86**).

The mesh is refined towards the sidewalls (in z direction). There is a portion of regular mesh on both sides of the centre plane then the mesh is refined towards the sidewalls down to $\Delta y_{sw,2}$ ensuring a continuity of the cell size in the span-wise direction. The refinement ratio in this direction is equal to 1.05. The cell size at these sidewalls is empirically determined to reach a wall resolved LES. The “theoretical” value to have a mean normal non-dimensional distance to the wall equal to 1 at the inlet.

$$\Delta n_{sw} = 2 \frac{v}{u_\infty} \sqrt{\frac{8}{C_f}} \text{ with } C_f = 0.184 Re_H^{-0.2}.$$

Due to the presence of the horseshoe vortices at the intersection of the sidewalls and the cylinders and to the acceleration the near-wall region of the sidewalls and after several tests, the value $\Delta y_{sw,2} = \Delta n_{sw}/8$.

One must notice that the constraint of refinement in the span-wise and wall-normal directions towards the sidewalls containing the half cylinders and the ones parallel to the span-wise direction are enough to have a wall-resolved LES at the inlet.

Finally, the mesh is coarsened towards the inlet and the outlet with a progressive unrefinement, followed by a constant one with the target $\Delta x/\Delta n_{sw}=15$.

The values of δ_{bl} , Δy_{cyl} , Δy_{out} and $\Delta x_{cyl}/\Delta y_{cyl}$ for five representative meshes of the tube bundle configurations are given in Table 26. The ones at $Q = 0.5$ l/s, 1.7 l/s and 2.9 l/s are the first reference three ones used in the rigid configuration. The reference mesh for $Q = 1.95$ l/s, the flowrate which exhibited in AMOVI experiment very large displacements with a standard deviation equal to 3.223 mm, gave a divergent solution for the configuration with a flexible cylinder. The coarse mesh for $Q = 1.95$ l/s allowed to have a stable case by coarsening the mesh; by increasing Δy_{out} , the characteristic length in the bulk. This modification led to an increase of the azimuthal cell-size around the cylinders.

Table 27 gives the number of grid-points in the span-wise direction n_z and in the azimuthal one n_q . It also gives the number of computational cells in the precursor domain n_{prec} and in the remaining computational domain n_{cells} in millions. The number of computational cells goes roughly from 30 million for the lowest flowrate to 500 million for the highest one.

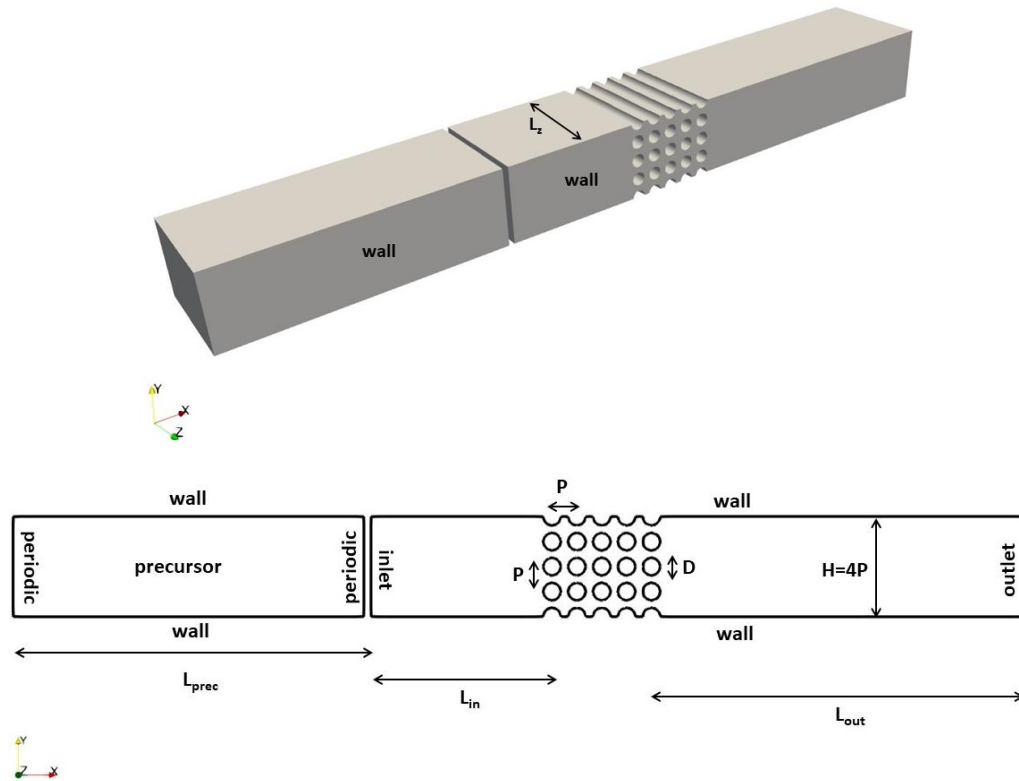


Figure 85: The computational domain and the boundary conditions for the tube bundle configuration. The frame axes is the one used in this figure, the origine being at the center of the central cylinder

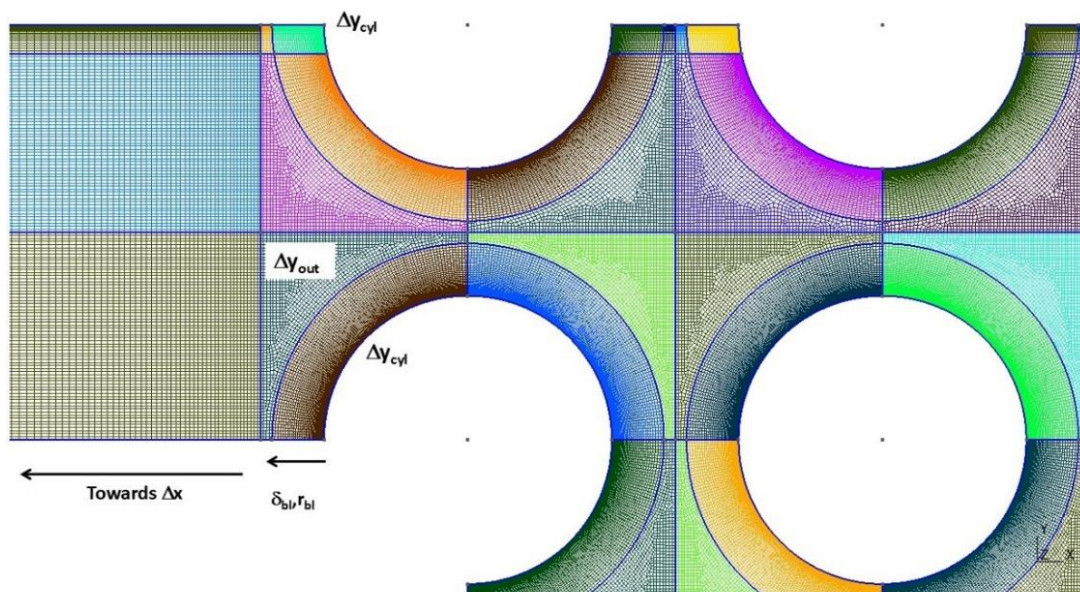


Figure 86: The main parameters for the mesh generation for the tube bundle configuration

Table 25: The gap velocity U_g , the upstream velocity U_∞ , the gap Reynolds number Re_g , the upstream Reynolds number Re_∞ and the hydraulic Reynolds number of the square duct at the inlet Re_H

Q (l/s)	U_g (m/s)	U_∞ (m/s)	Re_g	Re_∞	Re_H
0.5	0.234	0.071	2840	868	5882
1.7	0.795	0.243	9657	2951	20000
2.9	1.356	0.414	16474	5034	34118

Table 26: Representative lengths of the refinements shown in Figure 86

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
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
2.9	ref.	1.247E-03	1.300E-05	1.370E-04	10

Table 27: The number of discretization points in the spanwise and azimuthal direction around cylinders, the number of cells for the precursor simulation and the number of cells in the main computational domain (without the precursor domain)

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
1.95	ref.	444	268	22.3	174.4
1.95	coarse	444	212	16.6	124.5
2.9	ref.	602	356	59.8	430.6

5.3 Modelling and numerical approach

Standard water properties are utilized for all the computations; the density and the kinematic viscosity are $\rho = 1000 \text{ kg/m}^3$ and $\nu = 10^{-6} \text{ m}^2/\text{s}$, respectively.

Code_saturne is based on a co-located finite volume discretization and deals with unstructured meshes made up of polyhedral cells of any type. It can deal with incompressible and compressible flows. The solver is parallelized using MPI and OpenMP. The incompressible Navier-Stokes equations are solved. The pressure/velocity coupling is ensured thanks to a predictor-corrector algorithm. Rhie and Chow interpolation is used in the correction step while solving Poisson pressure equation. All the gradients are computed using an implicit Gauss method to take into account non-

orthogonal faces. LES is performed using the dynamic Smagorinsky subgrid-scale model and a filter-width equal to $2\Omega^{1/3}$ where Ω is the computational cell volume. LES approach has been successfully tested and validated and gave very encouraging results for numerous configurations among them the flow through a tandem of cylinders in [30] and for the flow in a matrix of pin-fin arrays [31]. The temporal discretization is centred in time: Crank-Nicolson for the diffusion and the transported velocity in the convection and Adams-Bashforth for the transporting velocity (the velocity and pressure are staggered in time). The spatial discretization is globally second order, the convection scheme being fully centred (outer iterations are imposed to make implicit the convection scheme and the corrections due to non-orthogonal faces). No outer iterations for pressure/velocity coupling are activated.

A precursor Large Eddy Simulation, giving a targeted mean bulk velocity thanks to an imposed pressure gradient, is embedded in the main computation. At each iteration, code_saturne solves simultaneously the precursor and the main computations and the transfer of variables from the periodic face to the inlet one (see Figure 85) is ensured thanks to a user subroutine. The actual inlet boundary condition of the main computation is given by the velocity of the periodic boundaries coming from the previous time step. The periodic and inlet faces are fully conformal by construction. The length of the domain used for the precursor computation in the stream-wise direction is $L_{prec} = 3D_H$. No-slip boundary conditions are utilized at all the walls; the cylinder and all the sidewalls. Figure 87 shows how the precursor computation of a square channel flow (left domain) feeds the main computational domain containing the tube bundle with the stream-wise velocity (and the other components) to impose a fully developed turbulent flow at the inlet face.

The outlet faces use a free outlet boundary condition and all the walls are “wall-resolved” (no-slip boundary conditions).

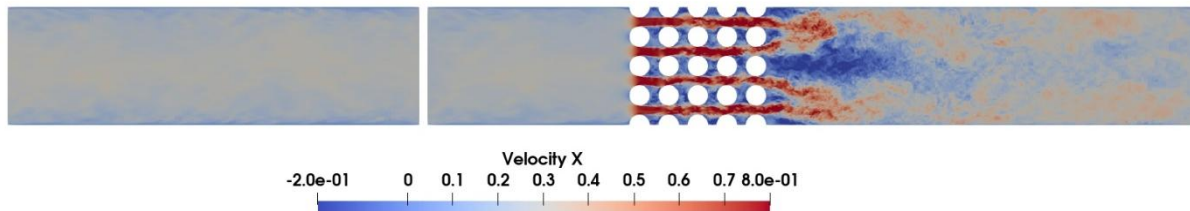


Figure 87: The stream-wise instantaneous velocity for $Q = 1.7$ l/s showing how the precursor computation (left part of the domain) feeds the main computational domain (right part) containing the tube bundle

When FSI is activated, a strong coupling is imposed with the Arbitrary Lagrangian Eulerian Method via outer iterations [32]. The maximum number of outer iterations between Navier-Stokes and the solid motion solver is 10 (this latter being never reached in the presented simulations) and the tolerance on the solid displacement tolerance is equal to 10^{-5} . A Newmark time scheme is utilized for solid displacement. At each iteration, the following equation is solved:

$$M\ddot{y}_i + C\dot{y}_i + Ky_i = f_y$$

The structure coefficients are set to $M = 0.04526$ kg, $C = 0$ N.s.m⁻¹ and $K = 1542.92$ N.m⁻¹. The natural frequency is $f_s = \frac{1}{2\pi}\sqrt{K/M} \approx 29.4$ Hz. f_y (N) is the lift force (y-direction) applied to the central cylinder. The cylinder moves only in the lift direction.

By default, the corrections due to non-orthogonal faces use an implicit Gauss algorithm. When the displacements are large, the algorithm using an implicit Gauss combined to a least square method is activated.

5.4 Single tube configuration results

5.4.1 Single rigid tube configuration

5.4.1.1 Results at $Q = 0.5$ l/s and at $Q = 1.7$ l/s with the reference mesh

Table 28 gives the time step and *a posteriori* quantifications of the CFL number and the non-dimensional distance to the wall y^+ for $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes (see Table 23). Note that the two meshes contain 129.4 and 205.3 million cells, respectively.

CFL_{max} , CFL_{max} Std. and $CFL_{sp,av}$ are the time average of the maximum of the CFL number over the computational domain, the standard deviation of CFL_{max} in time and the time average of the space averaged Courant number, respectively. Actually, the space average of the CFL is almost constant in time. y_{max}^+ , y_{max}^+ Std. and $y_{sp,av}^+$ are the time average of the maximum of y^+ over the domain, the standard deviation of y_{max}^+ in time and the time average of the space average of y^+ , respectively. The space average y^+ is also almost constant in time.

The time average of the maximum of the CFL doesn't exceed 1 for both $Q = 0.5$ l/s and $Q = 1.7$ l/s. Its standard deviation is almost null for $Q = 0.5$ l/s and is equal to 0.15 for $Q = 1.7$ l/s which shows that this quantity doesn't vary a lot in time. The space average of the CFL number is also reasonable with a time average value around 0.1 for both flowrates. The maximum of the non-dimensional distance to the wall y^+ remains around 2.1 to 2.2 for both flowrates as its standard deviation in time is around 0.2. The space average of non-dimensional distance has also a reasonable value around 0.4. Figure 88 shows the instantaneous non-dimensional distance to the wall y^+ , its time average and standard deviation the reference mesh at $Q = 1.7$ l/s. Figure 88 confirms Table 28. In addition to the fine structures, which can be observed on the walls thanks to the refinement characteristic of a wall-resolved LES, one can observe the acceleration between the cylinder and the upper wall, a detachment just downstream of this acceleration (observed with low values of y^+), then a redeveloping region. Although the high values of the non-dimensional distance are mainly on the walls of the acceleration region, the maximum values seem to occur in the redevelopment region downstream the tube. On the sidewalls, one may observe the trace of a horseshoe vortex (Figure 88 (top) and Figure 88 (bottom right)) and the turbulence in the wake of the cylinder (Figure 88 (bottom left)). Figure 89 gives the previous quantities, but restricted on one half of the cylinder. The values of non-dimensional distance to the wall y^+ are globally lower around the cylinder than on the top and bottom walls; the refinement required for the present configuration is then more constrained by the physical phenomena occurring in the top/bottom wall region. One can also observe in Figure 89 that the maximum y^+ is in the upstream region of the cylinder and that the turbulence activity in the wake leads to fluctuations on y^+ observable on the standard deviation of this latter. One can also observe on the standard deviation of y^+ a trace of the detachment which occurs on the cylinder; there is a thin line along the cylinder (Figure 89 (right)).

The variables are made non-dimensional using the reference velocity U_∞ given in Table 22.

Figure 90 shows an instantaneous snapshot of the stream-wise velocity and its time average value for $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes. A vortex shedding is observed in both instantaneous snapshots. However, the wake obtained at the lowest flow rate is much larger than the one obtained with the highest one. There is a detachment from the top and bottom walls

characterized by low or negative stream-wise velocities. This phenomenon is clear for $Q = 1.7$ l/s on Figure 92 on the mean pressure field where an adverse pressure gradient is observable.

Figure 91 shows the standard deviation of the non-dimensional velocity components. The turbulence is strongly anisotropic with a kind of bubble of turbulence for the normal fluctuation $\sqrt{v'^2}$. Moreover, turbulence develops far from the cylinder for the lowest flowrate following the long wake, which leads to low fluctuations around the cylinder compared to the highest flowrate (see Table 29).

Table 28: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes – single rigid cylinder configuration

Q (l/s)	Δt (s)	CFL _{max}	CFL _{max} Std.	CFL _{sp,av}	$y^+_{\max}$	$y^+_{\max}$ Std.	$y^+_{sp,av}$
0.5	0.0004	0.75	0.01	0.12	2.09	0.2	0.43
1.7	0.00005	0.96	0.15	0.09	2.17	0.21	0.38

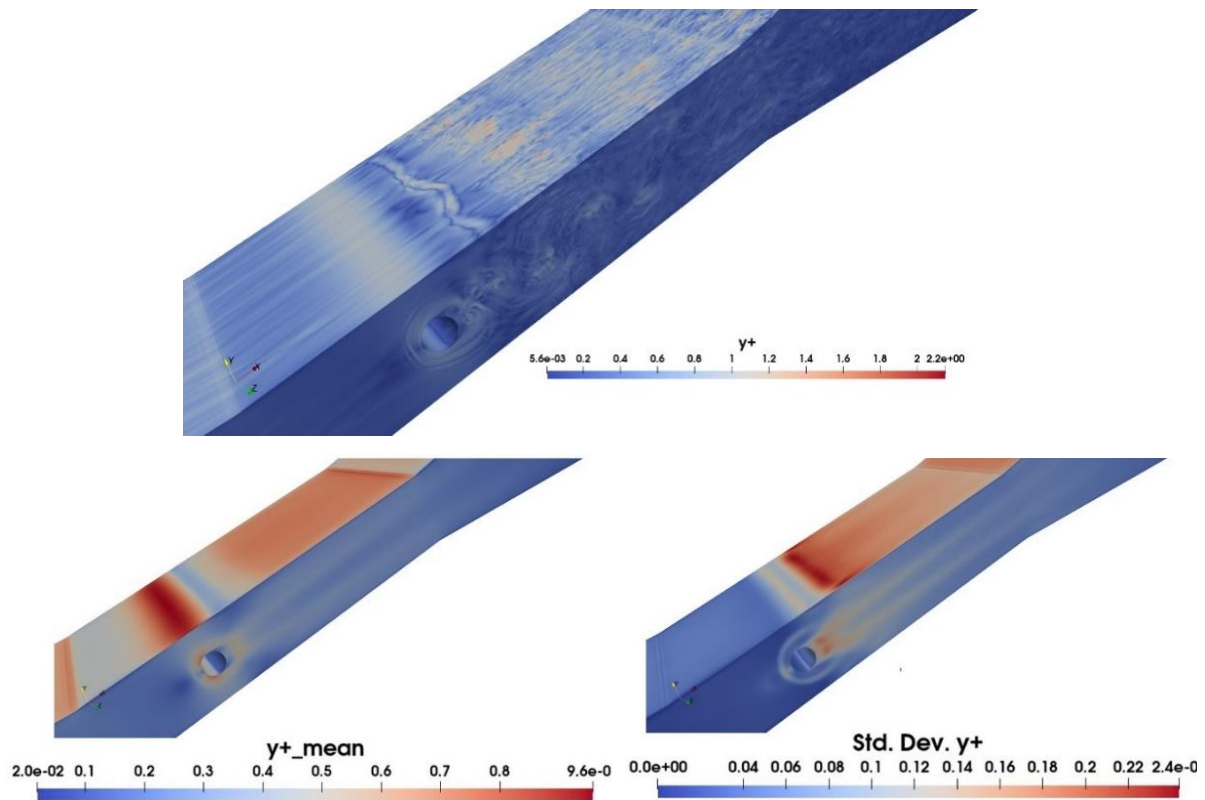


Figure 88: Instantaneous y^+ (top), time averaged y^+ (bottom left) and standard deviation of y^+ (bottom right) for the reference mesh at $Q = 1.7$ l/s – overview on the sidewalls

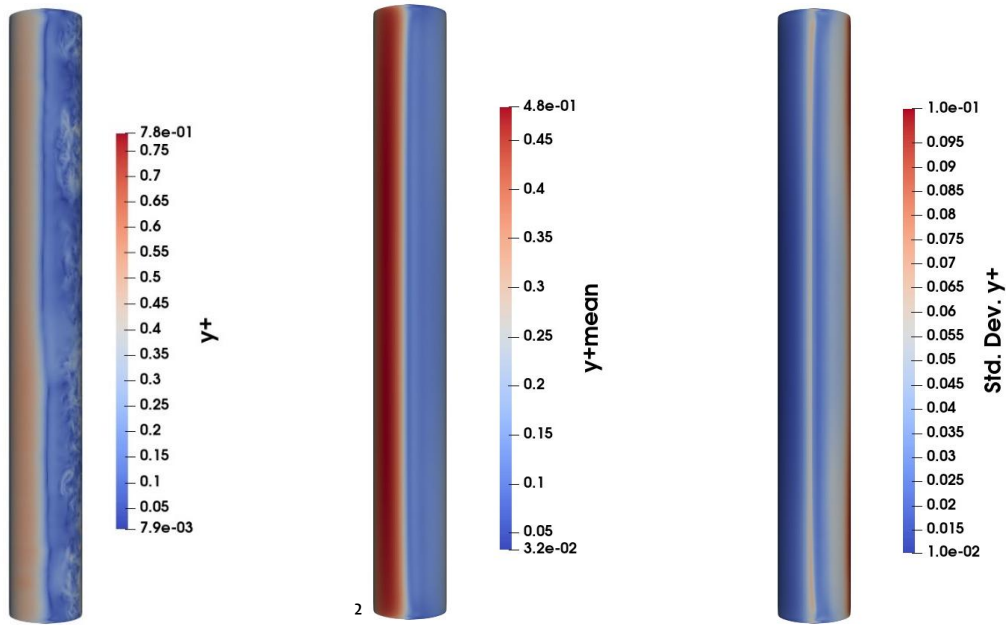


Figure 89: Instantaneous y^+ (left), time averaged y^+ (middle) and standard deviation of y^+ (right) on the tube wall for the reference mesh at $Q = 1.7$ l/s – top view on the cylinder, the flows goes from left to right

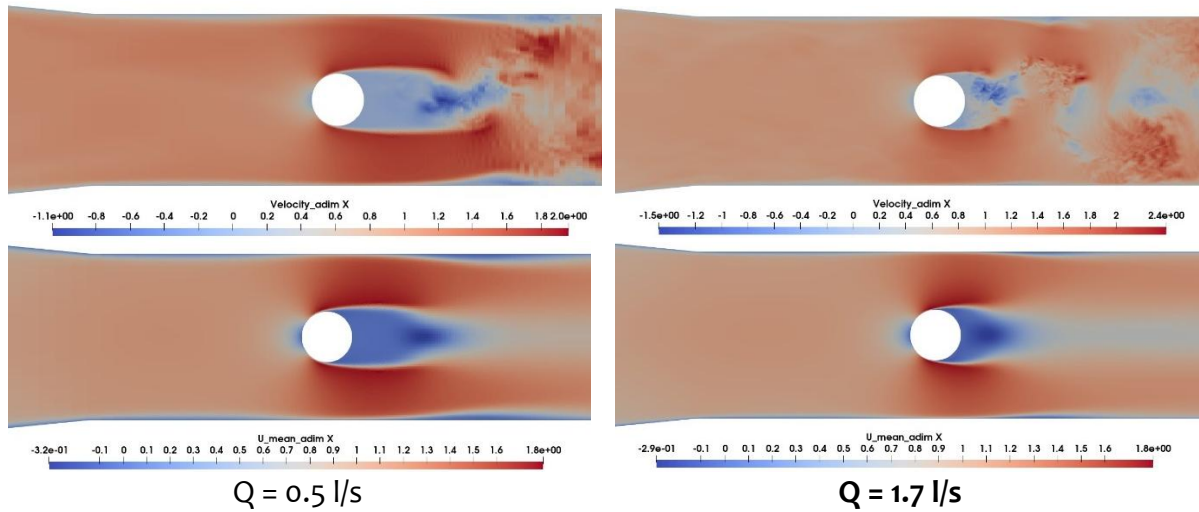


Figure 90: Instantaneous and time averaged non-dimensional stream-wise velocity in the center plane at $Q = 0.5$ l/s (left) and $Q = 1.7$ l/s (right) with the reference meshes - u/U_∞ (top), $\bar{u}/U_\infty$ (bottom)

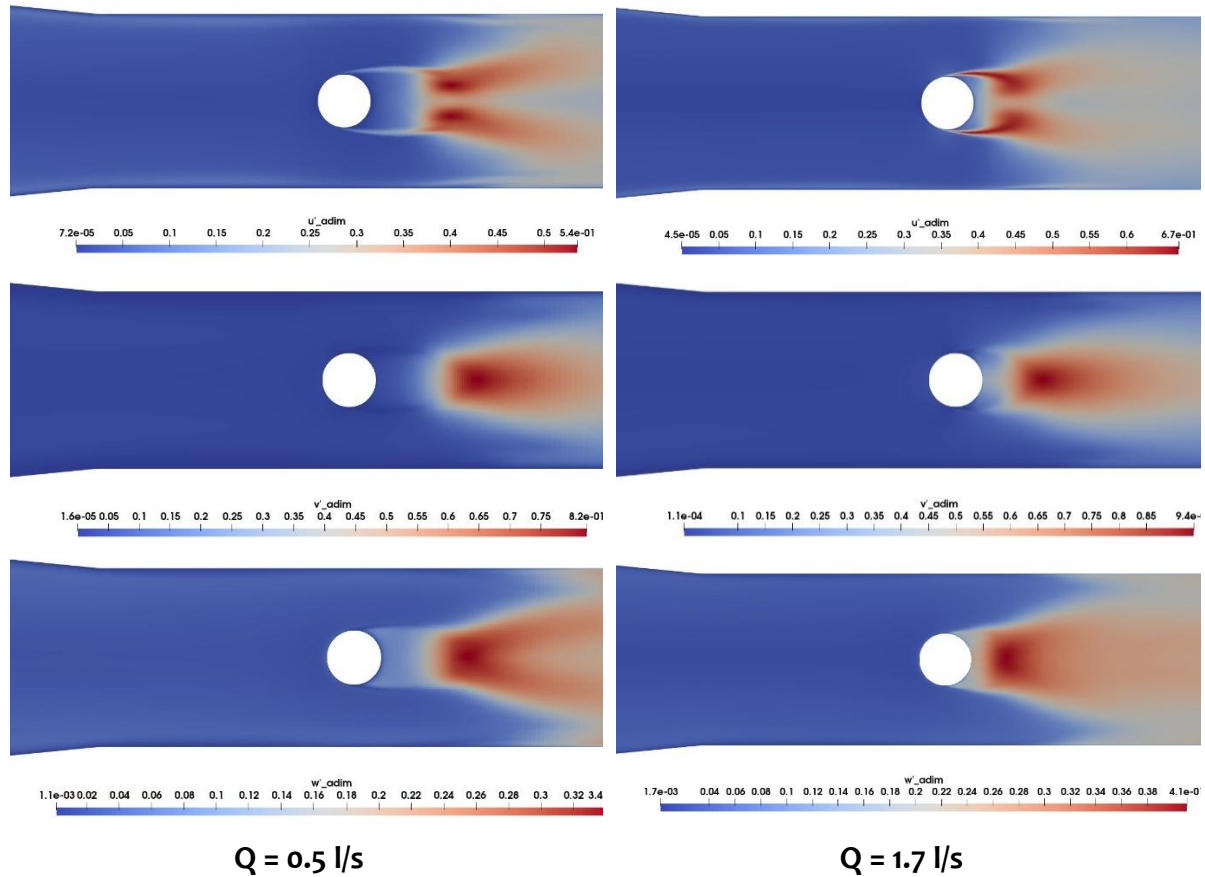


Figure 91: Standard deviation of the velocity components at $Q = 0.5$ l/s (left) and $Q = 1.7$ l/s (right) with the reference meshes, $\sqrt{u'^2}/U_\infty$ (top), $\sqrt{v'^2}/U_\infty$ (middle), $\sqrt{w'^2}/U_\infty$ (bottom)

Figure 92 shows the time average and the standard deviation of the non-dimensional pressure in the centre plane. The wake which is characterized by low pressure is much shorter for the high flowrate as this has been observed previously. Moreover, the pressure fluctuations downstream the cylinder affect a much longer region than with $Q = 0.5$ l/s, which is not obvious to deduce if one considers the non-dimensional fluctuations in Figure 91. One can also observe high pressure fluctuations in the top and bottom of the cylinder for $Q = 1.7$ l/s which will naturally lead to higher fluctuation of drag and lift forces (see Table 29).

Figure 93 shows the non-dimensional pressure fluctuations around the cylinder for the two flowrates $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes. The pressure fluctuations for $Q = 0.5$ l/s are dominated by the ones created by the horseshoe vortices. For $Q = 1.7$ l/s, these fluctuations are strong in the detachment region which creates von Kármán vortices. In the wake of the cylinder, one can also observe high fluctuations close to the sidewalls compared to those in the central region where there is a clear drop. For both flowrates, the fluctuations are small in the impingement region except close to the sidewalls.

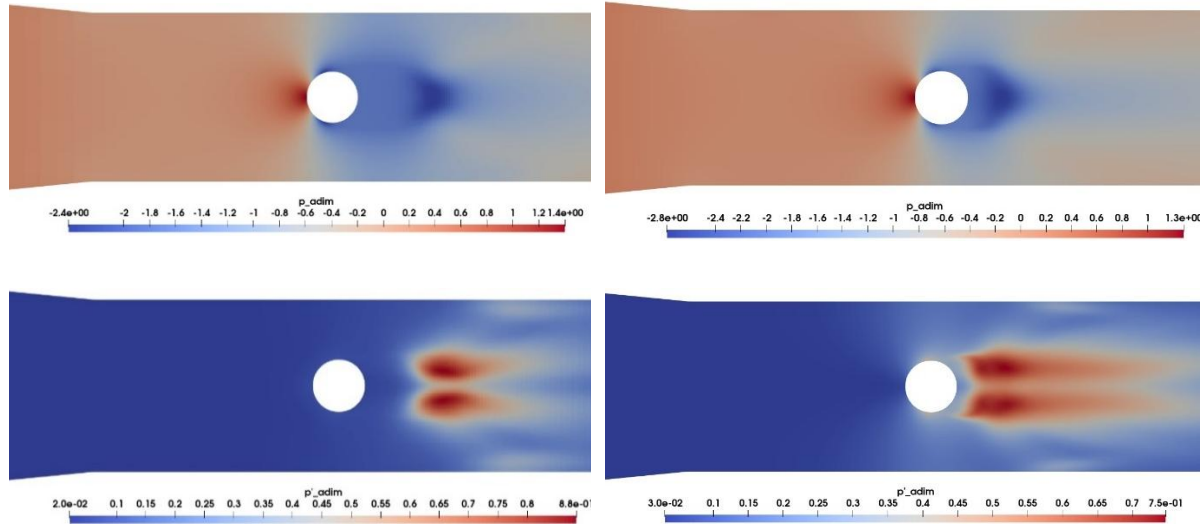


Figure 92: Time averaged (top) and standard deviation (bottom) of the pressure at $Q = 0.5$ l/s (left) and $Q = 1.7$ l/s (right) with the reference meshes - $\bar{p}/(0.5\rho U_\infty^2)$ (top), $\sqrt{p'^2}/(0.5\rho U_\infty^2)$ (bottom)

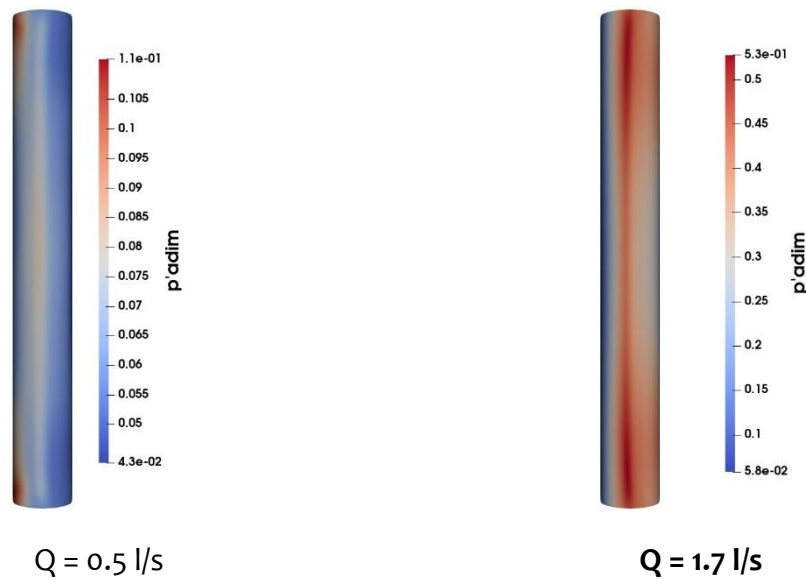


Figure 93: Standard deviation of the non-dimensional pressure $\sqrt{p'^2}/(0.5\rho U_\infty^2)$ on the cylinder surface for the reference mesh at $Q = 0.5$ l/s (left) and $Q = 1.7$ l/s (right) – top view on the cylinder, le flows goes from left to right

Figure 94: $0.5\rho U_\infty^2 DL_z$, respectively. If the index $fErr(\bar{C}_D) = \frac{\bar{C}_D - \bar{C}_{Df}}{\bar{C}_D} Err(\bar{C}_L'^2) = \frac{\bar{C}_L'^2 - \bar{C}_{Lf}^2}{\bar{C}_L'^2}$ Figure 95 gives these two quantities. The physical time needed to obtain a relative error lower that 2% for the mean drag coefficient is equal to 15 s and 30 s for $Q = 0.5$ l/s and $Q = 1.7$ l/s, respectively. The

error of this quantity is lower than 3% in less than 4.5 s for both flowrates. The first moment of the drag and lift forces converge relatively fast.

The physical time needed to obtain a relative error lower than 5% for the standard deviation of the lift coefficient is equal to 150 s and 130 s, for $Q = 0.5$ l/s and $Q = 1.7$ l/s, respectively. To obtain an error lower than 2%, the physical time increases to 210 s and 305 s. Even for 10% error, the physical times needed are 15 s and 40 s, respectively. These values are much higher than in usual CFD computations found in the literature. In terms of vortex shedding period and using the values of Fr_L in Table 29, 150 s corresponds to 532 and 1455 vortex shedding period for $Q = 0.5$ l/s and $Q = 1.7$ l/s! This corresponds to 31 and 106 flow through times, respectively. The flow through time is estimated with the distance from the inlet fed with the precursor computation and the outlet (≈ 0.6 m) and the reference velocity U_∞ given in Table 22. To reach a precision of 10% on the standard deviation of the lift, the computation needs to be run for 15 s and 40 s for $Q = 0.5$ l/s and $Q = 1.7$ l/s, respectively, which corresponds to 3 and 28 flow through times, which are usually representative of the computational time in standard CFD. Thus, according to the present estimates, one can't reach more than 10 % precision on the RMS values of the lift coefficient with 10 or 20 flow passes.

Table 29 gives the time used for averaging the drag and lift forces per unit of length. It also gives the time average and the standard deviation of these forces and the main frequency of the lift force spectrum obtained thanks to the power density spectra from Figure 96 (left). The spectra obtained from the experiment at $Q = 0.5$ l/s are not suitable for comparisons with CFD results. The standard deviation obtained in the computations at $Q = 0.5$ are largely underestimated for the drag and underestimated by a factor of 3 for the lift. A frequency at 3.55 Hz which is not observed experimentally is obtained in the numerical simulations. The computed standard deviation of the drag coefficient exhibits a reasonable value compared to the experimental one at $Q = 1.7$ l/s with a relative error of 28%. The computed standard deviation of the lift coefficient is however almost 5 times higher than the experimental one. These important gaps might be due to difference of inlet boundary conditions between the experiment and the computations.

Figure 96 compares the experimental and the computed spectra of the lift and drag forces per unit of length. As it has been already stated, the experimental results at $Q = 0.5$ l/s can't be exploited. At this flowrate, the lift force spectrum of the wall-resolved LES exhibits a dominant peak at 3.55 Hz and smaller peaks at lower frequencies. The experimental and the computational spectra compare fairly well at $Q = 1.7$ l/s for the lift force, but not for the drag one. The dominant frequencies are equal to 8.8 Hz and 9.7 Hz, which is a good agreement. The experimental drag force doesn't exhibit a dominant frequency which is not conform to what is observed usually. More investigations are needed, consulting also the existing literature. The computed drag spectra exhibit a dominant peak at 7.1 Hz and 13.1 Hz for $Q = 0.5$ l/s and $Q = 1.7$ l/s, respectively. A secondary peak (not observed for $Q = 0.5$ l/s) is observed on the computed lift spectra for $Q = 1.7$ l/s at around 29 Hz which is close to the eigen frequency of the mass-spring, which will be imposed later but the energy contained is very low.

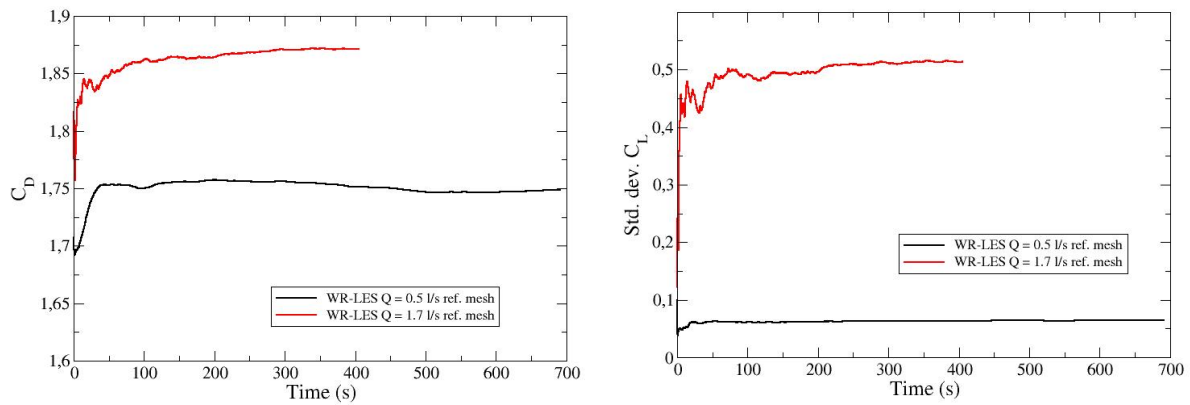


Figure 94: Time evolution of the mean drag coefficient $\overline{C_D}$ and the standard deviation of the lift coefficient $\overline{C_L'^2}$ for the reference meshes at $Q = 0.5$ l/s and 1.7 l/s – single rigid cylinder configuration

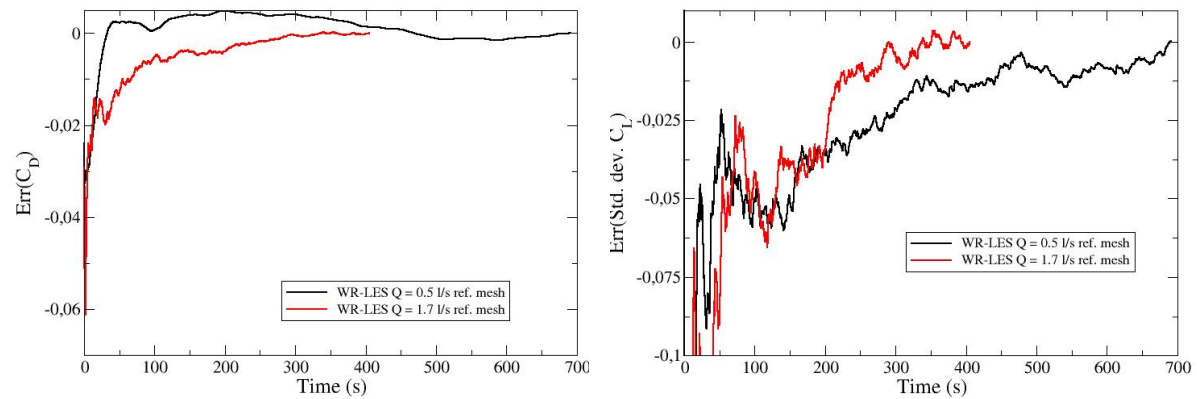


Figure 95: Time evolution of $\text{Err}(\overline{C_D})$ and $\text{Err}(\overline{C_L'^2})$ for the reference meshes at $Q = 0.5$ l/s and 1.7 l/s – single rigid cylinder configuration

Table 29: The computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length and the main frequency of the lift force - results of the reference mesh for $Q = 0.5$ l/s and $Q = 1.7$ l/s – single rigid cylinder configuration - comparison with experimental data

Q (l/s)	CFD/Exp.	mesh	Time (s)	$\overline{F_D}$ (N/m)	$\overline{F_L}$ (N/m)	$\overline{F_D'^2}$ (N/m)	$\overline{F_L'^2}$ (N/m)	Fr_L (Hz)
0.5	Exp.	-	410	-	-	0.022	0.016	-
0.5	CFD	ref.	691	0.166	0.00013	0.002	0.006	3.55
1.7	Exp.	-	410	-	-	0.106	0.122	8.8
1.7	CFD	ref.	406	2.054	0.00093	0.076	0.564	9.7

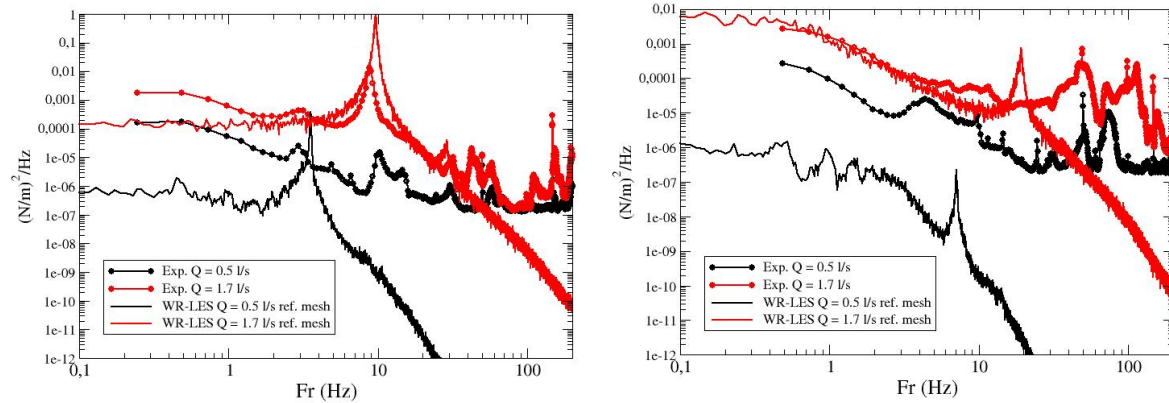


Figure 96: Power Density Spectra (PDS) of Lift (left) and drag (right) forces per unit of length for the references meshes at $Q = 0.5$ l/s and 1.7 l/s – comparison with the experimental data

5.4.1.2 Results at $Q=1.7$ and $Q = 2.9$ l/s with coarser meshes

Table 25 and Table 27 recall the refinements. Coarse (a) is a first level of refinement and coarse (b) a second one. For $Q = 1.7$ l/s, coarse (a) was used to test the sensitivity to the refinement. The refinement around the cylinders in coarse mesh (a) is two times coarser in the wall normal direction than in the reference mesh. The cell size in the bulk region (Δy_{out} in Figure 84) is also two times coarser. However, the refinement around the cylinders in the azimuthal direction is only 50% coarser than in the reference mesh. The aspect ratios in the span-wise direction are kept the same as in the reference mesh. Therefore, the span-wise direction is about 2 times coarser. The coarse mesh (a) has been used also with a moving cylinder at $Q = 1.7$ l/s, as it is shown in section 5.4.2.2, to also test the sensitivity to the mesh refinement in the presence of mesh deformation.

For $Q = 2.9$ l/s, coarse (a) mesh was too coarse to obtain a numerical solution for the moving cylinder configuration. A second level of refinement has thus been created. The expansion rate in the boundary layer is decreased which leads to a much larger boundary layer thickness δ_{bl} . The mesh is two times finer close to the wall but much coarser in the bulk. The resulting 2D mesh before extrusions in z direction has finally 2 times less computational cells. Note that the reference mesh for $Q = 2.9$ l/s is too fine to be computed with “reasonable” CPU resources.

Table 30 gives the time step and *a posteriori* quantifications of the CFL number and the non-dimensional distance to the wall y^+ at $Q = 1.7$ l/s with the reference mesh and the mesh coarse (a) and at $Q = 2.9$ l/s with the meshes coarse (a) and (b) (see Table 23 for the details of the refinements). The CFL numbers at $Q = 2.9$ l/s with both meshes coarse (a) and (b) are higher than the ones at $Q = 1.7$ but remain very reasonable for LES with space averaged values not exceeding 0.12. The refinement close to the walls are similar in terms of maximum values to what is reported for $Q = 1.7$ l/s using the reference mesh.

Figure 97 and Figure 98 show the time average non-dimensional streamwise velocity and the standard deviation of the non-dimensional pressure for the two flowrates, each on two levels of mesh refinement. Although slight differences might be observed between two successive refinements, the results are qualitatively the same.

Table 30: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes and the coarse ones at $Q = 1.7$ l/s and $Q = 2.9$ l/s – single rigid cylinder configuration

Q (l/s)	CFD/Exp.	mesh	Δt (s)	CFL_{max}	CFL_{max} Std.	$CFL_{sp,av}$	y^+_{max}	y^+_{max} Std.	$y^+_{sp,av}$
1.7	Exp	-	0.001	-	-	-	-	-	-
1.7	CFD	ref.	0.00005	0.96	0.15	0.086	2.17	0.21	0.38
1.7	CFD	coarse (a)	0.000075	0.88	0.16	0.066	2.1	0.18	0.42
2.9	Exp	-	0.001	-	-	-	-	-	-
2.9	CFD	coarse (a)	0.00004	1.56	0.23	0.12	2.2	0.18	0.37
2.9	CFD	coarse (b)	0.000025	1.73	0.28	0.08	2.25	0.22	0.43

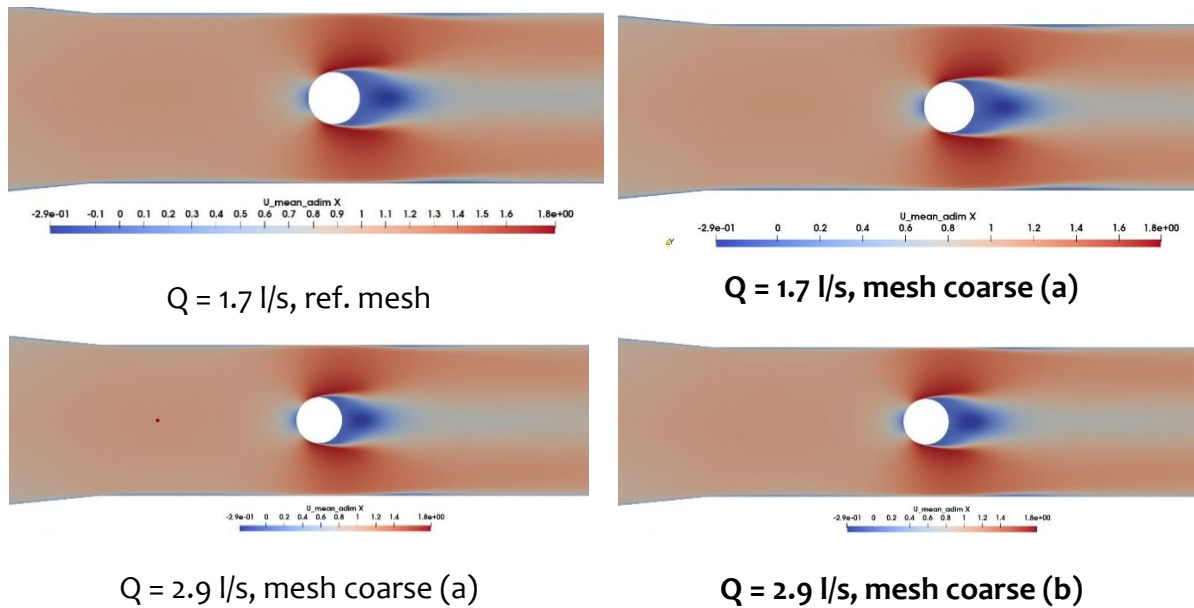


Figure 97: Time averaged non-dimensional stream-wise velocity in the centre plane at $Q = 1.7$ l/s (top) and $Q = 2.9$ l/s (bottom)

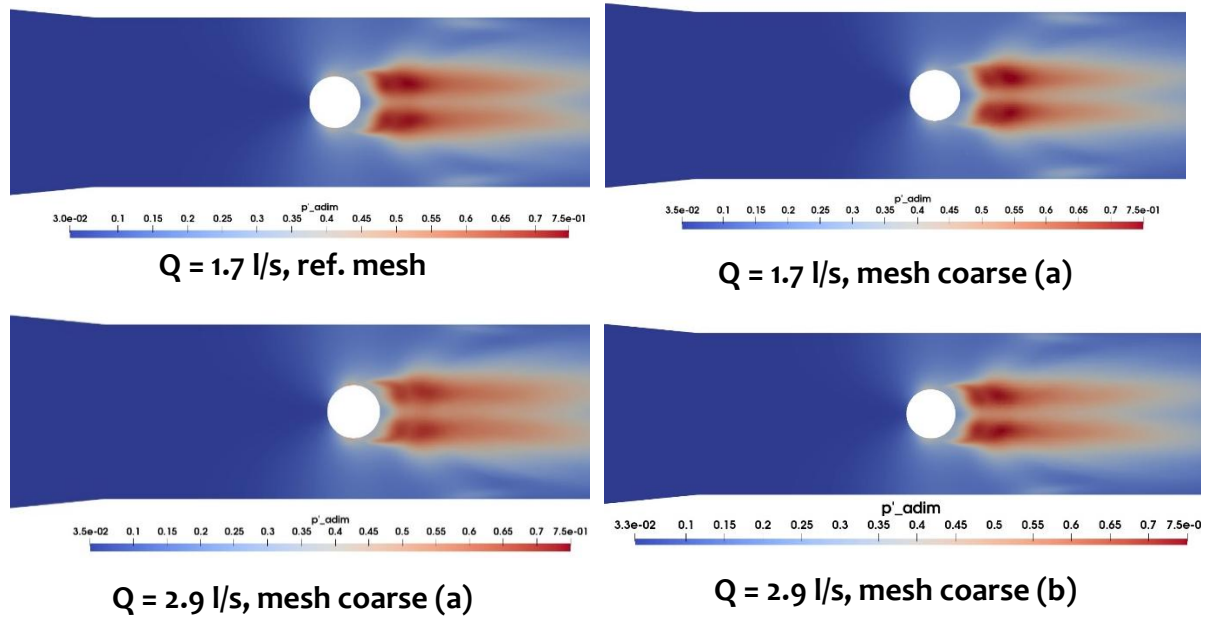


Figure 98: Time averaged non-dimensional pressure standard deviation in the centre plane at $Q = 1.7$ l/s (top) and $Q = 2.9$ l/s (bottom)

Figure 99 shows the time evolution of the mean drag coefficient and the standard deviation of lift coefficient. Despite the observed curves for the mean drag coefficient, its variations are low and it can be considered as converged quite fast. The standard deviation of the lift reaches a satisfactory convergence after at least 50 s of physical time for $Q = 2.9$ l/s and mesh coarse (a). The convergence seems to be reached for the same flowrate with mesh coarse (b) after “only” 25 s. This shows again the importance of checking the convergence for every case. The required physical time to obtain a decent convergence is of the order of 20 to 30 s.

Table 31 gives the computational and experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length and the main frequency of the lift force. Coarsening the meshes for $Q = 1.7$ l/s or for $Q = 1.9$ l/s only marginally affects the mean drag and the frequency peak. At $Q = 1.7$ l/s, going from the reference mesh to the coarse mesh (a) increases standard deviation of the drag by 6% and decreases the standard deviation of the lift by 14%. On the other hand, for $Q = 1.9$ l/s, going from the mesh coarse (a) to the mesh coarse (b) decreases the standard deviation of the drag by 8% and increases the standard deviation of the lift by 11.5 %. One can say that the change of the forces is of the order of 10% while coarsening the meshes. Finally, Figure 100 compares the power density spectra of the forces per unit of length with the experimental results. The frequencies of the lift force per unit of length are well predicted and the shapes of the spectra match with those of the experimental data. The experimental data do not exhibit a particular frequency for the drag force. One may notice that a secondary frequency is numerically predicted for the lift force for $Q = 2.9$ l/s as it is the case for $Q = 1.7$ l/s.

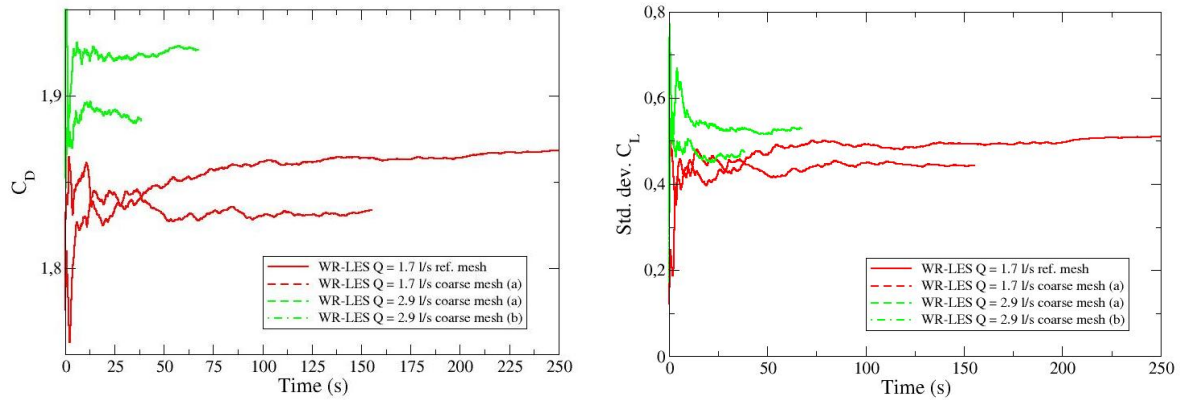


Figure 99: Time evolution of the mean drag coefficient $\overline{C_D}$ and the standard deviation of the lift coefficient $\overline{C_L'^2}$ at $Q = 1.7$ l/s for the reference and coarse (a) meshes and at $Q = 2.9$ l/s for coarse (a) and coarse (b) meshes

Table 31: The computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length and the main frequency of the lift force - sensitivity to mesh refinement at $Q = 1.7$ l/s and $Q = 2.9$ l/s for the single cylinder configuration

Q (l/s)	CFD/Exp.	mesh	time (s)	$\overline{F_D}$ (N/m)	$\overline{F_L}$ (N/m)	$\overline{F_D'^2}$ (N/m)	$\overline{F_L'^2}$ (N/m)	Fr_L (Hz)
1.7	Exp	-	410	-	-	0.106	0.122	8.8
1.7	CFD	ref.	406	2.054	0.00093	0.077	0.564	9.7
1.7	CFD	coarse (a)	155	2.012	0.00226	0.082	0.487	9.7
2.9	Exp	-	410	-	-	0.12	0.411	15.1
2.9	CFD	coarse (a)	67,5	6.152	-0.004	0.267	1.691	15.6
2.9	CFD	coarse (b)	39	6.021	0.00315	0.246	1.886	15.8

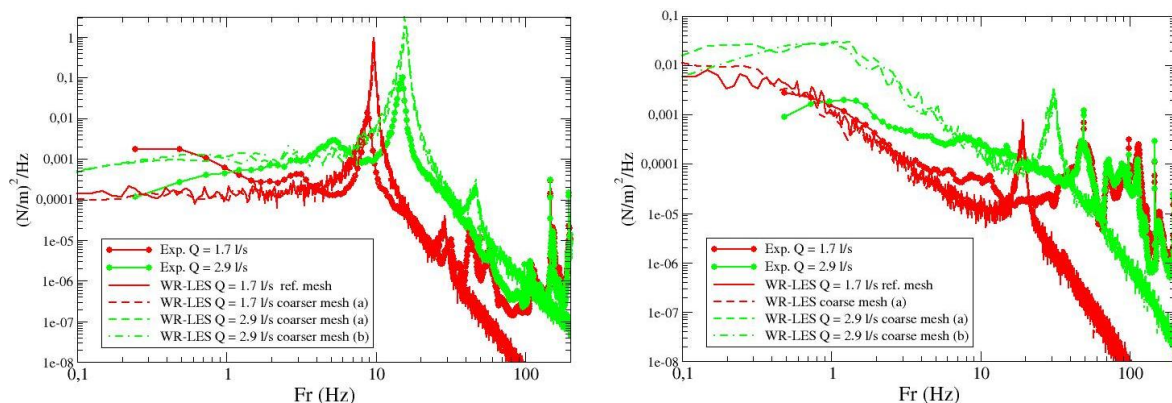


Figure 100: Power Density Spectra (PDS) of Lift (left) and drag (right) forces per unit of length at $Q = 1.7$ l/s for the reference and coarse (a) meshes and at $Q = 2.9$ l/s for coarse (a) and coarse (b) meshes – comparison with the experimental data

5.4.2 Single Flexible tube configuration results

5.4.2.1 Results at $Q = 0.5$ l/s and at $Q = 1.7$ l/s with the reference mesh

Table 32 gives the time step and *a posteriori* quantifications of the CFL number and the non-dimensional distance to the wall y^+ for $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes (see Table 23) for the single flexible tube configuration. Note that the two meshes contain 129.4 and 205.3 million cells, respectively. The quantities are in accordance with the previous observations made in 5.4.1.1. This is normal as the displacements are not that important.

Table 33 gives the computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length, the mean displacement of the cylinder and its standard deviation at $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes. The computed displacements are in better agreement than the forces in the rigid configuration with the experimental results for $Q = 1.7$ l/s. The displacements are overestimated by a factor of 3 for $Q = 0.5$ l/s and by “only” 60% for $Q = 1.7$ l/s.

Figure 101 and Figure 102 give the evolution in time of the mean displacement and its standard deviation for both flowrates with the reference meshes. The mean displacement goes very fast to small values (but non zero) whereas the standard deviation of displacement needs more than 20 s to converge at $Q = 1.7$ l/s. If the computation had been stopped after only 10 s, the standard deviation of the lift coefficient would have been underestimated by 20% (see Figure 102).

Table 34 gives the two main frequencies predicted by CFD for $Q = 0.5$ l/s and $Q = 1.7$ l/s. The frequencies of the lift and drag forces are also provided. The results of the experiment at $Q = 0.5$ l/s are not comparable to the CFD ones. For $Q = 1.7$ l/s there is a good agreement and this is confirmed by the spectra given in Figure 103.

Table 32: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes – single flexible cylinder configuration

Q (l/s)	Δt (s)	CFL _{max}	CFL _{max} Std.	CFL _{sp,av}	y^+_{max}	y^+_{max} Std.	$y^+_{sp,av}$
0.5	0.00055	1.06	0.012	0.16	2.08	0.21	0.43
1.7	0.00005	0.98	0.16	0.09	2.17	0.21	0.38

Table 33: The computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length, the mean displacement of the cylinder and its standard deviation at $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes – single flexible cylinder configuration

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)	$\overline{y}$ (mm)	$\overline{y'^2}$ (mm)
0.5	Exp	410	-	-	-	-	-	0.002
0.5	CFD	510	0.1651	0.00007	0.0020	0.00622	0.000004	0.0062
1.7	Exp	410	-	-	-	-	-	0.028
1.7	CFD	63.6	2.0643	-0.00070	0.0858	0.5983	0.000044	0.04403

Table 34: The main frequencies for the displacement, the drag and lift coefficient for $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes – comparison with the experimental data

Q (l/s)	CFD/Exp.	$Fr_{y,1}$	$Fr_{y,2}$	Fr_D	$Fr_{L,1}$	$Fr_{L,2}$
0.5	Exp	-	-	-	-	-
0.5	CFD	3.5	26	7.1	3.5	26
1.7	Exp	9.1	23.4	-	-	-
1.7	CFD	9.7	26	19.4	9.7	26

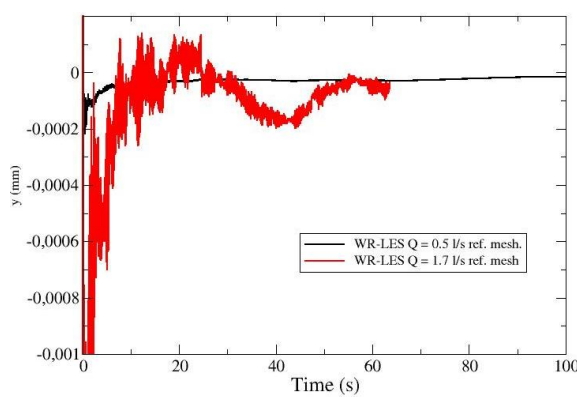


Figure 101: Time evolution of the cylinder displacement for the reference meshes at $Q = 0.5$ l/s and 1.7 l/s – single flexible cylinder configuration

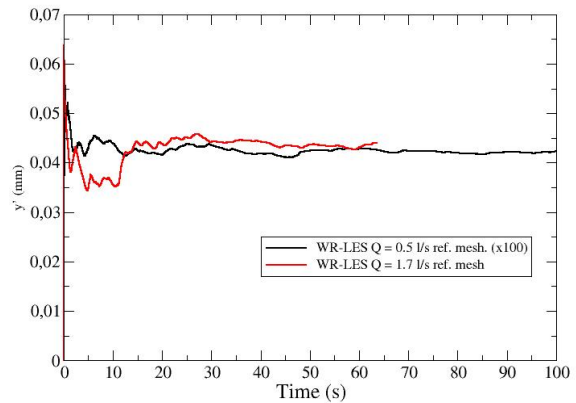


Figure 102: Time evolution of cylinder displacement standard deviation for the reference meshes at $Q = 0.5$ l/s and 1.7 l/s – single flexible cylinder configuration

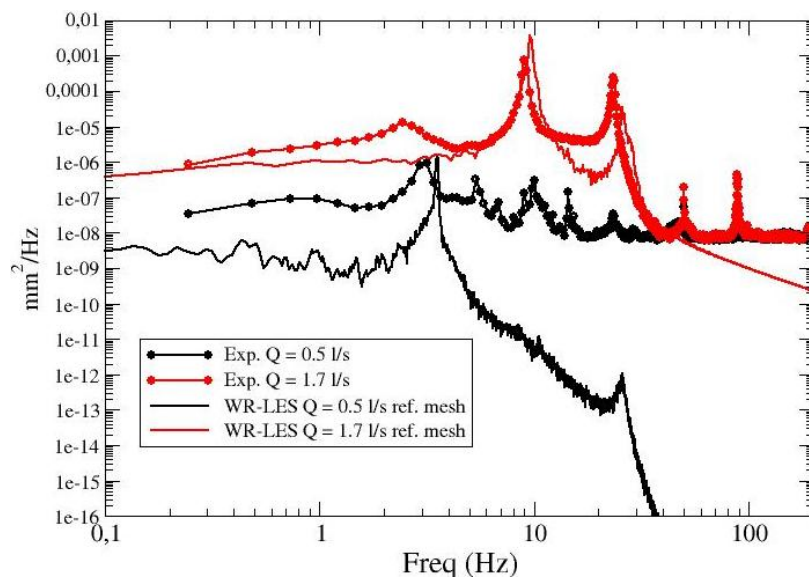


Figure 103: Power Density Spectra (PDS) of the tube displacement at $Q = 0.5$ l/s and $Q = 1.7$ l/s with the reference meshes – flexible single cylinder configuration – comparison with the experimental data

5.4.2.2 Results at $Q = 2.9$ l/s and $Q = 1.7$ l/s with a coarser mesh – sensitivity to the mesh refinement

Table 35 gives the time step and a posteriori quantifications of the CFL number and the non-dimensional distance to the wall y^+ at $Q = 1.7$ l/s with the reference mesh and mesh coarse (a) and at $Q = 2.9$ l/s with mesh coarse (b) (see Table 23) for the single flexible tube configuration. The quantities are in accordance with the previous observations made in 5.4.1.1. This is normal as the displacements are not that important. Note that the CFL has been decreased for $Q = 2.9$ with mesh coarse (b) compared to the single rigid configuration.

Figure 104 and Figure 105 give the evolution in time of the mean displacement and its standard deviation for both flowrates with the reference meshes. More than 20 s are again needed but this time for $Q = 2.9$ l/s to have a decent convergence.

Table 36 gives the computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length, the mean displacement of the cylinder and its standard deviation at $Q = 1.7$ l/s and $Q = 2.9$ l/s. Although the forces were largely overestimated at $Q = 2.9$ l/s in the rigid configuration, the standard deviation of the cylinder displacement is in very good agreement with the experimental data with a relative difference of 7%.

Table 37 gives the two main frequencies predicted by CFD for $Q = 1.7$ l/s and $Q = 2.9$ l/s. The frequencies of the lift and drag forces are also provided. For both flowrates there is a good agreement and this is confirmed by the spectra given in Figure 106. Note that for all the flowrates, the cylinder motion has the frequency of the lift force obtained for the rigid configuration (see sections 5.4.1.1 and 5.4.1.2), indicating the motion is due to the vortex shedding.

Table 35: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes and the coarse ones at $Q = 1.7$ l/s and $Q = 2.9$ l/s – single flexible cylinder configuration

Q (l/s)	CFD /Exp.	mesh	Δt (s)	CFL_{max}	$CFL_{max}^{Std.}$	$CFL_{sp,av}$	y^+_{max}	$y^+_{max}^{Std.}$	$y^+_{sp,av}$
1.7	Exp	-	0.001	-	-	-	-	-	-
1.7	CFD	ref.	0.00005	0.98	0.16	0.09	2.17	0.21	0.38
1.7	CFD	coarser (a)	0.00005	0.64	0.11	0.04	2.16	0.19	0.43
2.9	Exp	-	0.001	-	-	-	-	-	-
2.9	CFD	coarser (b)	0.00001	0.82	0.11	0.03	2.40	0.25	0.43

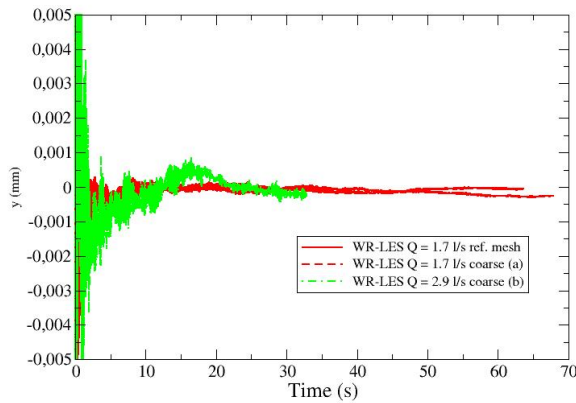


Figure 104: Time evolution of the cylinder displacement for the references meshes at Q = 1.7 l/s and 2.9 l/s – single flexible cylinder configuration

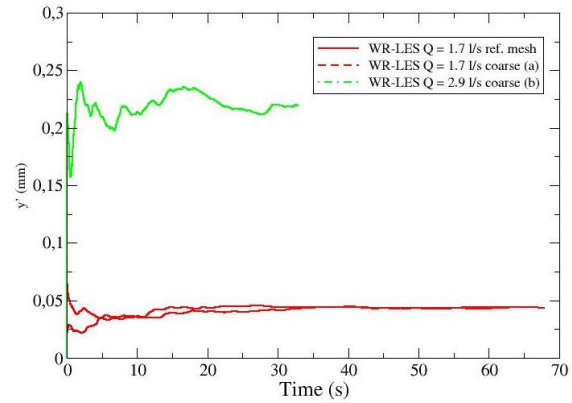


Figure 105: Time evolution of cylinder displacement standard deviation for the references meshes at Q = 1.7 l/s and 2.9 l/s – single flexible cylinder configuration

Table 36: The computational/experimental time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length, the mean displacement of the cylinder and its standard deviation at Q = 1.7 l/s and Q = 2.9 l/s – single flexible cylinder configuration

Q (l/s)	CFD/ Exp.	mesh	time (s)	$\overline{F_D}$ (N/m)	$\overline{F_L}$ (N/m)	$\overline{F_D'^2}$ (N/m)	$\overline{F_L'^2}$ (N/m)	$\overline{y}$ (mm)	$\overline{y'^2}$ (mm)
1.7	Exp.	-	410	-	-	-	-	-	0.028
1.7	CFD	ref.	63.6	2.064	- 0.0007	0.086	0.598	-0.00004	0.044
1.7	CFD	coarser (a)	67.9	2.126	- 0.0040	0.097	0.593	-0.00026	0.044
2.9	Exp.	-	410	-	-	-	-	-	0.205
2.9	CFD	coarser (b)	32	6.256	-0.0031	0.347	2.294	-0.00023	0.219

Table 37: The main frequencies for the displacement, the drag and lift coefficient for Q = 1.7 l/s and Q = 2.9 l/s with the reference meshes – comparison with the experimental data

Q (l/s)	CFD/Exp.	mesh	$Fr_{y,1}$ (Hz)	$Fr_{y,1}$ (Hz)	Fr_D (Hz)	$Fr_{L,1}$ (Hz)	$Fr_{y,2}$ (Hz)
1.7	Exp.	-	9.1	23.4	-	-	-
1.7	CFD	ref.	9.7	26	19.4	9.7	26
1.7	CFD	coarser (a)	9.9	26	19.4	10	26
2.9	Exp.	-	15.2	22.7	-	-	-
2.9	CFD	coarser (b)	15.4	25.6	30.6	15.5	25.6

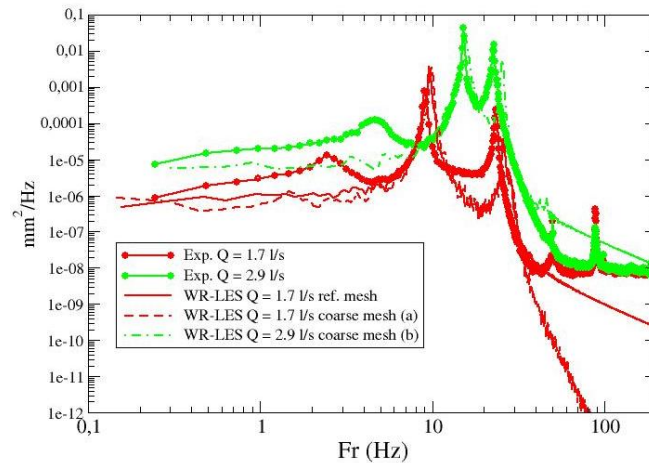


Figure 106: Power Density Spectra (PDS) of the tube displacement at $Q = 1.7$ l/s and $Q = 2.9$ l/s – flexible single cylinder configuration – comparison with the experimental data

5.5 Tube bundle configuration

Unless otherwise stated, all the post-processing in the following is made for the central moving/flexible cylinder (cylinder (3,2), with the first and second arguments being the column and the row of the bundle, respectively).

5.5.1 Rigid cylinder in a tube bundle

5.5.1.1 Results at $Q = 0.5$ l/s, $Q = 1.7$ l/s and $Q = 2.9$ l/s with the reference meshes

Table 38 gives the time step and *a posteriori* quantifications of the CFL number and the non-dimensional distance to the wall y^+ for the three main reference meshes (see Table 27 for more details about the mesh generation) of the three first AMOVI flow-rates for the flow through a rigid tube bundle. The description of the different values given in the table is available in section 5.4.1.1. The mean maximum CFL number is below 1 for the flowrates $Q = 0.5$ l/s and $Q = 1.7$ l/s. It is around 1.5 for the highest flowrate, which remains a very reasonable value, and the mean space average of the CFL is equal to 0.08.

The non-dimensional distances to the wall are also characteristic of a wall-resolved LES. The mean of the maximum non-dimensional distance to the wall is around 2 and its maximum standard deviation around 0.1, which shows that this maximum doesn't evolve a lot. The time-average of the space average of y^+ doesn't exceed 0.4.

Note that this section focuses on the flowrates $Q = 1.7$ l/s and $Q = 2.9$ l/s as the flow rate $Q = 0.5$ l/s seems to be too low, involving probably low Reynolds numbers effects making the analysis difficult at this flowrate.

Figure 107 shows instantaneous snapshots to confirm the mesh refinement at the wall. One recalls here that the aspect ratios of the near-wall cells allow also to capture very fine flow structures at the wall (see section 5.2).

Figure 108 shows an instantaneous snapshot and the time average of the non-dimensional streamwise velocity for $Q = 1.7$ l/s and $Q = 2.9$ l/s with the reference meshes. Figure 109 illustrates the standard deviation of the streamwise velocity component which gives an idea of the turbulence in this direction. One can first observe a bifurcation from one row to another. This can also be clearly observed on the non-dimensional pressure in Figure 110 (top) on which we observe alternating impinging jets from one row to another. For both flow rates the jets are on the top of the cylinders for rows 2 and 4 and on the bottom for rows 3 (the row of interest in the present report). The situation differs between the two flowrates for the last row (row 5). Indeed, the jets impinge on the top for $Q = 1.7$ l/s and on the bottom for $Q = 2.9$ l/s. This makes the wake anti-symmetrical with respect to the plane $y = 0$ for the two flowrates (however, the flow remains of the same kind). This kind of bifurcation has been observed in GOKSTAD experiment within the GO-VIKING project [33]. The analysis of this bifurcation goes beyond the scope of the present report. Note that this phenomenon requires to pay a particular attention to the convergence of the mean drag and lift forces and their standard deviation. There is no *a priori* reason to have a numerical solution maintained in a given stable position; it could switch from a stable position to another, and this switching may require a long time.

Comparisons with PIV experimental data are provided in section 6.4 for the two flowrates.

Figure 110 gives the mean pressure and its standard deviation for $Q = 1.7$ l/s and $Q = 2.9$ l/s. The alternating pattern is confirmed with the mean pressure. A maximum of pressure is reached at the impingement regions. One may observe, and this is particularly visible for $Q = 2.9$ l/s, that the tubes of the last two rows may have two regions upstream with high pressure fluctuations: the first one due to the impingement and the second one due to the shear turbulence coming from the upstream cylinder.

Table 38: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes at $Q = 0.5$ l/s, $Q = 1.7$ l/s and $Q = 2.9$ l/s – rigid tube bundle configuration

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

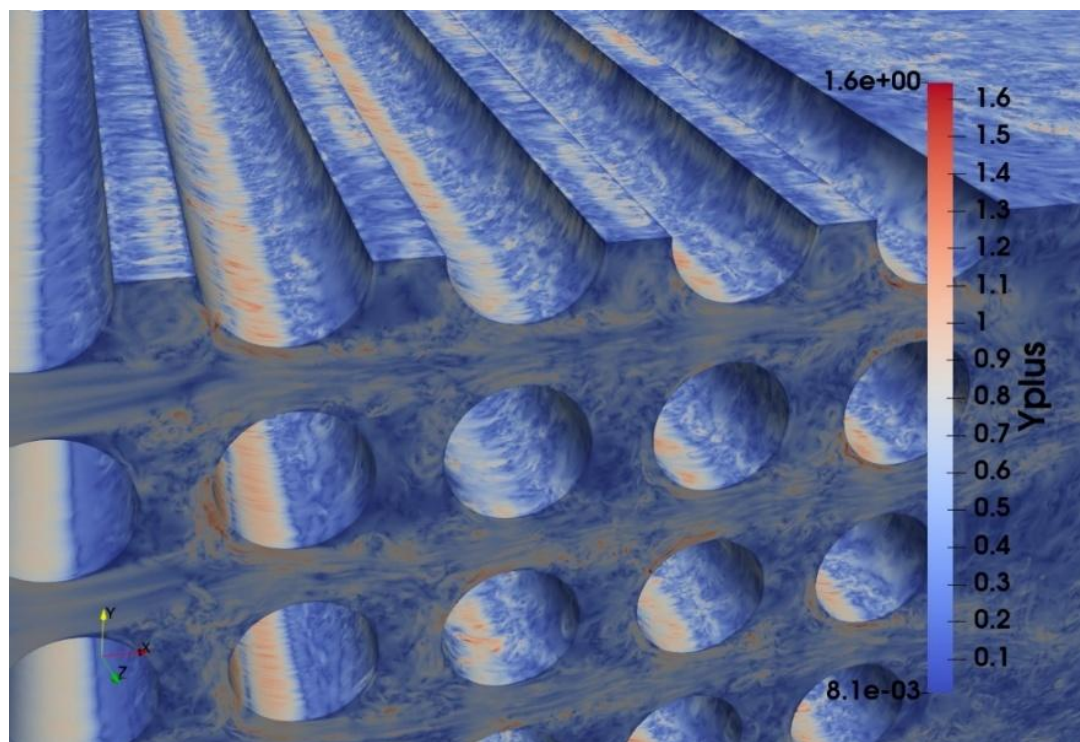
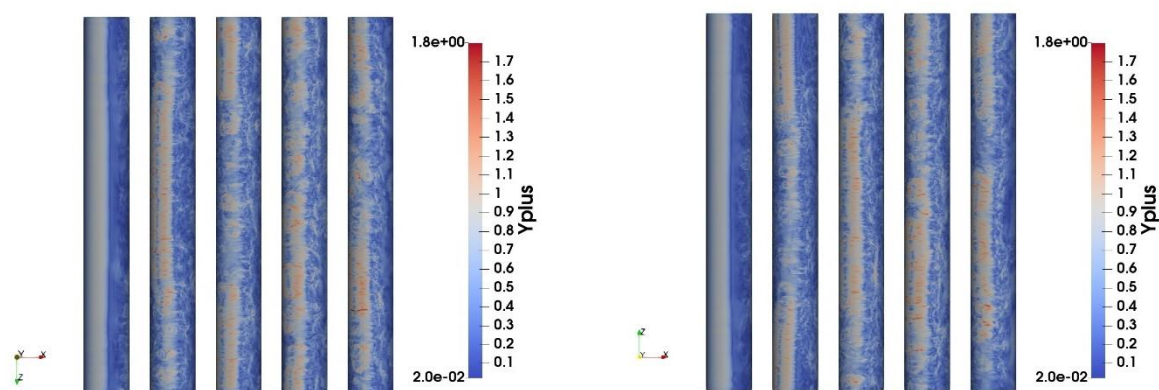


Figure 107: Instantaneous views of the non-dimensional distance at the wall – overview of the sidewalls (top) – view on the 5 central tubes the flow going from left to right (left: view from the bottom, right: view from the top)

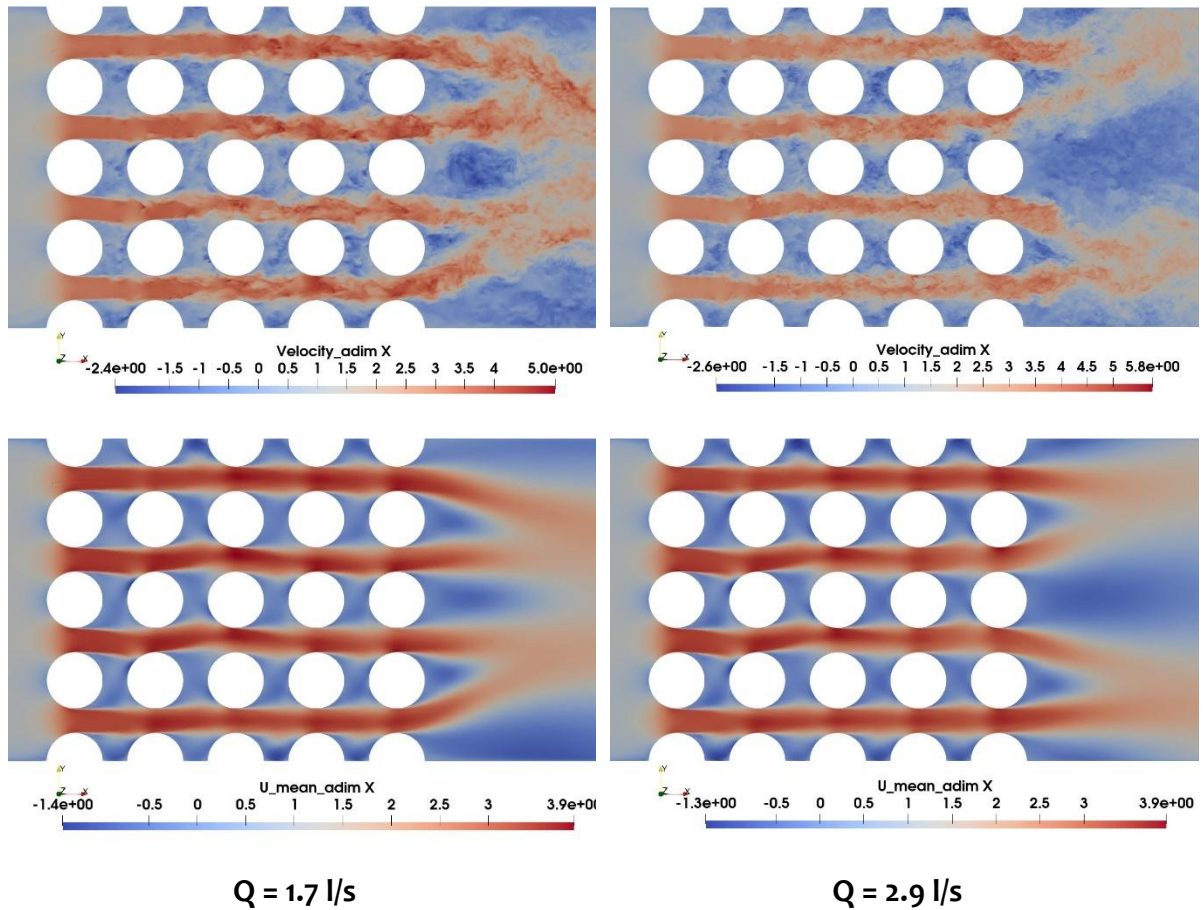


Figure 108: Instantaneous and time averaged non-dimensional stream-wise velocity in the center plane at $Q = 1.7$ l/s (left) and $Q = 2.9$ l/s (right) with the reference meshes - u/U_∞ (top), $\bar{u}/U_\infty$ (bottom)

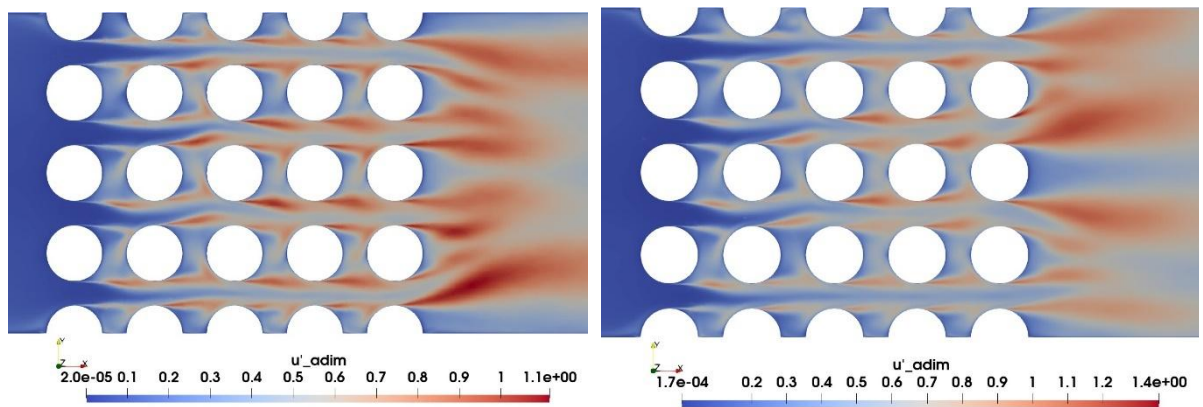


Figure 109: Standard deviation of the non-dimensional stream-wise velocity component at $Q = 1.7$ l/s (left) and $Q = 2.9$ l/s (right) with the reference meshes

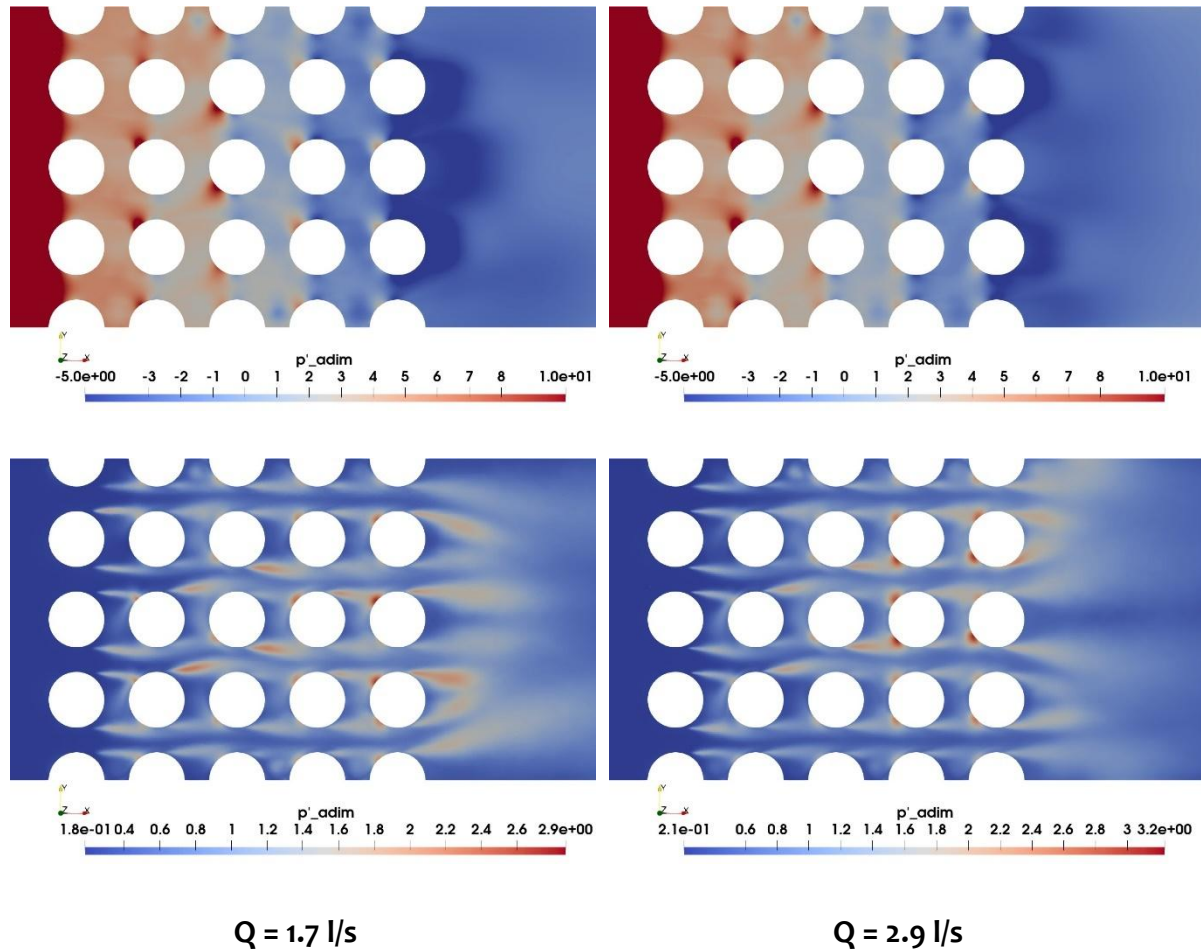


Figure 110: Time averaged (top) and standard deviation (bottom) of the pressure at $Q = 1.7 \text{ l/s}$ (left) and $Q = 2.9 \text{ l/s}$ (right) with the reference meshes – rigid tube bundle

$$\text{configuration} - \frac{\bar{p}}{0.5\rho U_\infty^2} \text{ (top)}, \frac{\sqrt{p'^2}}{0.5\rho U_\infty^2} \text{ (bottom)}$$

Based on what has been observed for the single cylinder configuration, a particular care must be given to the convergence of the drag and lift coefficients and their standard deviations. The flowrate $Q = 0.5 \text{ l/s}$ is first analysed. Figure 111 gives the time evolution of the standard deviation of the drag and lift coefficient. One may observe that after 500 s of physical time, the convergence of the drag and lift coefficients can be questionable. Figure 113 gives the time evolution of $\text{Err}(F'D)$ and $\text{Err}(F'L)$ (see section 5.4.1.1 for the definition of these errors). After 200 s, if one considers that the last value of the averaging is the right average, the standard deviation of the lift coefficient is estimated with less than 5% error, whereas 10% error is noticed for the drag. The same exercise, but with less computational physical time is performed for $Q = 1.7 \text{ l/s}$ and $Q = 2.9 \text{ l/s}$ in Figure 112 and Figure 114. A much longer computational time is needed to conclude at these two flowrates but one can say that a “plateau” is reached after 10 s of computation.

Table 39 gives the time used for averaging the drag and lift forces per unit of length. It also gives the time average and the standard deviation of these forces. 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 $Q = 1.7 \text{ l/s}$. The errors drop to 14% and 5% for $Q = 2.9 \text{ l/s}$. This is a satisfactory result, which shows that the more turbulent the case is, the better are the results of the wall-resolved LES concerning the forces applied to a cylinder in a tube bundle configuration.

Figure 115 and Figure 116 represent the power density spectra of the lift and drag force per unit of length applied to the central cylinder. The experimental spectra of $Q = 0.5$ l/s don't allow to exploit the result of this experiment. A particular behaviour is also obtained numerically, but the analysis of such a flow at a very low Reynolds number is beyond the scope of the present report. The predicted spectra of the central tube are in perfect agreement with the experimental data for $Q = 1.7$ l/s and $Q = 2.9$ l/s, which comforts the results obtained on the averaged quantities of Table 39.

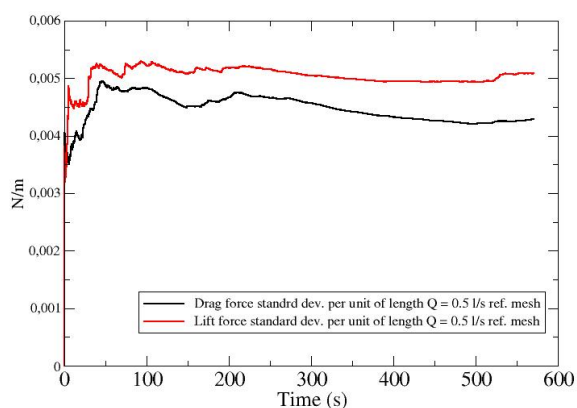


Figure 111: Time evolution of the drag and lift forces standard deviation per unit of length (F'_D and F'_L) as a function of time for $Q = 0.5$ l/s with the reference mesh

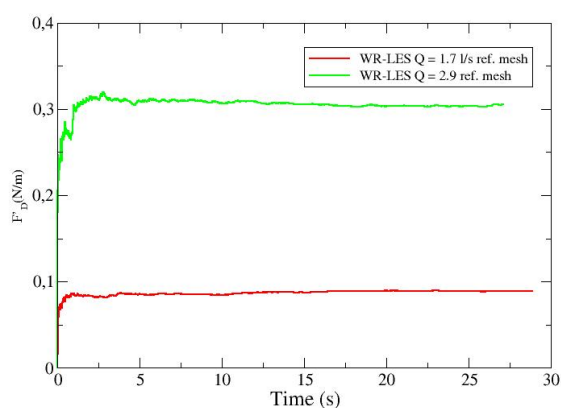


Figure 112: Time evolution of the drag force standard deviation per unit of length (F'_D) as a function of time for $Q = 1.7$ l/s and $Q = 2.9$ l/s with the reference meshes

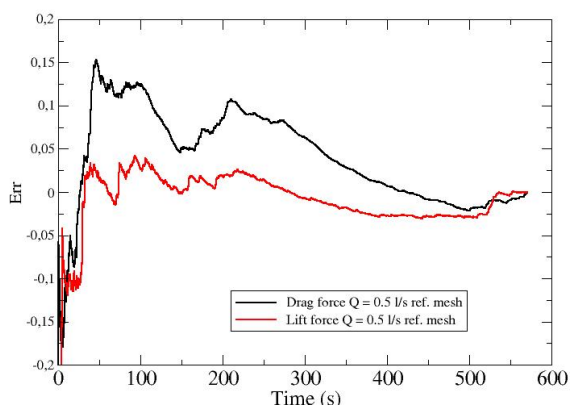


Figure 113: Time evolution of $\text{Err}(F'_D)$ and $\text{Err}(F'_L)$ for $Q = 0.5$ l/s with the reference mesh

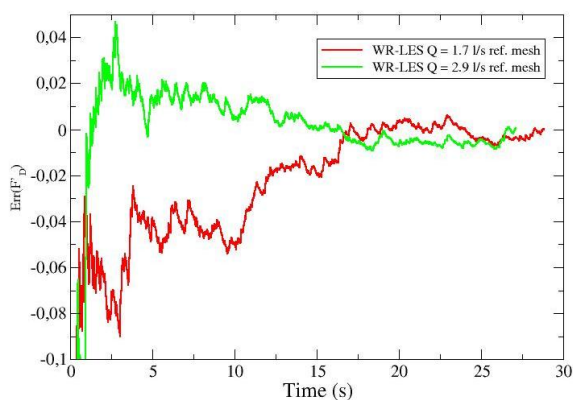


Figure 114: Time evolution of $\text{Err}(F'_D)$ for $Q = 1.7$ l/s and $Q = 2.9$ l/s with the reference mesh

Table 39: The computational/experimental time, the mean drag and lift forces per unit of length, and the standard deviation of the drag and lift forces per unit of length and the main frequency of the lift force - results of the reference mesh for $Q = 0.5$ l/s and $Q = 1.7$ l/s – rigid tube bundle configuration - comparison with 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

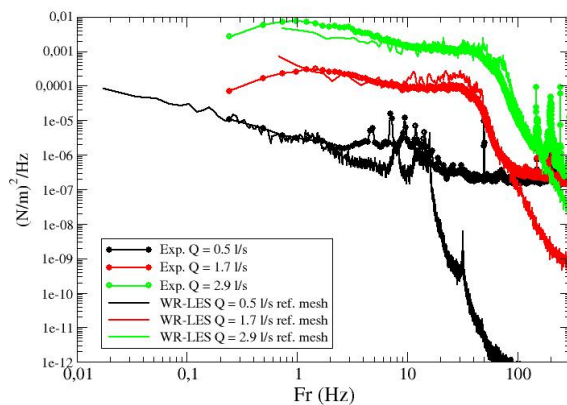


Figure 115: Power Density Spectra (PDS) of the lift forces per unit of length at $Q = 0.5$ l/s, $Q=1.7$ l/s and $Q=2.9$ l/s for the reference meshes – comparison with the experimental data

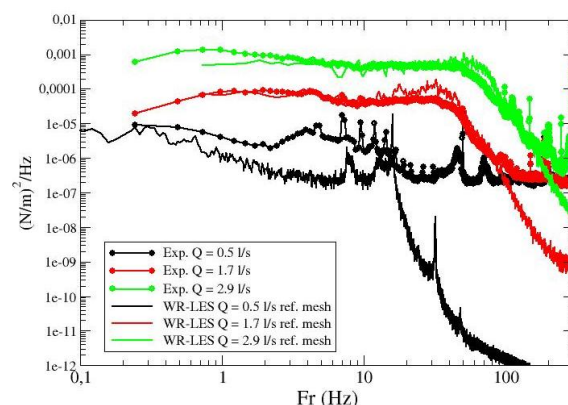


Figure 116: Power Density Spectra (PDS) of the drag forces per unit of length at $Q = 0.5$ l/s, $Q=1.7$ l/s and $Q=2.9$ l/s for the reference meshes – comparison with the experimental data

5.5.1.2 Results at intermediate flow rates

Nine additional flowrates have been computed with so-called reference meshes, adapted for every flowrate. Table 40 gives the physical computed time, the mean of the drag and lift forces per unit of length and their standard deviation.

The drag and lift coefficients have been computed using the pressure reference $0.5\rho U_\infty^2$ and the surface $4PL_z$ (see section 5.2).

Figure 117 gives the time average of the drag coefficient as a function of the flowrate. The drag coefficient increases from $Q = 0.5$ l/s to $Q = 1$ l/s or 1.1 l/s to reach a plateau around 0.65. A couple of intermediate flowrates should be computed between 1.95 and 2.9 l/s in order to conclude. Figure 118 gives the time average of the lift coefficient as a function of the flowrate. This coefficient should be zero if the flow is symmetrical with respect to the plane $y = 0$. However, it is not the case and values of the order of 0.01 are not negligible. The values might be positive or negative depending on the bifurcation of the flow. We notice that its sign is constant departing from $Q = 1.6$ l/s and its magnitude increases with the flowrate.

Figure 119 shows the standard deviation of the drag and lift coefficients as a function of the flowrate and compared to the experimental data. Figure 120 is the same figure, but with the forces per unit of length. There is a global increase of both coefficients, but stronger for the lift. The experiment also reflects this increase which doesn't depend only on the free-stream velocity. Again, intermediate points between 1.95 l/s and 2.9 l/s and above, when the computing power will allow it, should be computed to enrich these curves.

Table 40: The time-step, the computational time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length and the main frequency of the lift force - results of the reference mesh for several flow rates between $Q = 0.5$ l/s and $Q = 2.9$ l/s – rigid tube bundle configuration

Q (l/s)	Δt (s)	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	0.00025	570.9	0.072	-0.001	0.004	0.005
0.8	0.000125	60	0.244	0.008	0.014	0.016
1	0.0001	44	0.433	0.016	0.022	0.027
1.1	0.000075	55.7	0.591	-0.004	0.027	0.037
1.15	0.0001	116.6	0.640	-0.039	0.029	0.039
1.2	0.0001	25.5	0.680	0.035	0.031	0.042
1.6	0.00004	33.3	1.258	0.057	0.056	0.080
1.7	0.000035	28.9	1.394	0.017	0.063	0.089
1.85	0.000035	31.7	1.651	0.013	0.076	0.108
1.95	0.000025	26.7	1.846	-0.030	0.083	0.126
2.9	0.00002	27.1	3.802	-0.215	0.204	0.305

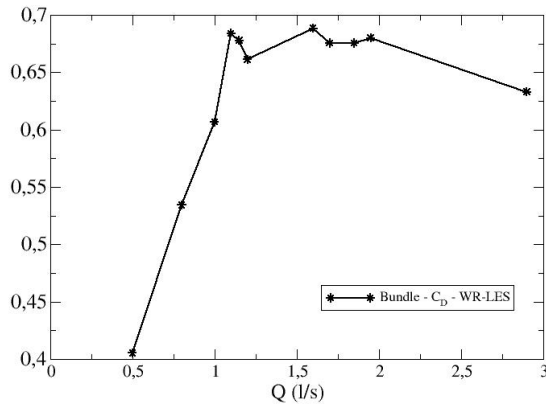


Figure 117: The time average of the drag coefficient of the central tube for the different flowrates of Table 40

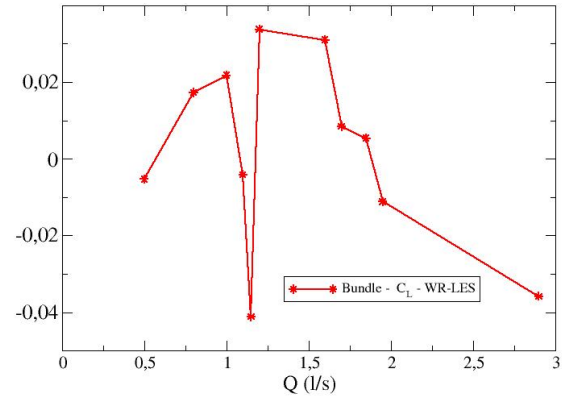


Figure 118: The time average of the lift coefficient of the central tube for the different flowrates of Table 40

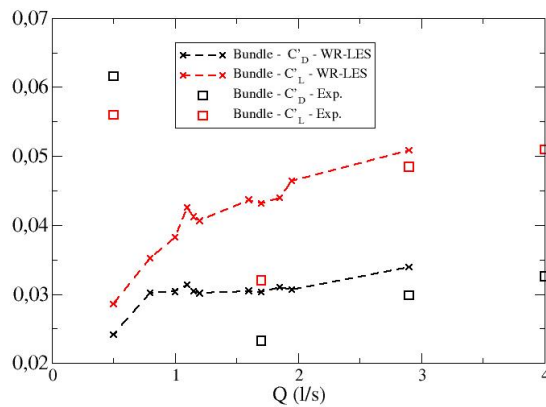


Figure 119: The standard deviation of the lift and drag coefficients for the different flowrates given in Table 40 – comparisons with the experimental data

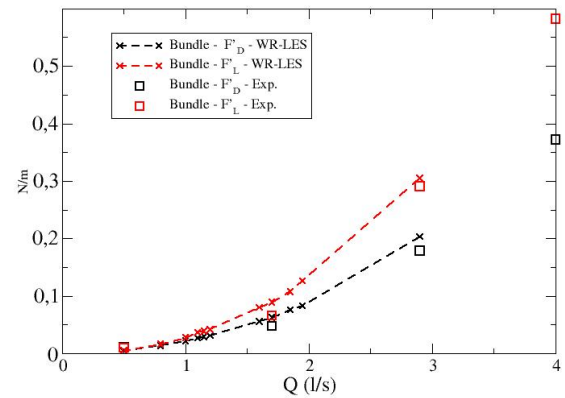


Figure 120: The standard deviation of the lift and drag forces per unit of length for the different flowrates given in Table 40 – comparisons with the experimental data

5.5.2 Single flexible tube in a rigid tube bundle

Four flowrates have been experimentally studied. $Q = 0.5$ l/s, $Q = 1.1$ l/s, $Q = 1.6$ l/s and finally $Q = 1.95$ l/s. The first subsection is dedicated to the results obtained for a moving cylinder at $Q = 0.5$ l/s and $Q = 1.6$ l/s with the reference meshes (see Table 26 for the refinements for $Q = 0.5$ l/s and $Q = 1.7$ l/s which is close to $Q = 1.6$ l/s).

A lock-in phenomenon is observed for the flowrate $Q = 1.1$ l/s. Understanding this topic goes beyond the scope of this report. This phenomenon occurs also at $Q = 1.05$ l/s and $Q = 1.15$ l/s at a frequency around 25 Hz, different from the natural frequency around 29 Hz of the dynamic system. This unexpected result has to be further investigated.

The reference mesh for $Q = 1.95$ l/s; the higher flowrate used with a moving cylinder in AMOVI experiment, is too fine to obtain a stable computation with a moving central cylinder having the mass and stiffness given in section 5.3. The tube displacements are too large for the mesh characteristic cell-size and negative volumes are obtained, causing the computations to crash. A coarser mesh is used later in section 5.5.2.2 for $Q = 2.9$ l/s.

5.5.2.1 Results at $Q = 0.5$ l/s, $Q = 1.6$ l/s with the reference meshes

The *a posteriori* maximum CFL and y^+ obtained with the two meshes in Table 41 are very similar to those obtained in the rigid configuration for $Q = 0.5$ l/s and $Q = 1.7$ l/s, respectively (see Table 32). The maximum standard deviation, obtained for $Q = 1.7$ l/s, is equal to $4 \cdot 10^{-5}$ m, which is only two times larger than the first computational at the cylinder wall and five times smaller than the characteristic cell size in the bulk regions (refer to Δy_{cyl} and Δy_{out} in Table 26). This explains why the figures are so close between the fully rigid tube bundle configuration and the one where a cylinder is vibrating.

Figure 121 shows the time average of the non-dimensional stream-wise velocity and the standard deviation of the non-dimensional pressure in the centre plane for $Q = 1.7$ l/s in the fully rigid configuration and for $Q = 1.6$ l/s in the tube bundle configuration having a central moving cylinder. The flow looks the same for the inner cylinder. However, the flow through the last row (row 5) exhibits two different wakes, but symmetrical with respect to the plane $y = 0$. The wakes are symmetrical, but exhibit two different possible patterns. The bi-stable issue doesn't only concern the inner cylinder, but also the wake around the bundle for which multiple solutions can be found or observed. This issue is behind the purpose of the present report, but should be kept in mind while dealing with tube bundle configurations.

Table 41: The time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for the reference meshes at $Q = 0.5$ l/s and $Q = 1.6$ l/s – flexible cylinder in a rigid tube bundle configuration

Q (l/s)	Δt (s)	CFL_{max}	$\text{CFL}_{\text{max}}^{\text{Std.}}$	$\text{CFL}_{\text{sp,av}}$	y^+_{max}	$y^+_{\text{max}}^{\text{Std.}}$	$y^+_{\text{sp,av}}$
0.5	0.00025	0.89	0.05	0.10	1.82	0.11	0.36
1.6	0.00004	1.05	0.12	0.07	2.04	0.11	0.37

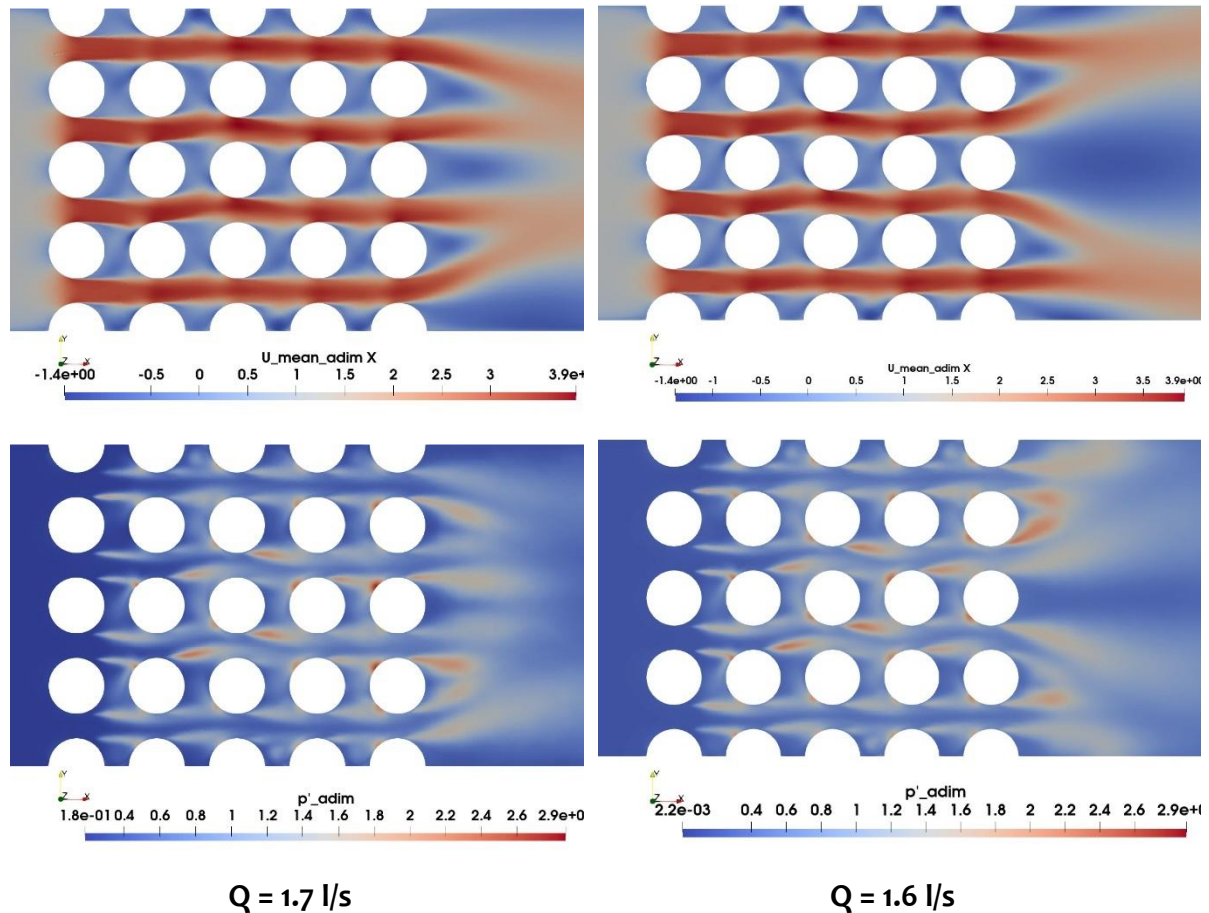


Figure 121: Time average of the non-dimensional mean stream-wise velocity (top) and standard deviation of the non-dimensional pressure (bottom) at $Q = 1.7$ l/s for the rigid tube bundle configuration (left) and at $Q = 1.6$ l/s for the single flexible cylinder in a tube bundle (right) – reference meshes – rigid tube bundle configuration

Figure 122 shows the time evolution of the standard deviation of the displacement for $Q = 0.5$ l/s and $Q = 1.6$ l/s. The displacement of the cylinder with $Q = 0.5$ l/s doesn't seem to be converged even after 400 s. This is confirmed by the plot of $Err(y')$ in Figure 123. If one considers the standard deviation at the last-time step as the final results, after 100 s, we can estimate the uncertainty to be around 8%! However, the error remains below this value after further simulation time (not shown here). The time available for $Q = 1.6$ l/s is much lower but the figures give an idea of the need for few tens of seconds to obtain an uncertainty lower than let's say 5%, at least 20 to 30 s in the present case. The computation should be run actually for a longer time such as a couple of minutes to refine the analysis.

Table 42 gives the physical time, the mean drag and lift forces per unit of length, the standard deviation of the drag and lift forces per unit of length, the mean displacement of the cylinder and its standard deviation at $Q = 0.5$ l/s and $Q = 1.6$ l/s with the reference meshes. The forces will be used later in the next section 5.5.2.2. While the computed lift coefficient was only underestimated by a factor of two compared to the experiment for $Q = 0.5$ l/s (see Table 39) in the rigid tube bundle configuration, the computed displacement is underestimated by a factor of 10. There is no explanation for the moment for this observation although one may suspect the different inlet boundary conditions between the experiment and the numerical simulation, in particular at the low

Reynolds number of the configuration (see Table 25). The numerical results are much closer to the experimental ones for $Q = 1.7$ l/s and have at least the right order of magnitude. However, one still has an inconsistency in the sense that the computations overestimate the standard deviation of the lift coefficients by 35% for $Q = 1.7$ l/s in the rigid tube bundle whereas the displacements are underestimated by 44% for the moving cylinder one.

Finally, Figure 124 shows the computed spectra for the lift and drag forces per unit of length compared to available experimental data. It is again clear that the results at the low flowrate $Q = 0.5$ l/s are not really usable. The computed spectrum at $Q = 1.6$ l/s shows a good agreement with the experimental data. The peak frequency of the experiment and the computation are located at 22.53 and 25 Hz, respectively. The natural experimental frequency of a tube as reported in the experiment and in the computation are equal to 27 Hz and 29.4 Hz, respectively. The trend between CFD and experiment is conforming; the oscillation frequency of the tube in a bundle 4.4 Hz to 4.5 Hz lower than the natural frequency.

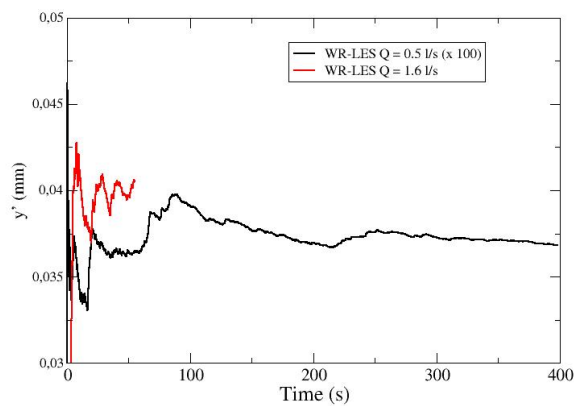


Figure 122: The standard deviation of the displacement as a function of time for $Q = 0.5$ l/s and $Q = 1.6$ l/s – reference meshes - flexible cylinder in a rigid tube bundle configuration

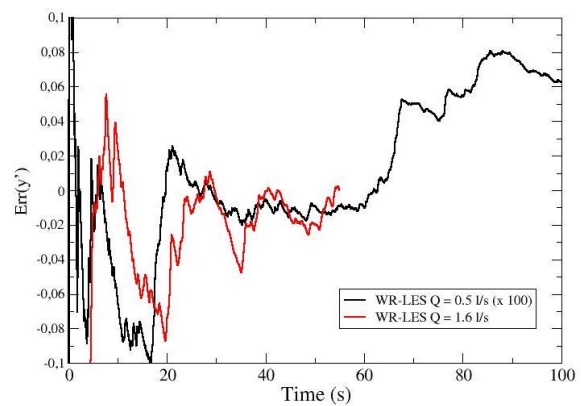


Figure 123: $Err(y')$ as a function of time for $Q = 0.5$ l/s and $Q = 1.6$ l/s – reference meshes - flexible cylinder in a rigid tube bundle configuration

Table 42: The computational/experimental physical time, the mean displacement of the cylinder and its standard deviation and the dominant frequency at $Q = 0.5$ l/s and $Q = 1.6$ l/s with the reference meshes – single flexible cylinder in a rigid tube bundle configuration

Q (l/s)	CFD/Exp.	time (s)	$\bar{y}$ (mm)	$\overline{y'^2}$ (mm)	Fr_y
0.5	Exp.	410		0.0030	-
0.5	CFD	398.3	-0.00008	0.0003	-
1.6	Exp.	410		0.0730	22.53
1.6	CFD	55	0.00360	0.0405	25

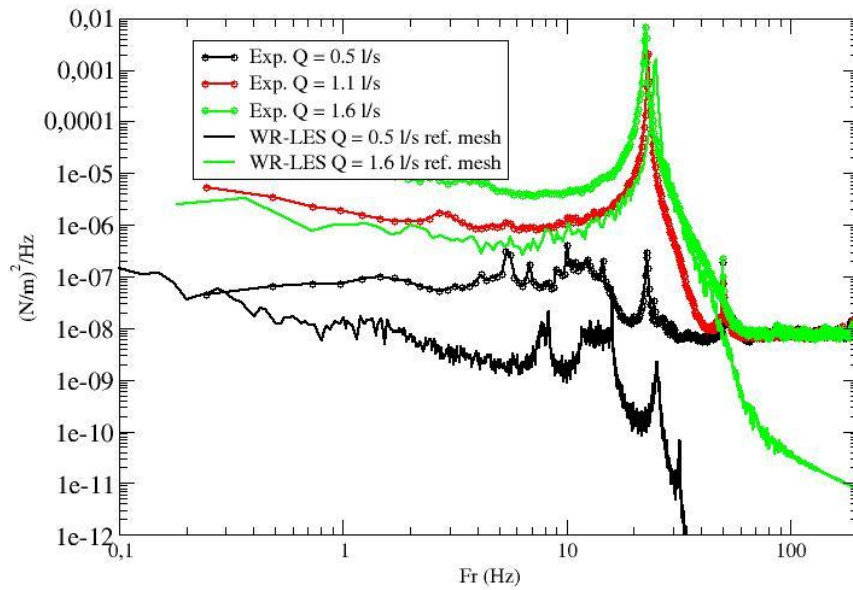


Figure 124: Power Density Spectra (PDS) of the tube displacement at $Q = 0.5$ l/s and $Q = 1.6$ l/s with the reference meshes – flexible cylinder in a rigid tube bundle configuration – comparison with the experimental data

5.5.2.2 Fluid elastic instability thanks to coarser meshes ?

A coarser mesh than the reference one has been used to obtain a result at $Q = 1.95$ l/s. The details are given in Table 27. The mesh is coarser in the azimuthal direction of the cylinders and this leads to wider cells in the bulk. This coarser mesh for $Q = 1.95$ l/s has not been created with the “frontal Delaunay for quad” option of Gmsh but only with the “frontal Delaunay”. As the computations stopped for $Q = 1.1$ l/s (so-called here “lock-in” phenomenon), other flowrates have been added in the analysis. The computations also stop with $Q = 1.05$ l/s and $Q = 1.15$ l/s. The mesh used for $Q = 1.95$ l/s has also been used for $Q = 1.05$ l/s, $Q = 1.1$ l/s and $Q = 1.15$ l/s without success. More investigations are needed for this particular point and no numerical results are available for $Q = 1.1$ l/s. The quality of all the computations are the same except for $Q = 1.85$ l/s and $Q = 1.95$ l/s for which the mesh had to be refined. The computation at $Q = 1.85$ l/s uses the same mesh as the one at $Q = 1.95$ l/s.

One has at this stage to check the quality of the results at $Q = 1.95$ l/s with the coarsened mesh.

Table 43 gives the details on the CFL and the wall refinement for the reference and coarsened meshes at $Q = 1.95$ l/s in the rigid tube bundle configuration as well as for the coarsened mesh with a moving cylinder. One can notice that the coarsening didn’t impact these figures.

Figure 125 shows the mean non-dimensional stream-wise velocity and the mean non-dimensional pressure fields for $Q = 1.95$ l/s with the reference and the coarsened mesh. The two solutions don’t bifurcate to the same pattern and are symmetrical with respect to the plane $y = 0$ with the same behaviour in the wake of the bundle downstream of the last row of cylinders.

Table 46 gives the mean drag and lift forces per unit of length and their standard deviation for the reference and the coarsened mesh at $Q = 1.95$ l/s. It is very difficult to quantify the difference on the mean lift. For the mean drag, its standard deviation and the lift standard deviation, the relative difference is equal to 10%, 14.5% and 11%, respectively. This is reasonable, in particular if one considers

the fact that the computations are not fully converged. Thus, the coarsened mesh is used with ALE method to compute fluid structure interaction.

Figure 126 represents the spectra of the tube displacement for $Q = 1.6$ l/s and $Q = 1.95$ l/s to compare them to the experimental results. The main computed frequency for $Q = 1.95$ l/s is equal to 24.8 Hz and the experimental one is equal to 23.56 Hz. One recalls that the obtained frequencies for $Q = 1.6$ l/s are 25 Hz and 22.53 Hz.

Table 45 gives the mean and standard deviation of the tube displacements for the different flowrates and the oscillation frequency. The available experimental data are added in the table for the standard deviation. All the CFD frequencies are close to 25 Hz whereas the experimental ones are rather close to 23 Hz. The agreement is good and the difference with the natural frequency is the same in the experiments and the computations.

The standard deviations of the tube displacements are reported in Figure 127. The measured experimental displacement for $Q = 1.95$ l/s is out of range (3.223 mm) and is not represented. Obviously, the computation at the same flowrate underestimates it drastically. The predicted values are reasonable for the intermediate flowrates in the range $[0.8$ l/s, 1.85 l/s]. They don't show a linear behaviour as in the experiment. The displacements at $Q = 1$ l/s and at 1.6 l/s are comparable which may be explained by the "lock-in" observed in the range $[1.05$ l/s, 1.15 l/s]. Moreover, there is a sudden increase departing from $Q = 1.85$ l/s, which is characteristic of a Fluid Elastic Instability. Table 46 gives the drag and lift forces per unit of length and their standard deviation applied to the central moving tube. Figure 128 and Figure 129 show the evolution of the mean and standard deviation of the drag force per unit of length for the different flowrate and compare them with the results obtained with the rigid tube bundle. One can see that the differences are marginal. However, the standard deviation of the lift force per unit on length (see Figure 130) differs between the two configurations and this quantity follows the behaviour of the tube displacement standard deviation. The Fluid Elastic Instability is thus captured with wall-resolved LES and a critical velocity can be estimated.

Table 43: The mesh type, the physical time, the time step, the maximum CFL number and the maximum non-dimensional distance to the wall y^+ for $Q = 1.95$ l/s – flexible tube in a rigid tube bundle configuration

Q (l/s)	mesh	Time (s)	Δt (s)	CFL_{max} x	CFL_{max} x Std.	$CFL_{sp,av}$	y^+_{max}	y^+_{max} Std.	$y^+_{sp,av}$
1.95	ref. rigid	26.7	0.000025	0.97	0.116	0.05	2.07	0.11	0.355
1.95	coarse r rigid	57.5	0.00003	1.17	0.16	0.054	2.04	0.117	0.343
1.95	coarse r flexible	27.1	0.00002	0.8	0.11	0.036	2.08	0.12	0.34

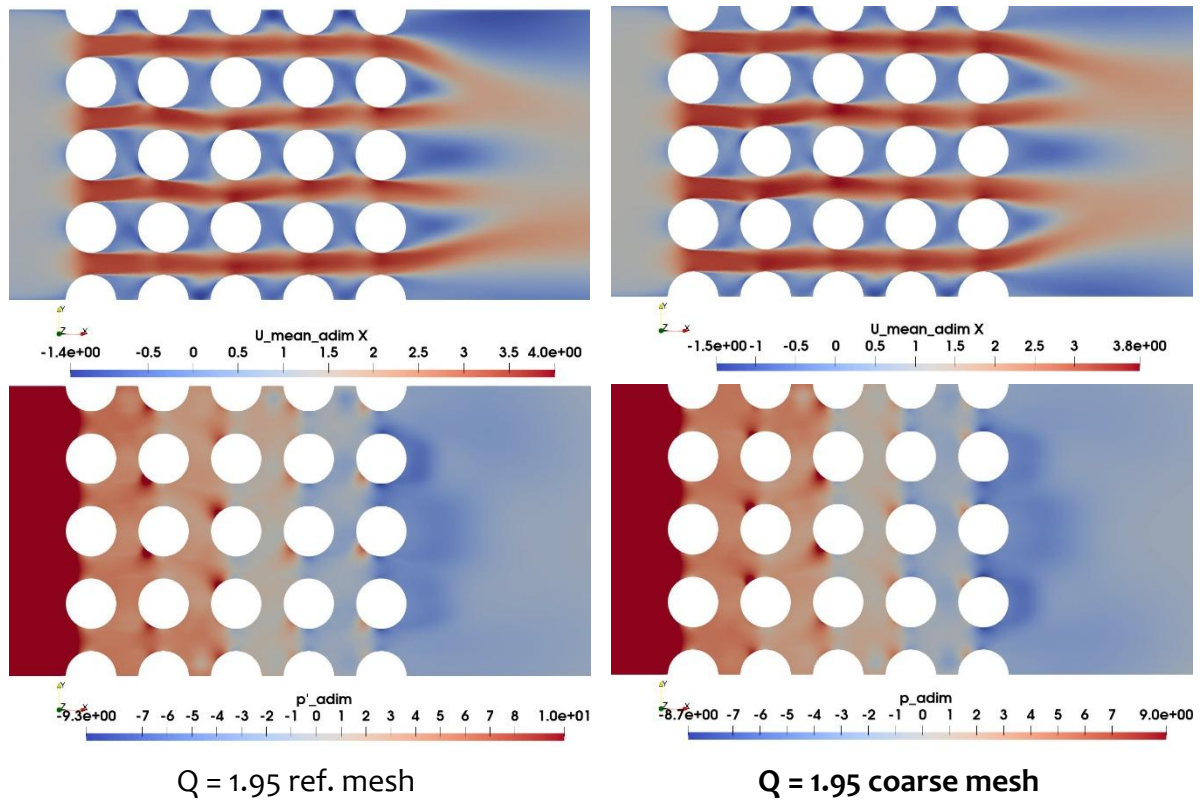


Figure 125: Time averaged non-dimensional stream-wise velocity and pressure in the center plane at $Q = 1.95$ l/s with the reference (left) and the coarse (right) meshes – rigid tube bundle

Table 44: The mesh type, the physical time, the mean drag and lift forces per unit of length, and the standard deviation of the drag and lift forces per unit of length – $Q = 1.95$ l/s with the reference and the coarsened mesh – rigid tube bundle configuration

Q (l/s)	mesh	Δt (s)	Time (s)	$\overline{F_D}$ (N/m)	$\overline{F_L}$ (N/m)	$\overline{F_D'^2}$ (N/m)	$\overline{F_L'^2}$ (N/m)
1.95	ref.	0.00025	26.7	1.846	-0.03	0.083	0.126
1.95	coarser	0.00003	57.5	1.661	-0.073	0.095	0.14

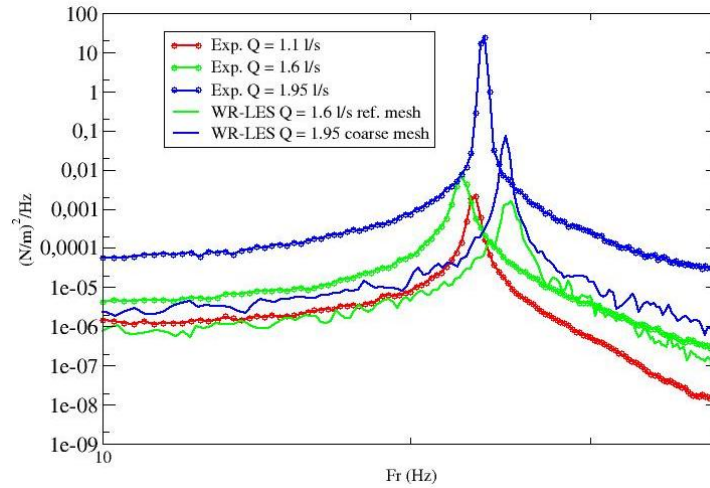


Figure 126: Power Density Spectra (PDS) of the tube displacement at $Q = 1.6$ l/s and $Q = 1.95$ l/s– flexible cylinder in a rigid tube bundle configuration – comparison with the experimental data

Table 45: The mean displacement of the cylinder, its standard deviation and its main frequency for different flowrates – the meshes for $Q = 1.85$ and $Q = 1.95$ l/s are coarsened compared to the reference one – flexible cylinder in a rigid tube bundle configuration

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

Table 46: The mean drag and lift forces per unit of length and their standard deviation for different flowrates – the meshes for $Q = 1.85$ and $Q = 1.95$ l/s are coarsened compared to the reference one – flexible cylinder in a rigid tube bundle configuration

Q (l/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	-	-	-	-
0.5	0.0721	-0.0012	0.0044	0.0051
0.8	0.2518	0.0078	0.0138	0.0225
1	0.4432	0.0179	0.02226	0.127
1.1	-	-	-	-
1.6	-	-	-	-
1.6	1.2603	0.0553	0.0566	0.1839
1.85	1.5413	-0.0569	0.0802	0.3669
1.95	-	-	-	-
1.95	1.668	-0.0613	0.0947	0.8071

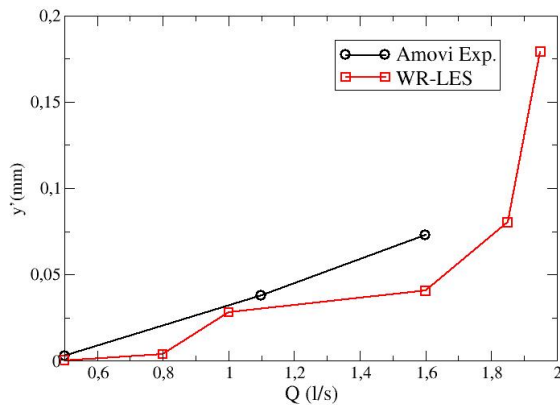


Figure 127: Standard deviation of the tube displacement for different flowrates compared with experimental data – flexible cylinder in a rigid tube bundle configuration

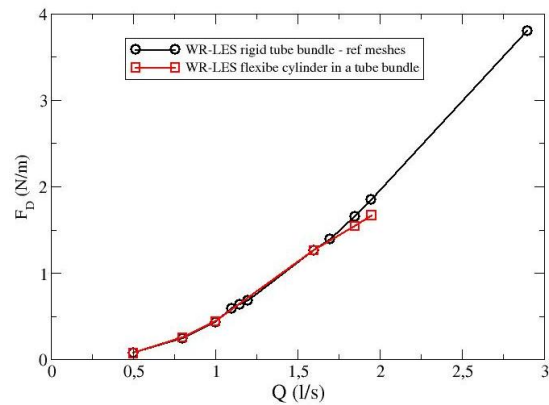


Figure 128: Mean drag force per unit of length on the central moving tube for different flowrates – comparison between ‘the flexible cylinder in a rigid tube bundle’ configuration and ‘the rigid tube bundle’ one

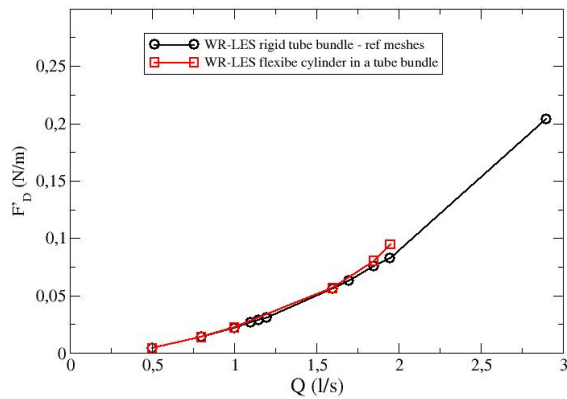


Figure 129: Standard deviation of the drag force per unit of length on the central moving tube for different flowrates – comparison between “the flexible cylinder in a rigid tube bundle” configuration and “the rigid tube bundle” one

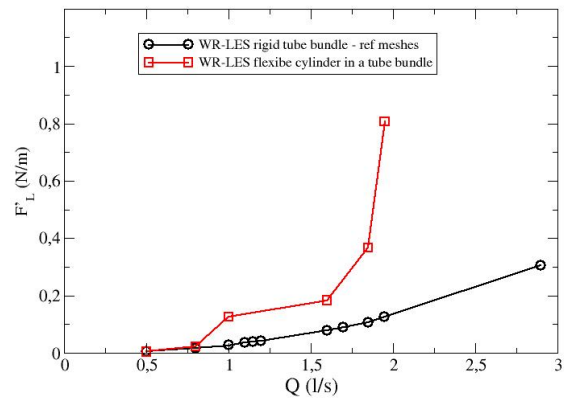


Figure 130: Standard deviation of the lift force per unit of length on the central moving tube for different flowrates – comparison between “the flexible cylinder in a rigid tube bundle” configuration and “the rigid tube bundle” one

5.6 Conclusions

Two common conclusions from the single cylinder and the tube bundle configurations can be drawn:

- The required physical time to obtain a decent convergence for the standard deviation of the lift force or the cylinder displacement is of the order of 20 to 30 s for both the rigid and flexible configurations. The required physical time to reach a decent convergence might be worse for low flowrates.
- Large displacements need the use of coarsened meshes and the quality of the results obtained for both the forces and the displacements remains reasonable within 10% to 15% of the ones obtained with the finer meshes.

The remaining conclusions are different.

The particular conclusions for the single cylinder configuration are the following:

- Large differences are obtained for the lift and drag standard deviations for the single rigid cylinder configuration (very large overestimation). However, the lift spectra for both $Q = 1.7$ l/s and $Q = 2.9$ l/s are satisfactory and show very satisfactory main frequencies. No clear frequency is obtained in the experiment for the drag force which might question the quality of inlet boundary conditions in AMOVI.
- A double peak is obtained for $Q = 1.7$ l/s and $Q = 2.9$ l/s on the displacement spectra for the single flexible cylinder configuration and this is in agreement with the experimental results.
- Although the forces are largely overestimated at $Q = 2.9$ l/s in the rigid configuration, the standard deviation of the cylinder displacement is in very good agreement with the

experimental data with a relative difference of 7%. The displacement prediction seems to be enhanced at high Reynolds numbers.

The conclusions for the tube bundle configuration are the following:

- The predicted standard deviations of the forces in the rigid configuration are in better agreement with the experimental data than in the single rigid cylinder configuration. The errors drop to 14% and 5% for $Q = 2.9$ l/s, for the drag and lift standard deviations, respectively. As it is the case for the displacements in the single flexible configuration, the more the flow is turbulent, the better are the results.
- A bifurcation depending on the tube column is observed in the numerical results. The bi-stable issue doesn't only concern the inner cylinders but also the wake around the bundle for which multiple solutions can be obtained. This issue is behind the purpose of the present report but should be kept in mind while dealing with tube bundles.
- In the tube bundle configuration containing a flexible cylinder, there is a good prediction of the frequencies and a reasonable prediction of the displacements.
- The Fluid Elastic Instability phenomenon is predicted with the present wall-resolved LES simulations.

6 Simulations results summary: the AMOVI benchmark

6.1 Introduction

To conduct the final comparative analysis within the benchmark, the participants agreed on a set of cases of interest. Table 47 summarizes which cases were performed by each Institution. From Table 47 it can be seen that, for further comparison and analysis, all types of rod configuration were provided by each institution participating in the AMOVI benchmarking. The main details of the benchmark cases performed have been summarized in the corresponding subsections.

Table 47: Performed cases

Institution	Cases Performed						
	Single Rigid Tube (SRT)	Single Flexible Tube (SFT)		Rigid Tube Bundle (RTB)		Flexible Tube in a rigid tube bundle (FTRTB)	
EDF	SRT-290 (Q=2.9l/s)	SFT-170 (Q=1.7l/s)	SFT-290 (Q=2.9l/s)	RTB-170 (Q=1.7l/s)	RTB-290 (Q=2.9l/s)	FTRTB-160 (Q=1.6l/s)	FTRTB-195 (Q=1.95l/s)
GRS	SRT-290 (Q=2.9l/s)	SFT-170 (Q=1.7l/s)	SFT-290 (Q=2.9l/s)	RTB-170 (Q=1.7l/s)	RTB-290 (Q=2.9l/s)	FTRTB-160 (Q=1.6l/s)	-
IPP	SRT-290 (Q=2.9l/s)	SFT-170 (Q=1.7l/s)	SFT-290 (Q=2.9l/s)	RTB-170 (Q=1.7l/s)	RTB-290 (Q=2.9l/s)	FTRTB-160 (Q=1.6l/s)	FTRTB-195 (Q=1.95l/s)

6.2 Single Rigid Tube cases

Among the available cases with the Single Rigid Tube configuration, participants agreed to perform a simulation for Q=2.9 l/s.

Table 48 – Table 51 provide the main information on the SRT-290 simulations performed by each involved Institution:

- Table 48 summarizes the participants' individual submissions reported by EDF, GRS and IPP on Single Rigid Tube cases.
- Table 49 provide the calculation domains details for Single Rigid Tube cases.
- Table 50 provides information on the Boundary Conditions applied for Single Rigid Tube cases.
- Table 51 summarizes the numerical settings applied by each participant.

Table 48: SRT - Key information of the submissions

Institution	Simulation Case Type	Approach	Formulation	Turbulence Modelling	Solid Modelling	Software
EDF	SRT	Pure CFD	3D	Wall-resolved LES	NA	Code Saturne Ver 8.0
GRS	SRT	Pure CFD	3D	Hybrid LES-URANS – SAS SST	NA	ANSYS CFX 2021 R2
IPP	SRT	Pure CFD	2D	URANS - SST	NA	ANSYS CFX 2018

Table 49: SRT - Calculation Domain Details

Institution	Case	Geometry Model	Mesh Cell Type	Mesh Cell Count, Cells	Wall Y+
EDF	SRT-290	3D Fluid Domain	Hexahedral	296M	<2 everywhere
GRS	SRT-290	3D Fluid Domain includes 255 mm upstream the tube and 137.5 mm downstream	Hexahedral	5.156M	Central Tube= 0.32; Walls = 0.37;
IPP	SRT-290	2D Fluid Domain represented by the central cross section and includes 57.5 mm upstream the central tube and 137.5 mm downstream.	Hexahedral	8.7k	Central Tube= 3.2; Walls = 23.0;

Table 50: SRT - Fluid Boundary Conditions

Institution	Case	Inlet BC	Outlet(s) BC	Tube Surface	All other walls
EDF	SRT-290	Fully developed using embedded precursor LES	Free Outlet	wall resolved (no-slip)	wall resolved (no-slip)
GRS	SRT-290	Fully developed inlet flow profile (x,y,z)	BOUNDARY TYPE = Opening; MASS AND MOMENTUM = Opening Pressure and Direction; TURBULENCE = Medium Intensity and Eddy Viscosity Ratio;	No slip, smooth wall	No slip, smooth wall
IPP	SRT-290	Fully developed inlet flow profile	“Opening” boundary type with Medium Intensity of Turbulence	No slip, smooth wall	No slip, smooth wall

Table 51: SRT - Numerical Settings

Institution	Case	Time Step Size (ms)	Total Simulation Time (s)	Order of Discretization	Other Provided Data
EDF	SRT-290	0.04	66.5	Spatial = 2 nd Temporal = 2 nd	
GRS	SRT-290	0.2	10	Spatial = 2 nd Temporal = 2 nd	Spatial=Second order High-Resolution schemes Temporal=Second Order Backward Euler
IPP	SRT-290	0.1	5	Spatial = 2 nd Temporal = 2 nd	Spatial=Second order High-Resolution schemes Temporal=Second Order Backward Euler

6.2.1 SRT-290 (Q=2.9 l/s). Simulations summary

Figure 131 and Figure 132 demonstrate the simulated lift forces on the tube and the corresponding calculated APSDs, respectively, along with their comparison with experimental data. The obtained amplitude of the lift forces has a quite good agreement with experiment in all three simulations, but only in simulations performed by EDF and GRS the modulation in time was captured. The calculated APSDs capture the peak present in the experimental spectrum with a sufficiently good accuracy, while the overall spectrum pattern obtained by EDF and GRS show a good match with the experimental one.

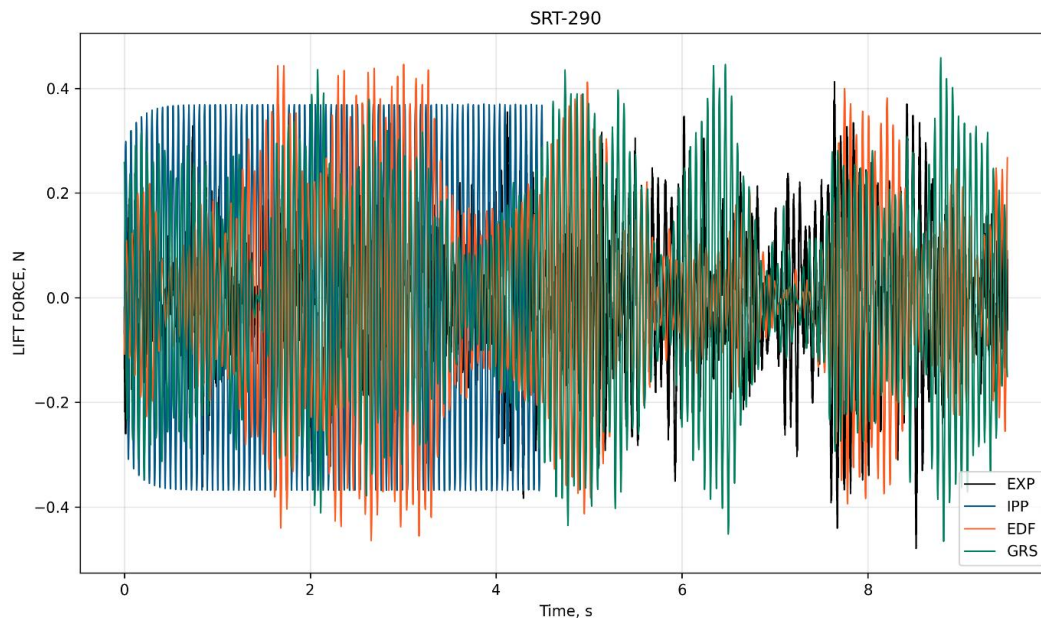


Figure 131: Comparison of the lift force simulated by the participants with experimental data for the single rigid tube with Q=2.9 l/s

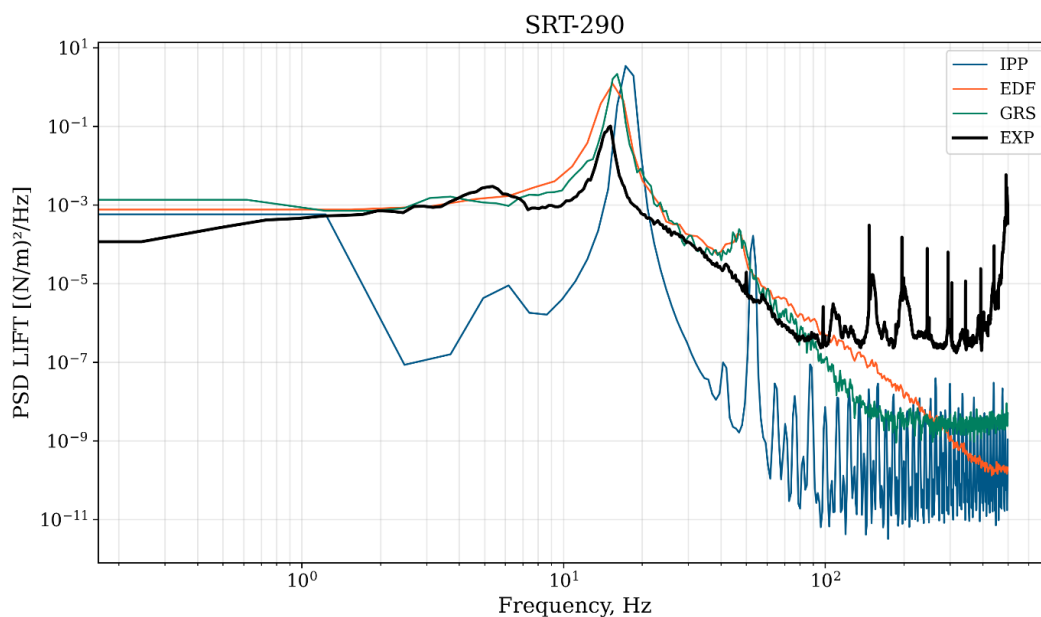


Figure 132: Comparison of resulting APSD of lift force with experimental APSD for the single rigid tube with Q=2.9 l/s

Figure 133 and Figure 134 demonstrate the drag forces obtained in simulations on the tube and the corresponding calculated APSDs, respectively, along with their comparison with experimental data. Obtained amplitude of the drag forces are lower than in experiment in all three simulations, and only in simulations performed by EDF and GRS the modulation in time was captured. Calculated APSDs captured visible peak in the interval $\sim 30\text{-}35\text{Hz}$, while in experimental spectrum this peak is at $\sim 50\text{Hz}$ frequency. Comparison of calculated RMS of Lift and Drag forces with appropriate experimental values is presented in Table 52.

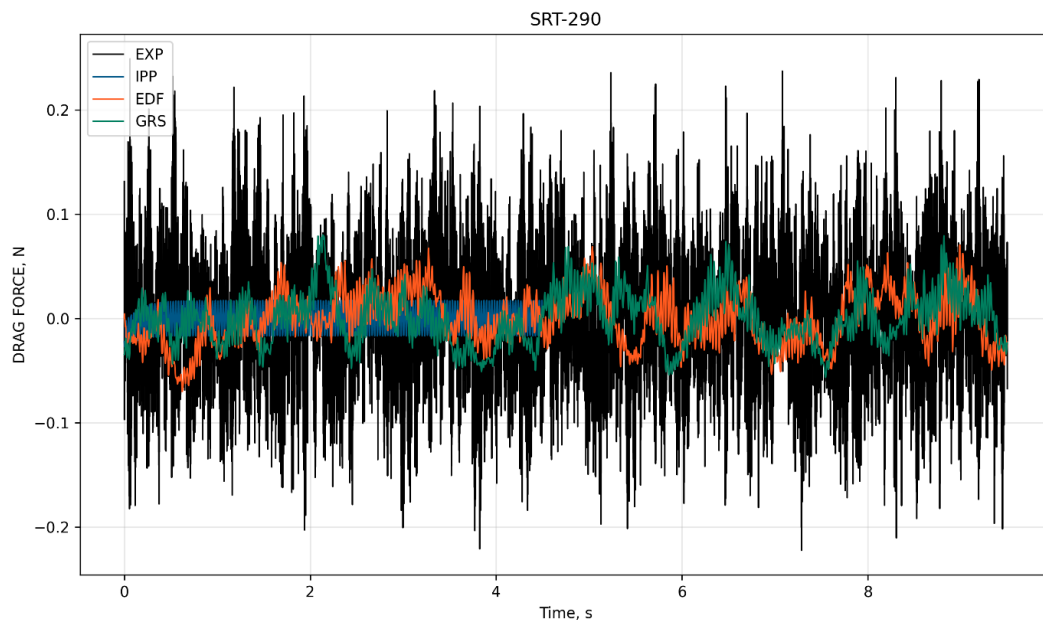


Figure 133: Comparison of the drag force simulated by the participants with experimental data for the single rigid tube with $Q=2.9\text{ l/s}$

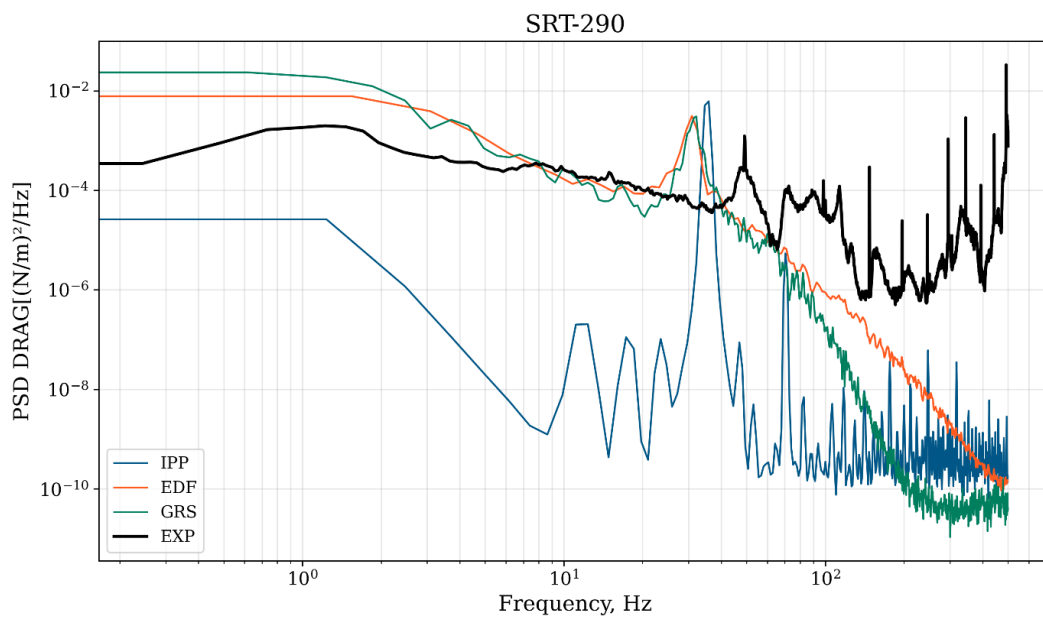


Figure 134: Comparison of resulting APSD of drag force with experimental APSD for the single rigid tube with $Q=2.9\text{ l/s}$

Table 52: Comparison of the resulting RMS values of LIFT and DRAG forces obtained by participants for the case SRT-290

Institution	Case	RMS LIFT	RMS DRAG
EXP	SRT-290	0.411	0.120
EDF		1.684	0.267
GRS		1.312	0.156
IPP		2.644	0.119

6.3 Single Flexible Tube cases

Among the available cases with Single Flexible Tube configuration participants agreed to perform the simulations of the cases with $Q=1.7$ l/s and $Q=2.9$ l/s.

Table 53 – Table 56 provide the main information on the simulations of Single Flexible Tube cases performed by each Institution:

- Table 53 summarizes the participants' individual submissions reported by EDF, GRS and IPP on Single Flexible Tube cases.
- Table 54 provides the calculation domains details for Single Flexible Tube cases.
- Table 55 provides information on the Boundary Conditions applied to Single Flexible Tube cases.
- Each participant has chosen the numerical settings for Single Flexible Tube cases. This information is presented in Table 56.

Table 53: SFT - Key information of the submissions

Institution	Simulation Case Type	Approach	Formulation	Turbulence Modelling	Solid Modelling	Software
EDF	SFT	FSI	3D	Wall-resolved LES	Rigid motion (subroutine in code_saturne)	Code Saturne Ver 8.0
GRS	SFT	FSI	3D	Omega Reynolds Stress Model	Rigid Body FSI	ANSYS CFX 2021 R2
IPP	SFT	FSI	2D	Hybrid LES-URANS – SAS SST	User Library (Mechanical model of Beam vibrations implemented in ANSYS CFX using FORTRAN – FDMK solver option)	ANSYS CFX 2018

Table 54:SFT - Calculation Domain Details

Institution	Case	Geometry Model	Mesh Cell Type	Mesh Cell Count, Cells	Wall Y+
EDF	SFT-170	3D Fluid Domain	Hexahedral	296M	<2 everywhere
	SFT-290				
GRS	SFT-170	3D Fluid Domain includes 255 mm upstream the tube and 137.5 mm downstream. The central flexible tube (tube + flexible blade) is modelled.	Hexahedral	3.950M	Flexible Tube= 2.2; Walls = 2.9; Outer Cylinder = 1.67
	SFT-290				
IPP	SFT-170	2D Fluid Domain represented by the central cross section and includes 57.5 mm upstream the central tube and 137.5 mm downstream.	Hexahedral	8.7k	Central Tube= 3.2; Walls = 23.0;
	SFT-290				

Table 55: SFT - Fluid Boundary Conditions

Institution	Case	Inlet BC	Outlet(s) BC	Tube Surface	All other walls
EDF	SFT-170	Fully developed using embedded precursor LES	Free Outlet	wall resolved (no-sip)	wall resolved (no-sip)
	SFT-290				
GRS	SFT-170	Fully developed inlet flow profile (x,y,z)	BOUNDARY TYPE = Opening; MASS AND MOMENTUM = Opening Pressure and Direction; TURBULENCE = Medium Intensity and Eddy Viscosity Ratio	Smooth Walls	Smooth Walls
	SFT-290				
IPP	SFT-170	Fully developed inlet flow profile	“Opening” boundary type with Medium Intensity of Turbulence	No slip, smooth wall. Mesh motion	No slip, smooth wall
	SFT-290				

Table 56: SFT - Numerical Settings

Institution	Case	Time Step Size (ms)	Total Simulation Time (s)	Order of Discretization	Other Provided Data
EDF	SFT-170	0.1	43.5	Spatial = 2 nd Temporal = 2 nd	
	SFT-290	0.004	14.8		
GRS	SFT-170	0.2	10.0	Spatial = 2 nd Temporal = 2 nd	Spatial=Second order High-Resolution schemes Temporal=Second Order Backward Euler
	SFT-290	0.2	12.0		
IPP	SFT-170	1.0	10.0	Spatial = 2 nd Temporal = 2 nd	Spatial=Second order Central Difference schemes Temporal=Second Order Backward Euler
	SFT-290	1.0	10.0		

6.3.1 SFT-170 (Q=1.7 l/s). Simulations summary

Figure 135 and Figure 136 demonstrate the tube displacements obtained in simulations and appropriate calculated APSDs, respectively, as well as their comparison with experimental data. The amplitudes obtained by GRS and EDF have good agreement with the experimental ones, while displacements obtained by IPP are overestimated. Comparison of calculated RMS of displacements with appropriate experimental values is presented in Table 57.

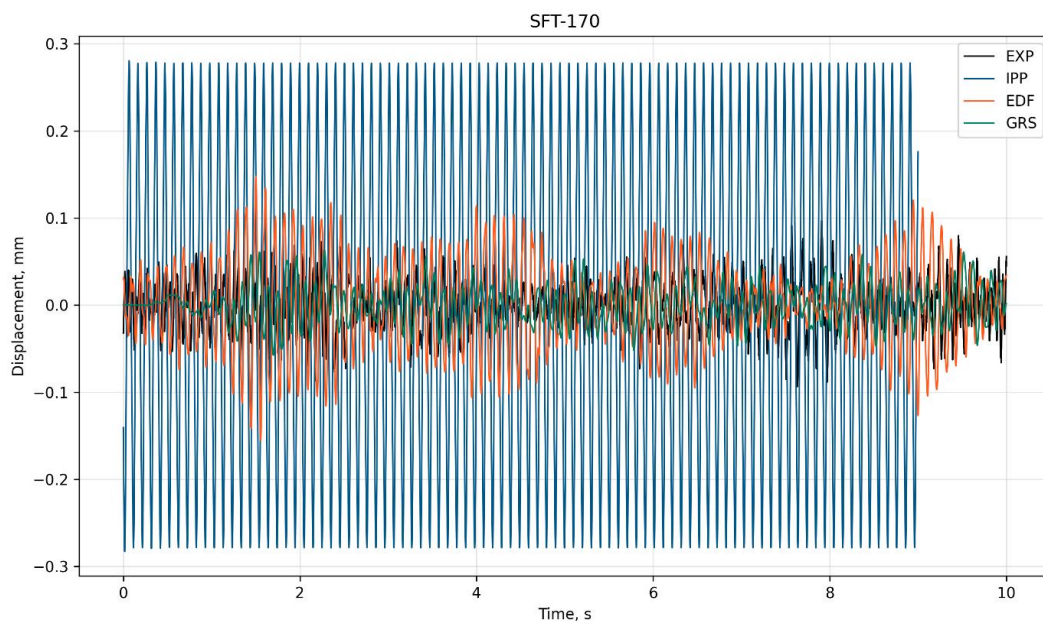


Figure 135: Comparison of the tube displacements simulated by the participants with experimental data for the single flexible tube with Q=1.7 l/s

Calculated APSD 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 (see Table 57).

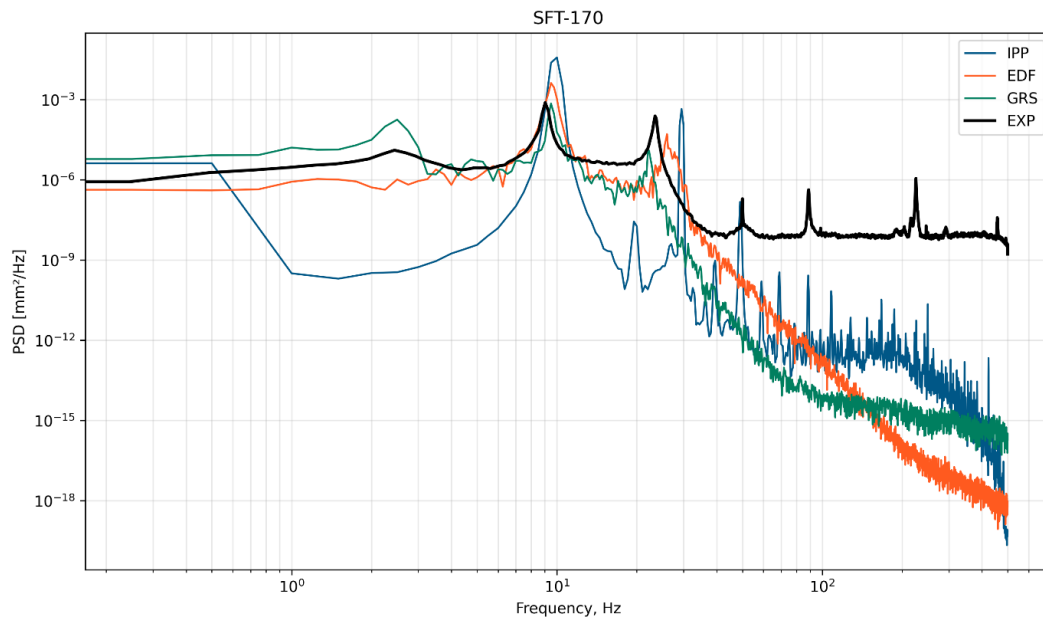


Figure 136: Comparison of the resulting displacements APSD calculated by the participants with experimental data for the single flexible tube with $Q=1.7$ l/s

Comparison of calculated RMS of Lift and Drag forces with appropriate experimental values is presented in Table 57.

Table 57: Comparison of the resulting RMS values of tube displacement and appropriate APSD peaks frequencies obtained by participants for the case SFT-170

Institution	Case	RMS	1 st Peak Frequency	2 nd Peak Frequency
EXP	SFT-170	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

6.3.2 SFT-290 ($Q=2.9$ l/s). Simulations summary

Figure 137 and Figure 138 demonstrate the rod displacements obtained in simulations and appropriate calculated APSDs, respectively, as well as their comparison with experimental data. The amplitudes obtained by GRS and EDF have good agreement with the experimental ones, while displacements obtained by IPP are overestimated as in the case SFT-170. Comparison of calculated

RMS of displacements with appropriate experimental values is presented in Table 58. Calculated APSD captured both peaks present in the experimental spectrum with a sufficiently good accuracy in all three simulations (see Table 58).

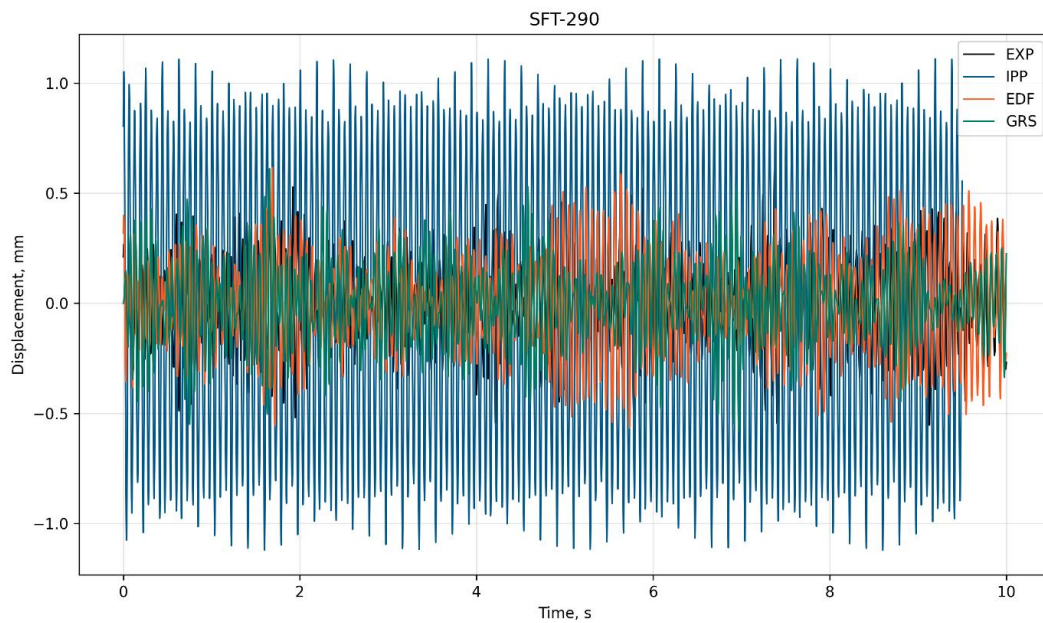


Figure 137: Comparison of the tube displacements simulated by the participants with experimental data for the single flexible tube with $Q=2.9$ l/s

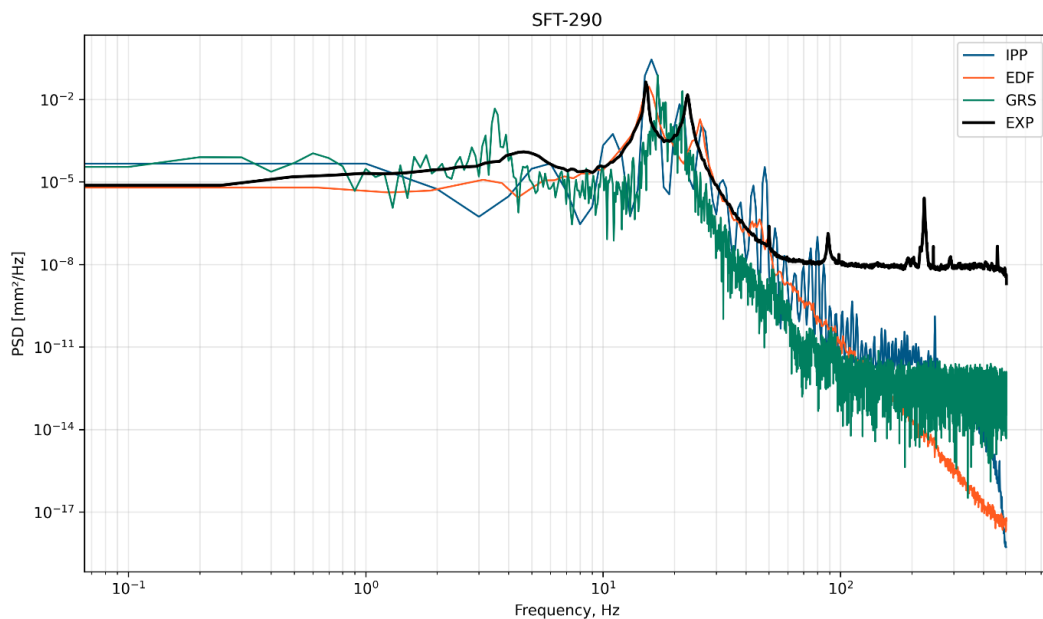


Figure 138: Comparison of the resulting displacements APSD calculated by the participants with experimental data for the single flexible tube with $Q=2.9$ l/s

Table 58: Comparison of the resulting RMS values of tube displacement and appropriate APSD peaks frequencies obtained by participants for the case SFT-290

Institution	Case	RMS	1 st Peak Frequency	2 nd Peak Frequency
EXP	SFT-290	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

6.4 Rigid Tube Bundle cases

Among the available cases with Rigid Tube Bundle configuration participants agreed to perform the simulations of cases with $Q=1.7$ l/s and $Q=2.9$ l/s. For both cases the comparison of measured in experiment (EXP) and calculated by participants (EDF, GRS, IPP) time-averaged streamwise velocity fields is provided (see Figure 143 and Figure 148).

Table 59 – Table 62 provide the main information on the simulations of Rigid Tube Bundle cases performed by each Institution:

- Table 59 summarizes the participants' individual submissions reported by EDF, GRS and IPP on Rigid Tube Bundle cases.
- Table 60 provides the calculation domains details for Rigid Tube Bundle cases.
- Table 61 provides information about the Boundary Conditions applied for Rigid Tube Bundle cases.
- Each participant chosen the numerical settings for the Rigid Tube Bundle cases. This information is presented in Table 62.

Table 59: RTB - Key information of the submissions

Institution	Simulation Type	Approach	Formulation	Turbulence Modelling	Solid Modelling	Software
EDF	RTB	Pure CFD	3D	Wall-resolved LES	NA	Code Saturne Ver 8.0
GRS	RTB	Pure CFD	3D	Hybrid LES-URANS - ZLES	NA	ANSYS CFX 2021 R2
IPP	RTB	Pure CFD	2D	URANS - SST	NA	ANSYS CFX 2018

Table 60: RTB - Calculation Domain Details

Institution	Case	Geometry Model	Mesh Cell Type	Mesh Cell Count, Cells	Wall Y+
EDF	RTB-170		Hexahedral	162M	<2 everywhere
	RTB-290			450M	
GRS	RTB-170	3D Fluid Domain includes 255 mm upstream the central tube and 137.5 mm downstream	Hexahedral	19.254M	Central Tube= 0.73; Other tubes = 1.63; Walls = 0.8;
	RTB-290			19.254M	Central Tube= 1.17; Other tubes = 2.52; Walls = 1.27;
IPP	RTB-170	2D Fluid Domain represented by the central cross section and includes 255 mm upstream the central tube and 345 mm downstream.	Hexahedral	34.1k	Central Tube= 2.09; Other tubes = 11.58; Walls = 8.3;
	RTB-290				

Table 61: RTB - Fluid Boundary Conditions

Institution	Case	Inlet BC	Outlet(s) BC	Tube Surface	All other walls
EDF	RTB-170	Fully developed using embedded precursor LES	Free Outlet	wall resolved (no-sip)	wall resolved (no-sip)
	RTB-290				
GRS	RTB-170	Fully developed inlet flow profile (x,y,z)	BOUNDARY TYPE = opening; MASS AND MOMENTUM = Opening Pressure and Direction; TURBULENCE = Zero Gradient;	Smooth Walls	Smooth Walls
	RTB-290				
IPP	RTB-170	Fully developed inlet flow profile	“Opening” boundary type with Medium Intensity of Turbulence	No slip, smooth wall	No slip, smooth wall
	RTB-290				

Table 62: RTB - Numerical Settings

Institution	Case	Time Step Size (ms)	Total Simulation Time (s)	Order of Discretization	Other Provided Data
EDF	RTB-170	0.035	39.7	Spatial = 2 nd	
	RTB-290	0.02	41.7	Temporal = 2 nd	
GRS	RTB-170	0.2	10	Spatial = 2 nd Temporal = 2 nd	Spatial=Second order High-Resolution schemes Temporal=Second Order Backward Euler
	RTB-290		16.6		
IPP	RTB-170	1.0	10.0	Spatial = 2 nd Temporal = 2 nd	Spatial=Second order High-Resolution schemes Temporal=Second Order Backward Euler
	RTB-290	1.0	10.0		

6.4.1 RTB-170 (Q=1.7 l/s). Simulations summary

Figure 139 and Figure 140 demonstrate the obtained in simulations Lift forces on the central tubetube and appropriate calculated APSDs, respectively, as well as their comparison with experimental data. Obtained amplitudes of the lift forces are underestimated slightly compared to the experiment in all three simulations. Calculated APSD have sufficiently good coincidence with experimental spectrum.

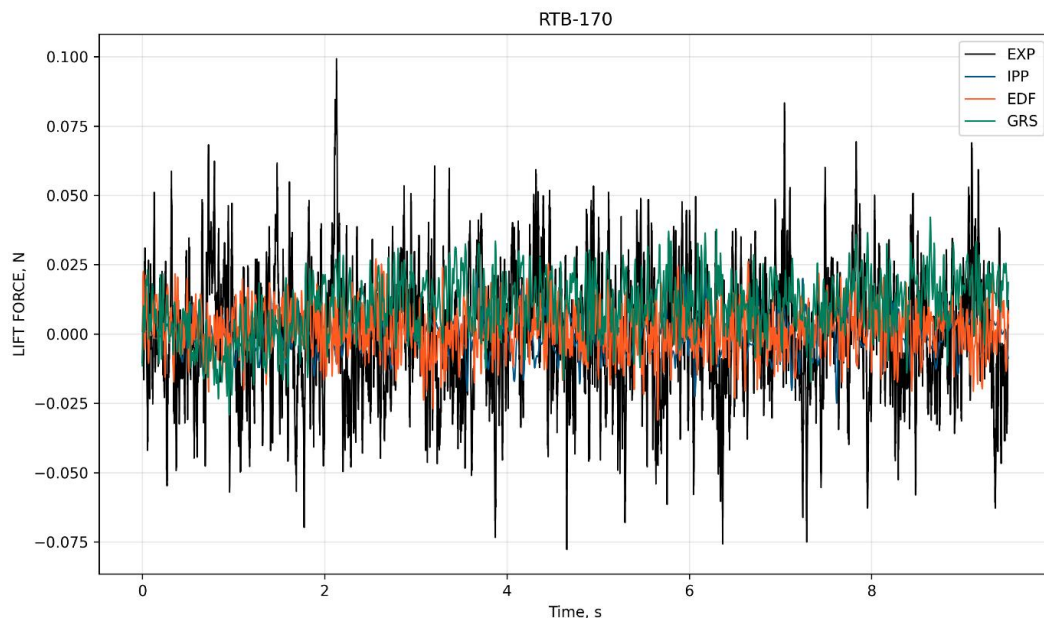


Figure 139: Comparison of the lift force on the central tube simulated by the participants with experimental data for the rigid tube bundle with Q=1.7 l/s

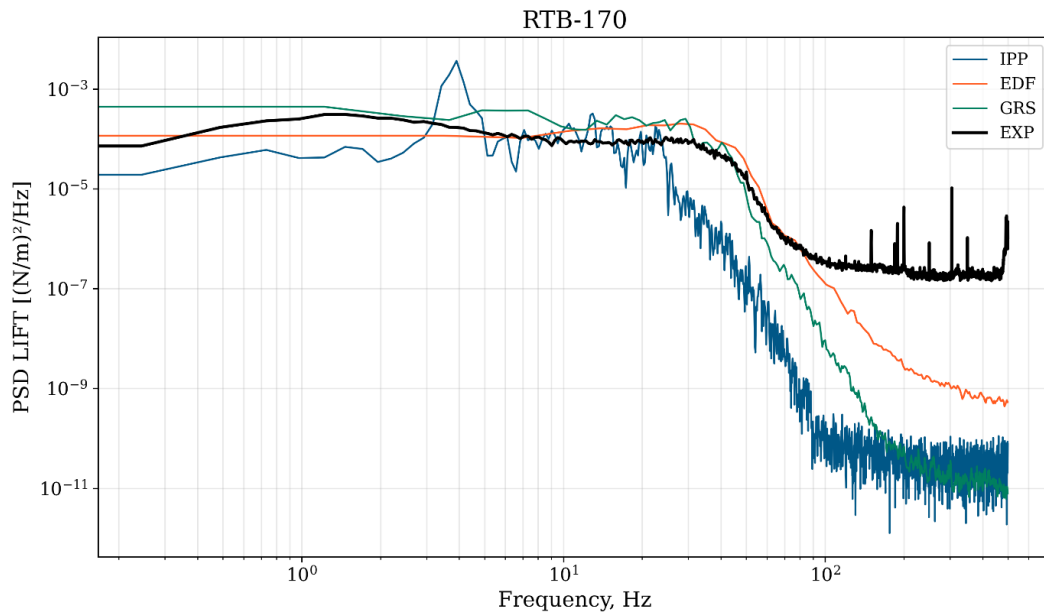


Figure 140: Comparison of resulting APSD of lift force for the central tube with experimental APSD for the rigid tube bundle with $Q=1.7$ l/s

Figure 141 and Figure 142 demonstrate the obtained in simulations Drag forces on the central tube and appropriate calculated APSDs, respectively, as well as their comparison with experimental data. Obtained amplitudes of the drag forces are also underestimated a little comparing to the experiment in all three simulations. Calculated APSD have sufficiently good coincidence with experimental spectrum, especially in simulations performed by GRS and EDF.

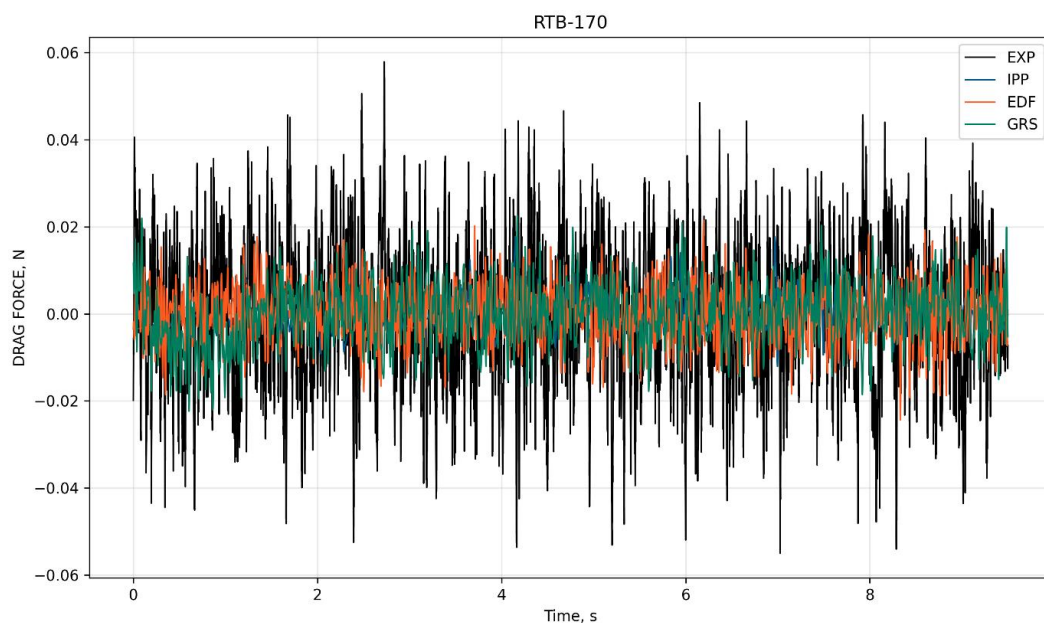


Figure 141: Comparison of the drag force on the central rod simulated by the participants with experimental data for the rigid tube bundle with $Q=1.7$ l/s

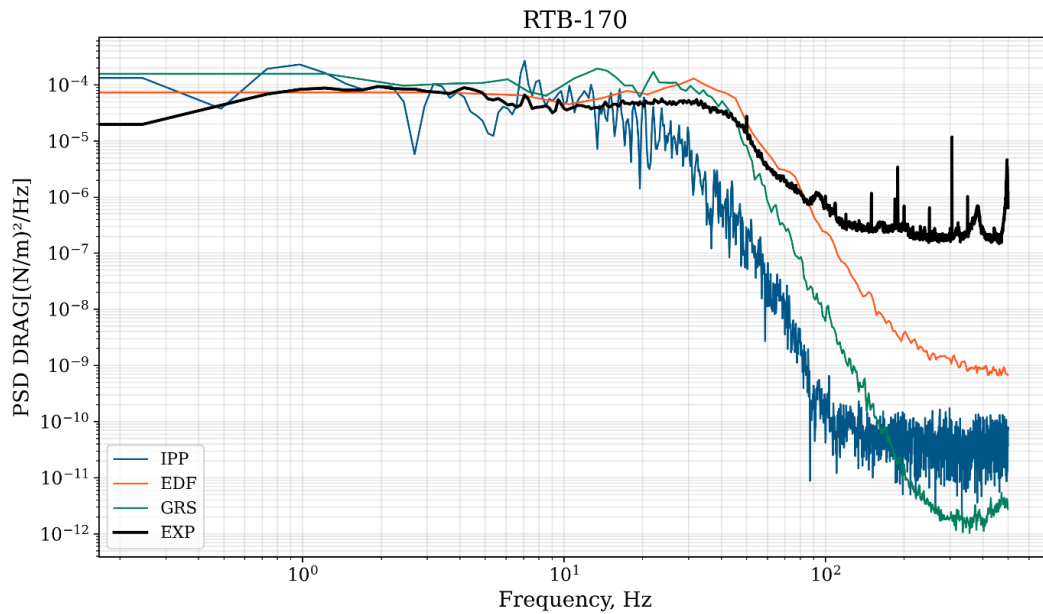


Figure 142: Comparison of resulting APSD of drag force for the central rod with experimental APSD for the rigid tube bundle with $Q=1.7$ l/s

Table 63 provide the comparison of calculated RMS of Lift and Drag forces with appropriate experimental values. It could be seen, that RMS of both Lift and Drag forces are predicted well in all three simulations.

Table 63: Comparison of the resulting RMS values of LIFT and DRAG forces on the central rod obtained by participants for the case RTB-170

Institution	Case	RMS LIFT	RMS DRAG
EXP	RTB-170	0.066	0.048
EDF		0.089	0.062
GRS		0.066	0.045
IPP		0.067	0.036

Figure 143 demonstrate the comparison of measured in experiment (EXP) and calculated by participants (EDF,GRS,IPP) time-averaged streamwise velocity fields for RTB-170. In order to provide a demonstrative comparison of the resulting fields, they were visualized within the same legend limits and color scheme. In all three simulations performed by EDF, GRS and IPP time-averaged velocity fields have definitely good agreement with the experimental picture.

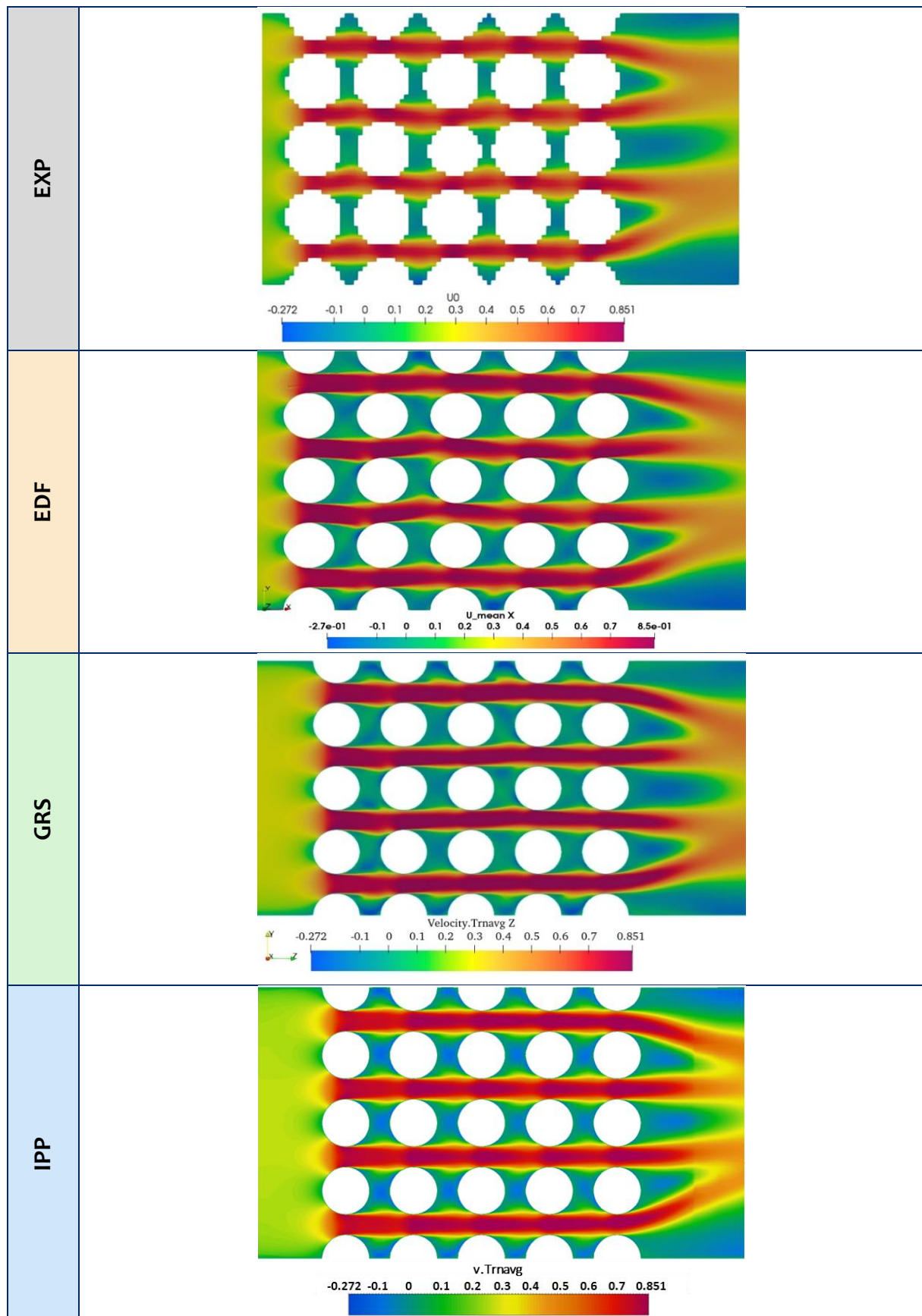


Figure 143: Comparison of the measured in experiment (EXP) and calculated by participants (EDF, GRS, IPP) time-averaged streamwise velocity in the rigid tube bundle with 1.7 l/s flow rate

6.4.2 RTB-290 ($Q=2.9$ l/s). Simulations summary

Figure 144 and Figure 145 demonstrate the obtained in simulations Lift forces on the central rod and appropriate calculated APSDs, respectively, as well as their comparison with experimental data. Obtained amplitudes of the lift forces are underestimated a little comparing to the experiment in all three simulations as in RTB-170 as well. Calculated APSD have sufficiently good match with experimental spectrum.

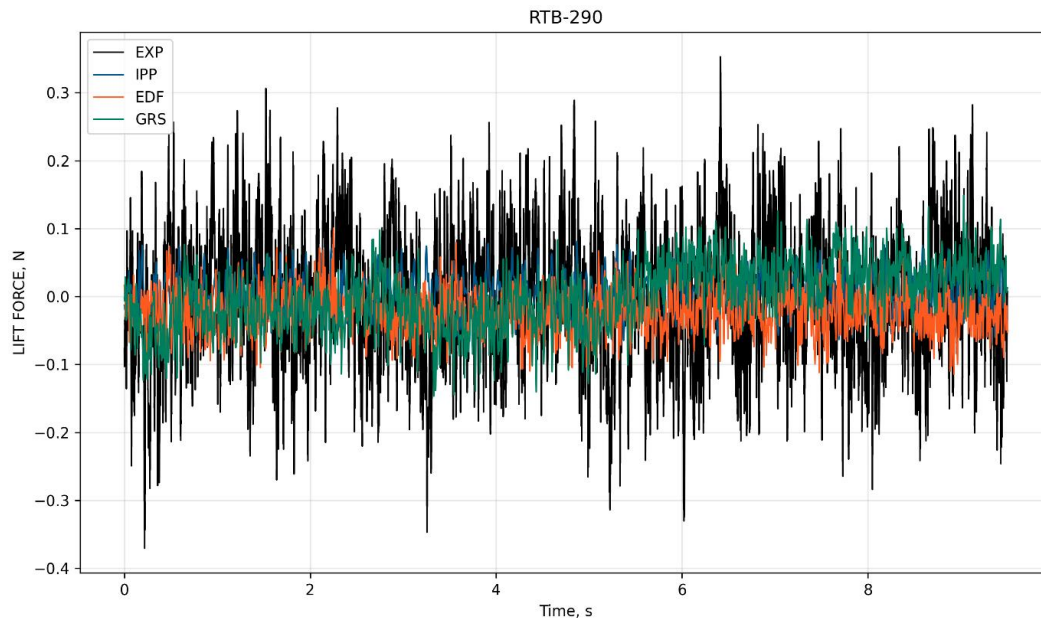


Figure 144: Comparison of the lift force on the central rod simulated by the participants with experimental data for the rigid tube bundle with $Q=2.9$ l/s

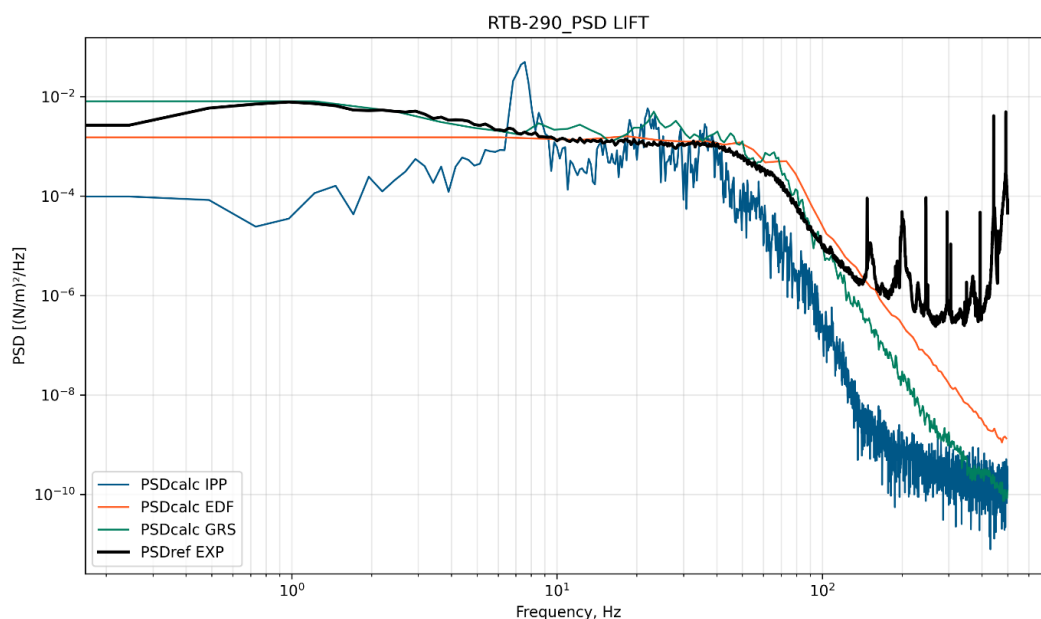


Figure 145: Comparison of resulting APSD of lift force for the central rod with experimental APSD for the rigid tube bundle with $Q=2.9$ l/s

Figure 146 and Figure 147 demonstrate the obtained in simulations Drag forces on the central rod and appropriate calculated APSDs, respectively, as well as their comparison with experimental data. Obtained amplitudes of the drag forces are also underestimated comparing to the experiment in all three simulations. Calculated APSD have sufficiently good coincidence with experimental spectrum, especially in simulations performed by GRS and EDF.

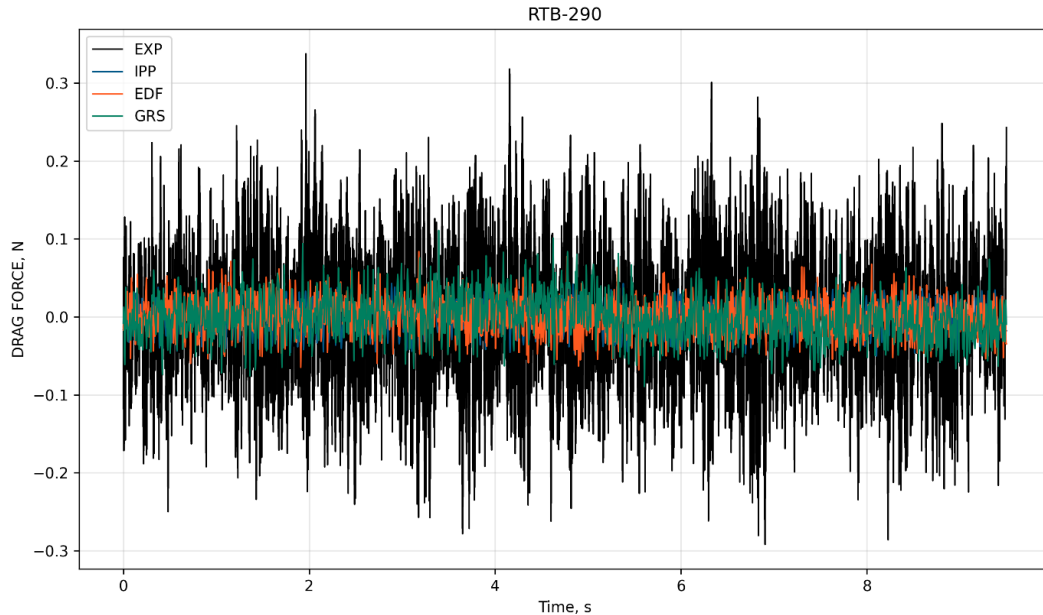


Figure 146: Comparison of the drag force on the central rod simulated by the participants with experimental data for the rigid tube bundle with $Q=2.9$ l/s

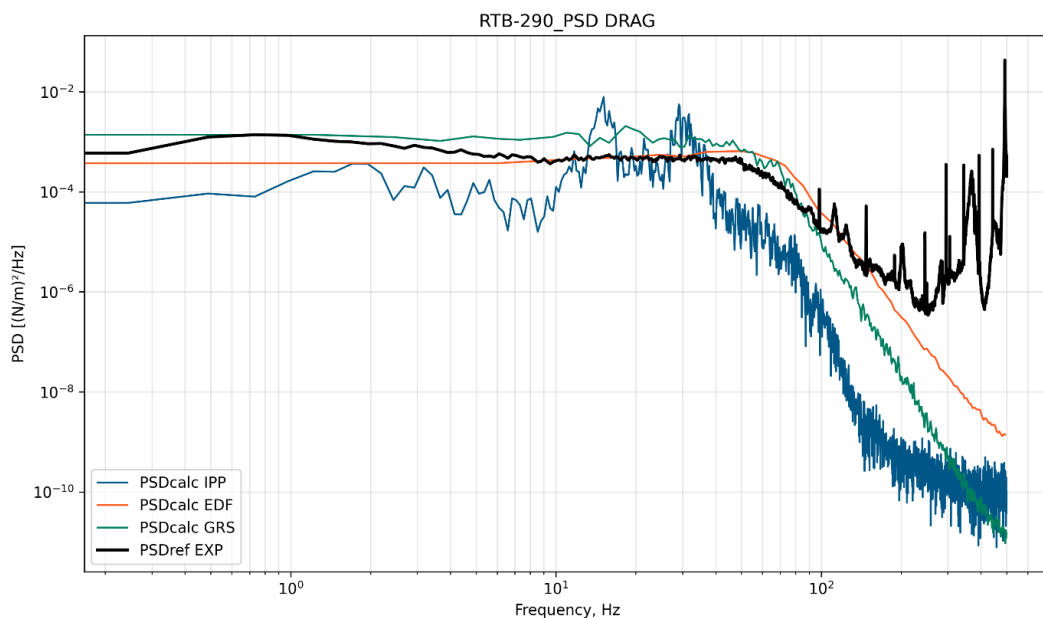


Figure 147: Comparison of resulting APSD of drag force for the central rod with experimental APSD for the rigid tube bundle with $Q=2.9$ l/s

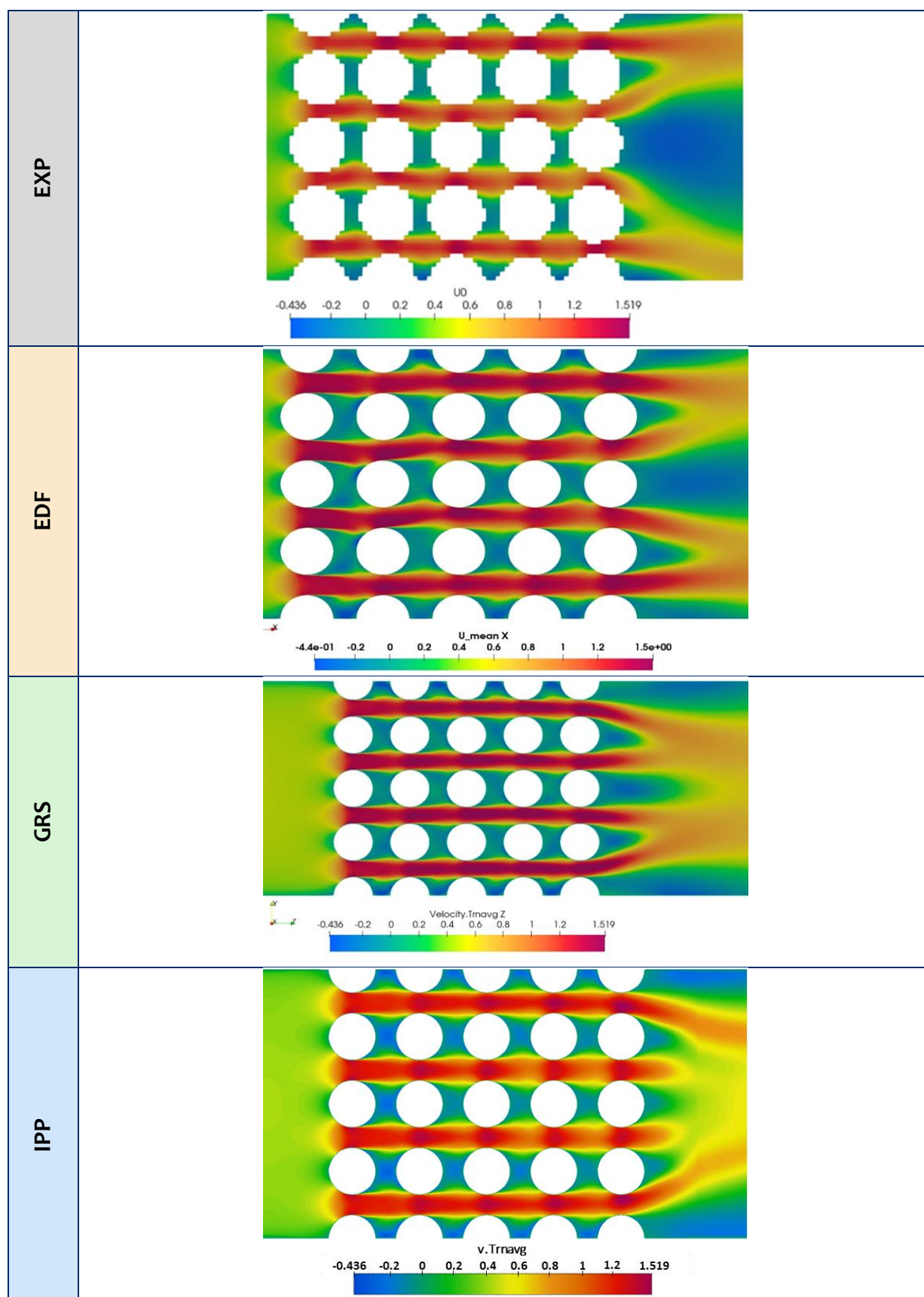


Figure 148: Comparison of the measured in experiment (EXP) and calculated by participants (EDF, GRS, IPP) time-averaged streamwise velocity in the rigid tube bundle with 2.9 l/s flow rate

Figure 148 demonstrate the comparison of measured in experiment (EXP) and calculated by participants (EDF, GRS, IPP) time-averaged streamwise velocity fields for RTB-290. In order to provide a demonstrative comparison of the resulting fields, they were visualized within the same legend limits and color scheme. 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. The reason of such difference could lie in the smaller simulation time of GRS and IPP analyses (16.6s and 10.0s), while EDF performed simulation of 41.7 seconds. In the case of GRS, it was found that the “Transient Statistics” option in CFX that averages the solution over all time steps was not activated. Instead, transient data was output at low frequency.

Table 64 provide the comparison of calculated RMS of Lift and Drag forces with appropriate experimental values. It could be seen, that RMS of both Lift and Drag forces are predicted well in all three simulations.

Table 64: Comparison of the resulting RMS values of LIFT and DRAG forces on the central rod obtained by participants for the case RTB-290

Institution	Case	RMS LIFT	RMS DRAG
EXP	RTB-290	0.291	0.179
EDF		0.305	0.204
GRS		0.252	0.178
IPP		0.292	0.184

6.5 Flexible Tube in a Rigid Tube Bundle cases

Among the available cases with Flexible Tube in a Rigid Tube Bundle configuration participants agreed to perform the simulations of the cases with $Q=1.6$ l/s and $Q=1.95$ l/s (EDF and IPP). For FTRTB-160 the comparison of measured in experiment (EXP) and calculated by participants (EDF, GRS, IPP) time-averaged streamwise velocity fields is provided (see Figure 151).

Table 65 – Table 68 provide the main information on the simulations of Flexible Tube in a Rigid Tube Bundle cases performed by each Institution:

- Table 65 summarizes the participants’ individual submissions reported by EDF, GRS and IPP on Flexible Tube in a Rigid Tube Bundle cases.
- Table 66 provide the calculation domains details for Flexible Tube in a Rigid Tube Bundle cases.
- Table 67 provides information about the Boundary Conditions applied for Flexible Tube in a Rigid Tube Bundle cases.
- Each participant has chosen the numerical settings for the Flexible Tube in Rigid Tube Bundle cases. This information is presented in Table 68.

Table 65: FTRTB - Key information of the submissions

Institution	Simulation Type	Approach	Formulation	Turbulence Modelling	Solid Modelling	Software
EDF	FTRTB	FSI	3D	Wall-resolved LES	Rigid motion (subroutine in code_saturne)	Code Saturne Ver 8.0
GRS	FTRTB	FSI	3D	Hybrid LES-URANS - ZLES	Rigid Body FSI	ANSYS CFX 2021 R2
IPP	FTRTB	FSI	2D	Omega Reynold Stress Model	Rigid Body FSI	ANSYS CFX 2018

Table 66: FTRTB - Calculation Domain Details

Institution	Case	Geometry Model	Mesh Cell Type	Mesh Cell Count, Cells	Wall Y+
EDF	FTRTB-160		Hexahedral	141M	<2 everywhere
	FTRTB-195				
GRS	FTRTB-160	Hexahedral	Hexahedral	19.157M	Flexible Tube= 0.79; Other tubes = 1.47; Walls = 0.7865; Outer cylinder = 0.28;
IPP	FTRTB-160	2D Fluid Domain represented by the central cross section and includes 255 mm upstream the central rod and 345 mm downstream.	Hexahedral	34.1k	Central Tube= 2.09; Other tubes = 11.58; Walls = 8.3;
	FTRTB-195				

Table 67: FTRTB - Fluid Boundary Conditions

Institution	Case	Inlet BC	Outlet(s) BC	Rod Surface	All other walls
EDF	FTRTB-160	Fully developed using embedded precursor LES	Free Outlet	wall resolved (no-slip)	wall resolved (no-slip)
	FTRTB-195				
GRS	FTRTB-160	Mass Flow Rate on the Inlet	BOUNDARY TYPE = opening; MASS AND MOMENTUM = Opening Pressure and Direction; TURBULENCE = Zero Gradient;	Smooth Walls	Smooth Walls
IPP	FTRTB-160	Fully developed inlet flow profile	“Opening” boundary type with Medium Intensity of Turbulence	No slip, smooth wall. Mesh motion	No slip, smooth wall
	FTRTB-195				

Table 68: FTRTB - Numerical Settings

Institution	Case	Time Step Size (ms)	Total Simulation Time (s)	Order of Discretization	Other Provided Data
EDF	FTRTB-160	0.04	55.0	Spatial = 2 nd	
	FTRTB-195	0.02	42.7	Temporal = 2 nd	
GRS	FTRTB-160	1.0	4.166	Spatial = 2 nd Temporal = 2 nd	Spatial=Second order High-Resolution schemes Temporal=Second Order Backward Euler
IPP	FTRTB-160	0.2	9.0	Spatial = 2 nd Temporal = 2 nd	Spatial=Second order High-Resolution schemes Temporal=Second Order Backward Euler
	FTRTB-195	0.5	4.5		

6.5.1 FTRTB-160 (Q=1.6 l/s). Simulations summary

Figure 149 and Figure 150 demonstrate the central rod displacements obtained in simulations and appropriate calculated APSDs, respectively, as well as their comparison with experimental data. The amplitudes obtained in simulations have good agreement with the experimental ones. Comparison of calculated RMS of displacements with appropriate experimental values is presented in Table 69. Calculated APSD captured the peak present in the experimental spectrum with a good accuracy in all three simulations (see Table 69).

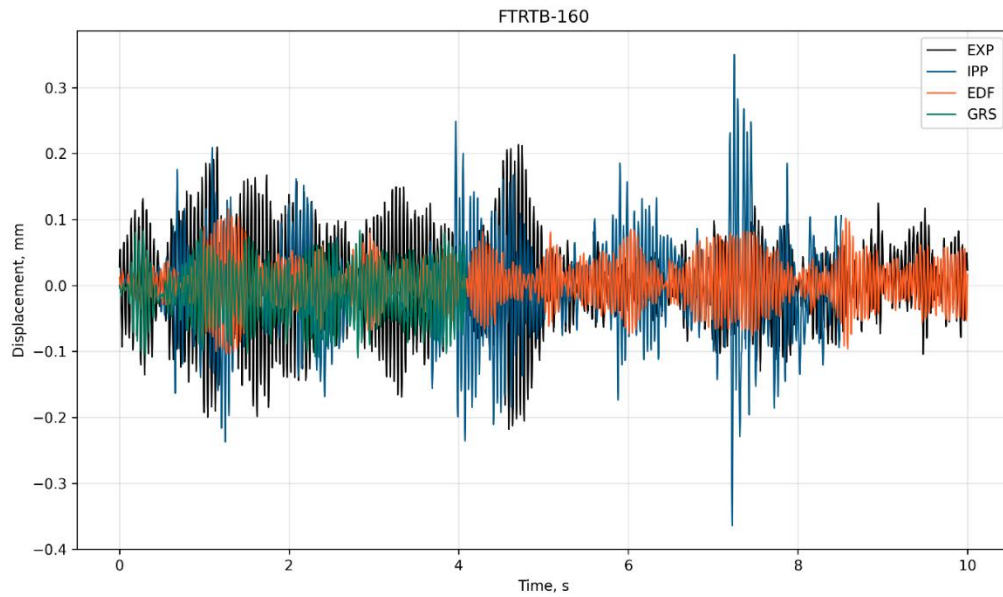


Figure 149: Comparison of the central rod displacements simulated by the participants with experimental data for the flexible tube in a rigid tube bundle with $Q=1.6$ l/s

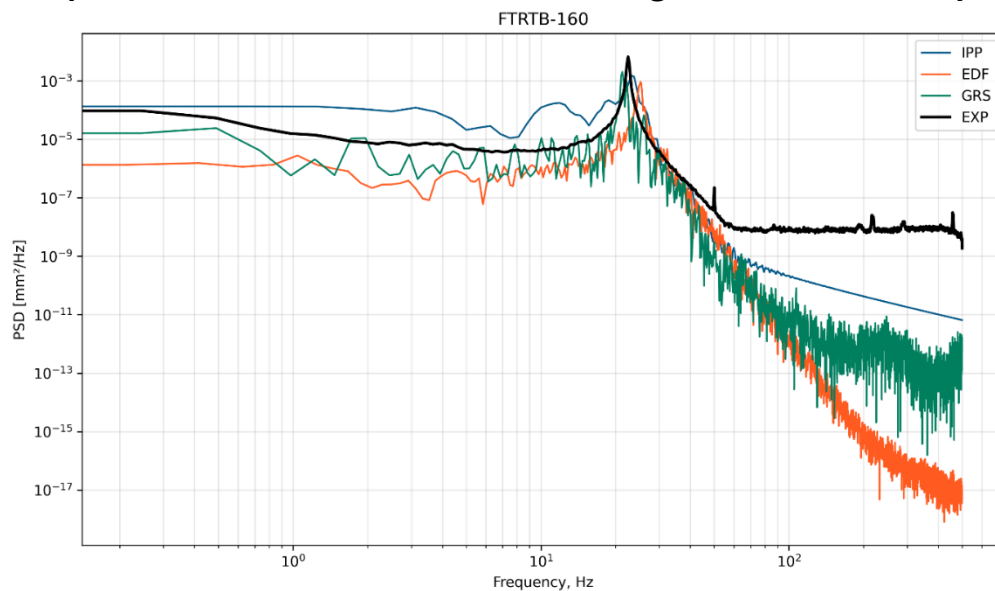


Figure 150: Comparison of the resulting central rod displacements APSD calculated by the participants with experimental data for the flexible tube in a rigid tube bundle with $Q=1.6$ l/s

Figure 151 demonstrate the comparison of measured in experiment (EXP) and calculated by participants (EDF,GRS,IPP) time-averaged streamwise velocity fields for FTRTB-160. In order to provide a demonstrative comparison of the resulting fields, they were visualized within the same legend limits and color scheme. 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. The reason of such difference could lie in the smaller simulation time of GRS and IPP analyses (4.16s and 9.0s), while EDF performed simulation of 55.0 seconds.

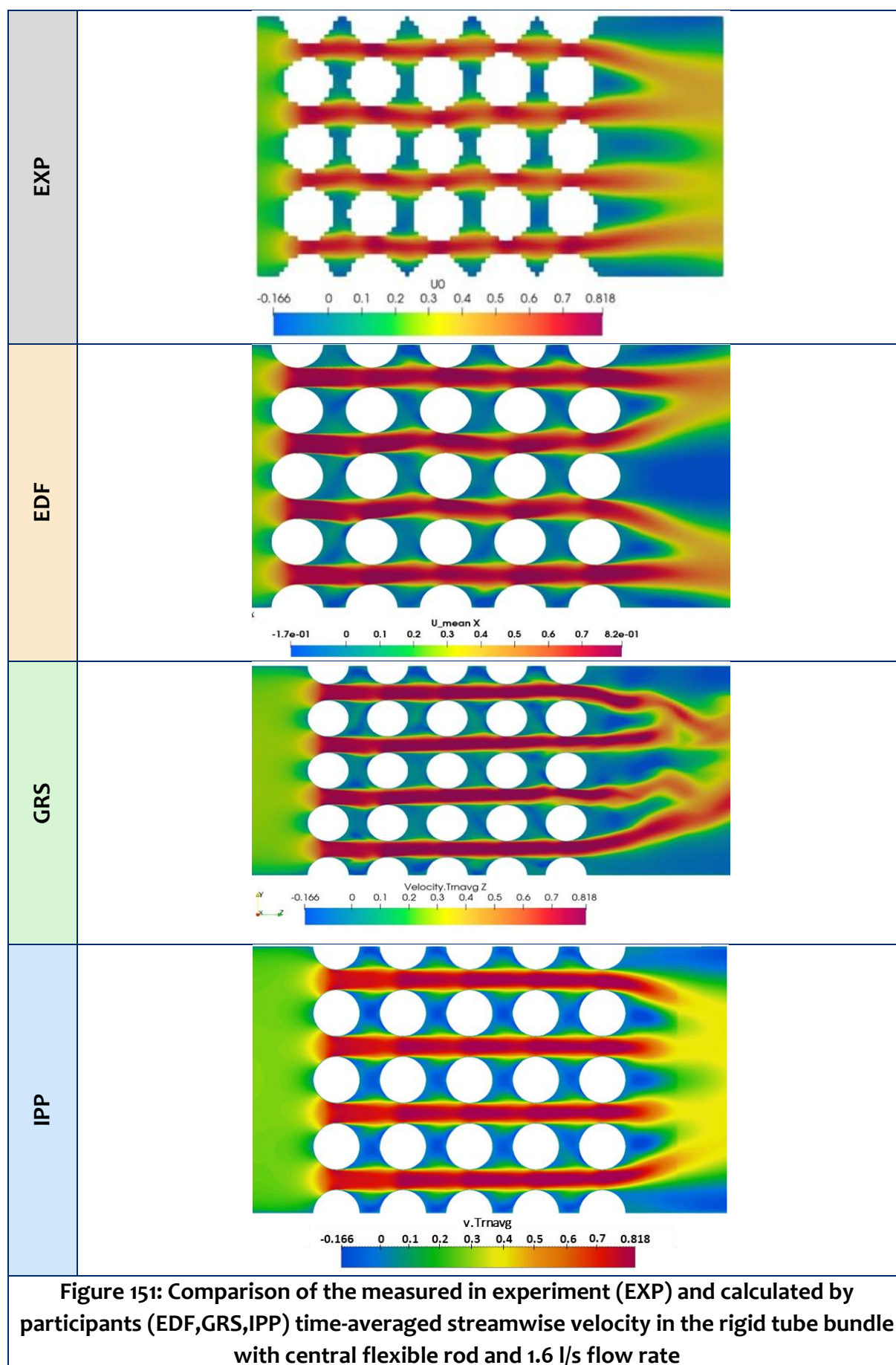


Table 69: Comparison of the resulting RMS values of central rod displacement and appropriate APSD peaks frequencies obtained by participants for the case FTRTB-160

Institution	Case	RMS	1 st Peak Frequency
EXP	FTRTB-160	0.073	22.46
EDF		0.04	25.21
GRS		0.044	21.31
IPP		0.069	22.59

6.5.2 FTRTB-195 (Q=1.95 l/s). Simulations summary

Figure 152 and Figure 153 demonstrate the central rod displacements obtained in simulations and appropriate calculated APSDs, respectively, as well as their comparison with experimental data. The amplitudes obtained in simulations are underestimated compared to the experimental ones in both EDF and IPP analyses. Comparison of calculated RMS of displacements with appropriate experimental values is presented in Table 70. Calculated APSD captured the peak present in the experimental spectrum with a good accuracy in both simulations (see Table 70), while the displacements were massively underestimated. One should keep in mind that in this particular case FEI occurs, and this is extremely challenging to be reproduced numerically. On top, the current FSI simulations are based on simplified structural representations of the AMOVI tube and its blade. Therefore, further analyses with more sophisticated mechanical methods are necessary in order to fully assess the predictive capabilities of the codes for such challenging phenomena such as FEI.

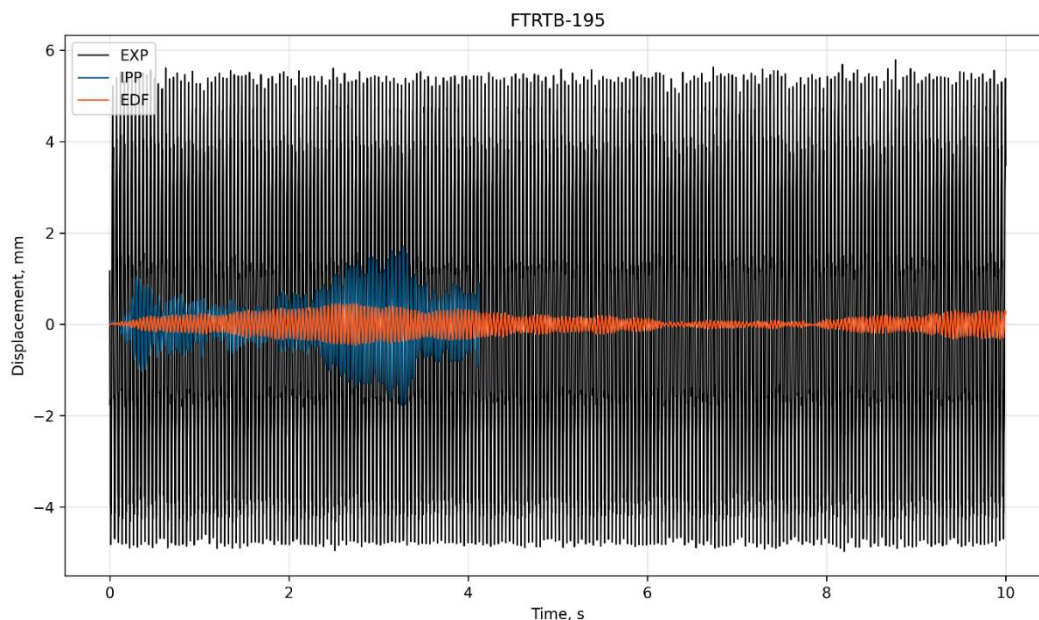


Figure 152: Comparison of the central rod displacements simulated by the participants with experimental data for the flexible tube in a rigid tube bundle with Q=1.95 l/s

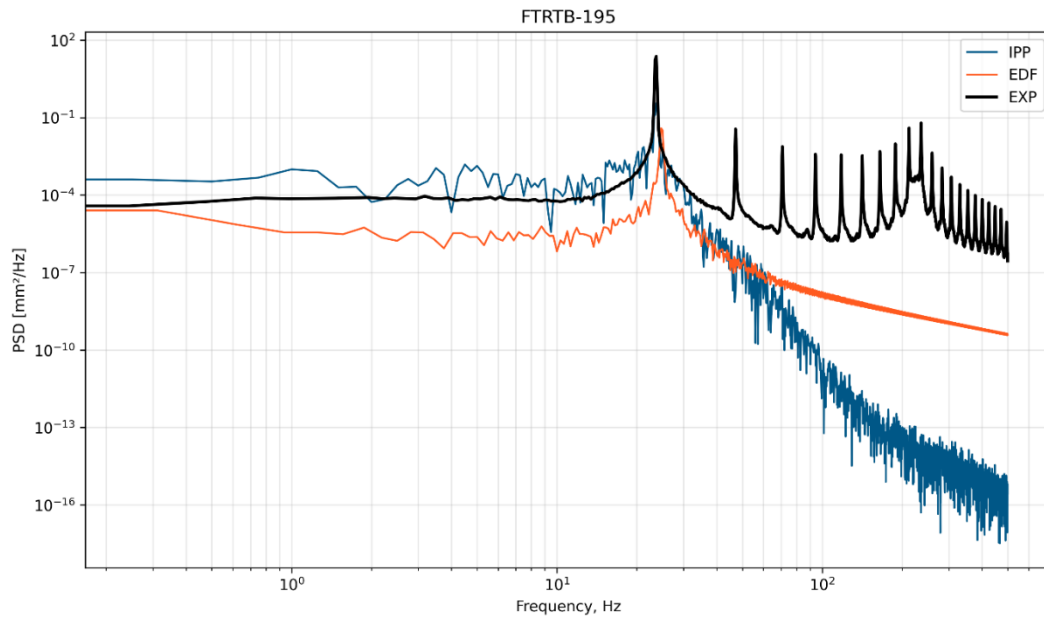


Figure 153: Comparison of the resulting central rod displacements APSD calculated by the participants with experimental data for the flexible tube in a rigid tube bundle with $Q=1.95$ l/s

Table 70: Comparison of the resulting RMS values of central rod displacement and appropriate APSD peaks frequencies obtained by participants for the case FTRTB-195

Institution	Case	RMS	1 st Peak Frequency
EXP	FTRTB-195	3.223	23.68
EDF		0.191	24.69
IPP		0.561	23.75

6.6 Overall Benchmark Summary

The AMOVI benchmark was performed by three participants of the GO-VIKING project (EDF, GRS and IPP) in order to investigate the abilities of different approaches and developed models to predict the Flow Induced Vibrations phenomenon as well as its main characteristics – amplitudes of rod vibrations, their RMS values and appropriate APSD. Two different configurations with Flexible Tube in a coolant flow with various flow rates were available – Single Flexible Tube (SFT) and Flexible Tube in a Rigid Tube Bundle (FTRTB). Among them participants agreed to perform simulations of two cases for each configuration – SFT with intermediate flow rates $Q=1.7\text{ l/s}$ and $Q=2.9\text{ l/s}$, as well as FTRTB with one intermediate $Q=1.6\text{ l/s}$ and one maximal $Q=1.95\text{ l/s}$ flow rate available in AMOVI test series. Also, configurations with absolutely rigid tubes were available for participants – the first one with the Single Rigid Tube, for which one case with $Q=2.9\text{ l/s}$ was agreed for simulation, and the second one with Rigid Tube Bundle, for which two cases with $Q=1.7\text{ l/s}$ and $Q=2.9\text{ l/s}$ were chosen for analysis.

It should be noted, that approaches and models developed and applied by participants for AMOVI simulations were quite different:

- EDF performed the high-fidelity simulations applying their CFD solver code `_saturne v8.0` and involving for this purpose supercomputers. All the models are three dimensional with fully hexahedral and conforming meshes. The wall-resolved LES with fully developed inlet boundary conditions and fully resolved walls is applied for all considered configurations. For moving cylinders configurations, the rigid body equations implemented in `Saturne 8.0` code are applied.
- GRS performed the medium-resolution simulations applying the ANSYS CFX code with Rigid Body Solution for flexible rods configurations. All the developed models are three dimensional with hexahedral meshes generated in ANSYS ICEM CFD tool. Application of different turbulence models (SST, Omega-RSM, SAS and ZLES) was investigated by GRS during the main series of simulations (described in Chapter 3) and the ones which provided the best agreement (mostly ZLES and Omega-RSM) with experimental data were applied for the benchmarking.
- IPP was aimed to test the possible application of the least resource-intensive models for simulations – for this purpose ANSYS CFX code with two-dimensional formulation was applied for all simulated cases. Hexahedral meshes for calculation models were prepared in ANSYS ICEM CFD tool. Application of different turbulence models (SST, SAS and Omega-RSM) was also investigated by IPP and described in Chapter 4. For simulation of the flexible rod configurations two options were applied – Rigid Body Solution available in ANSYS CFX and User Libraries with different options for the Beam Solution implemented using FORTRAN language. The most successful combinations of the turbulence models and options for the beam solution in terms of agreement with the experimental data were applied for the main series of simulations during benchmarking.

The described differences in the approaches used are directly related to the computing resources required for modeling. Table 71 presents the computational resources consumed by each participant during simulations.

Table 71: Computational Resources used for calculations

Institution	Case	Total Simulation Time, s	Wall Clock Time, s	Wall Clock Time needed for 1 second of simulation, s	Equivalent CPU/Node hours	RAM	No. of (CPU/GPU) Cores
EDF	SRT-290	66.5	2.37×10^6	35.6×10^3	4 M	57.7 Tb	CPU=64
	SFT-170	43.5	2.12×10^6	48.7×10^3	3.7 M	45 Tb	
	SFT-290	14.8	1.42×10^6	94.7×10^3	1.8 M	36 Tb	
	RTB-170	39.7	2.87×10^6	72.3×10^3	2.7 M		CPU=37
	RTB-290	41.7	3.67×10^6	88.0×10^3	13 M	155 Tb	CPU=64
	FTRTB-160	55.0	3.02×10^6	54.9×10^3	4 M	42 Tb	
	FTRTB-195	42.7	6.08×10^6	142.4×10^3	6.5 M	30 Tb	
GRS	SRT-290	10.0	8.39×10^5	83.9×10^3	233	10.46 Gb	CPU=30
	SFT-170	10.0	1.35×10^6	135.0×10^3	374.75	23.0 Gb	CPU=50
	SFT-290	12.0	3.23×10^4	2.69×10^3	336.16	31.83 Gb	CPU=96
	RTB-170	10.0	2.02×10^6	202.0×10^3	562.08	44.77 Gb	CPU=70
	RTB-290	16.6	3.51×10^6	211.4×10^3	975.96	44.77 Gb	
	FTRTB-160	4.16	9.61×10^5	231.0×10^3	267.166	37.81 Gb	CPU=50
IPP	SRT-290	5.0	7.45×10^3	1.49×10^3	24.83	1.23 Gb	CPU=12
	SFT-170	10.0	3.55×10^4	3.55×10^3	118.3	0.49 Gb	CPU=12
	SFT-290	10.0	4.07×10^4	4.07×10^3	135.6	0.49 Gb	CPU=12
	RTB-170	10.0	4.08×10^4	4.08×10^3	136.0	2.59 Gb	CPU=12
	RTB-290	10.0	5.67×10^4	5.67×10^3	189.0	2.59 Gb	CPU=12
	FTRTB-160	9.0	2.41×10^5	26.77×10^3	803.3	1.82 Gb	CPU=12
	FTRTB-195	4.5	8.42×10^4	18.71×10^3	280.6	1.82 Gb	CPU=12

Analyzing the obtained results, it could be seen that 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 (see Figure 154), while for the Rigid Tube Bundle cases the corresponding results demonstrate much better agreement with the experiment (see Figure 155). Comparison of RMS values of displacements obtained in simulations both for Single Flexible Tube and Flexible Tube in a Rigid Tube Bundle cases is presented in Figure 156. From the demonstrated results, excellent agreement with the experiment was obtained by EDF and GRS for cases with SFT, while IPP have slightly overestimated predictions for the corresponding RMS values (Figure 156, left). For the FTRTB cases (Figure 156, right), all participants obtained excellent agreement in the displacements RMS for an average flow rate of 1.6 l/s. For 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.

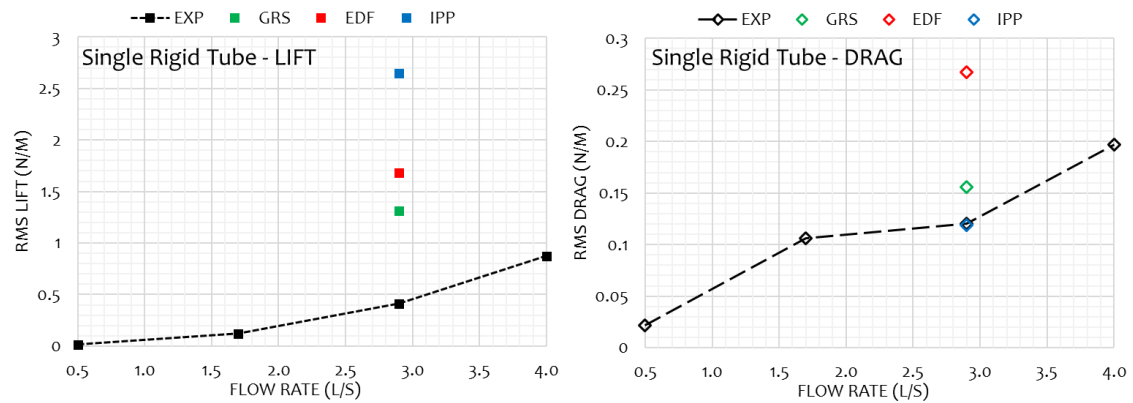


Figure 154: Comparison of simulated and experimental values of RMS value of the turbulent force per unit length as a function of the flow rate (Single Rigid Tube cases)

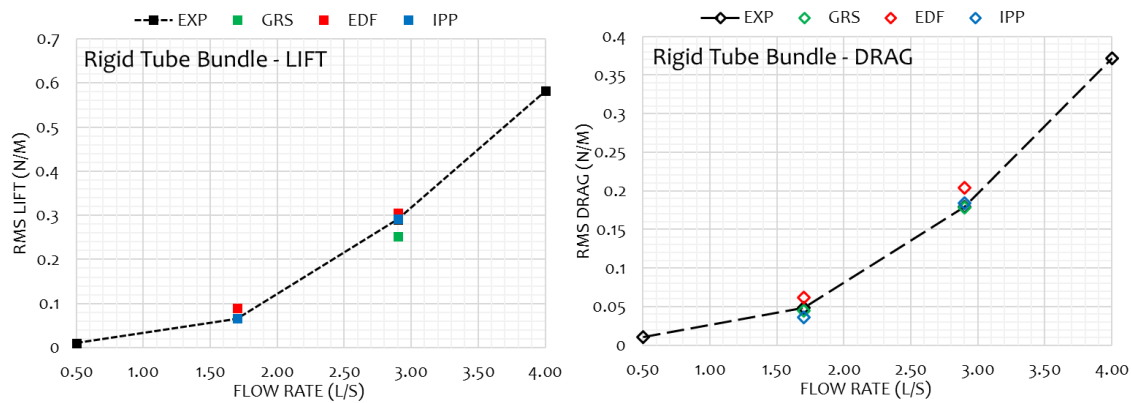


Figure 155: Comparison of simulated and experimental values of RMS value of the turbulent force per unit length as a function of the flow rate (Rigid Tube Bundle cases)

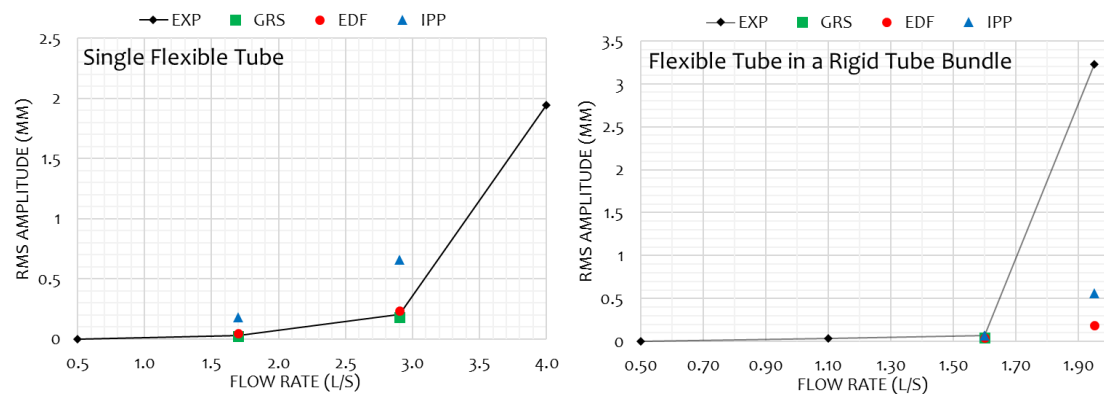


Figure 156: Comparison of simulated and experimental values of RMS value of the displacement as a function of the flow rate

Overall, 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, as 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. This indicates that, 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.

Conclusion

This work package is dedicated to development, investigation and identification of existing methods to provide reliable assessment of structural vibrations, occurring in heat exchangers and steam generators under cross-flow conditions. The overview of Flow Induced Vibrations phenomena as well as examples of its occurrence in nuclear power plants equipment and the possible impact on safety and operability issues are discussed in Chapter 1.

In the framework of current Deliverable both high-resolution computations as well as simplified models and simulation approaches are investigated on representative geometries for SGs – single tubes and tube bundles exposed to a single-phase cross-flow. The main numerical efforts are focused on modeling the AMOVI experiment, the short description of which is provided in Chapter 2 with indication of calculation cases simulated by involved partners (EDF, GRS and IPP). Each partner was aimed to simulate the time dependent vibration amplitudes and forces acting on the rod for selected test cases. The analysis was performed by each participant for four different configurations: single rigid tube, rigid tube bundle, single flexible tube and flexible tube in a rigid tube bundle.

In Chapter 3 the calculation models and tools used by GRS to perform the simulations of the selected AMOVI test cases are described. The simulations performed using ANSYS CFX R21.2 in 3D formulation for all considered cases with application of the Rigid Body Approach for flexible tube configurations. Different turbulence models – SST, Omega-RSM, SAS and ZLES for the calculation of the drag and lift fluid-induced forces on the tube surface in rigid tube configurations and SST, Omega-RSM and ZLES for the cases with flexible tube configuration were investigated during AMOVI simulations. For the single rigid tube case the SAS hybrid model demonstrated the best agreement with experimental data. While, for the rigid tube bundle case, the ZLES model was found to perform best: 13% deviation from experiment for the case with 2.9 l/s at the test section inlet 6% deviation for the $Q=1.7$ l/s case. During the FSI analyses with single flexible tube configuration Omega-RSM showed best results by not only capturing first vortex shedding frequency and the reduced natural frequency of vibration, but also the RMS displacement. Among all other turbulence models, the ZLES model showed best agreement with experiment for the flexible tube in a rigid tube bundle case. Vibration frequency and tube displacement were predicted well by the code using the Rigid Body Motion approach.

Chapter 4 provides the description of the approaches and models used by IPP to calculate both fluid-induced forces on the tube surface as well as the tube vibrations characteristics in selected AMOVI test cases. ANSYS CFX code is also used for all simulations. The main difference between these simulations and those presented above is the use of a quasi-two-dimensional formulation to achieve the least resource-intensive models, which should allow for predicting FIV characteristics with an accuracy sufficient for engineering analysis. For the cases with a flexible rod, two options for the mechanical modeling are considered: the Rigid Body Solution or a user FORTRAN subroutine with the Beam equation solvers. The SST turbulence model is considered as the main one for such type of calculations, however the application of SAS and Omega-RSM models is also investigated. Despite the two-dimensional formulation, for which the use of eddy-resolving approaches does not provide a fundamental eddy structure, it improved the predictive capabilities in regimes where, when using SST model, oscillations were not modeled at all (i.e. in the bundles at low flow rates). Results of performed quasi-2D URANS simulations demonstrated relatively good predictive capabilities with capturing the fluid elastic instability phenomena, which can be useful primarily for engineering assessments.

The high-resolution simulations of AMOVI in code_Saturne v.8.0 were performed by EDF and described in Chapter 5. For this purpose the supercomputers Cronos and Selena have been used. Simulations performed with application of wall-resolved LES and ALE approach to ensure mesh deformation for FSI. The rigid body motion equations implemented in the code are applied to simulate the rods vibrations. Both rigid tube and flexible tube cases with single rod and rod bundle configurations were analyzed at different flow rates. Important conclusions regarding the required physical time to reach a decent convergence was made for both single cylinder and tube bundle configurations. Good agreement of calculated lift and drag forces RMS and spectra as well as the displacements PSD and RMS with the experimental data was obtained in almost all analyzed cases, with the exception of single rigid tube configuration, for which the large differences in lift and drag forces standard deviations were obtained. The Fluid Elastic Instability phenomenon was predicted with the applied wall-resolved LES simulations.

In Chapter 6 the overall benchmark comparison of AMOVI simulations done by GRS, IPP and EDF for chosen cases is performed. This allowed to investigate and compare the abilities of different approaches and developed models to predict FIV phenomenon. The comparison of computing resources needed to perform both simulations with rigid tube and flexible tube configurations is presented as well. It 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.

Bibliography

- [1] EPRI (2008) Fuel Reliability Guidelines: PWR Grid-to-Rod Fretting.
- [2] International Atomic Energy Agency (2019) Review of Fuel Failures in Water Cooled Reactors (2006–2015): An Update of IAEA Nuclear Energy Series No. NF-T-2.1. IAEA, Vienna.
- [3] Paidoussis, M.P. (1980) Flow-induced vibrations in nuclear reactors and heat exchangers. Practical Experiences and State of Knowledge in Practical Experiences with Flow-Induced Vibrations, 1–81.
- [4] Blevins, R.D. (1979) Flow-induced vibration in nuclear reactors: A review. Progress in Nuclear Energy, 4, 25–49. <https://www.sciencedirect.com/science/article/pii/0149197079900088>
- [5] Kotthoff, K., Haunhorst, H., Schleifer, F., Trambauer, K. (1984) Erkenntnisse aus dem Ablauf ausländischer Vorkommnisse mit Dampferzeuger-Heizrohrbruch, GRS. Technische Mitteilungen, Bd. 77, Nr. 1.
- [6] US NRC (2015) <https://www.nrc.gov/docs/ml1501/ML15015A419.pdf>.
- [7] MacDonald, P.E., Shah, V.N., Ward, L.W. and Ellison, P.G. (1996) Steam generator tube failures.
- [8] R. D. Blevins (1979) Formulae for Natural Frequency and Mode Shapes.
- [9] OECD/NEA (2023) The FSI Benchmark Based on OKBM Experiments to Validate Coupled CFD and CSD Calculations. https://www.oecd-neo.org/jcms/pl_92333/the-fsi-benchmark-based-on-okbm-experiments-to-validate-coupled-cfd-and-csd-calculations-nea-benchmark-exercise?details=true
- [10] Dolfen, H., Bertocchi, F., Rohde, M. and Degroote, J. (2019) Numerical simulations of vortex-induced vibrations in a 7-rod bundle compared to experimental data. Proc. of the SESAME International Workshop, Petten, The Netherlands.
- [11] Vivaldi, D. and Baccou, J. (2023) URANS and Hybrid URANS/LES Coupled Fluid-Structure Simulations of Fluid-Induced Vibrations in a Square-Pitch Tube Bundle Subjected to Water Cross-Flows.
- [12] Benhamadouche, S., Benguigui, W. (2023) Wall-Resolved LES and Low-Reynolds Number URANS Combined to an Arbitrary Lagrangian Eulerian Approach for Predicting Water Cross-Flow Induced Vibrations of a Single Flexible Tube in a Normal Square Tube Array. Proc. of the NURETH-20 conference, Washington, USA.
- [13] Zwijsen, K., Tajfirooz, S., Roelofs, F., Papukchiev, A., Vivaldi, D., Hadžić, H., Benhamadouche, S., Benguigui, W., Norddine, T., Iacovides, H., Cioncolini, A., Nabawy, M., Angele, K., Lillberg, E., Chazot, B. and Iyamabo, E. (2025) Outcome of the VIKING project: status and perspectives of numerical modeling of flow-induced vibrations of nuclear power plant components.
- [14] Papukchiev, A., Mistry, H. and Herb, J. Eds. (2024) Flow-induced vibrations in nuclear steam generators: simulation of the AMOVI experiment with different FSI approaches.
- [15] Panunzio, D. (2022) Single-phase AMOVI experiments and benchmark: GO-VIKING deliverable D3.1. <https://go->

viking.eu/images/resources/GO_VIKING_D3_1_Single_phase_AMOVI_experiments__benchmark_submitted_on_21_11_2023.pdf

- [16] OECD Nuclear Energy Agency (2015) Best Practice Guidelines for the Use of CFD in Nuclear Reactor Safety Application - Revision.
- [17] Menter, F.R. (1994) Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32, 1598–1605.
- [18] Menter, F.R. (2015) Best Practice: Scale-Resolving Simulations in ANSYS CFD.
- [19] Papukchiev, A. and 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, Busan, Südkorea.
- [20] ANSYS, Inc. (2025) ANSYS CFX User's Manuals R2025R1.
- [21] Kachanov, Y.S., Kozlov, V.V. & Levchenko, V.Y. Occurrence of Tolman-Schlichting waves in the boundary layer under the effect of external perturbations. *Fluid Dyn* 13, 704–711 (1978). <https://doi.org/10.1007/BF01050234>
- [22] Abhishek Mishra, Lyazid Djenidi, Amit Agrawal; Dynamics of wall jet flow under external pulsation. *Physics of Fluids* 1 September 2021; 33 (9): 095103. <https://doi.org/10.1063/5.0057695>.
- [23] Jaiman, R., Li, G., & Chizfahm, A. (2023). *Mechanics of Flow-Induced Vibration: Physical Modeling and Control Strategies*. Springer Nature.
- [24] Borna, A., Habashi, W. G., McClure, G., & Nadarajah, S. K. (2013). CFD-FSI simulation of vortex-induced vibrations of a circular cylinder with low mass-damping. *Wind Struct*, 16(5), 411-431.
- [25] Muhamad Pauzi, A. (2024). URANS simulation and experimental validation of axial flow-induced vibrations on a blunt-end cantilever rod for nuclear applications. *Research Square* (Research Square). <https://doi.org/10.21203/RS.3.RS-4014582/V1>
- [26] Alexander van Zuijlen, Zwijsen K., Filonov V., Filonova Y., Papukchiev A. (2026). Validation of fast-running FSI methods: GO-VIKING deliverable D.5.2.
- [27] Bistany S., Norddine T., Muaaz-Hussain M., Harris E., Filonov V., Filonova Y., Dubyk Y. (2025). Simulations of the single-phase Cantilever Rod benchmark: GO-VIKING deliverable D.2.2
- [28] Blevins, R. D. (2015). *Formulas for dynamics, acoustics and vibration*. John Wiley & Sons.
- [29] J. Cardolaccia and F. Baj. An Experimental Study of Fluid-Structure Interaction in Basic In-Line Arrangements of Cylinders. *Proceedings of the ASME 2015 Pressure Vessels & Piping Division Conference PVP2015*. July 19-23. 2015. Boston. Massachussets. USA.
- [30] Afgan. I., Kahil. Y., Benhamadouche. S., & Sagaut. P. (2011). Large eddy simulation of the flow around single and two side-by-side cylinders at subcritical Reynolds numbers. *Phys. Fluids*. 23.7.
- [31] S. Benhamadouche. I. Afgan. R. Manceau. Numerical Simulations of Flow and Heat Transfer in a Wall-Bounded Pin Matrix. *Flow Turbulence and Combustion*. 104(1). pp. 19–44 (2020).

- [32] Huvelin. F. (2008). Couplage de codes en interaction fluide-structure et applications aux instabilités fluide-élastiques. Thèse de l'Université de Sciences et Technologies de Lille.
- [33] Zwijsen K., Franklin T., Carasik L., Benhamadouche S. (2026). GOKSTAD model preparation and simulation results: GO-VIKING deliverable D.3.4