



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



Validation of fast-running FSI methods

Authors : Alexander VAN ZUIJLEN (TUD), Mohammed Muaaz M.D. Hussain (NRG PALLAS, TUD), Kevin Zwijsen (NRG PALLAS), Vladislav Filonov (IPPC), Yuliia Filonova (IPPC), Angel Papukchiev (GRS)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	Validation of fast-running FSI methods
Author(s)	Alexander VAN ZUIJLEN, Mohammed Muaaz M.D. Hussain (NRG PALLAS, TUD), Kevin Zwijsen (NRG PALLAS), Vladislav Filonov (IPPC), Yuliia Filonova (IPPC), Angel Papukchiev (GRS)
Number of pages	169
Document type	Deliverable
Work Package	WP5
Document number	D5.2
Issued by	TUD
Date of completion	2026-06-03 14:26:29
Dissemination level	Public

Summary

Description of the results from the validation of the fast-running methods.

Approval

Date	By
2026-06-03 14:41:45	Papukchiev ANGEL (GRS)
2026-06-03 14:42:24	Papukchiev ANGEL (GRS)



D5.2 Validation of fast-running FSI methods

Month 48

Version N°1
June / 2026

Alexander van Zuijlen¹, Mohammed Muaaz M. D. Hussain^{1,2}, Kevin Zwijsen², Vladislav Filonov³, Yuliia Filonova³, Angel Papukchiev⁴

¹Delft University of Technology, Delft, The Netherlands

²NRG PALLAS, Petten, The Netherlands

³IPP Centre, Kiev, Ukraine

⁴Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) gGmbH, Garching, Germany

EU DuC = oE001



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.

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

Document information

Grant Agreement / Proposal ID	101060826
Project Title	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
Project Acronym	GO-VIKING
Scientific Coordinator	Angel Papukchiev, angel.papukchiev@grs.de , GRS Kevin Zwijsen, kevin.zwijsen@nrgpallas.com , NRG PALLAS
Project starting date (duration)	1st June 2022 – 31st May 2026 (48 Months)
Related Work Package	WP5
Related Task(s)	T5.1: Fast-running FSI methods for FIV
Lead Organisation	TU Delft
Contributing Partner(s)	NRG PALLAS, GRS, IPP, TUD
Due Date	30-11-2025
Submission Date	02-06-2026
Dissemination level	PU

History

Date	Version	Submitted by	Reviewed by	Comments
02.06.2026	1.0	Alexander H. van Zuijlen	Kevin Zwijsen, Angel Papukchiev	First submitted version



Table of contents

1	Validation of the Anisotropic Pressure Fluctuation Model (AniPFM) for fast running FSI	18
1.1	Introduction	18
1.1.1	Updates to AniPFM	18
1.1.2	Experimental Case.....	20
1.2	Validation Case Studies	21
1.2.1	Vibration Study in Air/Vacuum	22
1.2.2	Vibration Study in Still Water.....	24
1.2.3	Flow-only simulation with AniPFM at annular $Re=21.2k$	28
1.2.3.1	Challenges specific to the cantilever rod problem.....	28
1.2.3.2	Flow-only case setup and results	30
1.2.4	1-way FSI vibration study with AniPFM at annular $Re=21.2k$	41
1.3	Conclusions	45
1.3.1	Best Practices	46
1.3.1.1	Deal II	46
1.3.1.2	Vibration in still water with gravity.....	48
1.3.1.3	Synthetic Turbulence	50
1.3.1.4	General Advice	50
1.3.2	Future Work.....	51
2	Validation of the ANSYS Model Order Reduction for fast-running FSI simulations.....	52
2.1	The AMOVI Experiment	52
2.2	FSI Approaches.....	53
2.2.1	Full-order model ANSYS CFX-Mechanical	53
2.2.2	Reduced-order model ANSYS CFX-MOR.....	54
2.3	AMOVI structural model and modal analysis.....	54
2.4	Generation and Implementation of the MOR Model in ANSYS CFX	58
2.5	Model Preparation for the Validation on the AMOVI Single Flexible Tube Experiments	60
2.5.1	CAD and mesh generation	60
2.5.2	Generation of the ANSYS CFX model and setup	63
2.6	Results and Comparison Between FOM and ROM for the $Q=2.9$ l/s experiment...	63
2.6.1	Flow pattern.....	63
2.6.2	Vibration analysis.....	66

2.6.3	Further investigations to improve the numerical results through the application of a hybrid turbulence model	67
2.6.4	Further investigations on the number of modes considered in the MOR approach	75
2.7	Results for the Q=1.7 l/s experiment	77
2.7.1	Flow pattern.....	78
2.7.2	Vibration analysis.....	78
2.8	Validation of ANSYS CFX-MOR on AMOVI single flexible tube in a rigid tube bundle experiment.....	80
2.8.1	Mesh generation	81
2.8.2	Simulation of the flexible tube in a rigid tube bundle experiment with 1.6 l/s inlet flow	81
2.8.2.1	Simulation setup and boundary conditions	81
2.8.2.2	Analysis of the Flow Pattern in the AMOVI Test Section	83
2.8.2.3	Vibration analysis.....	88
2.8.3	Simulation of the flexible tube in a rigid tube bundle experiment with 1.95 l/s inlet flow	90
2.8.3.1	Simulation setup and boundary conditions	90
2.8.3.2	Analysis of the Flow Pattern in the AMOVI Test Section	91
2.8.3.3	Vibration Analysis.....	92
2.8.3.4	Further efforts to improve the numerical results	93
2.8.4	Simulation of the flexible tube in a rigid tube bundle experiment with 1.1 l/s inlet flow	98
2.8.5	Simulation of the flexible tube in a rigid tube bundle experiment with 0.5 l/s inlet flow	101
2.9	Conclusions on the AMOVI validation activities	104
3	Validation of the Synthetic Turbulence procedure implemented based on Stable Fluid Algorithm for fast-running FSI	106
3.1	Extending the capabilities of the Stable Fluid Algorithm based on deterministic vorticity source and Synthetic Turbulence for FSI engineering practice.....	106
3.2	Definition of oscillators and forces calculation method	113
3.3	Organization of calculations and available options	117
3.4	Validation of the developed Synthetic Turbulence Generator	123
3.4.1	Validation of STG on AMOVI Single Flexible Tube Experiment	124
3.4.1.1	Simulation of the Single Flexible Tube experiment with Q=4.0 l/s	124
3.4.1.2	Simulation of the single flexible tube experiment with Q=2.9 l/s and Q=1.7 l/s	129
3.4.2	Validation of STG on AMOVI Single Flexible Tube in a Rigid Tube Bundle Experiment (Q=1.95 l/s and Q=1.6 l/s)	131

3.4.3	Overall summary of STG validation on AMOVI cases.....	134
4	Validation of the Projection Based Fluid Model for fast running FSI.....	137
4.1	Introduction	137
4.2	Building ROM for cantilever rod case	139
4.2.1	Meshing and material properties.....	139
4.2.2	Solution procedure	140
4.3	Verification.....	145
4.4	Validation	146
4.4.1	Wall turbulence pressure fluctuation model	146
4.4.2	Results.....	149
4.5	Conclusions	151
5	Conclusions.....	153
6	Bibliography.....	158
	Appendix A: Matlab script to create .pre macro to define mode shape interpolations	160
	Appendix B: Matlab script to create .pre macro to define .def simulation definition files for each mode	166
	Appendix C: Matlab script to create .cse macro for CFD-Post to export fluid surface response modes	167

List of figures

Figure 1. A flow-chart of the different computational steps of the proposed AniPFM	20
Figure 2. Experimental test section: (a) full test piece with a closeup showing the internal structure of the flow straightener; (b) union cross internal configuration, and schematic and dimension of hemispherical end-piece of rod (Hatching represents the rod pipe) (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022)	21
Figure 3. Solid Mesh: front view (left), bottom view (top right); Blocking Strategy: hollow cylinder (bottom right: left) and hemispherical tip (bottom right: right)	23
Figure 4. Time history of vibrations for the experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022) (left) and numerical results ($E=197.822$ GPa) (right)	24
Figure 5. Fluid geometry and sectional view of the mesh	25
Figure 6. Time history of vibrations for the experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022) (left) and numerical results (S1Y1G1) (right)	27
Figure 7. Horizontal distribution of <i>kstart</i> , <i>ke</i> , and <i>kcut</i> in the pipe section ($y=-1.5$ m, left) and annulus section ($y=-0.5$ m, right). Note: $y=0$ m is at the outlet.	28
Figure 8. Sample <i>p'</i> obtained during testing	30
Figure 9. Time-averaged non-dimensional axial velocity 10D ahead of rod tip.....	31
Figure 10. Mean non-dimensional radial velocity 10D ahead of rod tip (right: Zoomed in)...	32
Figure 11. Time-averaged non-dimensional Reynolds stress component ($u'v'$) 10D ahead of rod tip	33
Figure 12. Comparison of base URANS <i>k</i> – <i>ω</i> SST results with experimental (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022), and high fidelity (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025) results for time-averaged non-dimensional axial component of velocity	34
Figure 13. Comparison of base URANS <i>k</i> – <i>ω</i> SST results with experimental (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022), and high fidelity (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025) results for time-averaged non-dimensional radial component of velocity	34
Figure 14. Comparison of AniPFM results with experimental (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022), and high fidelity (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025) results for non-dimensionalized stress components.....	35
Figure 15. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 15.63786 mm upstream of the rod tip	36

Figure 16. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 3.786006 mm upstream of the rod tip	36
Figure 17. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 0.493836 mm upstream of the rod tip	37
Figure 18. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 0.164619 mm downstream of the rod tip	37
Figure 19. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 6.74898 mm downstream of the rod tip	37
Figure 20. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 29.135799 mm downstream of the rod tip	38
Figure 21. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 15.63786 mm upstream of the rod tip.....	39
Figure 22. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 3.786006 mm upstream of the rod tip.....	39
Figure 23. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 0.493836 mm upstream of the rod tip.....	40
Figure 24. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 0.164619 mm downstream of the rod tip.....	40
Figure 25. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 6.74898 mm downstream of the rod tip.....	40
Figure 26. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 29.135799 mm downstream of the rod tip.....	41
Figure 27. Time history of the non-dimensional rod tip front (left) and side (right) displacements	42
Figure 28. 3D time history of non-dimensional rod tip displacements.....	42
Figure 29. 2D polar time history of non-dimensional rod tip displacements	43
Figure 30. PSD of the tip displacement in the front (x-) direction	44
Figure 31. PSD of the tip displacement in the side (y-) direction.....	45
Figure 32. Spurious velocities observed for 'rest' conditions	49
Figure 33. AMOVI test section with two different configurations: single tube (left) and tube bundle (right) in a flow channel.....	53
Figure 34: Numerical mesh of the AMOVI flexible tube and its blade.....	55

Figure 35: First bending mode of the flexible tube and its blade.	55
Figure 36: Second bending mode of the flexible tube and its blade.	56
Figure 37: Third bending mode of the flexible tube and its blade.....	56
Figure 38: Fourth bending mode of the flexible tube and its blade.	57
Figure 39: Fifth bending mode of the flexible tube and its blade.....	57
Figure 40: Sixth bending mode of the flexible tube and its blade.	58
Figure 41: Input file for MOR approach providing data for the first bending mode of the AMOVI tube and blade.....	59
Figure 42: CEL expressions for the displacements in x, y, z directions, provided to ANSYS CFX for the calculation of the new, deformed mesh.....	59
Figure 43: CAD model of the AMOVI test section with a single vibrating tube.	61
Figure 44: Top view on the CFD mesh with single vibrating tube.....	62
Figure 45: Bottom view on the CFD mesh with single vibrating tube.....	62
Figure 46: Horizontal and vertical planes through the test section showing mesh density around the vibrating tube.	63
Figure 47: Velocity magnitude distribution in a longitudinal plane (water flows bottom-top) through the AMOVI test section calculated with MOR.	64
Figure 48: Velocity vectors in XY plane showing flow acceleration, deceleration and recirculation zones as well as the formation of von Karman vortex street behind the cylinder.	65
Figure 49: Formation of a von Kármán vortex street behind the AMOVI tube visualized with the turbulence kinetic energy.	65
Figure 50: Tube displacements in lift direction for a single vibrating tube exposed to a $Q=2.9$ l/s flow rate.	67
Figure 51: APSD of the tube displacements for a single vibrating tube exposed to a $Q=2.9$ l/s flow rate.	67
Figure 52: Mesh with 8 million elements.....	68
Figure 53: Mesh with 8 million elements (longitudinal view).....	69
Figure 54: Mesh with 8 million elements (longitudinal view).....	69
Figure 55: Tube displacements in lift direction.....	70

Figure 56: APSD of the tube displacements in lift direction.	70
Figure 57: Separation of the flow domain in pre-defined LES (red) and URANS (blue) zones for the needs of the ZLES turbulence approach.....	72
Figure 58: Y^+ values on the FSI surface for the single tube case with $Q=2.9$ l/s at the inlet of the test section.	72
Figure 59: Velocity magnitude in longitudinal direction, calculated with ZLES.	73
Figure 60: Velocity-colored streamlines in the vicinity of the vibrating tube.....	74
Figure 61: Turbulence kinetic energy in a horizontal cross-section close to the end of the vibrating tube (large displacement).	74
Figure 62: Tube displacement in lift direction with the SST (MOR) and the ZLES (ZLES MOR) turbulence models.....	75
Figure 63: Auto power spectral density of the tube displacements calculated with the SST (MOR) and the ZLES (ZLES MOR) turbulence models.	75
Figure 64: Tube displacements calculated with the consideration of 1 and 6 modes.....	76
Figure 65: APSD of the tube displacements calculated with the consideration of 1 and 6 modes.	77
Figure 66: Velocity distribution in a vertical plane through the middle of the test section ($Q=1.7$ l/s).....	78
Figure 67: Tube displacement in lift direction with the ZLES turbulence model with $Q=1.7$ l/s at the test section inlet.....	79
Figure 68: Auto power spectral density of the tube displacements ($Q=1.7$ l/s).	80
Figure 69: Numerical mesh of the tube bundle with the flexible tube in its center.	81
Figure 70: Separation of the AMOVI tube bundle domain in pre-defined LES (red) and URANS (blue) zones for the needs of the ZLES turbulence approach.....	82
Figure 71: Y^+ values on the tube bundle surfaces.	83
Figure 72: Velocity distribution in the AMOVI test section equipped with a tube bundle ($Q=1.6$ l/s).....	84
Figure 73: Velocity distribution in the AMOVI test section in a vertical plane ($Q=1.6$ l/s).	85
Figure 74: Velocity distribution in the AMOVI test section in a horizontal plane ($Q=1.6$ l/s).	85
Figure 75: Recirculation zones visualized with velocity-colored vectors ($Q=1.6$ l/s).	86
Figure 76: Zoom-in of the top left corner of the picture above ($Q=1.6$ l/s).....	86

Figure 77: Formation of two recirculation zones behind certain tubes rotating in opposite directions (flow from bottom to top of the picture) ($Q=1.6$ l/s).	87
Figure 78: Measured (top) and calculated (bottom) time-averaged streamwise velocity in the tube bundle ($Q=1.6$ l/s).	88
Figure 79: Measured and calculated tube displacements for the 1.6 l/s case.....	89
Figure 80: Measured and calculated tube displacements for the 1.6 l/s case.....	90
Figure 81: Y^+ value on the walls of the tubes ($Q=1.95$ l/s).	91
Figure 82: Instantaneous velocity distribution at $t=6.2$ s in the AMOVI test section equipped with a tube bundle ($Q=1.95$ l/s).....	92
Figure 83: Time-averaged velocity distribution in the AMOVI test section in a vertical plane ($Q=1.95$ l/s).	94
Figure 84: Time-averaged velocity distribution in the AMOVI test section in a horizontal plane ($Q=1.95$ l/s).	94
Figure 85: Recirculation zones visualized with velocity-colored vectors at $t=6.2$ s ($Q=1.95$ l/s).	95
Figure 86: Recirculation zones (time-averaged) visualized with velocity-colored vectors ($Q=1.95$ l/s).	95
Figure 87: Measured and calculated tube displacements ($Q=1.95$ l/s).	96
Figure 88: Comparison of the tube displacements in both cases.	96
Figure 89: Measured and calculated APSD for the 1.95 l/s case.	97
Figure 90: Measured and calculated tube displacements for the 1.95 l/s case.....	98
Figure 91: Velocity distribution in the AMOVI test section in a horizontal plane ($Q=1.1$ l/s) at $t=5.06$ s.	99
Figure 92 Measured and calculated with ANSYS CFX – MOR and ANSYS CFX – Mechanical tube displacements for the 1.1 l/s case.	100
Figure 93 Measured and calculated APSD for the $Q=1.1$ l/s case.	101
Figure 94 Velocity distribution in the AMOVI test section in a horizontal plane ($Q=0.5$ l/s) at $t=9.27$ s.	102
Figure 95 Measured and calculated tube displacements for the 0.5 l/s case.....	103
Figure 96 Measured and calculated APSD for the $Q=0.5$ l/s case.	103
Figure 97: Dynamic Mesh Block.....	112

Figure 98: General view of the two-part AMOVI beam	115
Figure 99: The influence of discretization and time step on the characteristics of free oscillations	116
Figure 100: Boundary conditions and typical meshes applied in the calculation procedure	120
Figure 101: The basic block diagram of the calculation procedure	121
Figure 102: Results of displacements calculation for the single flexible tube with $Q = 4.0$ l/s	125
Figure 103: Beam displacements over time at different reference points and the range of deviations obtained for the single flexible tube case with $Q = 4.0$ l/s using SFA+ST	126
Figure 104: Visualization of the main instantaneous fields calculated using SFA+ST for S-Mesh and F-Mesh meshes (case $Q = 4.0$ l/s, automatic legend)	127
Figure 105: Visualization of the main instantaneous fields calculated using ANSYS CFX + ST for the single flexible tube case with $Q = 4.0$ l/s (a – integration time step 2.5 ms, K10; b – integration time step 1.0 ms, K31)	128
Figure 106: Results of displacements calculation for the single flexible tube with $Q = 2.9$ l/s	130
Figure 107: Results of displacements calculation for the single flexible tube with $Q = 1.7$ l/s	131
Figure 108: Results of displacements calculation for the case with a flexible tube in a rigid tube bundle and $Q = 1.95$ l/s	132
Figure 109: Results of displacements calculation for the case with a flexible tube in a rigid tube bundle and $Q = 1.6$ l/s	133
Figure 110: Tube displacements over time at different reference points and the range of deviations obtained for the flexible tube in a rigid tube bundle cases with $Q = 1.95$ l/ and $Q=1.6$ l/s using SFA+ST	133
Figure 111: Comparison of simulated and experimental values of displacement RMS as a function of the flow rate.....	135
Figure 112: Comparison of the specific time spent for calculations of analyzed AMOVI cases	136
Figure 113: Full order model.....	137
Figure 114: Modal reduced order model.....	138
Figure 115: Modal reduction of the fluid-structure interaction system	138
Figure 116: Unified modal reduction for turbulence induced vibration.....	139

Figure 117: Domain cantilever rod	139
Figure 118: Mesh block structure cantilever rod	139
Figure 119: Structure model tip (left) and full rod (right)	140
Figure 120: Structural bending mode pairs 1 and 2.....	141
Figure 121: Structural bending mode pairs 3 and 4.....	141
Figure 122: RANS convergence and resultant force cantilever rod	143
Figure 123: Comparison of full and reduced order model in vacuum and in water	145
Figure 124: Periodic annular flow mesh (left) and wall pressure and Q-criterion iso-surfaces coloured by velocity magnitude (right)	147
Figure 125: Turbulence wall pressure fluctuations obtained from OpenFOAM periodic annular flow: pressure field at odd time steps (left), at even time steps (middle) and averaged pair (right)	148
Figure 126: Reduced order fluid and structure model forced by turbulence pressure fluctuations sampled from fully developed annular channel flow; (a) fully correlated, (b), uncorrelated, frozen turbulence with (c) $Uc = 1.04\text{m/s}$, (d) $Uc = 1.67\text{m/s}$, (e) $Uc =$ 4.17m/s	150
Figure 127: Cantilever front displacement [taken from D2.1] with ROM tip displacement using full correlation (blue), uncorrelated (red) and advective $Uc = 1.67\text{m/s}$ (light green), $Uc = 4.17\text{m/s}$ (dark green)	151

List of tables

Table 1. Solid Material Properties.....	22
Table 2. Obtained results: (experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022): $f_n, a = 4.10$ Hz, $\zeta_n, a = 0.9 \times 10^{-3}$).....	24
Table 3. Fluid Material Properties.....	24
Table 4. Tested Cases.....	26
Table 5. Obtained results: (experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022): $f_n, w = 3.72$ Hz, $\zeta_n, w = 15 \times 10^{-3}$)	27
Table 6. Observed rod tip offset	43
Table 7. Obtained displacement results	43
Table 8. Frequencies of the numerical and experimental front displacement PSD peaks compared to the beam natural frequencies obtained analytically	44
Table 9. Frequencies of the numerical and experimental front displacement PSD peak #1 compared to the beam natural frequencies obtained analytically	45
Table 10. Obtained results: (experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022): $f_n, w = 3.72$ Hz, $\zeta_n, w = 15 \times 10^{-3}$)	50
Table 11: Calculated eigenfrequencies with Mechanical, used as input for the MOR approach	58
Table 12: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube with 2.9 l/s flow rate	77
Table 13: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube with 1.7 l/s flow rate	80
Table 14: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube in a rigid tube bundle ($Q=1.6$ l/s).	90
Table 15: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube in a rigid tube bundle ($Q=1.95$ l/s).	97
Table 16: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube in a rigid tube bundle ($Q=1.1$ l/s).	101
Table 17 Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube in a rigid tube bundle ($Q=0.5$ l/s).	103
Table 18: Algebraic relations for calculating the pressure fluctuation field	111

Table 19: Mathematical formulation of the combinations of basic boundary conditions used for the beam ends in procedure.....	114
Table 20: Mathematical formulation of the two ends conjunction of the uniform segments in the beam	114
Table 21: Main parameters of the calculation procedure	117
Table 22: User input parameters for Lib ANSYS CFX.....	122
Table 23: Overview of available AMOVI test cases with FIV	123
Table 24: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube ($Q=4.0$ l/s)	129
Table 25: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube ($Q=2.9$ l/s)	130
Table 26: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube ($Q=1.7$ l/s)	131
Table 27: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle ($Q=1.95$ l/s and $Q=1.6$ l/s)	134
Table 28: Mesh block parameters cantilever rod	139
Table 29: Effective material properties cantilever rod	140
Table 30: Natural frequencies first 8 bending modes.....	141
Table 31: Operating and boundary conditions cantilever rod model.....	142
Table 32: RMS tip displacement for reduced order fluid and structure model forced by turbulence pressure fluctuations, same conditions as Figure	150

Abbreviations and Acronyms

Acronym	description
AniPFM	Anisotropic Pressure Fluctuation Model
CFD	Computational fluid dynamics
DES	Detached eddy simulation
DNS	Direct numerical simulation
FEM	Finite element method
FOM	Full-order model
GO-VIKING	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
WP	Work package
LES	Large eddy simulation
MOR	Model order reduction
PWR	Pressurized water reactor
ROM	Reduced-order model
SAS	Scale adaptive simulation
SRS	Scale-resolving simulations
TIV	Turbulence-Induced Vibrations
URANS	Unsteady Reynolds-Averaged Navier-Stokes

Summary

For an accurate prediction of the impact of flow induced vibrations in nuclear power generation, computational methodologies are necessary that can simulate the interaction between fluids and structures. Within different components of the nuclear power generation system, interaction effects like turbulence induced vibration (mostly for axial flow cases) and flow/vortex induced vibration (mostly for cross flow cases) in single rod and rod-bundle configurations occur. Simulation of such cases using direct CFD-FEM coupling approaches with high fidelity models may provide an accurate prediction, but at a prohibitive computational cost.

In this work package, fast-running methods are identified that improve computational speed for such cases by reducing computational complexity. Each of the methods described in this report aims to reduce the computational complexity by simplifying a specific part of the flow induced vibration problem. Two methods target in particular the effect of turbulence and turbulence pressure fluctuations: by introducing a synthetic turbulence model, the goal is to provide an alternative to the use of LES with a certain degree of accuracy, which will allow for engineering analysis of complex equipment configurations with acceptable computational costs. Two levels of simplification are investigated: 1) considers the anisotropy of the Reynolds stresses and solves a Poisson equation to reconstruct pressure fluctuations from the synthetic turbulence velocity fluctuations, 2) uses an algebraic expression to correlate pressure fluctuations directly to the synthetic turbulence velocity fluctuations. Two other approaches use the decomposition of the structure dynamics into decoupled modes to: 1) simplify the fluid-structure coupling by introducing a reduced order model of the structure directly into the CFD solver, and 2) building a linear fluid response model for a subset of modal deformation shapes to replace the CFD solver.

In this report, these fast-running methods are validated with experimental data obtained within the GO-VIKING project.

Keywords

Synthetic turbulence, turbulence induced vibration, turbulence pressure fluctuation model, reduced order model, flow induced vibration, fluid-structure interaction

1 Validation of the Anisotropic Pressure Fluctuation Model (AniPFM) for fast running FSI

This chapter presents the application of the fast-running FSI model called AniPFM under development by NRG PALLAS, in collaboration with the Delft University of Technology (TUD). The details of the model, as well as first validation results, both for pure fluid-flow cases and an FSI case were presented in (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024). This chapter is organized as follows: Section 1.1 provides a short reintroduction to AniPFM, including mention of small changes made to the version of AniPFM described in (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024), as well as a short introduction to the Cantilever Rod experiment, section 1.2 provides the efforts made towards the numerical simulations, while the conclusions, including best practices and future work, are provided in section 1.3.

1.1 Introduction

In the fuel assemblies, the flow is in axial direction and the driving FIV mechanism is Turbulence-Induced Vibrations (TIV), as the vibrations of the fuel rods are driven by the local, small-scale, turbulent velocity and pressure fluctuations. These fluctuations can numerically be resolved by, for example, the use of Large-Eddy Simulations (LES) or Direct Numerical Simulations (DNS). The drawback of these simulations though is that they require a large amount of computational power and time, and hence cannot be applied to most industrial applications. Cheaper numerical tools, such as those based on the Unsteady Reynolds-Averaged Navier-Stokes (URANS), which are widely used in the industry, do not resolve the fluctuations causing the structural vibrations.

To overcome this, NRG PALLAS in the Netherlands has been working on the development of the Anisotropic Pressure Fluctuation Model (AniPFM) (Zwijsen, van den Bos, Frederix, Roelofs, & van Zuijlen, 2024) to address the shortcomings found in the legacy Pressure Fluctuation Model (PFM) (Kottapalli, Shams, van Zuijlen, & Pourquie, 2019). The new model tackles several limiting assumptions of the PFM, such as that of isotropic turbulence and the method of time correlation used in PFM. With these extensions to the PFM, the AniPFM aims to improve the accuracy of the prediction of vibration amplitudes through FIV-simulations of TIV.

The model has been reported in detail in (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024). However, since then, a few small but noteworthy modifications have been made to AniPFM which are provided in section 1.1.1. The cantilever rod experiment, used to further validate AniPFM, is briefly described in section 1.1.2.

1.1.1 Updates to AniPFM

The first modification is made to the first mode wavenumber, k_{start} . Previously, it was defined as

$$k_{start} = \max \left[\min \left(\frac{\epsilon}{\max(\|\mathbf{u}\|)}, \frac{1}{2} k_e \right), \frac{2\pi}{l_{user}} \right] \quad (1)$$



while it is now updated to

$$k_{start} = \min \left[\max \left(\frac{\epsilon}{\max(\|\mathbf{u}\|)}, \frac{2\pi}{\min(l_{user}, 2d_w)} \right), \frac{1}{2}k_e \right] \quad (2)$$

where d_w is the distance to the wall, while the other terms are as defined before. This rearrangement gives a higher priority to the criterion that the mode sampling should begin before the wavenumber at which the energy is maximum, k_e . Some user control via the l_{user} still remains but it also limits the eddy size based on the wall distance.

The next modification is made to k_e , by similarly limiting it to not exceed the cutting wavenumber k_{cut} , given by

$$k_e = \min(k_e, \frac{1}{2}k_{cut}) \quad (3)$$

to ensure that the highest energies are still represented by the synthetic turbulence. Note that this is a current modeling choice that deviates from true turbulence formulations, which would be mesh independent.

Thirdly, to construct a divergence free synthetic turbulence, the requirement of $\mathbf{k}_n \cdot \boldsymbol{\sigma}_n = 0$, was mentioned where $\boldsymbol{\sigma}_n$ was previously defined as the normalized cross-product between a random vector $\boldsymbol{\zeta}_n$, and the wavenumber vector:

$$\mathbf{k}_n = k_n [\sin \theta_n \cos \psi_n, \sin \theta_n \sin \psi_n, \cos \theta_n] \quad (4)$$

$$\boldsymbol{\sigma}_n = \frac{\boldsymbol{\zeta}_n \times \mathbf{k}_n}{|\boldsymbol{\zeta}_n \times \mathbf{k}_n|} \quad (5)$$

An analytical formulation was derived and implemented for $\boldsymbol{\sigma}_n$, and is presented below

$$\boldsymbol{\sigma}_n = \begin{bmatrix} \cos \alpha_n \cos \theta_n \cos \phi_n - \sin \alpha_n \sin \phi_n, \\ \cos \alpha_n \cos \theta_n \sin \phi_n + \sin \alpha_n \cos \phi_n, \\ -\cos \alpha_n \sin \theta_n \end{bmatrix} \quad (6)$$

This formulation has two reasons of superiority to the former random vector multiplication approach. Firstly, while $\boldsymbol{\zeta}_n$ was distributed with equal probability on a unit sphere, $\boldsymbol{\sigma}_n$ was not guaranteed to be distributed with equal probability on a plane. Secondly, if $\boldsymbol{\zeta}_n$ is randomly aligned or anti-aligned with $\mathbf{k}_n$, the cross product would result in a null vector. These deficiencies do not exist in the current formulation.

Fourthly, the Wilcox correction (Wilcox, 1993) previously proposed to convert the isotropic tensor obtained from linear eddy viscosity models into an anisotropic tensor, based on the nonlinear eddy viscosity model of Wilcox (Wilcox, 1993):

$$\overline{u'u'} = \frac{8}{9}k, \quad \overline{v'v'} = \frac{4}{9}k, \quad \overline{w'w'} = \frac{6}{9}k \quad (7)$$

has not been utilized in the current study. This stems from the fact that the theory was developed from flat plate theory, and is mainly intended for constant pressure gradient flows, which is not the case for the Cantilever Rod flow geometry. Furthermore, from a

practical perspective, the correction was hard-coded in the AniPFM file assuming cartesian flows, while pipe and annular flows still need a cylindrical coordinate implementation.

Finally, instead of the convection and exponential correlation method of time correlation, the simpler frozen wake approximation of convecting turbulence at a constant uniform speed was used. All other implementations and formulae reported in (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024) have been preserved including the model overview shown in Figure 1.

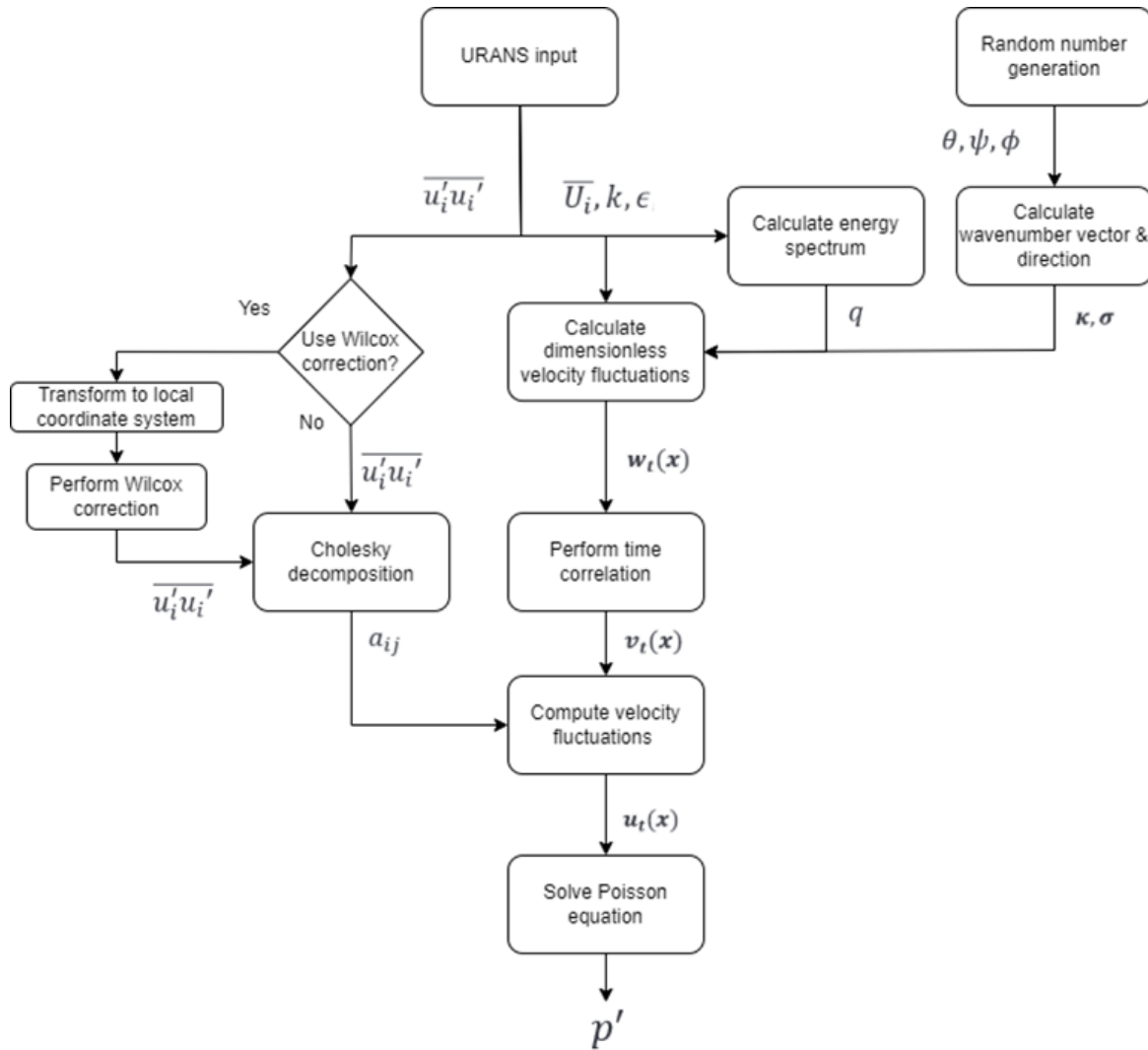


Figure 1. A flow-chart of the different computational steps of the proposed AniPFM

1.1.2 Experimental Case

The experiment of interest is the FIV of a cantilevered rod subjected to axial flow. This was performed at the University of Manchester, UK. Readers interested in full details of the same are referred to (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022). The test section and the rod tip is shown in Figure 2. The case of interest for the current cross validation of AniPFM is the free-clamped flow at an annular $Re = 21.2k$ for the rod with a hemispherical tip. Vibration tests of the rod in air and still water were also experimentally performed, and are used in the numerical studies for validation.

Key measurements include PIV data for axial and horizontal velocity components as well as Reynolds stress components in the vicinity of the rod tip, and FSI data for the rod tip displacement time history, from which vibration amplitudes and frequencies are extracted.

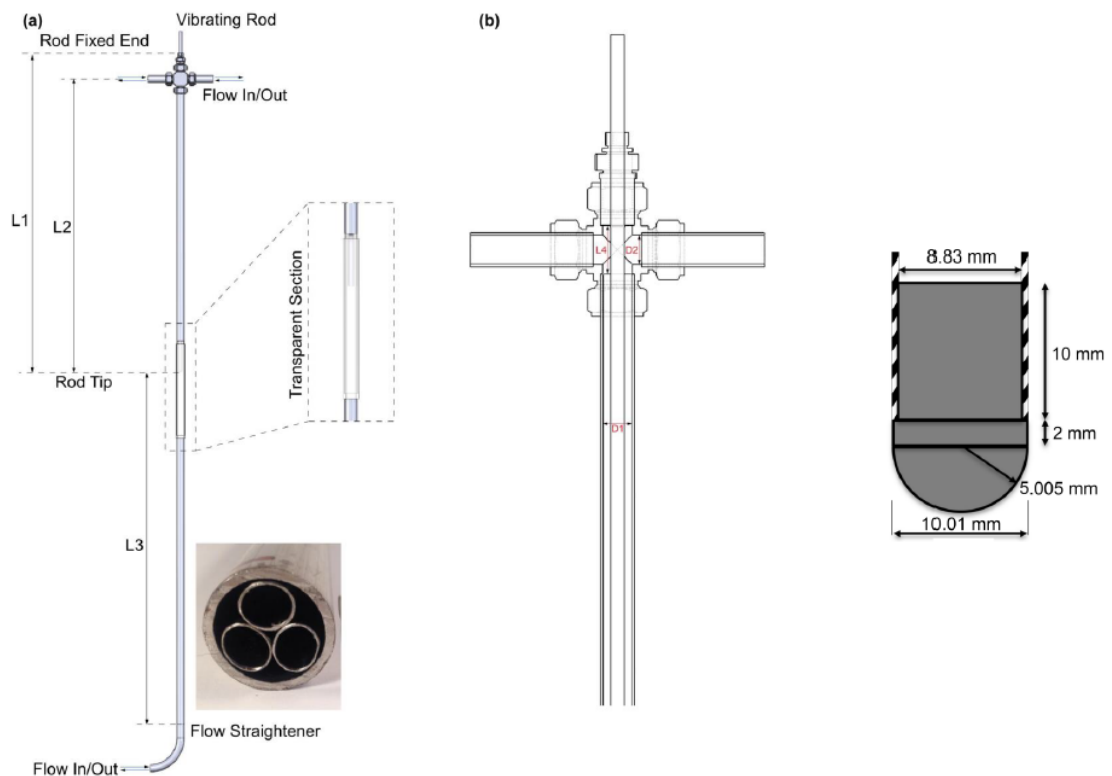


Figure 2. Experimental test section: (a) full test piece with a closeup showing the internal structure of the flow straightener; (b) union cross internal configuration, and schematic and dimension of hemispherical end-piece of rod (Hatching represents the rod pipe) (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022)

1.2 Validation Case Studies

In this section, the new AniPFM is attempted to be validated against the Cantilever Rod experimental data as well as select high fidelity data available in (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025).

To meet this goal, the following studies are performed and are subsequently reported in sections 1.2.1-1.2.4, respectively:

1. Free vibration simulation in air/vacuum
2. Free vibration simulation in still water
3. Flow-only simulation with AniPFM at annular $Re=21.2k$
4. 1-way FSI vibration study with AniPFM at annular $Re = 21.2k$

Originally, it was envisioned that after item 1 & 2, i.e, after the preliminary fluid and solid models were deemed satisfactory, a two-way FSI run would be performed for AniPFM at

annular $Re = 21.2k$. However, technical and practical difficulties were faced in the application of AniPFM, which has triggered further investigation. A few of these will be touched upon in section 1.2.3. Unfortunately, the necessary development work could not be achieved within the timeframe of the GO-VIKING project, thus items 3 & 4 were carried out as an alternative to satisfy the requirements of the current work package 5.

1.2.1 Vibration Study in Air/Vacuum

The rod geometry and mesh was created using GMSH v4.13.1. The rod geometry is simplified compared to the experiment. Firstly, the lead shots filling the rod are geometrically neglected, while the long hollow cylinder is retained. For the end plug, the real geometry could be envisioned as a Boolean CAD union of two cylinders and a hemisphere. However, in the current numerical model, the end plug is a thin-shelled envelope connecting with the hollow cylinder. Thus, the end tip shell maintains the same thickness of 0.59 mm as the hollow cylinder while topologically maintaining the hemispherical shape. This is done for ease of meshing and the possibility to assign a single material for the entire geometry. The latter stems for ease of coding the setup in Deal II v9.7.0, the open source finite element library used to solve the structural dynamics in the current study.

The material properties used in the simulation are provided in Table 1. The base material is assumed to be that of the hollow cylinder in the experiment, i.e, stainless steel (AISI316). The absence of lead shots is compensated by dramatically increasing the density to achieve the same linear mass density of 588 g/m as the experiment. The Youngs modulus of AISI316 at 20° C is 193 GPa. This was later intentionally increased in order to get perfectly matching numerical results with the experiments for the free vibration in air/vacuum. Thus, two sets of numerical results are available for this test.

Table 1. Solid Material Properties

Material	E (GPa)	ρ (kg/m ³)	ν (-)
Modified Stainless Steel	I. 193	33685.62393	0.26
	II. 197.822646		

A transfinite hexahedral structured mesh is created in GMSH by using a total of 12 blocks: 4 for the hollow cylinder and 8 for the hemispherical tip. The blocking strategy for the hemispherical tip and the resulting mesh are shown in Figure 3. Mesh refinement is controlled for the rod by prescribing the number of elements in the axial, radial and azimuthal directions. By virtue of the blocking strategy of the tip, setting the azimuthal elements also sets the discretization of the 4 innermost blocks. An additional control is on the number of elements along the edges of the 4 outermost blocks of the tip that curve from the cylinder towards the tip. The number of elements is set to 800, 64, 1, and 10, for the axial, radial, and azimuthal directions, and along the curvature of the outer blocks' curved tip edges, respectively, leading to a total cell count of 52.096k cells. This choice stemmed from the mesh independence study performed on a slightly different geometry as reported in (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025).

The mesh was put to a vibrational test in vacuum, i.e., a pure structural analysis, to test its suitability for the final FSI cases. The top flat end of the rod is fixed and the rod is subjected to a time-dependent acceleration given by

$$a = \begin{cases} [-1, -9.81, 0] \text{ m/s}^2, & t < 0.12 \text{ s} \\ [0, -9.81, 0] \text{ m/s}^2, & t \geq 0.12 \text{ s} \end{cases} \quad (8)$$

where gravity is also accounted for. The time instant of 0.12 s, when the body force is removed, is carefully chosen to match the time instant the maximum rod displacement is achieved.

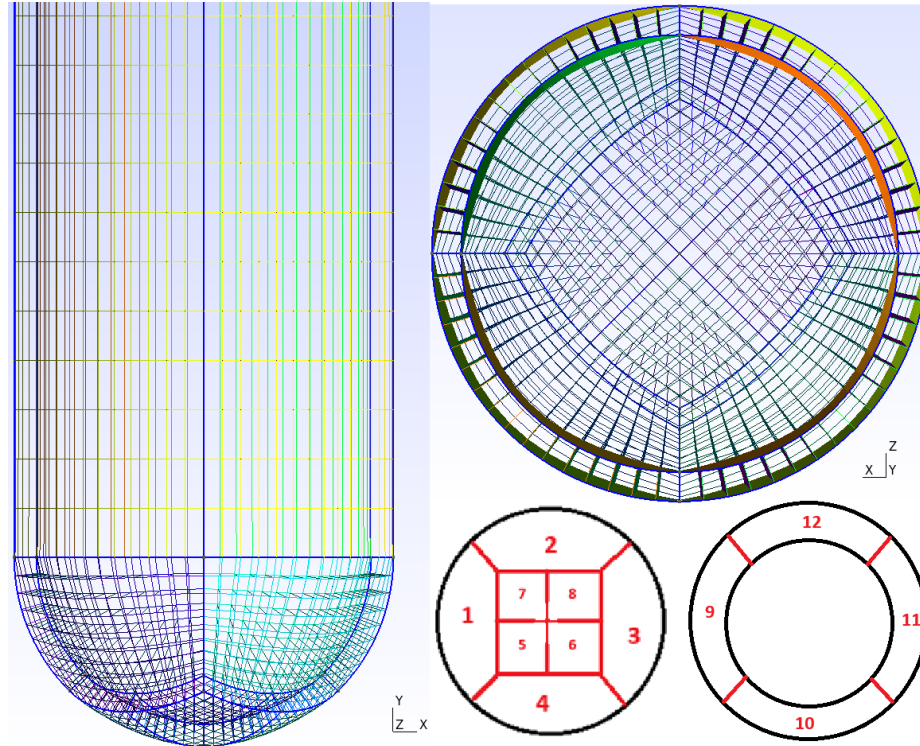


Figure 3. Solid Mesh: front view (left), bottom view (top right); Blocking Strategy: hollow cylinder (bottom right: left) and hemispherical tip (bottom right: right)

To incorporate the natural damping of the rod in air, the Rayleigh damping model is implemented in Deal II. During the run, the mass and stiffness damping coefficients are set to 0 Hz and $6.9729018e-05$ s, respectively, targeting the desired experimental damping ratio for an expected vibrational frequency. The θ -time integration is implemented and utilized with a model constant of $\theta = 0.5$, implying a second order accuracy. The discrete governing equation is solved using the Direct method via LU decomposition. This was done using the ‘SparseDirectUMFPACK’ in Deal II. There is another method available for solving the discrete governing equation, which is iterative (Conjugate Gradient), but this was found to be slower than the direct method and was thus not used moving forward.

For this test, the linear geometry model and linear finite elements are employed and the simulations (testing two Youngs Moduli on the same mesh) are run, starting at rest conditions, for 10 s using a time step size of 2.4 ms. The simulations, after making the source

code thread-able, are run with 32 threads. The tip displacement history for the fine-tuned Young's modulus case is presented in Figure 4 along with the experimental results. The time history itself is not directly comparable to the experiment as the exact forcing and initial conditions are not known. However, the natural frequency and damping ratio extracted from the time history can be compared and is done so in Table 2 for both cases. To extract this data from the raw time history, a least error fit is made with a decaying sine wave ($A_0 \sin\left(\frac{2\pi t}{T} + \phi\right) \exp(-\lambda t) + offset$) as seen in Figure 4 for the numerical results.

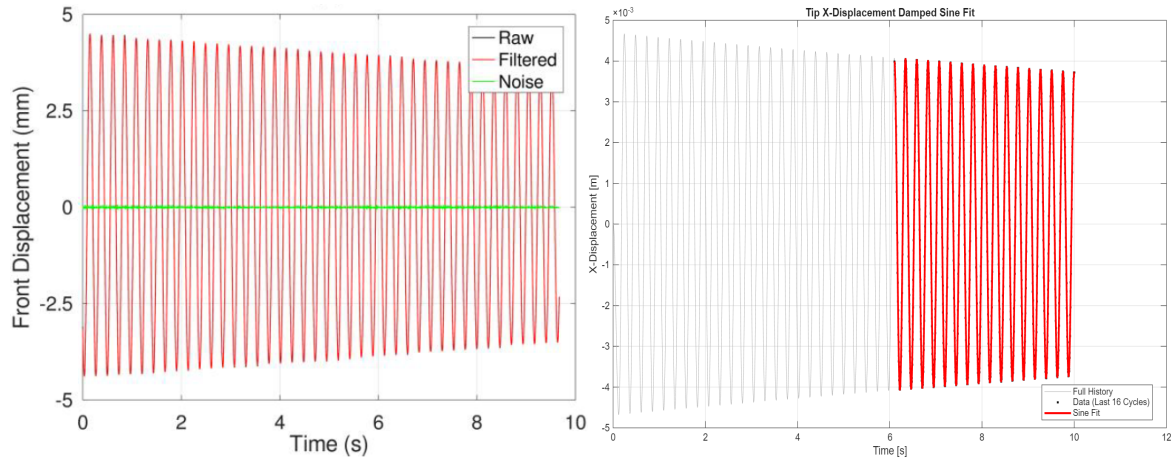


Figure 4. Time history of vibrations for the experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022) (left) and numerical results ($E=197.822$ GPa) (right)

Table 2. Obtained results: (experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022): $f_{n,a} = 4.10$ Hz, $\zeta_{n,a} = 0.9 \times 10^{-3}$)

Case E (GPa)	$f_{n,a}$ (Hz)	Error (%)	$\zeta_{n,a} (\times 10^{-3})$	Error (%)	Wall Clock Time
193	4.050574	1.21	0.889	1.20	7h 18m 47 s
197.822	4.100000	0.00	0.900	0.00	7h 19m 14s

The base results are already considerably good with only 1.2% error, requiring an increase of only $\sim 2.5\%$ in Young's modulus to get perfect results. This brings confidence in the material properties and chosen modeling strategy which, while clearly differing from the experiment, still results in a similar performance. The current mesh, material properties, and Deal II implementations are thus finalized for further testing.

1.2.2 Vibration Study in Still Water

To prepare for the FSI run, there is a need to set up a working simulation for the fluid, and testing the coupling between codes. The working fluid is water with chosen material properties at 20°C provided in Table 3.

Table 3. Fluid Material Properties

Material	$\mu (\times 10^{-3})$ (kg/(ms))	ρ (kg/m ³)
Water	1.002	998.23

The fluid geometry and mesh is prepared using STAR-CCM+ V2306 and then later imported into OpenFOAM v2312 for solving. The fluid geometry considers the entire test section, as available in the experiment. A simplification is made where the union cross, and two-outlet configuration of the experiment is not retained. Rather, the very top of the confining pipe serves as a single outlet for the fluid. The fluid mesh is unstructured hexahedral dominant, as generated using the automatic mesher of STAR-CCM+. The mesh has 15 prism layers on all walls with a stretching ratio of 1.25 on the confining pipe and 1.23 on the rod surface.

A total cell count of 8.486M cells was obtained. Note that the mesh was created intending for the with-flow FSI run, but was also used for the vibration study in still water. Given the low-speed oscillations of the rod in still water, the fluid mesh could have used few prism layers without any significant impact on the solution, but this was not done in the current study. The fluid geometry and mesh are presented in Figure 5.

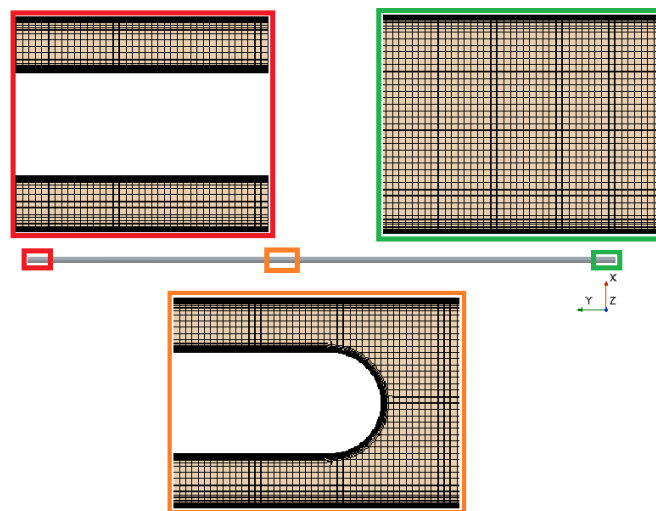


Figure 5. Fluid geometry and sectional view of the mesh

For the solid, the model reported in section 1.2.1 is reused. For the fluid, the pimpleFoam solver of OpenFOAM is used which discretizes the URANS equations in space using the PIMPLE algorithm, which is a combination of the classical PISO and SIMPLE algorithms. The laminar flow model is used. The fluid inlet is set to receive an inflow of 0 m/s, the outlet is set to zeroGradient, the confining pipe wall uses no-slip, while the rod surface uses movingWallVelocity for the velocity boundary conditions. For the pressure, the outlet is set to a fixed value of 0 Pa, the inlet and confining pipe wall are set to zeroGradient, while the rod surface is set to fixedFluxPressure to better account for the pressure gradient during motion. The fluid mesh morphing is handled by OpenFOAM using the exponential inverseDistance method, which keeps prism layer cells close to the FSI interface stiff.

The fluid and solid solvers are coupled through the preCICE library v3.2.0 for solving the FSI problem. Since the fluid and solid meshes are not conforming at the FSI interface, forces and displacements must be mapped from one grid to the other. To this end, radial basis functions implemented in preCICE are used to map the displacements from the solid to the fluid interface in a consistent way, and to map the forces from the fluid to the solid interface in a conservative way.

The parallel implicit coupling solver within preCICE is utilized to solve the FSI problem. In implicit approaches, the flow and structural problems are solved iteratively until convergence of the FSI problem is satisfied within one time step. Here, convergence means that the equality of displacements and stresses/forces is guaranteed up to a certain tolerance. In this study, the interface quasi-Newton technique with an approximation for the inverse of the Jacobian from a least-squares model (IQN-ILS) is used as an acceleration technique for convergence. This method was used as it proves to be more robust than Aitken's under-relaxation methods for strongly coupled problems.

The time-dependent acceleration applied to the rod is given by

$$a = \begin{cases} [0, g, -0.135] \text{ m/s}^2, & t < 0.1326\text{s} \\ [0, g, 0] \text{ m/s}^2, & t \geq 0.1326\text{s} \end{cases} \quad (9)$$

The time instant of 0.1326s, when the body force is removed, is carefully chosen to match the time instant the maximum rod displacement is achieved. The simulation starts with both domains at rest and is run for a simulation time of 5 s using a time step size of 2.6 ms. The consistent second-order backward difference scheme (BDF2) developed for moving grids is used to integrate the equations in time.

A number of cases are tested. These are listed in Table 4. For the fluid, since the mesh is not perfectly orthogonal (average non-orthogonality $\sim 2.5^\circ$, maximum non-orthogonality $\sim 52^\circ$), a more sophisticated scheme is available for discretizing for diffusive terms (Laplacian operator) and surface normal gradients via settings for `laplacianSchemes` and `snGradSchemes`, respectively, in OpenFOAM. The effect of these settings are tested, along with the effect of the rod Young's modulus and gravity. For the cases with gravity, the effect of the same on the structure and fluid was incorporated in Deal II. The self-weight of the Solid was captured by setting `g` in Deal II as per equation 9. As for the hydrostatic pressure force, this was also accounted for in Deal II analytically for each FSI interface cell based on its location in space. All cases use the same computational resources: 32 threads for Deal II, and 60 cores for OpenFOAM.

Table 4. Tested Cases

Case ID	Fluid Scheme	E (GPa)	g (m/s ²)
S1Y1G1	Uncorrected	193	0
S1Y2G1	Uncorrected	197.822646	0
S2Y1G1	Corrected	193	0
S2Y2G1	Corrected	197.822646	0
S2Y1G2	Corrected	193	-9.81
S2Y2G2	Corrected	197.822646	-9.81

The rod displacement time history for case S1Y1G1 is shown in Figure 6 along with the experimental results. As with the vibration in vacuum, the time history itself is not directly comparable to the experiment as the exact forcing and initial conditions are not known. However, the natural frequency and damping ratio extracted from the time history can be compared and is done so in Table 5 for all meshes, using the least error fit technique for

extraction. Unlike the vibration in vacuum, a small dependency was observed on the number of cycles used for the extraction. For consistency, 14 cycles are used for all cases.

The computational effort is similar across cases, except S1Y1G1. This comes from the distribution of resources: this case utilized 1 node for all 92 processes, while all other cases used 1 node each for each solver with communication via InfiniBand. For the other cases, the difference partly arises from slight variation from the unequal performance of computational nodes since the simulations were not all performed using the same nodes, but mainly because of the difference in the coupling iterations needed for each time step.

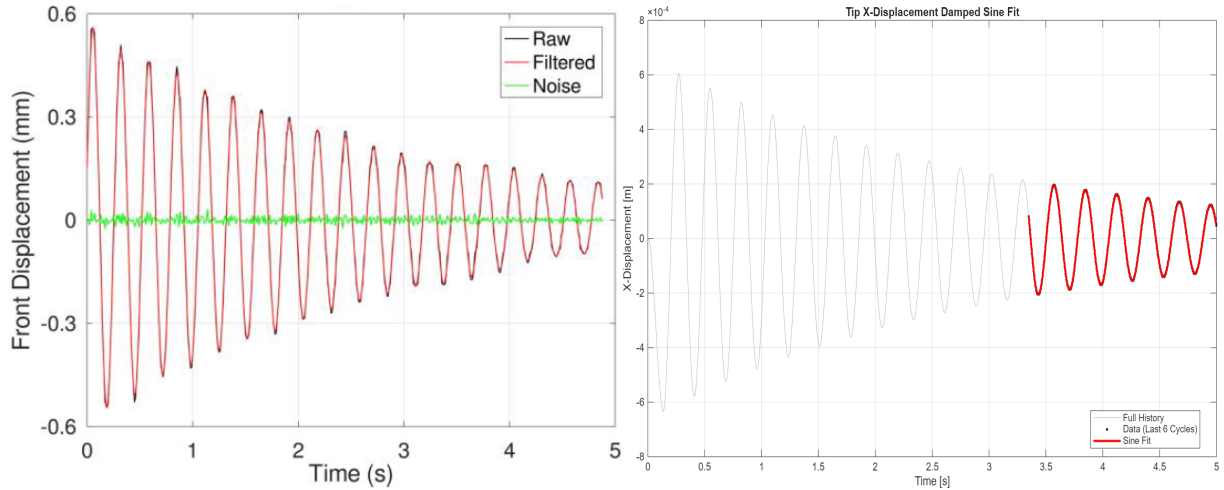


Figure 6. Time history of vibrations for the experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022) (left) and numerical results (S1Y1G1) (right)

Table 5. Obtained results: (experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022): $f_{n,w} = 3.72 \text{ Hz}$, $\zeta_{n,w} = 15 \times 10^{-3}$)

Case ID	$f_{n,w} \text{ (Hz)}$	Error (%)	$\zeta_{n,w} (\times 10^{-3})$	Error (%)	Wall Clock Time
S1Y1G1	3.63407	2.31	14.9703	0.20	3d 13h 33m 34s
S1Y2G1	3.67946	1.09	14.8894	0.74	1d 23h 50m 41s
S2Y1G1	3.63354	2.32	14.7506	1.66	2d 2h 58m 15s
S2Y2G1	3.67892	1.10	14.6702	2.20	2d 2h 28m 45s
S2Y1G2	3.65550	1.73	14.3460	4.36	2d 4h 19m 11s
S2Y2G2	3.70062	0.52	14.2714	4.86	2d 1h 24m 56s

Considering the obtained results, all cases yield good results with errors less than 5%, however, some observations are made on the effect of the tested parameters. Considering the G1 cases, the effect of switching to the more sophisticated corrected scheme yields barely any change in the frequency and a slight reduction in damping ratio, indicating an impact on the viscous drag prediction on the vibration of the rod without any change on the normal pressure forces.

The effect of increasing the Young's modulus or including gravity has a similar effect of increasing the natural frequency and decreasing the damping ratio, albeit to different degrees. This stems from increasing the apparent stiffness (k_{eff}) of the rod. Since $f_{n,w} \propto$

$\sqrt{k_{eff}}$ and $\zeta_{n,w} \propto \frac{1}{\sqrt{k_{eff}}}$, the outcome is as expected. While it is straightforward to understand the increased stiffness when the Young's modulus is increased, the effect of gravity is indirect. When considering gravity, the rod is subjected to its own self-weight, and an opposing buoyancy force, which are strictly oriented vertically. A third contribution kicks in, which is the follower-load effect, a deformation-dependent surface force owing to the hydrostatic pressure always acting normal to the surface. As the rod bends, the surface orientation rotates and the pressure force vector tilts, creating a net force distribution that changes depending on how far the rod has bent. This net force always tries to return the rod to its original rest position, making the rod effectively stiffer.

1.2.3 Flow-only simulation with AniPFM at annular Re=21.2k

As mentioned in section 1.2, it was originally planned to continue with the finalized solid and fluid models for the validation study of AniPFM against experimental data at annular Re=21.2k by performing two-way coupled FSI simulations. However, applying AniPFM to this scenario resulted in unexpected observations which triggered investigations. Some improvements and modifications were made and listed in section 1.1.1, but some open concerns remained until the end of the duration of the project, hampering the intended calculation. The key feature that exposed the issue was the type of geometry at hand, in particular, the change in flow cross-section from a pipe to annulus. A few of the consequences are presented in section 1.2.3.1. Owing to these concerns, a flow only simulation was first performed with AniPFM. The setup and obtained results are described in section 1.2.3.2.

1.2.3.1 Challenges specific to the cantilever rod problem

As detailed in (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024), AniPFM involves first establishing the energy spread assuming a spectrum based on the Shur method. This involves setting up k_{start} , k_e , k_{cut} , and k_η at each cell in the domain. Since there is a clear difference in possible eddy sizes in the domain, the spectra significantly shifts moving close to a wall, or downstream after the rod tip. This is shown in Figure 7.

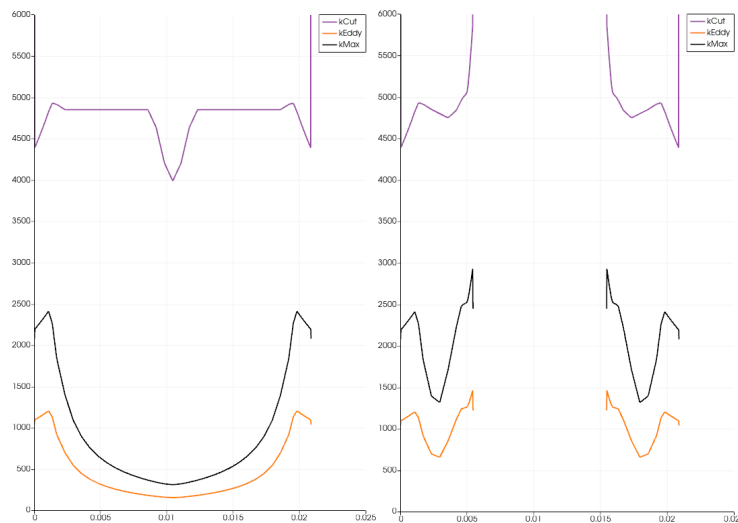


Figure 7. Horizontal distribution of k_{start} , k_e , and k_{cut} in the pipe section ($y=-1.5$ m, left) and annulus section ($y=-0.5$ m, right). Note: $y=0$ m is at the outlet.

Based on these quantities, the coefficient for each mode, $q_n = \frac{E(k_n)\Delta k_n}{\sum_n E(k_n)\Delta k_n}$ is computed. Two issues arise here. The first is achieving a value of $\sum_n q_n = 1$, which is necessary for the correct scaling of the non-dimensional velocity fluctuations to dimensional velocity fluctuations by the Cholesky method. The integral/summation term in the denominator needs to be performed using the same wavenumber samples as those used to sample the numerator. Furthermore, the sampling for the fraction must be done on the same spectrum (either cut at k_{cut} or uncut proceeding until k_η), otherwise the turbulent kinetic energy (TKE) will not be perfectly respected post the Cholesky operation.

The second issue lies in the fact that we now have a spatially varying q_n field. This implies that it is no longer sufficient for the direction vector of non-dimensional velocity fluctuation, σ_n , to be perpendicular to the wave vector, k_n , in order for the generated non-dimensional velocity fluctuation to be divergence free. There were attempts made to construct a different formulation for the non-dimensional velocity fluctuation: as a curl of a potential field, to utilize the mathematical property of the divergence of a curl of a potential field in the continuous world would be 0. However, in the discrete world of the mesh, this was difficult to achieve. Furthermore, even if it was achieved, the Cholesky operation moving to the dimensional velocity fluctuations would again reintroduce a spatial variation and thereby a non-divergence free solution.

This divergence free property is crucial when dealing with the pressure Poisson equation governing the pressure fluctuations needed for the FSI calculations. The governing equation in the forms presented below, assumes this property during its derivation and in the conversions between the conservative (equation 10) and non-conservative forms (equation 11).

$$\frac{1}{\rho} \nabla^2 p' = \nabla \cdot \nabla \cdot (\mathbf{R} - [\mathbf{U} \otimes \mathbf{u}' + \mathbf{u}' \otimes \mathbf{U}] - \mathbf{u}' \otimes \mathbf{u}') \quad (10)$$

$$\frac{1}{\rho} \nabla^2 p' = \nabla \cdot \nabla \cdot (\mathbf{R}) - 2\nabla \mathbf{U} : (\nabla \mathbf{u}')^T - \mathbf{u}' : (\nabla \mathbf{u}')^T \quad (11)$$

A related outcome of this is that the constructed $\mathbf{u}'$, though in a time-averaged sense reflects the $\mathbf{R}$ tensor ($\overline{\mathbf{u}' \otimes \mathbf{u}'} = \mathbf{R}$) via the Cholesky operation, does not perfectly balance the $\mathbf{R}$ in the source term every time instant.

A technique already implemented in AniPFM to account for this is the source adjustment, where, after the RHS of the governing equation is constructed, it is centered around its volume average before the equation is solved. For the test cases reported in (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024), the flow cross section was always constant, thus if an error was made in the source term, the volume average centering made sure that the p' solution did not get any spurious parabolas.

In the current scenario, where the geometry changes from a pipe to an annulus flow, the volume average centering could not fully correct the asymmetric offsets in the pipe section and annulus section. Thus, a spurious double parabola in the solution exists at each time step under which the actual small perturbations necessary for enhancing the FSI

performance are buried. This is shown in Figure 8 for a sample test run where the solution is sampled at a radial location of 7.7175 mm spanning the whole distance of 2.145 m between the inlet and outlet. While the double parabola bounds appear small, it is to be realized that these are in kinematic units. Thus, the actual load in Pascal is about 1000 times this value.

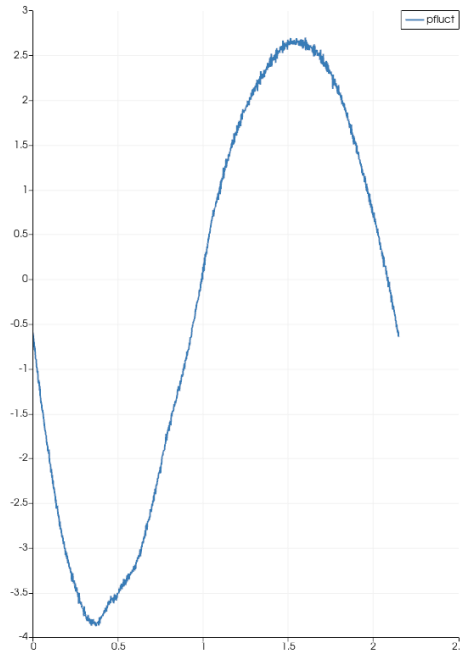


Figure 8. Sample p' obtained during testing

1.2.3.2 Flow-only case setup and results

The challenges reported in section 1.2.3.1 were attempted to be resolved within the timeframe of the project, however, they could not be resolved. Investigations on this will continue beyond the scope of this project. This also prevented the two-way FSI simulations from being tested. As an alternative, one-way FSI simulations were performed with the flow-only comparisons made in this section.

The fluid model and mesh reported in section 1.2.2 was used as the starting point. The boundary conditions were first changed. For the velocity, the inlet was prescribed with mapped velocity boundary condition from a location 0.5 m downstream of the inlet, with a fixed average setting of $[0, 1.509666389, 0]$ m/s corresponding to annular $Re=21.2k$. The rod surface was set to noSlip since the problem was now a pure CFD run. For the pressure, the fixedFluxPressure was also changed to zeroGradient for the same reason. Next, the laminar flow model was swapped with the two-equation $k - \omega$ SST model. Appropriate boundary conditions were also set for the model specific variables: k , ω , and v_t assuming a turbulence intensity of 5%. Gravity was excluded and the uncorrected schemes were used.

For AniPFM, 256 modes were used to compute the Fourier contributions to the turbulent spectrum and thereby velocity fluctuations. The user defined parameter (l_{user} , equation 2) was set to the confining pipe diameter 20.86 mm, while the turbulence convection speed was set to $[0, 1.96129, 0]$ m/s.

Before activating AniPFM, a RANS simulation was first run using the SIMPLEC pressure-velocity coupling scheme by employing the simpleFOAM solver as available in OpenFOAM, for 5000 iterations before switching to pimpleFOAM. Then the transient URANS simulation was run for 4.475 s using a time step size of 0.05 ms to respect $CFL < 1$. This established the steady state URANS solution. Since this solution would not change with the inclusion of AniPFM, the URANS base solution was frozen and only AniPFM was allowed to run.

Once AniPFM was activated, the simulation was continued from 4.475 s until 10.22325 s using a time step size of 0.25 ms with data collection for time averaging beginning at 4.476 s until the end of the calculation. The following data was extracted for comparison: horizontal profiles 10D ahead of rod tip to confirm the inflow conditions, qualitative contour plots in the vicinity of the rod tip, horizontal profiles 15.63786 mm, 3.786006 mm and 0.493836 mm upstream of the rod tip, and similar horizontal profiles 0.164619 mm, 6.74898 mm, and 29.135799 mm downstream of the rod tip. The data is time-averaged and can be considered so for the velocity fluctuations. Since the URANS solution was frozen, the time-averaging yields the same instantaneous frozen field for velocity. Note that comparisons are made to experimental data but also to high fidelity simulation data from EDF (WR-LES: FSI) and PSU (DNS: CFD) where possible, details of which are available in (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025).

Before diving into the validation, a first look is taken at the verification of the inflow profile at a significant distance upstream of the rod. There are experimental data available at 10D upstream of the rod tip which can be used for comparison. The LES data from EDF comes from a pre-cursor simulation which implies perfectly developed pipe flow without any influence of the downstream rod. The available data is plotted and displayed in Figure 9, Figure 10, and Figure 11 for the non-dimensional axial velocity, non-dimensional radial velocity and non-dimensional Reynolds stress component $u'v'$ respectively.

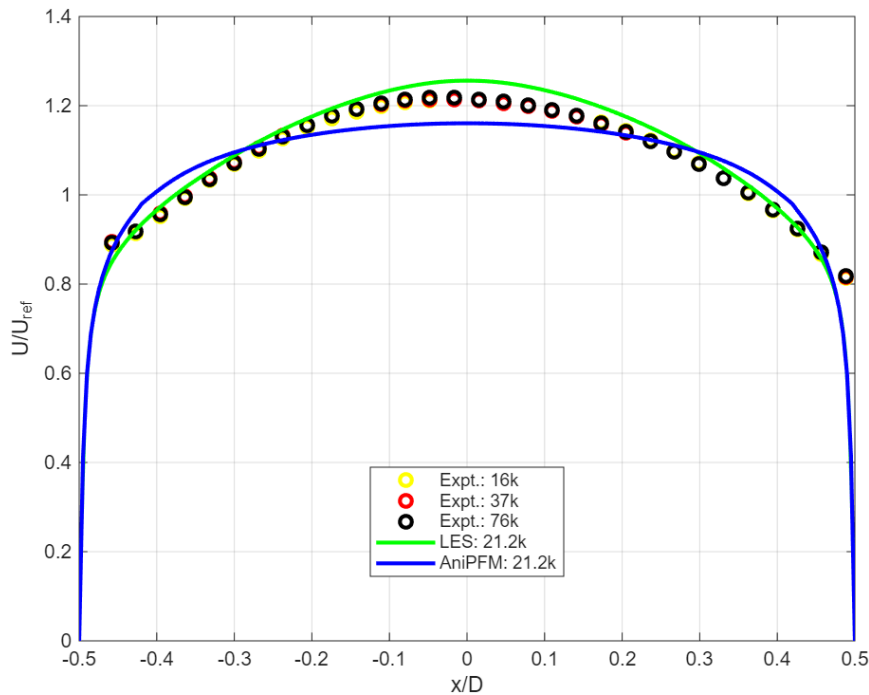


Figure 9. Time-averaged non-dimensional axial velocity 10D ahead of rod tip

Focusing on Figure 9, what is seen from the experimental results is that the non-dimensional velocity profile is independent of the inflow velocity. It is thus possible to use it to compare against numerical results at $Re=21.2k$. The maximum disagreement between the current and LES solutions is $\sim 8\%$, at the center of the confining pipe where the maximum non-dimensional velocity is captured. Despite this difference in the solution, it can still be considered to be a good match against the experiment.

From Figure 10, it is observed that the experimental results appear to show some dependency on the inflow Re . However, considering the scale of the results and the uncertainty in the U_{ref} , the experimental results can still be considered to suggest Reynolds number independence and can be used for the comparison to numerical results. The trend of the experiment reveals the fluid to move to the left (negative values) in the left half while move to the right (positive values) in the right half, with some scattered data close to the confining pipe walls.

The numerical results all underpredict the radial component of velocity compared to the experiment. Observing the zoomed plot, the current results predicts an even smaller magnitude (3 orders lower) than the LES. The LES data predicts mostly a positive radial velocity except for a distance of $0.1D$ at the left pipe wall, which can be explained from the fact that the solution data is taken from the precursor domain where the situation is turbulent pipe flow with no influence of the cantilever rod. The current results indicate that no significant effect of the rod tip is felt at this location. Since the velocity at the inlet is mapped from a location 0.5 m downstream, there may be an error creeping in as a feedback that prevents the radial component of velocity from reflecting reality $10D$ ahead of the rod tip.

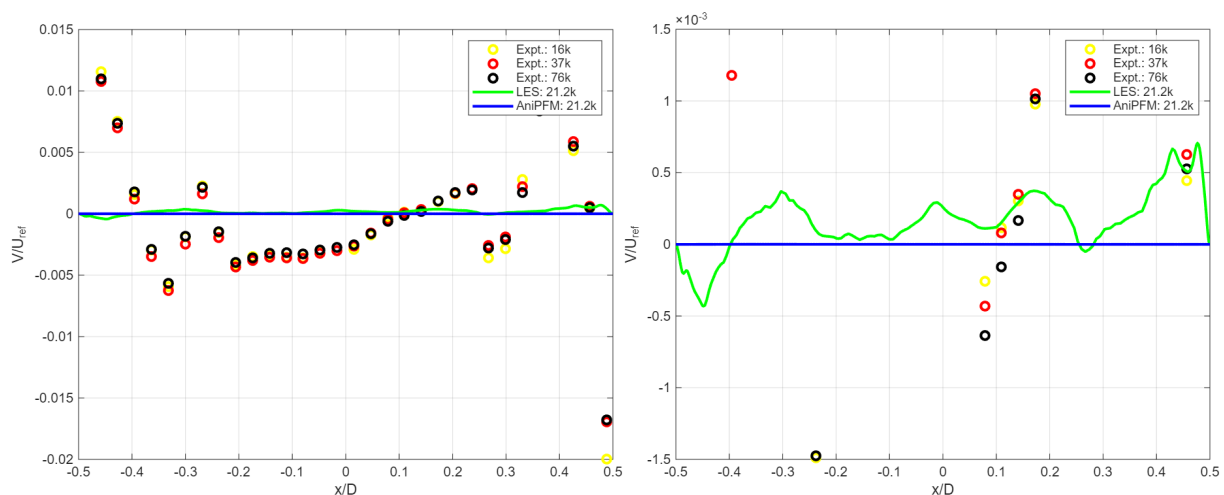


Figure 10. Mean non-dimensional radial velocity 10D ahead of rod tip (right: Zoomed in)

Figure 11 shows the second order statistics. For reasons similar to that presented for the non-dimensional radial velocity, the experimental solution can be considered to be effectively Re independent. The LES and AniPFM solutions predict a trend similar to the experiment but overpredict the magnitude compared to the experiment. The lower experimental values may be a result of the spatial filtering done on the interrogation areas

which effectively averages out the small, high-frequency turbulent structures, and potentially also due to noise/bias, which could arise from velocity bias (peak locking) and poor time resolution (aliasing) (Lavoie, Avallone, Gregorio, Romano, & Antonia, 2007) (Sarno, et al., 2019). Comparing the AniPFM solution to LES, it is observed to show magnitudes larger than LES at the peaks. Furthermore, the solution though time-averaged, appears slightly asymmetric which indicates that, most likely, the time window considered for time averaging is short and the simulation needs to be run longer.

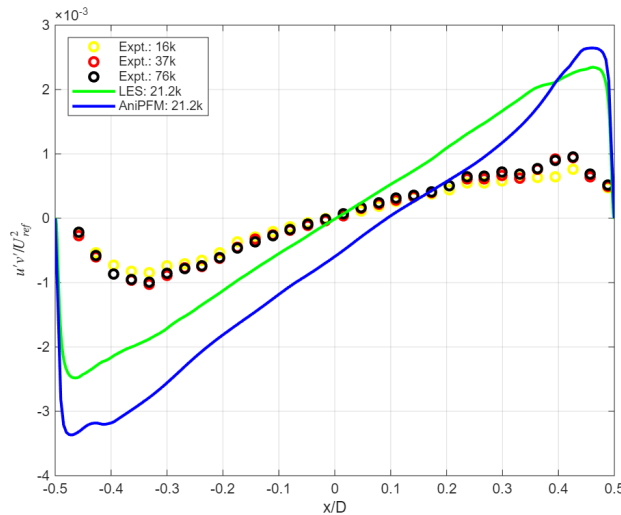


Figure 11. Time-averaged non-dimensional Reynolds stress component ($u'v'$) 10D ahead of rod tip

The time-averaged non-dimensional velocity contour plots are shown in Figure 12 and Figure 13. From these results, the flow in the vicinity of the rod tip is revealed. For this upward flow case, the flow upstream of the rod starts to change from a typical fully developed pipe flow owing to the obstruction in flow path caused by the vibrating rod. The flow smoothly curves around the tip of the rod and proceeds upwards through the annular gap between the rod and the confining pipe. Due to a decrease in available cross-section, the flow is seen to speed up in this region. A stagnation point at the rod tip is also observed. The current results are found to be in good agreement with the experimental and high fidelity results. It must be noted that these figures also revealed that the DNS data did not collect sufficient data for a converged time averaged solution (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025).

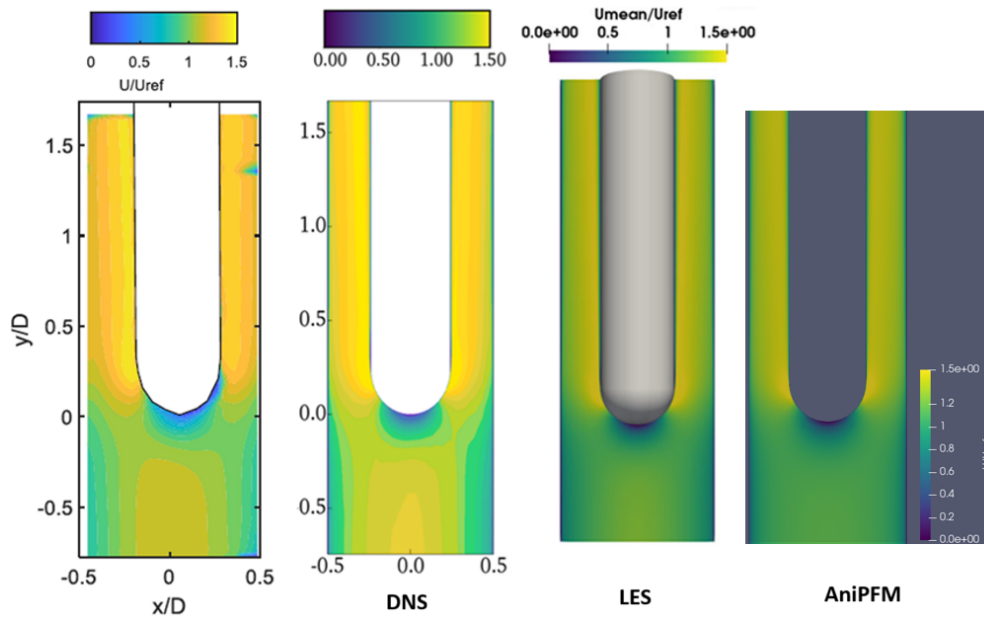


Figure 12. Comparison of base URANS $k-\omega$ SST results with experimental (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022), and high fidelity (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025) results for time-averaged non-dimensional axial component of velocity

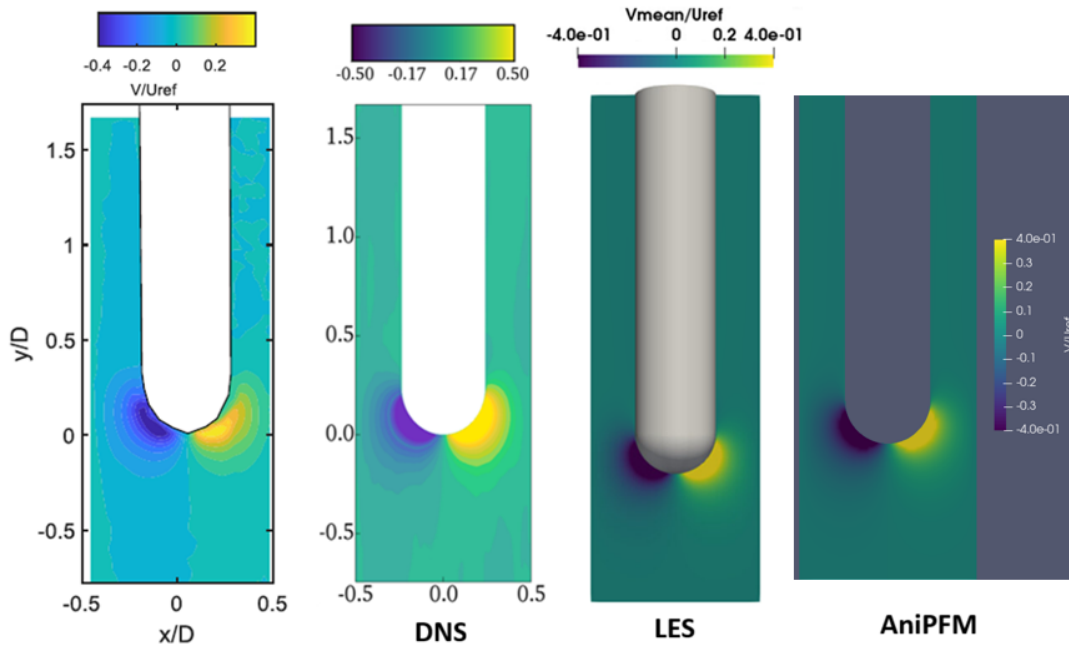


Figure 13. Comparison of base URANS $k-\omega$ SST results with experimental (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022), and high fidelity (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025) results for time-averaged non-dimensional radial component of velocity

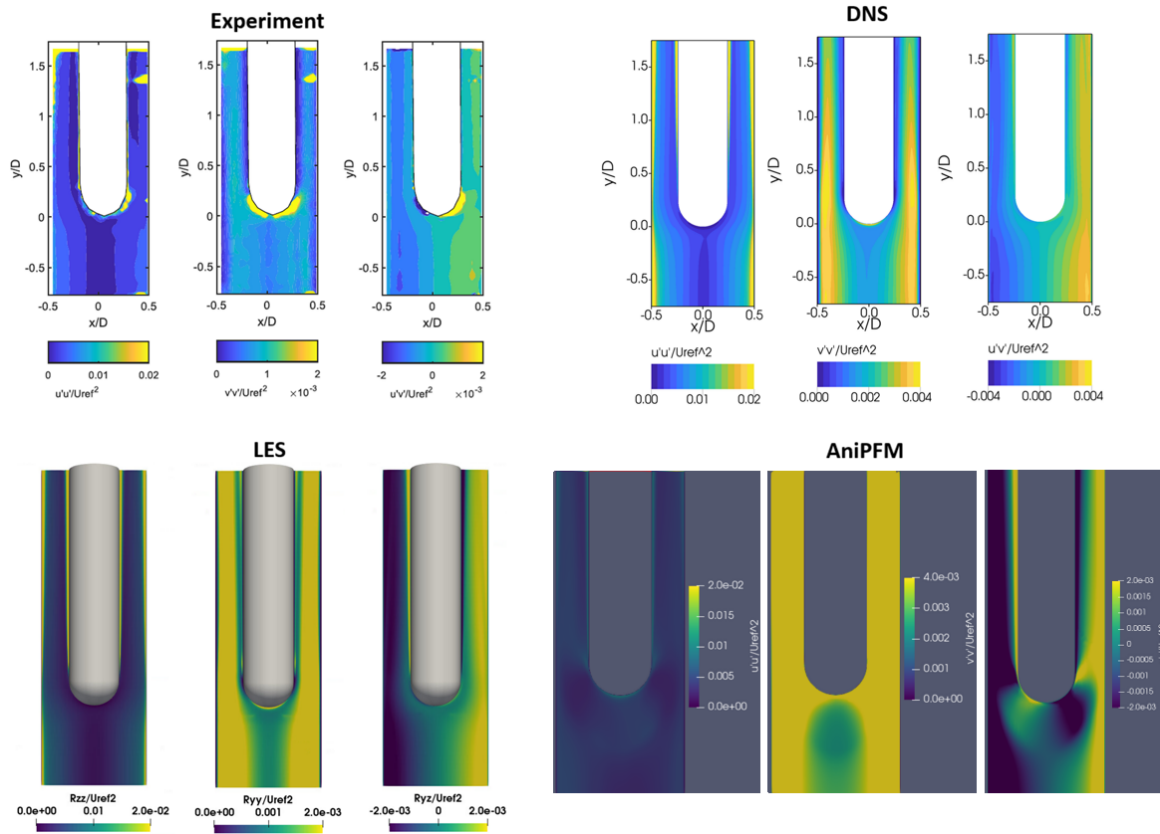


Figure 14. Comparison of AnIPFM results with experimental (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022), and high fidelity (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025) results for non-dimensionalized stress components

The time-averaged non-dimensional stress component contours are shown in Figure 14. The AnIPFM results are constructed from the synthetic velocity fluctuations rather than the base Reynolds Stress Tensor available from URANS for a more meaningful comparison, though they were found to be qualitatively the same, which is expected owing to the Cholesky transformation of scaling the non-dimensional fluctuations to respect this tensor. Clear differences are observed in the obtained solution compared to the experiment and high fidelity data. This is more of an outcome of the choice of the underlying URANS turbulence model rather than the performance of the AnIPFM, which can only be as good as this initial choice. The two-equation $k - \omega$ SST model utilizes the Boussinesq approximation and still creates an isotropic stress tensor. Since the Wilcox correction for introducing anisotropy within AnIPFM could not be utilized, the outcome is in line with expectations. Improvements are envisioned by implementing and utilizing the same, or switching to a more sophisticated URANS model such as RSM which captures anisotropy.

Several radial profiles of the time-averaged axial and transverse velocity at the PIV measurement locations are provided in Figure 15 to Figure 20. The PIV data inside of the beam are discarded owing to metallic reflections. The experimental results for the axial velocity component show that as the fluid approaches the rod tip, the standard pipe flow profile transitions to that expected for annular flow. However, the experimental profiles also show clear asymmetry which was reasoned by the potential offset in the vibrating rod tip from the axis of the confining pipe, as was also confirmed in a separate rod vibration analysis reported in (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025).

Concerning the experimental results for the radial velocity component, at a large distance (15.6 mm) before the rod tip, the magnitudes are revealed to be quite low similar to that observed for the location 10D (Figure 10) before the rod tip as well as some scatter in the data close to the confining pipe wall. As the rod tip is approached, this component of velocity understandably increases indicating the flow accommodating itself in the annular cross-sectional area due to the obstruction of the rod. As with the axial velocity component, asymmetry in the profiles are observed which could be reasoned with the rod tip offset. In particular, for dissimilarity of the left and right profiles of Figure 19 and Figure 20, this is reasoned to be from a potential measurement error.

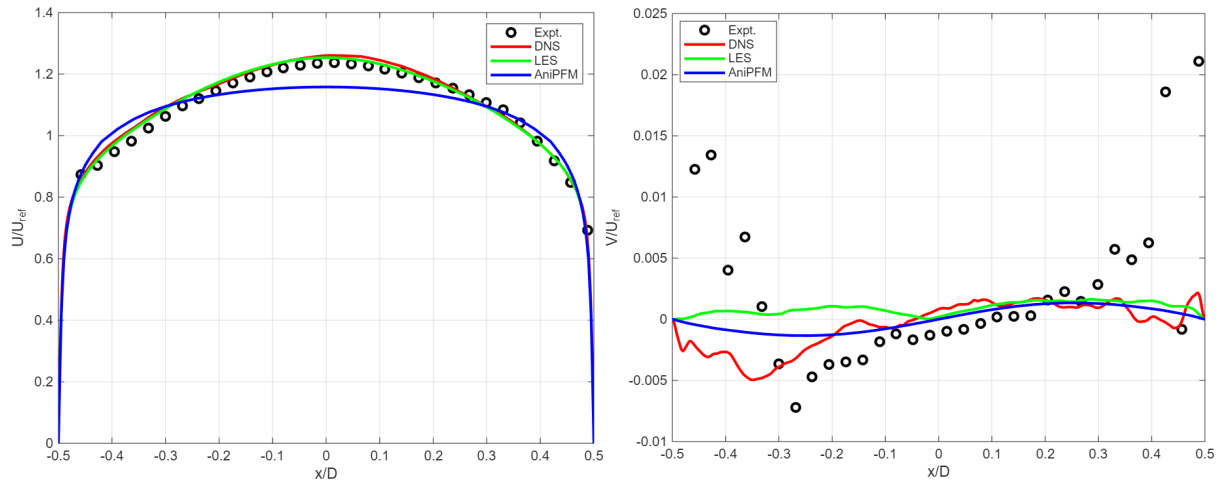


Figure 15. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 15.63786 mm upstream of the rod tip

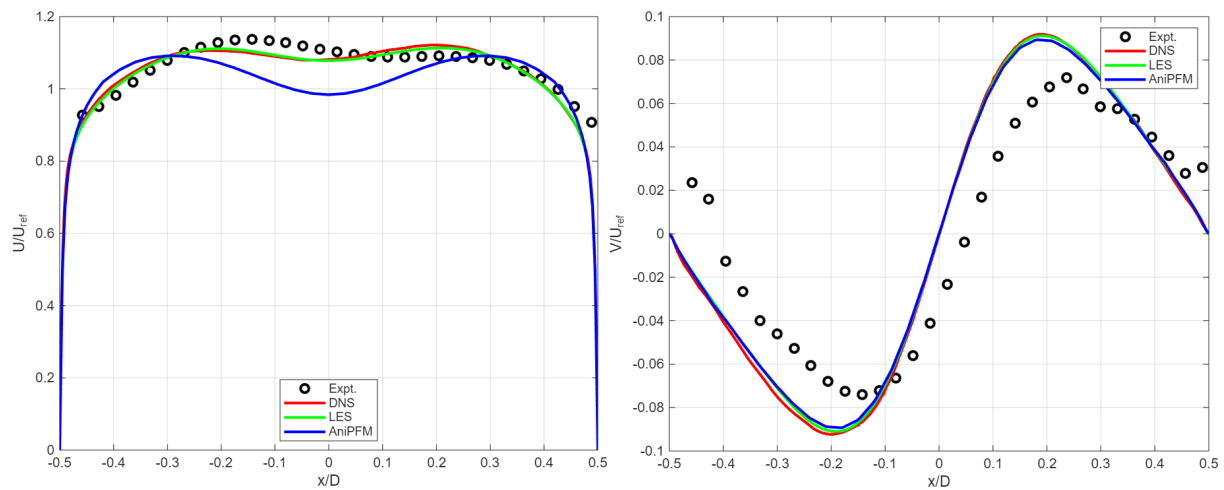


Figure 16. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 3.786006 mm upstream of the rod tip

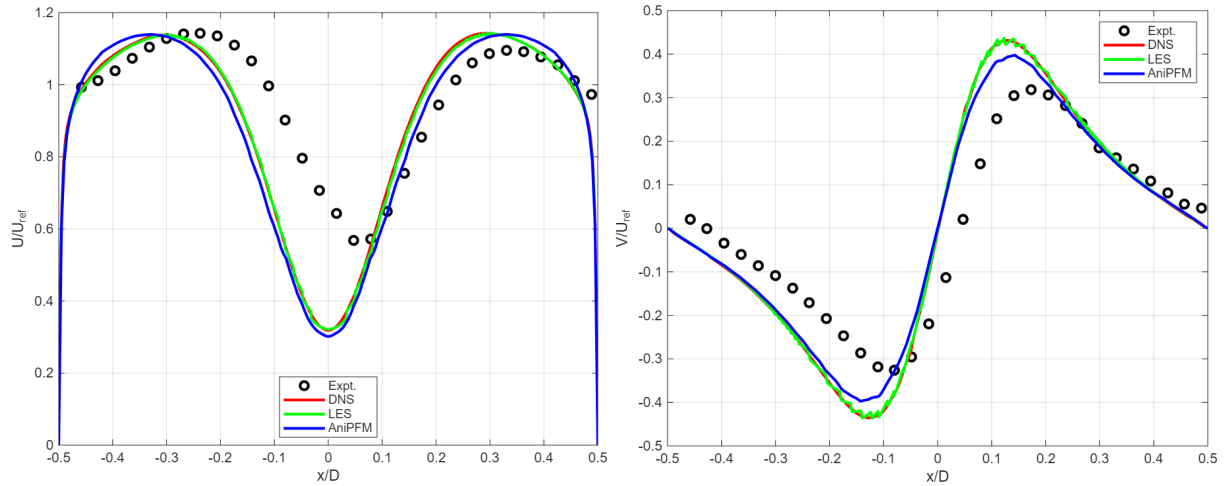


Figure 17. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 0.493836 mm upstream of the rod tip

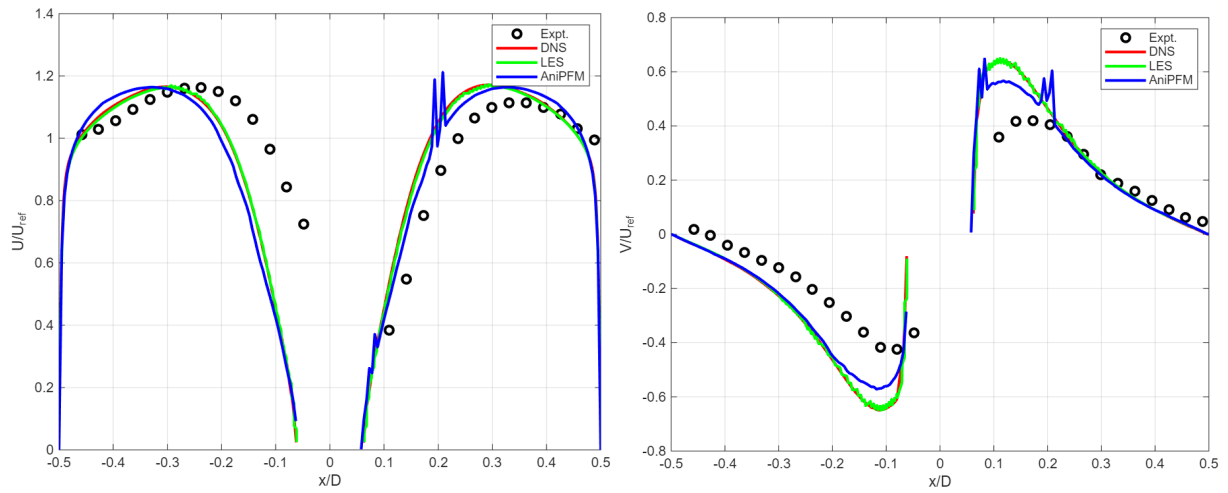


Figure 18. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 0.164619 mm downstream of the rod tip

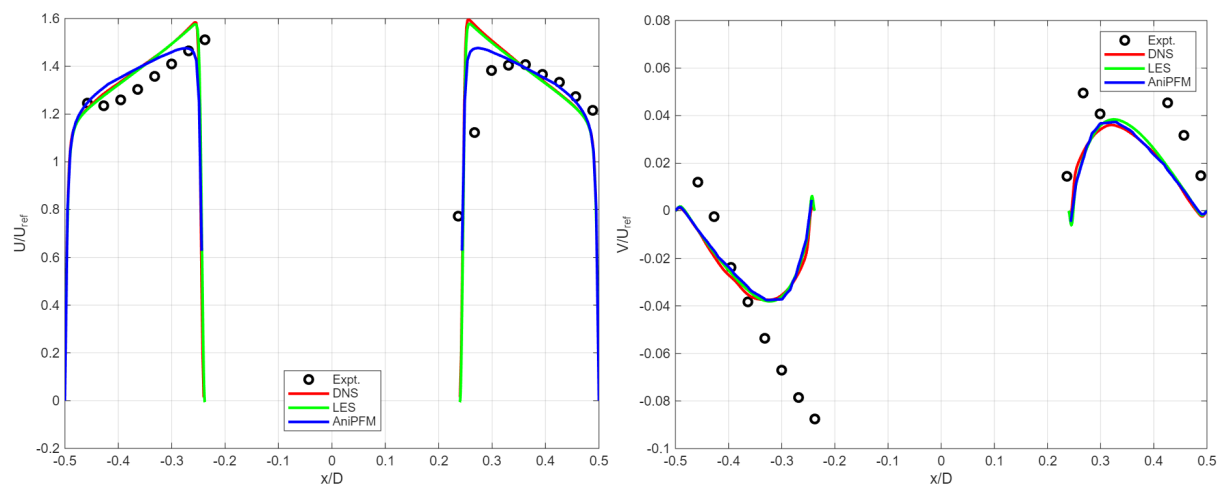


Figure 19. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 6.74898 mm downstream of the rod tip

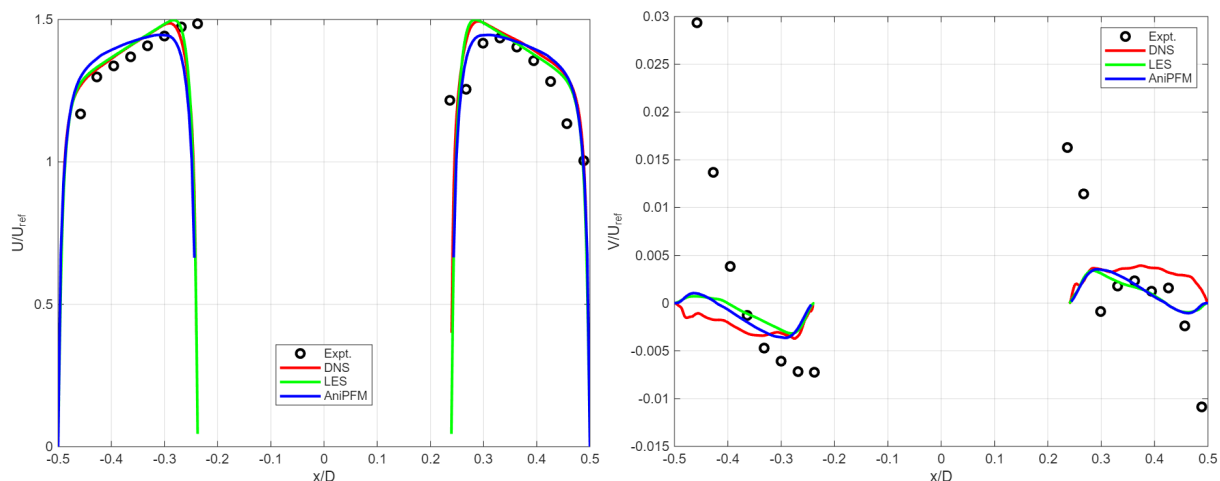


Figure 20. Time-averaged non-dimensional axial (left) and transverse (right) velocity profiles 29.135799 mm downstream of the rod tip

Concerning the numerical results, all numerical results predict a similar evolution of velocity component profiles moving towards the rod. Concerning the flow profiles, the effect of the choice of turbulence modelling appears to dictate its prediction.

The attention is first brought to the axial component of velocity. AniPFM (or more precisely, base URANS $k - \omega$ SST) is found to closely predict results similar to the high fidelity simulations with a few notable differences. As the flow transitions from pipe flow to annular flow, the maximum axial velocity at the center of the pipe is underpredicted compared to experimental and high fidelity data at a significant distance ahead of the rod tip (15.6 mm [Figure 15] and 3.8 mm [Figure 16] upstream). In the close vicinity of the rod tip (0.49 mm upstream [Figure 17] and 0.16 mm downstream [Figure 18]), the current results are in very good agreement with high fidelity data while not to the same extent with experimental results which appear to be impacted by the offset. Moving further downstream (6.7 mm [Figure 19] and 29.1 mm [Figure 20]), the match with high fidelity data and experiment remains good, though underpredicting the near-rod peaks predicted by high fidelity results.

Observing the radial component of velocity, at a distance of 15.6 mm upstream of rod tip (Figure 15), the high fidelity results appear wiggly with DNS results showing positive values in the right half (flow moving right) and negative values in the left half (flow moving left), and LES results show small but positive values throughout. The current results show a smooth S-shaped profile with magnitudes in the range of the experimental and high fidelity results. This also indicates that the effect is rod obstruction is numerically felt here but was not at a very large distance of 10D ahead of rod tip (Figure 10).

At the location 3.8 mm upstream of rod tip (Figure 16), the current results match very well with high fidelity data. The agreement remains good in the vicinity of the rod tip (0.49 mm upstream [Figure 17] and 0.16 mm downstream [Figure 18]), though the current results underpredict the peak magnitudes compared to high fidelity data. Moving further downstream (6.7 mm [Figure 19] and 29.1 mm [Figure 20]), the match with high fidelity results is very good, with the exception of DNS data at 29.1 mm downstream which was observed to be wiggly.

The profiles concerning the time averaged second order statistics ($u'v'$ and turbulent kinetic energy, TKE or k) are presented in Figure 21 through Figure 26. Considering the experimental PIV results for the $u'v'$ data, they are observed to measure general trends, however owing to the aforementioned reasons, their magnitudes are hypothesized to be undermined with respect to reality. The underestimation is likely most severe in the regions of highest energy and smallest scales, and least severe where turbulent scales are larger and more diffused. This was also confirmed by observing the high fidelity numerical results which provide similar profile trends but much higher magnitudes.

Considering the secondary statistics results, the AniPFM results do not agree so well on a plot-to-plot basis, however, it does evolve similar to the high fidelity data with the final profile 29.1 mm downstream of the rod tip agreeing decently well. Two other notable observations are slight asymmetry in the current solution, which could come from insufficient time averaging, and underpredictions in TKE peaks close to confining pipe walls, overprediction of the same in the center and rod wall (in the vicinity of the rod tip).

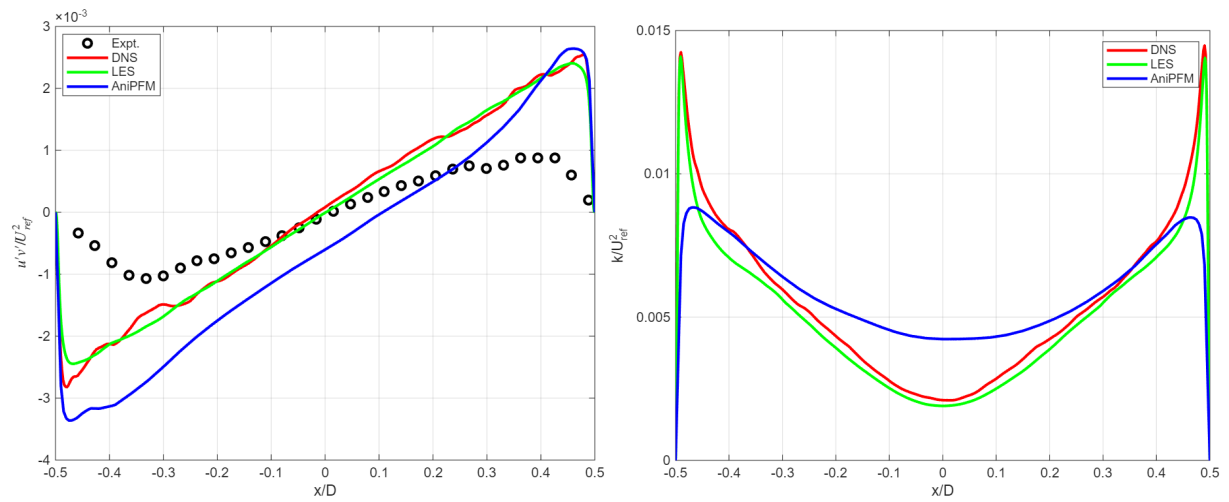


Figure 21. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 15.63786 mm upstream of the rod tip

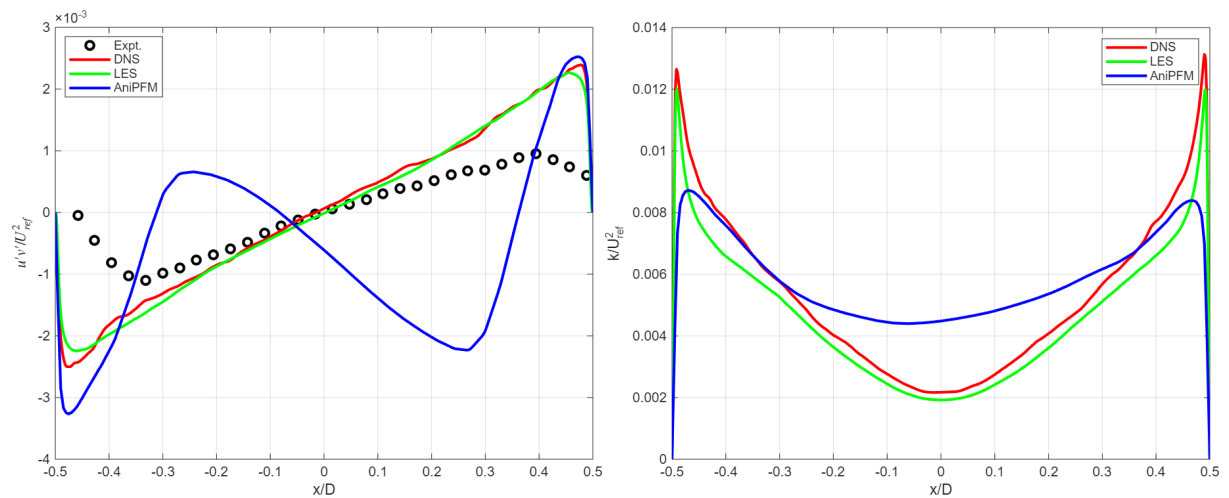


Figure 22. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 3.786006 mm upstream of the rod tip

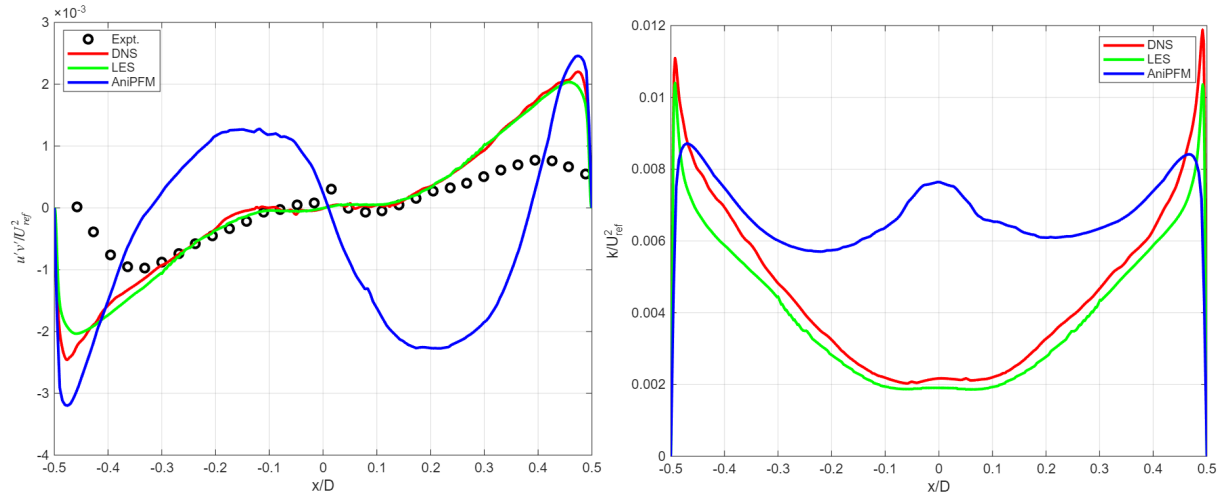


Figure 23. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 0.493836 mm upstream of the rod tip

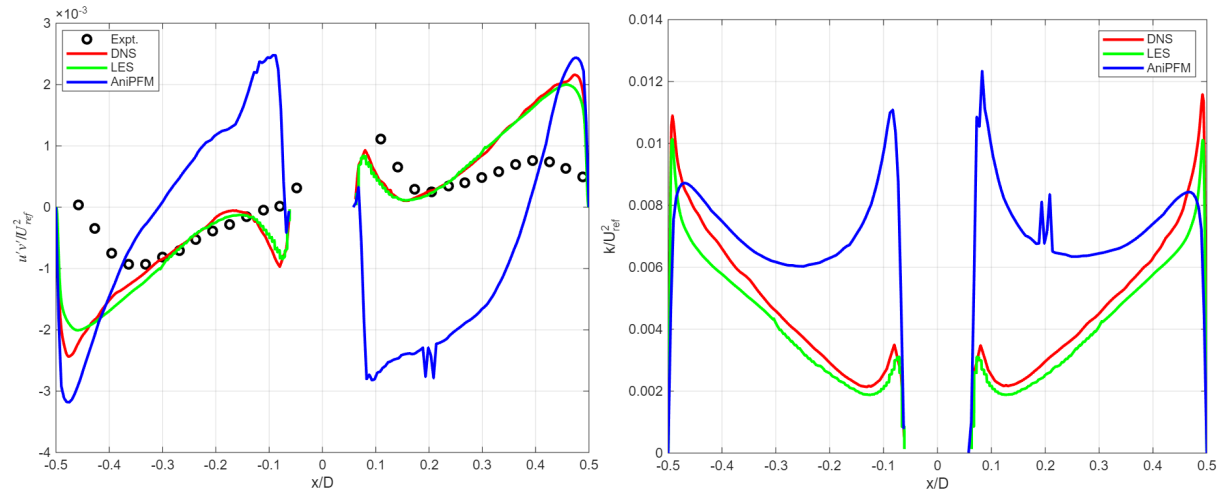


Figure 24. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 0.164619 mm downstream of the rod tip

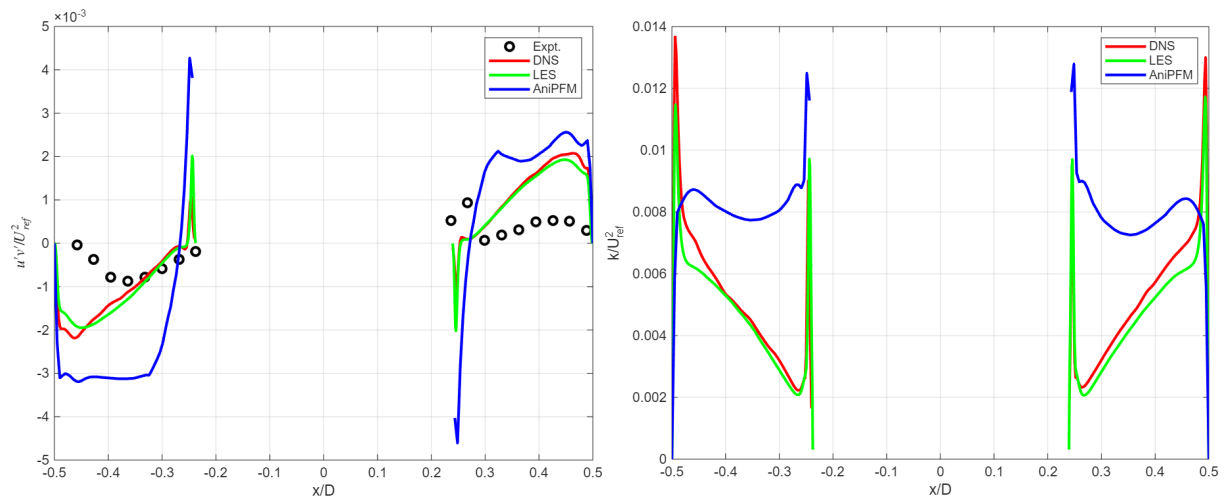


Figure 25. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 6.74898 mm downstream of the rod tip

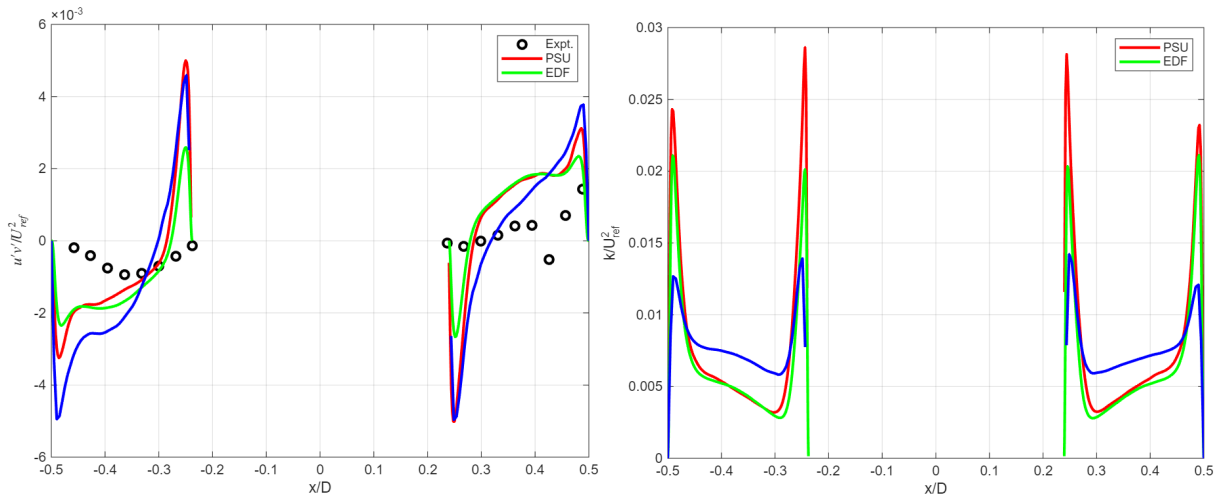


Figure 26. Time-averaged non-dimensional Reynolds stress component ($u'v'$) (left) and turbulent kinetic energy (right) profiles 29.135799 mm downstream of the rod tip

The latter observation could be interpreted as a shortcoming of the AniPFM procedure itself, but, as was also mentioned previously, the AniPFM relies on the Cholesky transformation for anisotropy, which scales non-dimensional velocity fluctuations based on the Reynolds stress tensor (thereby TKE) of the underlying URANS turbulence model. Thus, it could also be interpreted as the lack of anisotropy introduced either by an analytical approach (Wilcox correction) or by virtue of an anisotropic URANS turbulence model such as RSM. Improvements are thus anticipated once the switch is made and further development is carried out in the future to address the challenges reported in section 1.2.3.1.

1.2.4 1-way FSI vibration study with AniPFM at annular $Re=21.2k$

With the flow-only comparison complete, a 1-way FSI vibration study was performed. The same flow simulation reported in section 1.2.3.2 is considered. The flow-only simulation was run longer until an end time of 13 s, and was used to sample the p' over the rod surface.

Only the sampled p' was used to drive the structure in the simulation. The viscous contribution was thus not accounted for in this run. For the structural model, the ROM prepared by TU Delft as reported in section 4.2 was used. The interpolation of p' to the structure was done in a consistent way. A total vibration time history of 8.525 s (4.475 to 13 s) is thus available.

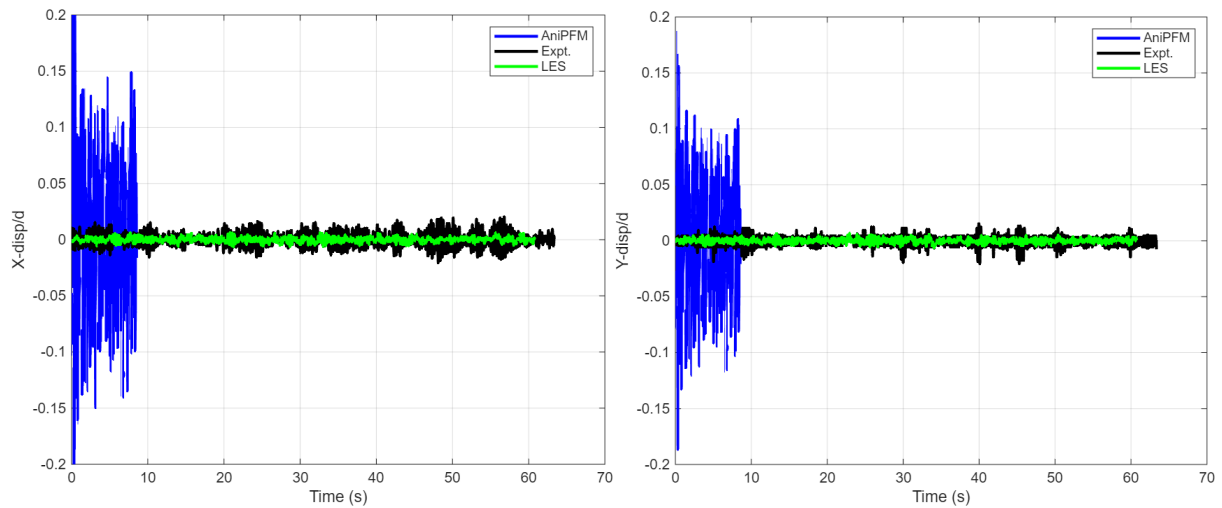


Figure 27. Time history of the non-dimensional rod tip front (left) and side (right) displacements

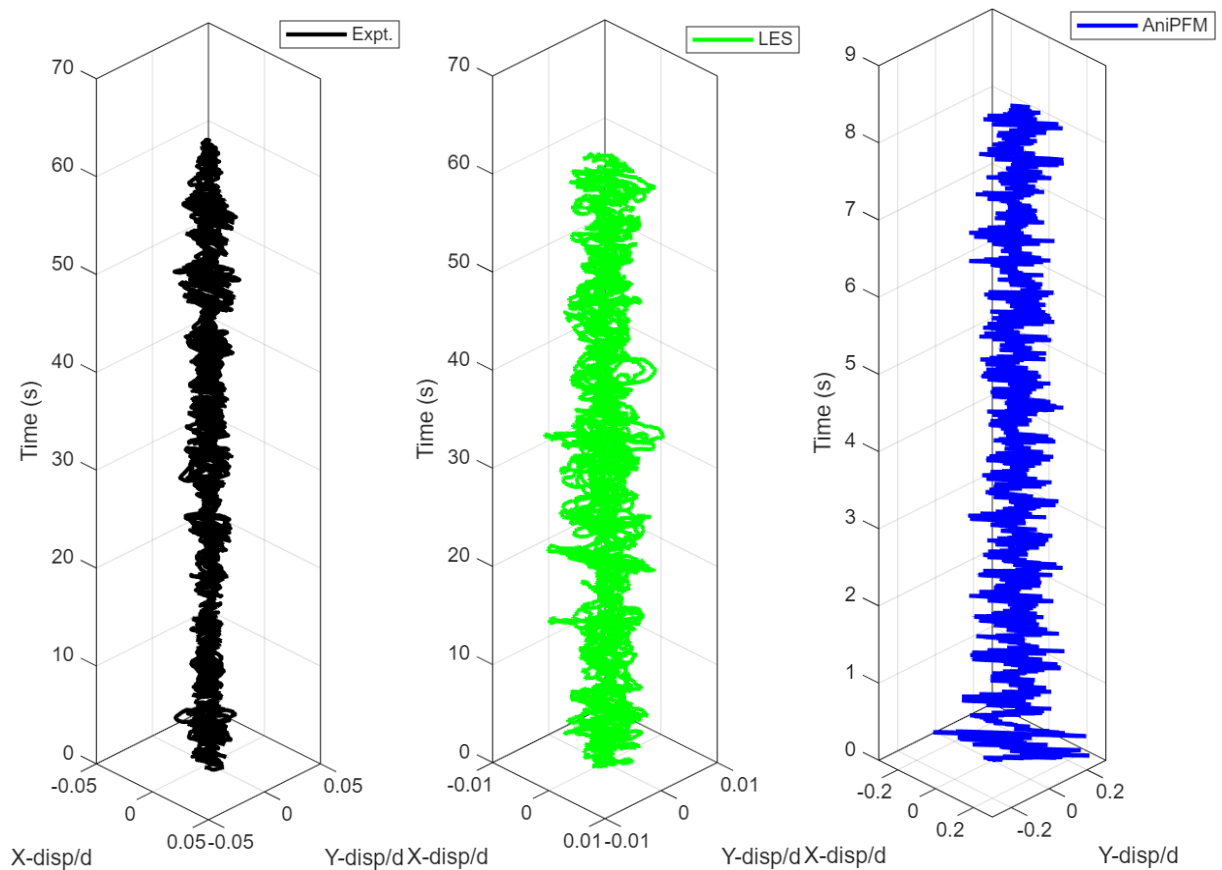


Figure 28. 3D time history of non-dimensional rod tip displacements

The time histories obtained from the simulation is compared against the experiment and LES in Figure 27. The displacements are non-dimensionalized using the rod diameter. Both the experimental and numerical results were centered about the 0 axis by removing the mean of the data excluding the initial transient which was about 1 s for the current result. A first glance already reveals much higher vibration amplitude predictions by the current 1-way FSI run with AniPFM compared to experimental and LES data.

The obtained time histories are also presented as 3D plots in Figure 28 without removing the mean for the numerical results. Looking at the same data in polar displacement plots, as shown in Figure 29, reveals strong directionality of the current results instead of the expected chaotic pattern seen in the experimental and LES results. This is unexpected as the simulated domain is axisymmetric, and indicates a biased excitation of modes.

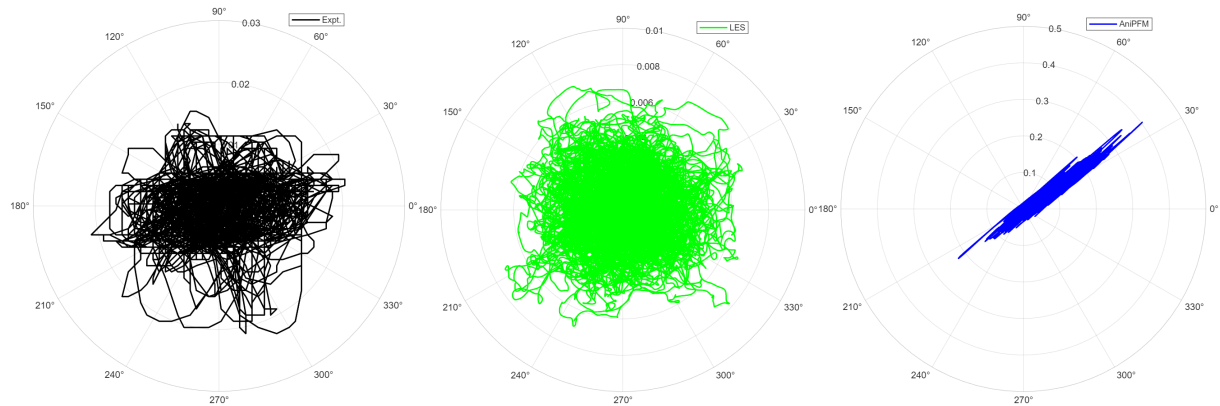


Figure 29. 2D polar time history of non-dimensional rod tip displacements

A clear offset is visible for the current results but was also the case in the experiment, the latter coming from a potential installation misalignment. This offset is tabulated in Table 6 for all considered data. The LES offsets are much smaller in comparison to the experiment and are considered to be a numerical artifact. The current results unexpectedly show offsets in the same order of magnitude as the experiment. This is hypothesized to come from a systematic offset in the net force acting on the rod.

Table 6. Observed rod tip offset

Case	Front-Offset (mm)	Side-Offset (mm)	Net Offset (mm)
Experiment	1.47675	0.02654	1.4770
LES	0.00088232	0.001089	0.0014
AniPFM	0.8115	0.5112	0.9591

Key results obtained from the time history are presented in Table 7. The RMS polar displacement is numerically predicted to be 7.775 times the experimental value, which is a severe overprediction, while the LES underpredicts compared to the experiment by a factor of 2.87. The RMS front (x-) and side (y-) displacements are found to be very dissimilar for the experiment and the current numerical predictions, while this is not the case for LES. The current simulation uses an axisymmetric domain, so the observation is surprising.

Table 7. Obtained displacement results

Case	RMS X-Disp. (μm)	RMS Y-Disp. (μm)	RMS Polar Disp. (μm)
Experiment	64.7558	48.138	80.6882
LES	20.4478	19.2663	28.0945
AniPFM	488.1401	386.2039	622.4422

The Power Spectral Density (PSD) of the rod tip front (x-) displacement is compared to the experimental and LES data in Figure 30. In general, the numerical trends themselves agree well with the experiment and with each other. However, the LES results are mostly found to be lower in energy content compared to the experiment, while the AniPFM results are found to be higher in energy content and also noisier.

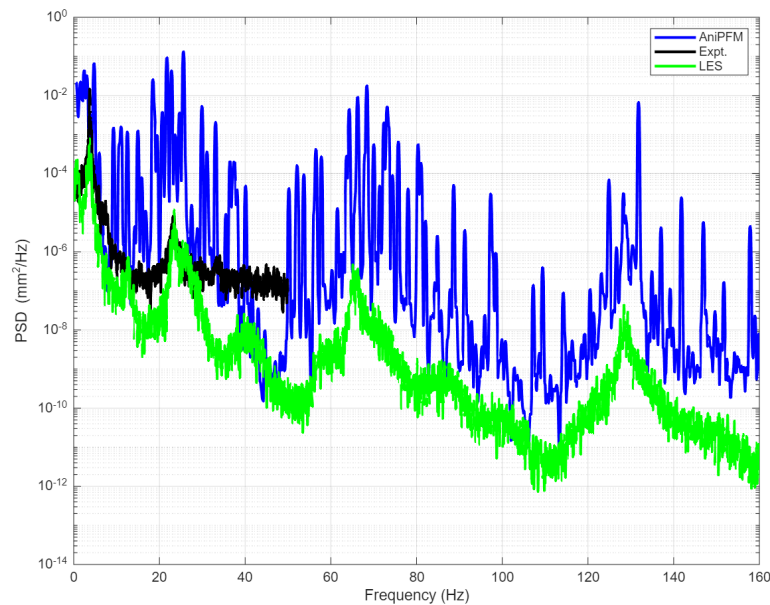


Figure 30. PSD of the tip displacement in the front (x-) direction

The frequencies of the peaks obtained numerically, experimentally and analytically via classical Euler-Bernoulli beam theory are provided in Table 8. For the first two peaks, the current results overpredict the frequency by ~2% compared to the experiment. For the higher frequency peaks, the current results agree well with LES. In all cases, the peak frequencies significantly differ from the analytical solution at higher modes of natural frequencies.

Table 8. Frequencies of the numerical and experimental front displacement PSD peaks compared to the beam natural frequencies obtained analytically

Case	#1 (Hz)	#2 (Hz)	#3 (Hz)	#4 (Hz)
Experiment	3.76	23.47	NA	NA
Analytical	3.71	25.6	71.8	141
LES	3.70	23.42	65.13	128.4
AniPFM	3.85	23.65	66.18	128.2

A similar look is also taken for the rod tip side (y-) displacement as shown in Figure 31. Only the first peak is distinctly visible in the experimental results while the numerical results show a very similar trend as the front displacement (Figure 30). The first mode frequency for both directions is summarized in Table 9. The frequencies obtained from the front and side displacement are similar in the experiment and current results, with slight dissimilarity

for LES. For the average frequency, the numerical predictions show a higher overprediction compared to experiment than LES but is still within the ballpark. It can thus be concluded that the vibration amplitudes are more affected by the p' generated by AniPFM than the vibration frequencies, which appear to be more influenced by the chosen structural model.

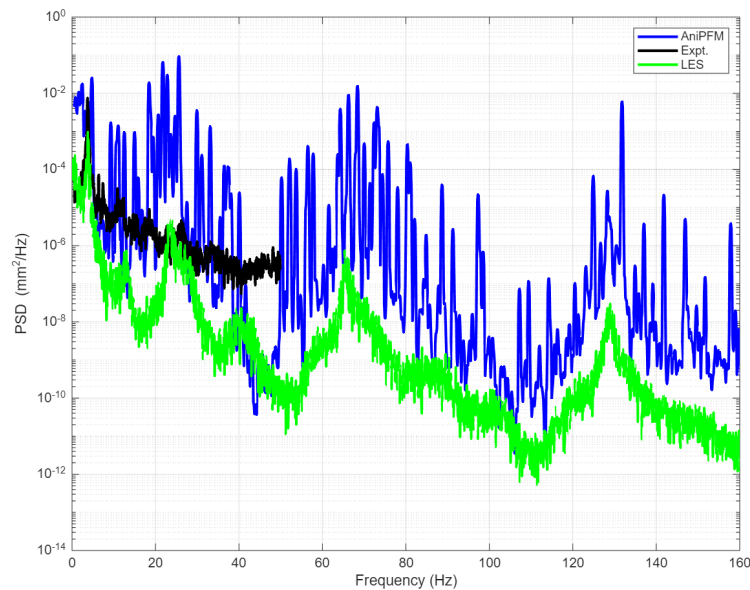


Figure 31. PSD of the tip displacement in the side (y-) direction

Table 9. Frequencies of the numerical and experimental front displacement PSD peak #1 compared to the beam natural frequencies obtained analytically

Case	Front (Hz)	Side (Hz)	Average (Hz)
Experiment	3.76	3.71	3.73
Analytical	3.71	3.71	3.71
LES	3.70	3.80	3.75
AniPFM	3.85	3.85	3.85

1.3 Conclusions

In this chapter, the current AniPFM model was attempted to be validated against the cantilever rod experiment at an annular $Re=21.2k$. A few updates of the model were first presented along with the current challenges that could not be addressed within the timeframe of the project. Ultimately, the consistent observation of a ‘double parabola’ persisted in the p' field. This is known to exist due to a net source in the pressure Poisson equation that, though centered around zero for the whole domain, has local offsets before the rod tip and after the rod tip. Due to these limitations, a safer approach was taken where a pure CFD was performed on the still rod to compare flow field data, and a 1-way FSI was performed to compare vibration data.

Initial progress was first made with a 2-way FSI run in mind. To this end, vibration tests of the cantilever rod were made in vacuum using standalone Deal II, and in still water using coupled OpenFOAM-preCICE-Deal II FSI solver. The results for the solid and fluid model were calibrated and validated using the same, but as mentioned earlier, were not

transferred for the final validation study. Only the fluid model was re-used for the pure-CFD study as the starting point with the inclusion of the AniPFM model. Comparisons were made with experimental data as well as high fidelity DNS and LES data reported in (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025).

The pure CFD results revealed good predictions of the 1st order statistics, i.e the mean axial and radial components of velocity at several elevations in the vicinity of the rod tip. This was directly an output of the base URANS $k - \omega$ SST. For validation of AniPFM, 2nd order statistics, namely the mean $u'v'$ Reynolds stress tensor component and the mean turbulent kinetic energy were compared. The experimental results coming from PIV were considered mainly to realize trends while the high fidelity data was taken as the standard for the magnitudes. The comparison revealed that the AniPFM predicted significantly different trends when compared on a location basis, but it could also be interpreted as a different evolution of the quantities in a holistic sense moving downstream where ultimately the match becomes decent as the flow becomes purely annular.

The comparison also revealed the current gap that must be improved with future work. The main limitation was reasoned to come not from the AniPFM methodology but the exclusion of introducing anisotropy either via the Wilcox correction or using a 2-equation model that uses the Boussinesq approximation for the Reynolds stress tensor which the generated synthetic turbulence replicates.

The 1-way FSI study revealed vibration predictions that were, unexpectedly, directional instead of chaotic, and at an offset instead of centered. Furthermore, the vibration amplitudes were grossly overpredicted by ~8 times compared to the experiment. In terms of reactor safety and operation, this means that the AniPFM predictions of the vibration amplitude are very conservative.

The frequency of vibrations, however, were correctly predicted to be the same for front and side displacements and only slightly higher than the experiment (within 2%). It is concluded that the vibration amplitudes are more affected by the p' generated by AniPFM than the vibration frequencies, which appear to be more influenced by the chosen structural model.

1.3.1 Best Practices

Based on the experience of this work, a number of ‘best’, or more realistically, ‘advisory’ practices can be identified. These are listed below based on the code/tool/case involved.

1.3.1.1 Deal II

Working with Deal II, by virtue of the possibility of tweaking the setup code provides the user (with the right know-how) immense flexibility to setup the same case n-number of ways. One of this know-how involves parallelization of the working code. Two options are possible: Threads and MPI.

The current code used threads since the structural mesh (and thereby degrees of freedom) was large, but not so large that there was a need to split the problem across multiple interconnected servers. The advice is to pick the right one that suits your needs, or if you

are dealing with a very coarse mesh, a single core could do the job as was the case in past validation works (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024).

To implement threads, Deal II has WorkStream available which avoids users to jump in to standard threading libraries (like `std::thread` or OpenMP), which can be a nightmare. WorkStream abstracts all the scary parts of multithreading (mutex locks, thread creation, load balancing) away from you. There are two distinct phases:

1. The Scratch Phase (Parallel): Each thread gets its own isolated bucket of memory (ScratchData). Threads pick a mesh cell, calculate the local cell matrix/vector using their own local FEValues, and do the heavy math completely independent of each other.
2. The Copy Phase (Sequential/Thread-Safe): Once a thread finishes calculating a single cell, it hands its local result (CopyData) to a supervisor. The supervisor manages a single-file line to safely dump the local matrix into the global SparseMatrix. This prevents two threads from writing to the same global matrix slot at the same time.

To carry this out, create two small structs inside your class. The first is ScratchData which holds objects that are expensive to create, but can be reused from cell to cell (like FEValues or Mapping). The rule is threads can read/write to this, but only for their currently assigned cell. The other is the CopyData which holds the final output of a single cell's calculation before it gets shipped to the global matrix. It is temporary, short-lived data.

Instead of one massive loop, you break your assembly into two distinct member functions. The first is the local worker which replaces the inner math of your old cell loop. Do not touch global matrices/vectors here. See sample below.

```
template <int dim>
void MySolver<dim>::assemble_local_cell(const typename
DoFHandler<dim>::active_cell_iterator &cell, ScratchData &scratch, CopyData &data)
{
    const unsigned int dofs_per_cell = fe.n_dofs_per_cell();

    // Reinitialize the thread's local storage for this cell
    data.cell_matrix.reinit(dofs_per_cell, dofs_per_cell);
    data.local_dof_indices.resize(dofs_per_cell);
    scratch.fe_values.reinit(cell);

    // ... DO ALL YOUR FEM MATH HERE ...
    // Use scratch.fe_values to calculate data.cell_matrix(i,j)

    // Cache the local-to-global DoF indices
    cell->get_dof_indices(data.local_dof_indices);
}
```

The other is the transporter which is a function automatically locked by Deal II. Only one thread can execute this at a time, making it perfectly safe to write to the global matrix. See sample below.


```
template <int dim>
void MySolver<dim>::copy_local_to_global(const CopyData &data)
{
    // Safely distribute the single cell matrix into the global matrix
    hanging_node_constraints.distribute_local_to_global(data.cell_matrix,
data.local_dof_indices, system_matrix);
}
```

Now, wipe out your old `for(auto cell : dof_handler.active_cell_iterators())` loop (typical for 1 core jobs) completely. In its place, call `WorkStream::run`. See sample below.

```
template <int dim>
void MySolver<dim>::assemble_system()
{
    // 1. Reset your global matrix to zero
    system_matrix = 0;

    // 2. Hand everything over to WorkStream
    WorkStream::run(dof_handler.begin_active(),    // Start cell
dof_handler.end(),    // End cell
*this,    // This class instance
&MySolver<dim>::assemble_local_cell, // Math function
&MySolver<dim>::copy_local_to_global, // Safe copy function
ScratchData(mapping, fe, quadrature), // Template scratch bucket
CopyData());    // Template copy bucket
}
```

Two final advises here: If you have hanging nodes or boundary conditions, do not call `constraints.distribute_local_to_global` inside your `assemble_local_cell` function. That must happen exclusively inside the `copy_local_to_global` stage. Inside `assemble_local_cell`, never modify class variables (e.g., trying to count total iterations with a global counter variable `cell_counter++`). If multiple threads write to `cell_counter` at the same time, your code will experience a race condition and produce garbage numbers.

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

1.3.1.2 Vibration in still water with gravity

There was a significant challenge faced in the inclusion of gravity in the two-way FSI test with OpenFOAM-preCICE-Deal II for the cantilever rod vibration in still water. Two attempts were made as described below, before the final approach of implementing effect of fluid

gravity in Deal II (see section 1.2.2), along with advice from this experience. Both these attempts made efforts to include gravity in OpenFOAM.

The first attempt was sticking with pimpleFoam solver. Gravity can be included via fvOptions as a body force. The catch here is that boundary conditions need to be modified for 'p' to account for it. The aforementioned fixedFluxPressure boundary condition was used, where previously zeroGradient was used, to better account for gravity. However, setting gravity via fvOptions applies it only to cell volumes, but does not share the information with the boundary patches, essentially reverting to zeroGradient boundary conditions. It is not correct to use zeroGradient, especially on patches that have their normal oriented such that a component aligns in the vertical direction.

There is thus a need to set the gradient correctly for each patch with Neumann boundary conditions. This can be a challenge, especially for the curved end patch, requiring more advanced knowledge of coded boundary conditions. The latter can still fail, if the discretization scheme selection is not optimal and for severely non-orthogonal meshes. Failure to perfectly balance the boundary conditions leads to leaky flow that should not exist in this case with still water, as shown in Figure 32. The advise here is not use pimpleFoam for this scenario.

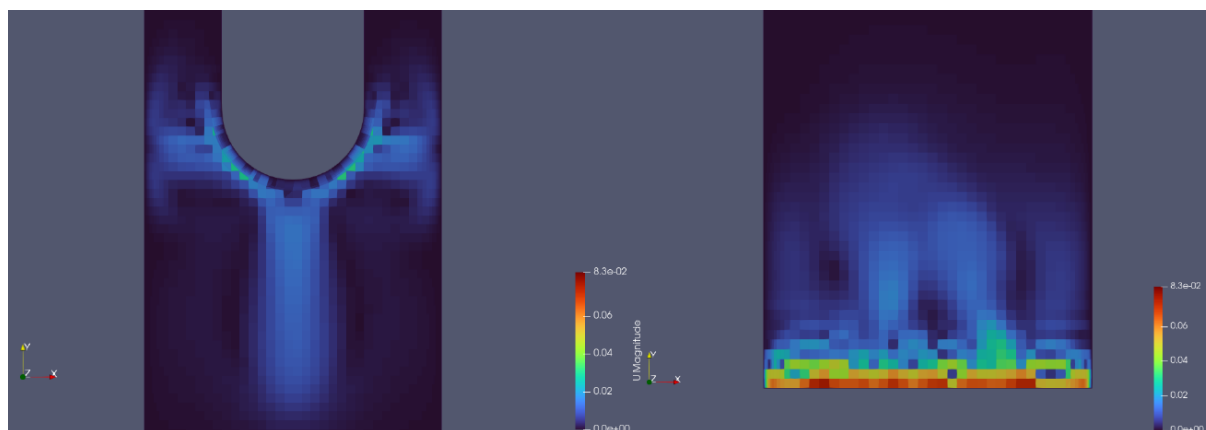


Figure 32. Spurious velocities observed for 'rest' conditions

The next attempt was made by switching to buoyantPimpleFoam. Now, gravity can be more easily added via constant/g file. The pressure boundary conditions are also easy to handle since you can use the same 'p' file and rename it as 'p_rgh' that is solved by buoyantPimpleFoam. The hydrostatic component is automatically handled by buoyantPimpleFoam by constructing 'p' from 'p_rgh' as $p = p_{rgh} + \rho gh$. The good outcome is that the flow is really at rest before the rod is pulled to vibrate and the hydrostatic pressure is well established throughout the calculation. There is a catch that buoyantPimpleFoam is compressible, meaning it additionally solves for density, temperature, and internal energy depending on the sub-model selected. Furthermore, the pressure is now dynamic and not kinematic, likewise dynamic viscosity is used.

There was an issue realized after the simulation was run, and results presented in Table 10 were analyzed. There is a clear improvement in vibration frequency from the case without gravity, however, the damping ratio was severely impacted. This indicates that something

has gone wrong with the computation of viscous forces on the rod. No apparent issues were found in OpenFOAM setup as well as Deal II. A closer look at preCICE revealed that the cause lied in the preCICE adapter for OpenFOAM for compressible flow solvers.

The off-the-shelf adapter uses `devRhoReff()` which returns the viscous deviatoric stress tensor. However, `buoyantPimpleFoam` uses a rho-based/buoyant formulation where the density fields fluctuate or are handled inside specialized thermophysical/multiphase models. This means that the advice for those attempting this approach is that they must correct this adapter locally to use the more general `devTau()`. This was only discovered towards the end of the project and was thus not pursued.

Table 10. Obtained results: (experiment (Cioncolini, Zhang, Li, Nabawy, & Iacovides, 2022): $f_{n,w} = 3.72 \text{ Hz}$, $\zeta_{n,w} = 15 \times 10^{-3}$)

Case	$f_{n,w}$ (Hz)	Error (%)	$\zeta_{n,w} (\times 10^{-3})$	Error (%)
Without Gravity (S1Y1G1)	3.634072	2.31	14.9703	0.20
With Gravity (S1Y1G2)	3.65550	1.73	14.3460	4.36
With Gravity (buoyantPimpleFoam)	3.748560	0.77	10.7246	28.5

1.3.1.3 Synthetic Turbulence

The challenges faced in the development of AniPFM have been reported and its current performance against the validation study. The advice here for those developing a fast working method via generation of synthetic turbulence is to not lose sight of the ultimate goal – predict vibration amplitudes accurately. By extension, this means the primary interest is to obtain ‘correct’ pressure fluctuations on the rod from (synthetic) turbulence in the flow. In terms of capturing the turbulent structures and scales, it is expected that the vibration contribution of the tiny scales very close to the wall are insignificant owing to the extremely high frequencies involved – the structure won’t be able to react. Rather the intermediate/larger scales passing by the structure are the ones more critical and the ones that a synthetic turbulence model should predict accurately.

1.3.1.4 General Advice

A few, more general, advices can be given to those performing this simulation:

- It is good practice to validate structural and fluid models before applying them to the final application. This is done via validation against free vibration tests in air/vacuum and still water.
- Use of AI: This always has to be done carefully. These tools will always have an answer/advice for you – whether it is good/bad is up to you to judge. This also means having good arguments for why something suggested by AI should actually be used rather than just applying it and hoping for the best. For example, during the parallelization efforts of Deal II model, the actual solving done by

SparseDirectUMFPack is a single core effort. When AI was asked about making this operation parallel, it said it could be done, just use MKL which is possible to get by setting the flag `DEAL_II_WITH_MKL` ‘on’ during installation. Now while MKL might be a good alternative, the flag it suggested does not exist.

1.3.2 Future Work

The future work mainly lies in further advancing and testing AniPFM. These will be taken primarily to address the challenges faced during the current validation study.

1. Better representation of k_{start} and k_e : The largest eddy that can exist in the Cantilever Rod domain is different in the pipe section and annulus section, and while these maybe respected in a more straightforward way with few changes to the code, a more challenging aspect is the transition from pipe to annulus.
2. Construct divergence-free non-dimensional velocity fluctuations: The curl of a potential formulation is promising and will be pursued to see if mesh improvements or better scheme choices can better serve to respect this property.
3. Inclusion of anisotropy: This may be by implementing the Wilcox correction or adopting the RSM URANS model.
4. Not conserve the turbulent kinetic energy of the underlying URANS model: This seems counter-intuitive to suggest, but what is currently done is to fully energize whatever scales we can resolve with the mesh. The mesh effectively filters out all the small scales it cannot represent via the k_{cut} . To compensate, the surviving scales get energized more than they should by the Cholesky operation. What will be tested is to give the scales only the fraction of the energy that can be resolved by the mesh. Likewise, the governing pressure Poisson equation will also be modified to handle this change and give more reasonable p' results.
5. Test other time correlation methods for the synthetic turbulence: Currently, only the constant convection was tested. It may be worthwhile to investigate the effect of the Exponential Correlation & Convection method described in (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024).

2 Validation of the ANSYS Model Order Reduction for fast-running FSI simulations

Large cost reduction in a FSI simulation in terms of computational time can be achieved with reduced-order models for the fluid domain. This results from the fact that in an FSI simulation the CFD part is significantly more expensive than the structural one. Therefore, almost all fast-running models in GO-VIKING's WP5 are focused on the development and implementation of fluid-based ROMs. It is still worth investigating the savings in computational time when the model reduction is based on the structural part of the computational domain. In GO-VIKING D5.1 "Development and implementation of fast-running FSI methods", Model order Reduction (MOR) approach, developed by ANSYS (Einzinger, 2014), was shortly introduced and information was provided on how to implement this in ANSYS CFX along with some comparative results between reduced- and full-order model for a simplified test case. Further, its efficiency was evaluated through comparison with a full-order model (FOM). In this chapter, the focus is put on the validation of the ANSYS CFX-MOR code on the AMOVI experiment.

2.1 The AMOVI Experiment

AMOVI (Piteau, X., Panunzio, Lagrange, & Antunes, 2022) 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 $P/D = 1.44$. 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 and a full tube bundle with central flexible tube in the 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 33), allowing it to vibrate primarily in lift (perpendicular to the main flow) direction, with a natural frequency in air $f_n = 27.5$ Hz and a damping ratio 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. In the rigid arrangement, a tri-axial force sensor measures the forces acting on the central tube. 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.

Twenty-one 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. The analyses presented here are based on AMOVI Test 27 with single flexible tube exposed to a volumetric flow rate of 2.9 l/s at 25 °C. This case was selected as a first step before the simulation of the bundle configurations.

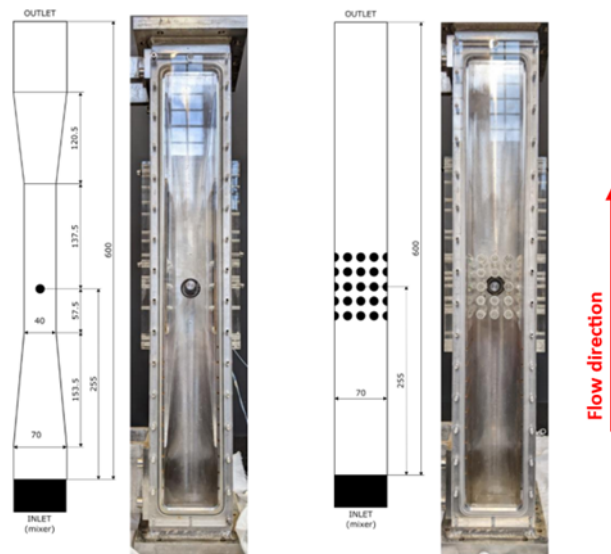


Figure 33. AMOVI test section with two different configurations: single tube (left) and tube bundle (right) in a flow channel

2.2 FSI Approaches

In a first step, the FSI programs ANSYS CFX-MOR and the full-order model (FOM) ANSYS CFX-Mechanical were validated on two different AMOVI experiments. These are existing and part of the used ANSYS code package. The study here is focused not only on the validation of these methods, but also on their comparison. Below is a short reminder on the reduced- and full-order models used in this analysis.

2.2.1 Full-order model ANSYS CFX-Mechanical

ANSYS CFX 22.1 and ANSYS Mechanical 22.1 were coupled through System Coupling (SYSC) service. SYSC can integrate multiple individual analyses, enabling to leverage different physics solvers and/or static external data sources in a single multiphysics simulation. Both programs are coupled using stagger loops, in which ANSYS CFX calculates the fluid forces acting on the structures, while Mechanical provides back the mesh displacement. This allows for implicit coupling of the fields in the SYSC solution. Within each step in the time step loop, the field solutions are exchanged in the stagger loop until convergence is

reached. In the presented analyses, 3 staggered iterations were performed, and the total amount of ANSYS CFX iterations within one time step was around 10. Large displacements, strains and non-linear material behavior can be considered during FSI simulations with SYSC. The interface tracking in all simulations presented in this paper are based on Arbitrary Lagrangian Eulerian (ALE) method with moving meshes.

2.2.2 Reduced-order model ANSYS CFX-MOR

In the ANSYS CFX-MOR 22.1 code ANSYS CFX is coupled to a mechanical reduced-order model. MOR is a fast-running system of mechanical equations, which uses results from a modal analysis, performed in advance with a CSM code. The objective of the MOR is to describe the static or dynamic responses of the mechanical system in an efficient and sufficiently accurate way. ANSYS MOR is based on the mode-superposition method, which uses the eigenfrequencies and mode shapes generated from a modal analysis to characterize the dynamic response of a structure to transient or steady harmonic excitations. Since it is a simplification of the mechanical system, it is important to remember that FSI simulations with ANSYS CFX – MOR are subject to some restrictions. The approach is valid for cases with linear dynamics and this implies moderate displacements, small strains, linear contacts and linear material behavior (Einzinger, 2014). More information on MOR can be found in (Papukchiev, Numerical Analysis of Reactor Relevant Vibrations Using Advanced Multiphysics CFD-CSM Methods, 2019).

2.3 AMOVI structural model and modal analysis

The vibrating tube was produced of stainless steel (density of 8300 kg/m³, 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, 39773 elements were used in ANSYS Mechanical to generate an unstructured mesh for the 0.2 m long tube and its blade (see Figure 34). Further refinement had no noticeable effect on the results. The right end of the structure was fixed in x, y, z directions, while the left end of the tube can freely vibrate. This was realized with DISPLACEMENT boundary conditions. The outer tube and blade surfaces were specified as fluid-solid boundary, which is necessary for the FSI analyses.

The modal analysis is used to determine the vibration characteristics of the analysed structure. The modal analysis was performed for six modes. The mode shapes, frequencies and the total mesh displacement for each node in x, y and z directions were extracted and used as input for ANSYS CFX-MOR code. The calculated first eigenfrequency of 29.30 Hz agrees with the experimental result of 27.5 Hz, reported by CEA. The CEA own calculations also resulted in 29.30 Hz. Currently, there is no clear explanation why the measurements reveal 29.30 Hz for the natural frequency of all tubes, but only for the central (vibrating) one the value is 27.5 Hz despite the same geometry and materials. Figures Figure 35 - Figure show the shapes of the first six modes, while Table 11 provides information on the calculated eigenfrequencies used as input for the MOR approach.

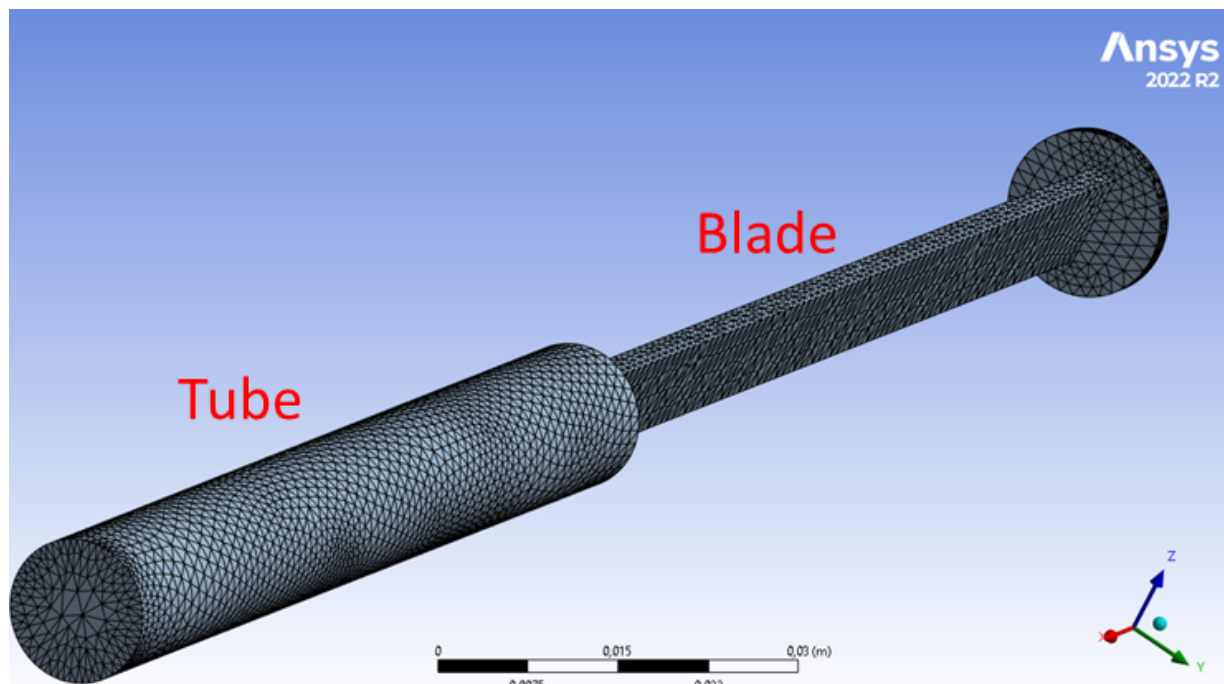


Figure 34: Numerical mesh of the AMOVI flexible tube and its blade.

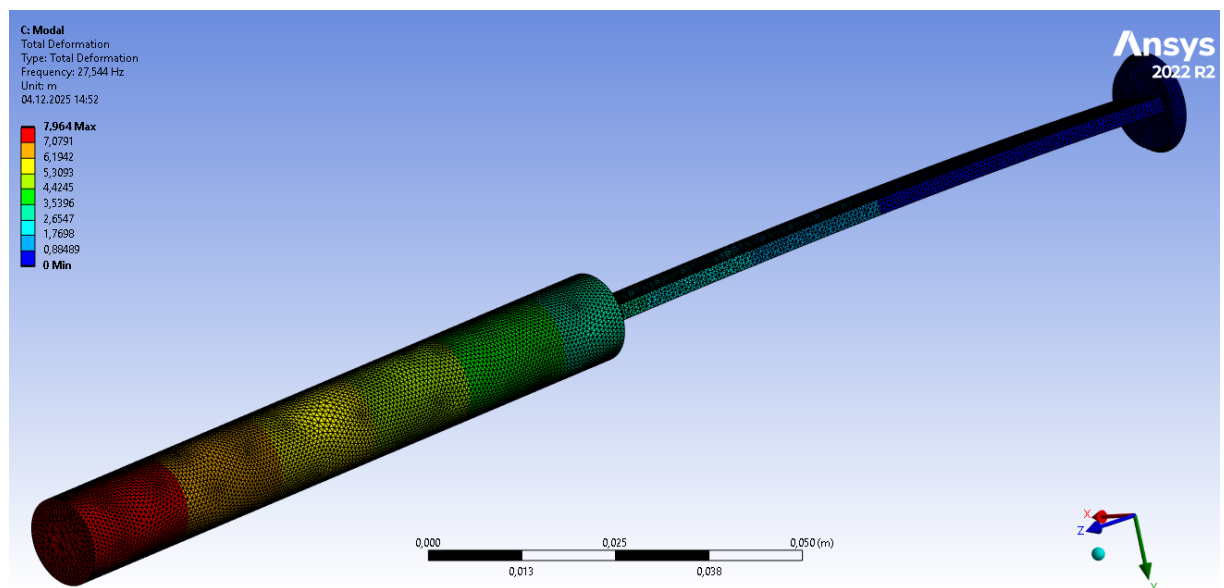


Figure 35: First bending mode of the flexible tube and its blade.

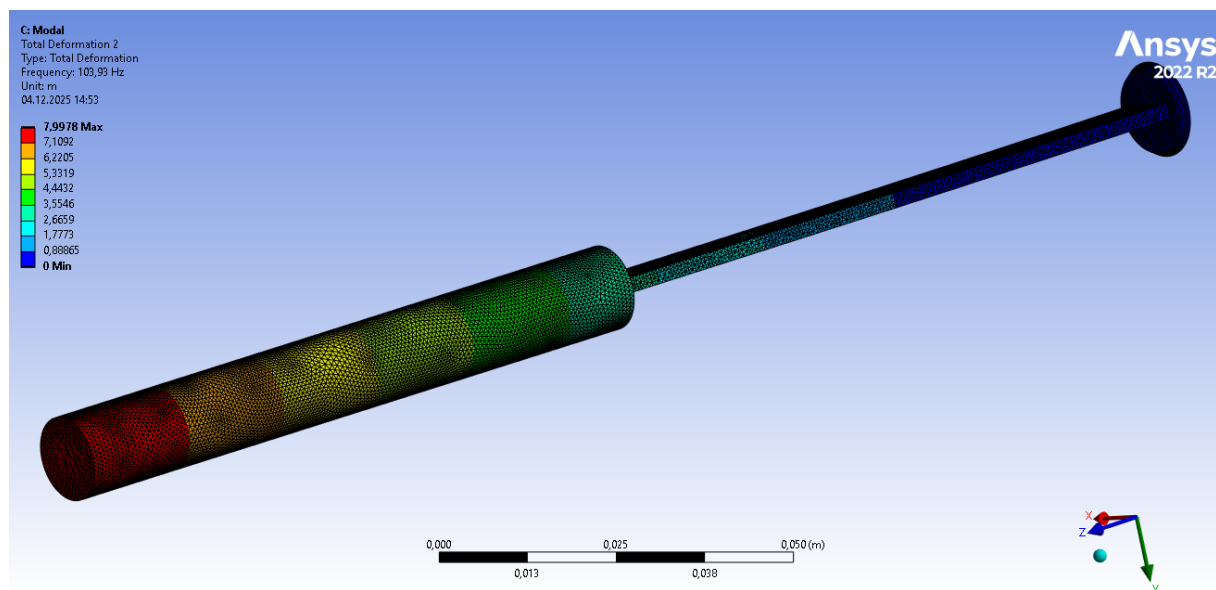


Figure 36: Second bending mode of the flexible tube and its blade.

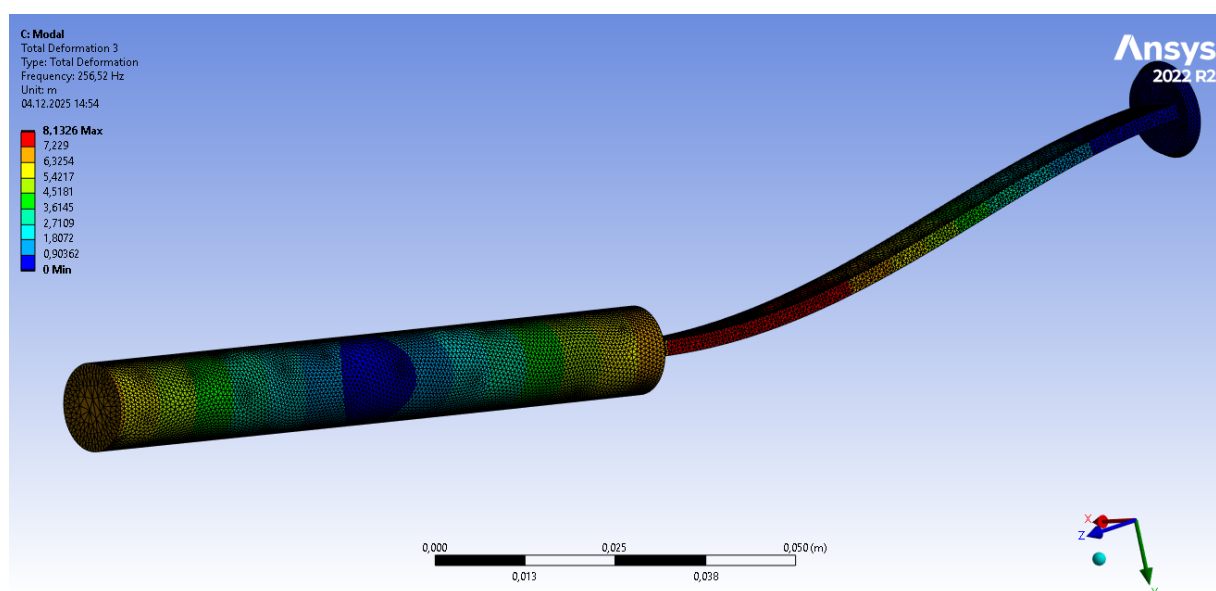


Figure 37: Third bending mode of the flexible tube and its blade.

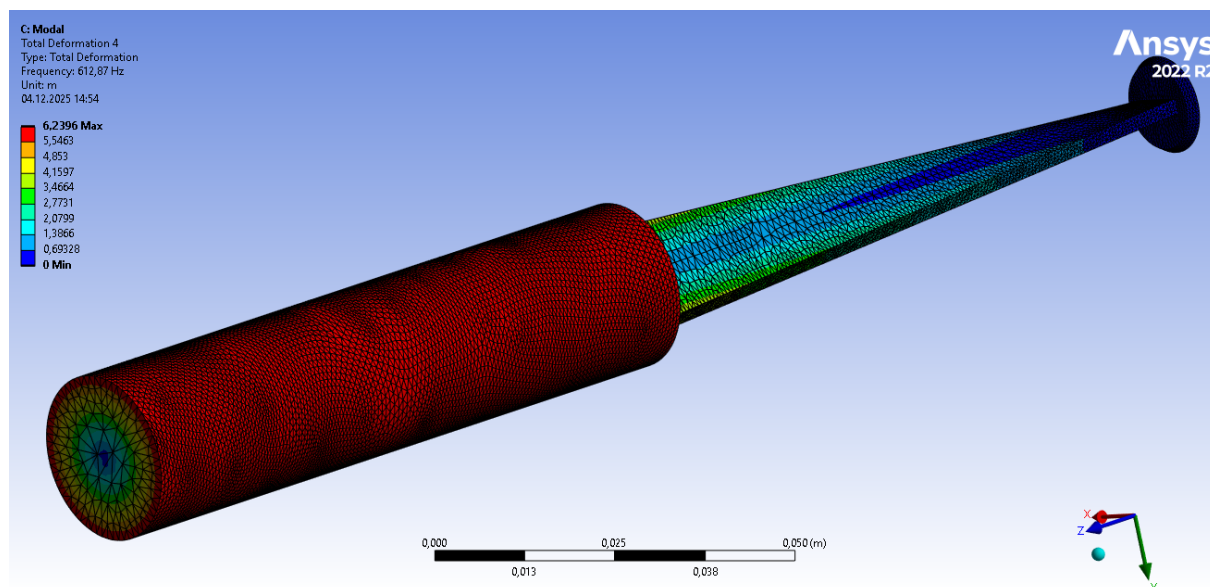


Figure 38: Fourth bending mode of the flexible tube and its blade.

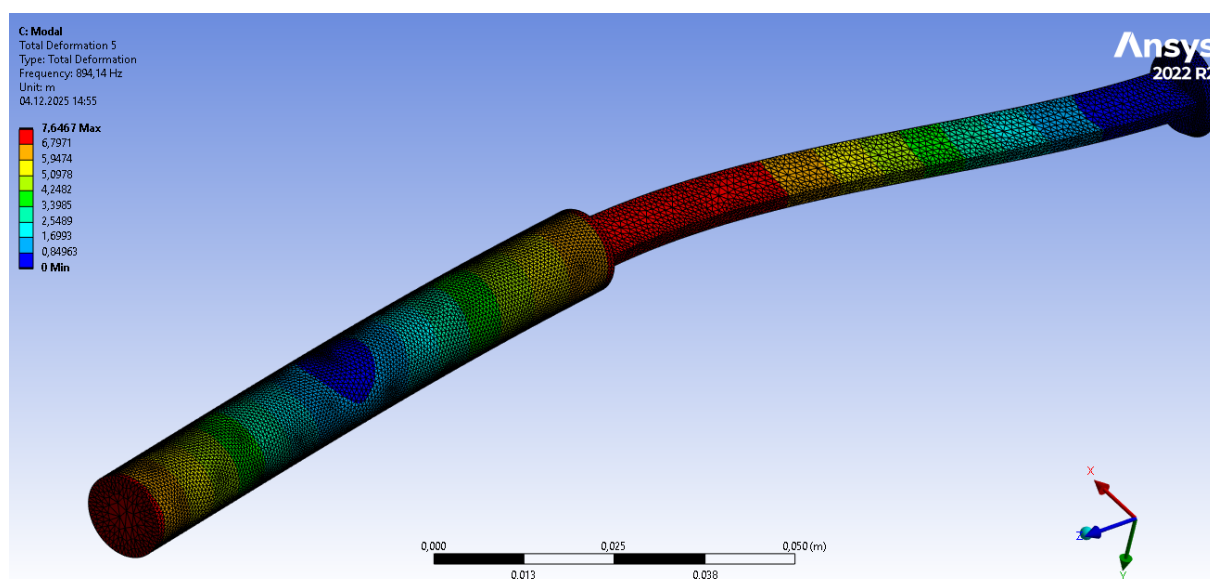


Figure 39: Fifth bending mode of the flexible tube and its blade.

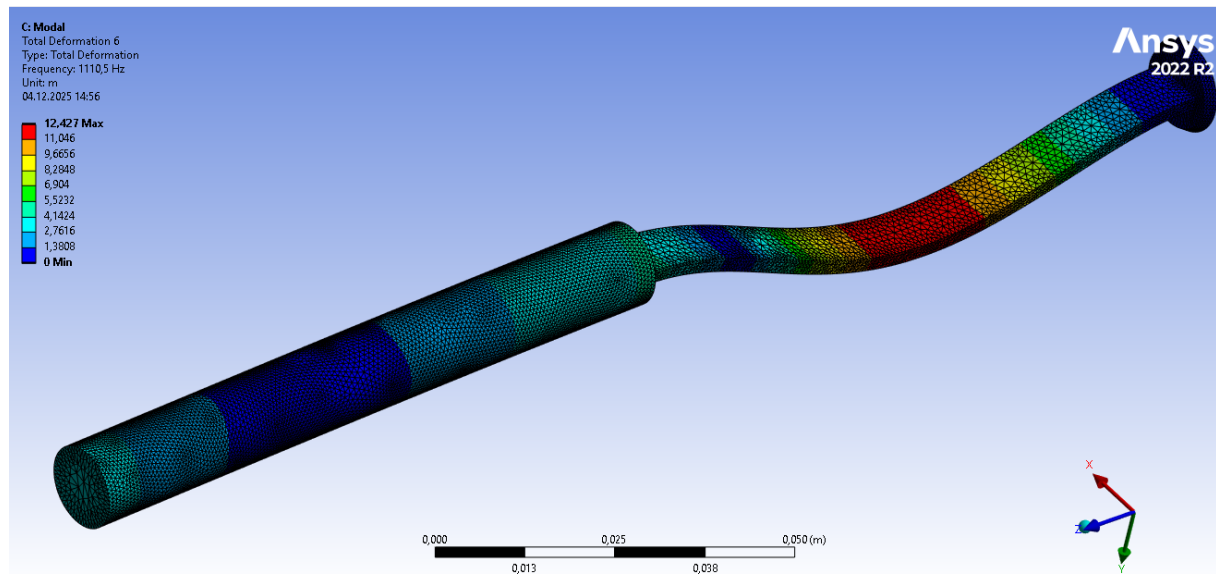


Figure 40: Sixth bending mode of the flexible tube and its blade.

Table 11: Calculated eigenfrequencies with Mechanical, used as input for the MOR approach

Mode number	1	2	3	4	5	6
Frequency, Hz	29.3	110.7	273.0	652.4	052.2	1180.9

2.4 Generation and Implementation of the MOR Model in ANSYS CFX

As already mentioned before, a modal analysis with 6 modes was performed with the cylinder to obtain the fundamental vibration mode shapes and frequencies. Six files were generated, and these were used as an input for the MOR approach in the ANSYS CFX-MOR calculation. These files contain the fundamental frequency for each vibration mode together with the initial x, y, z coordinates and the total mesh displacement for each node of the defined FSI interface. This comprises the outer surfaces of the tube and the blade.

In a next step, the MOR model was built and integrated in the ANSYS CFX input deck. This was done with the help of CEL (CFX Expression Language) expressions. Additional variables and expressions are generated for the fluid forces acting on the FSI interface surface, the nodal displacements, the nodal velocities, etc. To calculate the displacement of each node of the FSI surface in x, y and z directions, the information provided in the six MOR input files (Figure 41 shows the one for the first vibration mode) is used as six field functions, containing the total mesh displacements from the modal analysis. This information in conjunction with the method of mode-superposition is utilized for the calculation of the three displacement components in x, y, z directions, which are provided to the FSI interface surface in ANSYS CFX using CEL (Figure 42). The CFD code calculates the new mesh and then performs with the new, deformed mesh its simulation, generating new results for the updated pressure distribution on the FSI surface. This data is then passed back to the MOR,

which provides the new nodal displacements to ANSYS CFX. User Defined Routines are called at the end of each time step to store the calculated data, which is also used for the computations in the next time step.

```

# EXPROFILE,SURF,DISP, 1,TubeAndBlade01,..\\..\\TubeAndBlade01.csv,
[Name]
TubeAndBlade01

[Parameters]
Ncompt = 3
Nnodes = 4173
Mass = 4.71787247E-002 [kg]
Frequency = 27.566792 [Hz]
Maximum Displacement = 7.9646732 [m]

[Spatial Fields]
Initial X [m], Initial Y [m], Initial Z [m]

[Data]
Initial X [m], Initial Y [m], Initial Z [m], meshdisptot x [m], meshdisptot y [m], meshdisptot z [m], node
1.08870654E-002, 3.06978002E-002, 0.25928914, 0.21851994, 3.4779665, -1.91391650E-004, 380
1.04782555E-002, 2.98447838E-002, 0.25821393, 0.26184058, 3.4572107, -1.93055189E-004, 381
1.20080227E-002, 3.03504469E-002, 0.25890990, 0.23617522, 3.5349124, -1.94286406E-004, 382
9.04853789E-003, 2.94239972E-002, 0.25741119, 0.28319217, 3.3845986, -1.91150803E-004, 383
1.02875512E-002, 2.92460842E-002, 0.25694887, 0.29224545, 3.4475342, -1.91484531E-004, 384
1.31748756E-002, 3.00454374E-002, 0.25851539, 0.25168138, 3.5941927, -1.96566220E-004, 385
1.35250824E-002, 3.09209758E-002, 0.25950191, 0.20721014, 3.6119757, -1.94890540E-004, 386
1.12543726E-002, 3.17404907E-002, 0.26012652, 0.16556261, 3.4966169, -1.87479269E-004, 387
9.82862524E-003, 3.10715644E-002, 0.25963390, 0.19952503, 3.4241993, -1.88237382E-004, 388
1.30949704E-002, 2.94251434E-002, 0.25741383, 0.28318865, 3.5901417, -1.95859761E-004, 389
1.15634451E-002, 2.91397481E-002, 0.25660096, 0.29766400, 3.5123464, -1.92023163E-004, 390
1.31355198E-002, 2.90501318E-002, 0.25622666, 0.30223738, 3.5922085, -1.92678113E-004, 391
1.03799311E-002, 2.89595088E-002, 0.25564660, 0.30680075, 3.4522337, -1.87005452E-004, 392
1.32487372E-002, 2.89250144E-002, 0.25501323, 0.30859304, 3.5979642, -1.87889536E-004, 393
1.45875297E-002, 2.89827526E-002, 0.25583568, 0.30567948, 3.6659761, -1.92991060E-004, 394
1.46196155E-002, 2.98074688E-002, 0.25815329, 0.26378633, 3.6675933, -1.98504301E-004, 395
5.98574844E-003, 2.92388039E-002, 0.25307276, 0.29254700, 3.2290690, -1.65760215E-004, 396
6.72172685E-003, 2.95218208E-002, 0.25237413, 0.27818297, 3.2664392, -1.64388650E-004, 397
7.04996349E-003, 2.90733886E-002, 0.25366550, 0.30096413, 3.2831125, -1.71878088E-004, 398
6.44743599E-003, 3.02067691E-002, 0.25126763, 0.24339214, 3.2525029, -1.59422045E-004, 399
7.42124812E-003, 2.99321688E-002, 0.25164997, 0.25735201, 3.3019586, -1.64334842E-004, 400
8.21279084E-003, 2.89788370E-002, 0.25419302, 0.30578460, 3.3421665, -1.76966069E-004, 401
7.42711455E-003, 2.89259027E-002, 0.25510472, 0.30846234, 3.3022648, -1.80760439E-004, 402
5.21770781E-003, 2.90707189E-002, 0.25367741, 0.30107477, 3.1900689, -1.68098689E-004, 403
5.00046942E-003, 2.94809696E-002, 0.25246112, 0.28023562, 3.1790334, -1.59720357E-004, 404

```

Figure 41: Input file for MOR approach providing data for the first bending mode of the AMOVI tube and blade.

```

mq DISPX = \
mqpos01new*mqDISPX01+mqpos02new*mqDISPX02+mqpos03new*mqDISPX03+mqpos04\
new*mqDISPX04+mqpos05new*mqDISPX05+mqpos06new*mqDISPX06
mq DISPX Wall = mq \
DISPX*if(citern>0,SetHistory(mqpos01new,mqpos02new,mqpos03new,mqpos04\
new,mqpos05new,mqpos06new,mqvel01new*1[s],mqvel02new*1[s],mqvel03new*\
1[s],mqvel04new*1[s],mqvel05new*1[s],mqvel06new*1[s]),1.0)
mq DISPY = \
mqpos01new*mqDISPY01+mqpos02new*mqDISPY02+mqpos03new*mqDISPY03+mqpos04\
new*mqDISPY04+mqpos05new*mqDISPY05+mqpos06new*mqDISPY06
mq DISPY Wall = mq \
DISPY*if(citern>0,SetHistory(mqpos01new,mqpos02new,mqpos03new,mqpos04\
new,mqpos05new,mqpos06new,mqvel01new*1[s],mqvel02new*1[s],mqvel03new*\
1[s],mqvel04new*1[s],mqvel05new*1[s],mqvel06new*1[s]),1.0)
mq DISPZ = \
mqpos01new*mqDISPZ01+mqpos02new*mqDISPZ02+mqpos03new*mqDISPZ03+mqpos04\
new*mqDISPZ04+mqpos05new*mqDISPZ05+mqpos06new*mqDISPZ06
mq DISPZ Wall = mq \
DISPZ*if(citern>0,SetHistory(mqpos01new,mqpos02new,mqpos03new,mqpos04\
new,mqpos05new,mqpos06new,mqvel01new*1[s],mqvel02new*1[s],mqvel03new*\
1[s],mqvel04new*1[s],mqvel05new*1[s],mqvel06new*1[s]),1.0)

```

Figure 42: CEL expressions for the displacements in x, y, z directions, provided to ANSYS CFX for the calculation of the new, deformed mesh.

2.5 Model Preparation for the Validation on the AMOVI Single Flexible Tube Experiments

In this chapter, models based on different FSI methods (MOR and SYSC) are prepared for the simulation of the AMOVI single flexible tube tests with $Q=2.9$ l/s and $Q=1.7$ l/s flow rate at the test section inlet.

2.5.1 CAD and mesh generation

CAD models of the test section channel with single tube were generated with SpaceClaim from the CEA drawings. The resulting geometry is seen in Figure 43. Good initial mesh quality is essential for the accuracy of the FSI results. During FSI calculations, mesh deformation might occur (like in the present analysis), and this directly affects the mesh quality during the simulation run by changing important mesh parameters (min. orthogonality angle, expansion factor, aspect ratio, etc.). This, in turn, has an impact on the CFD solver convergence, and in some cases might lead to instability and other numerical issues. The model has been sliced in several sub-parts and a multizone meshing method was applied. Several hexahedral meshes for the test section with single tube and tube bundle were generated using ANSYS Meshing. Mesh studies were performed within GO-VIKING WP3 Task 3.1, where AMOVI was simulated with the Rigid Body Motion approach in ANSYS CFX. The final mesh used in the ANSYS simulations with a single tube consisted of 4 million elements (Figure 44 - Figure 46). The overall mesh quality is good, although for some elements the aspect ratio reaches 115, and this is beyond the best practice suggestions. One of the most important mesh quality parameters are the min. orthogonality angle (38°) and the expansion factor (21), both of these are within the prescribed limits. Mesh independent studies with 6 and 8 million elements proved that the CFD results are mesh-independent.

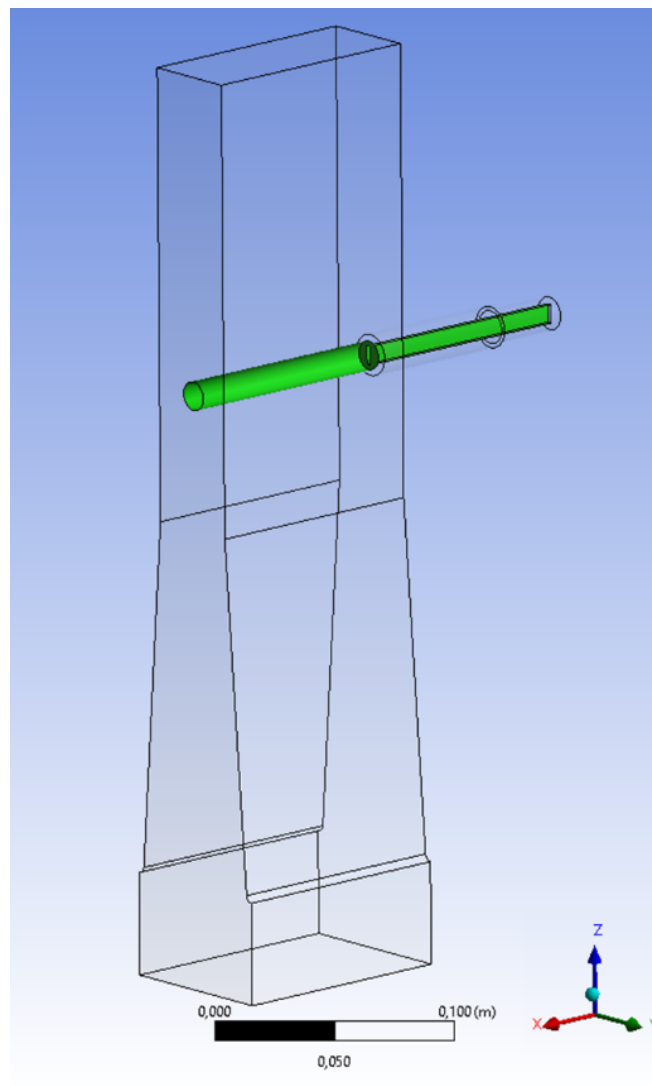


Figure 43: CAD model of the AMOVI test section with a single vibrating tube.

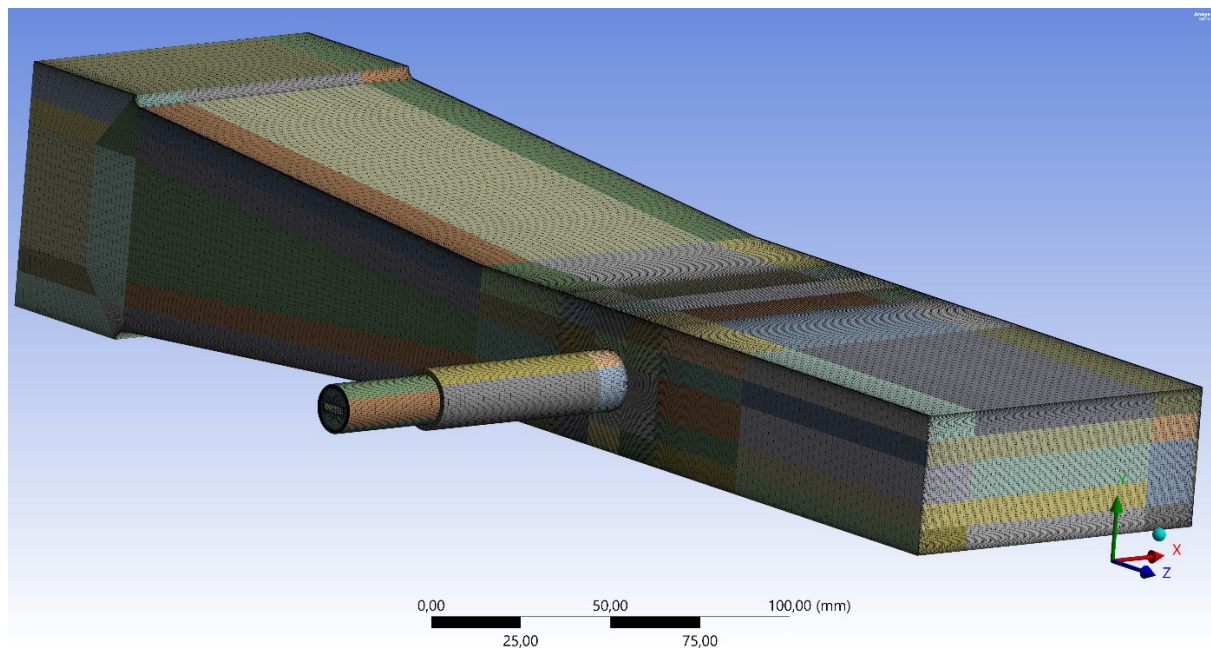


Figure 44: Top view on the CFD mesh with single vibrating tube.

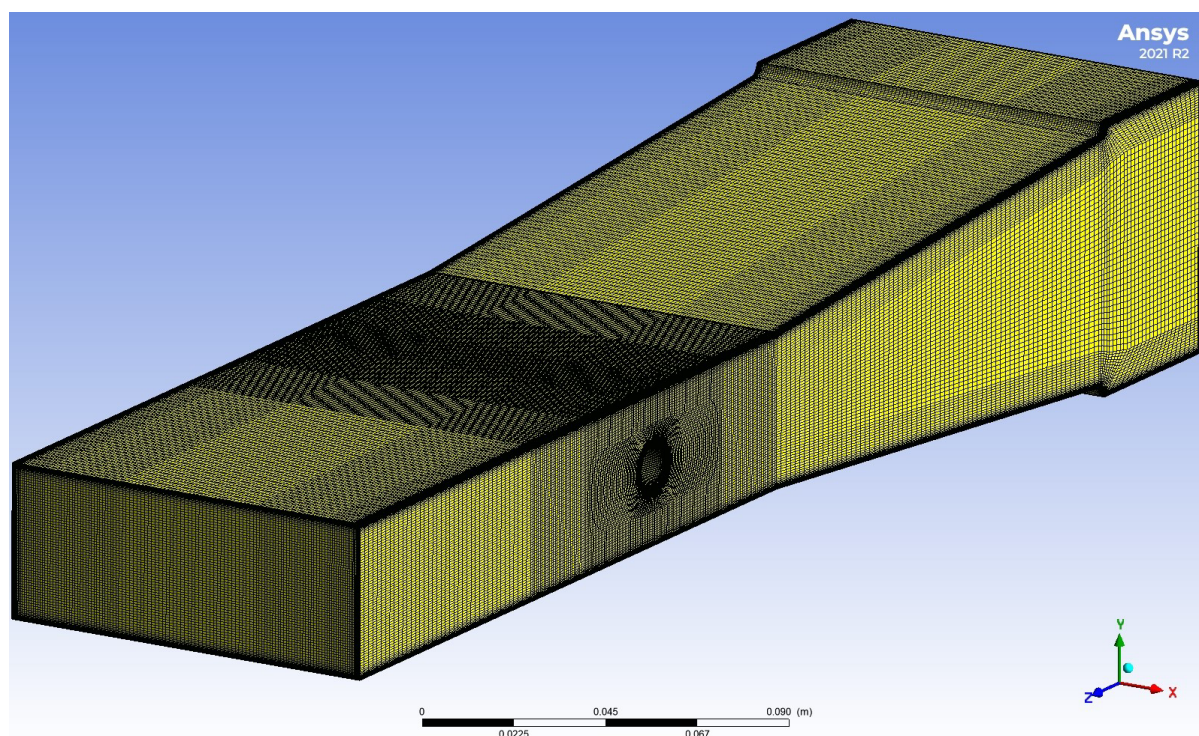


Figure 45: Bottom view on the CFD mesh with single vibrating tube.

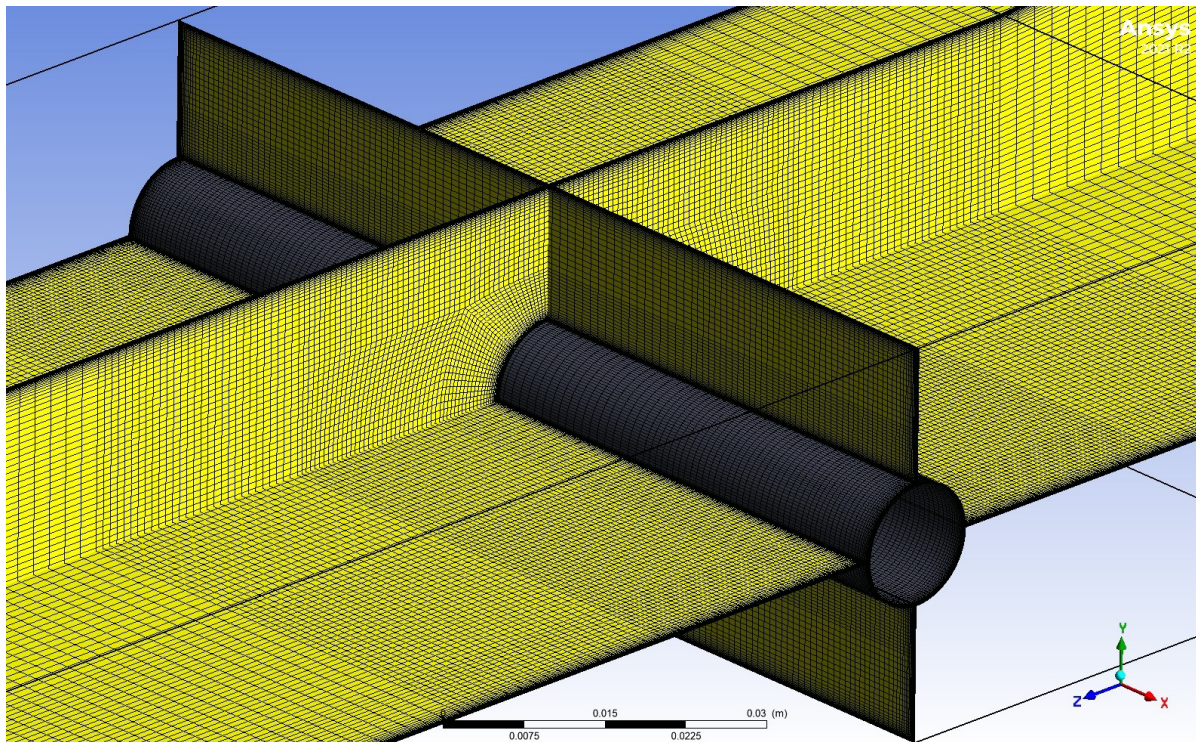


Figure 46: Horizontal and vertical planes through the test section showing mesh density around the vibrating tube.

2.5.2 Generation of the ANSYS CFX model and setup

IAPWS IF97 water material data were used from the ANSYS CFX material libraries. The CFD simulations with ANSYS CFX were isothermal, because the AMOVI experiment was performed at constant 25 °C fluid temperature. For the simulation cases, 2.9 l/s resp. 1.7 l/s flow rate and 5 % turbulence intensity were specified at the inlet of the test section inlet chamber. Since no information on the inlet flow profile was available, a flat velocity profile was specified. In the current CFD model, this is located 16 tube diameters upstream the vibrating tube, which allows the flow to develop before it reaches the tube surface. Gravity is applied in $-z$ direction and reference pressure is set to 3.2 bar. At the outlet patch the pressure is set to zero.

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. The SST turbulence model was selected for the FSI simulations. The time step size used in the reduced-order model (MOR) and the full-order model (SYSC) analyses was $2.0\text{E-}4$ s. The SYSC analysis was performed only for the 2.9 l/s case.

2.6 Results and Comparison Between FOM and ROM for the $Q=2.9$ l/s experiment

2.6.1 Flow pattern

Figure 47 shows the calculated instantaneous velocity with MOR at $t = 9.2$ s. The mean free stream velocity in the channel is 0.74 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 48, which also depicts the flow re-circulation zones behind the tube. The maximum velocity of 1.4 m/s is observed in the 1 mm gap between the left tube end and the channel wall (see Figure 47). The flow in the right part of the blade channel, close to the blade fixation point is stagnant. Figure 47 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 49 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.

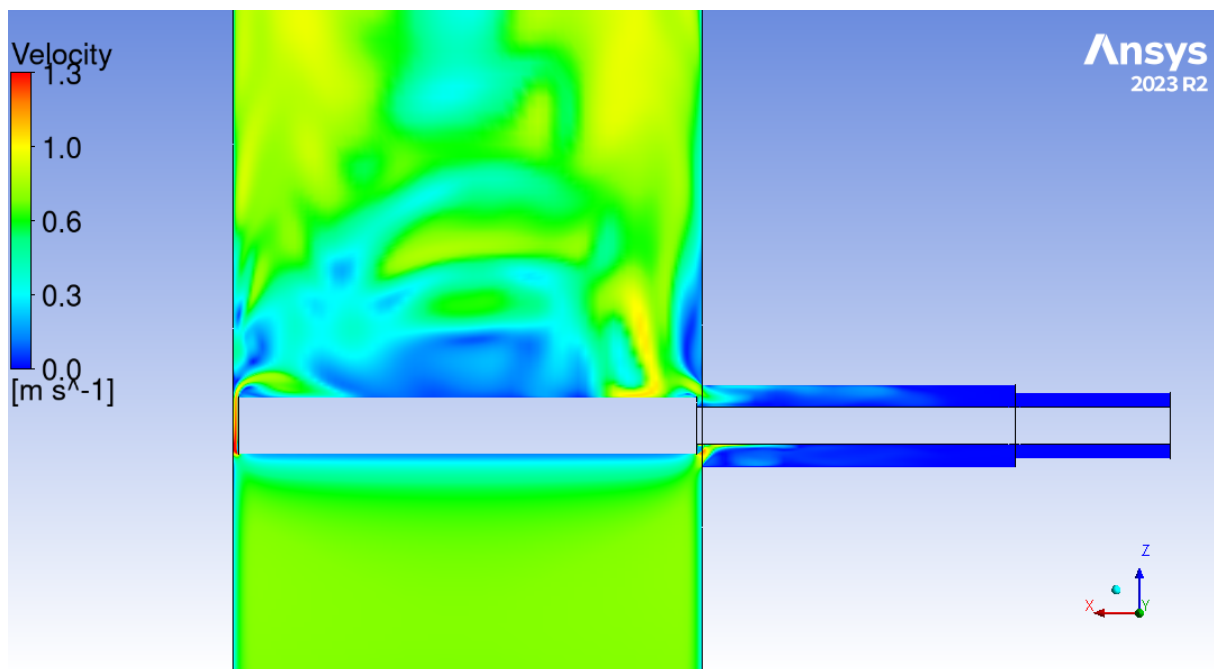


Figure 47: Velocity magnitude distribution in a longitudinal plane (water flows bottom-top) through the AMOVI test section calculated with MOR.

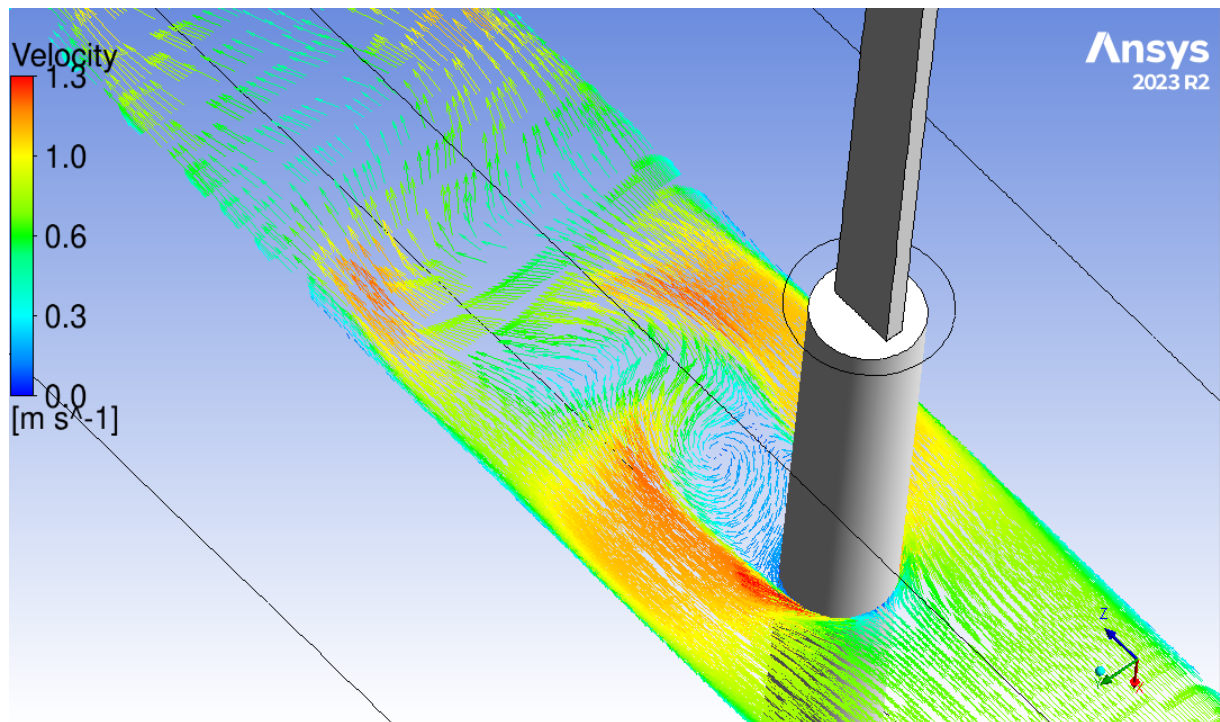


Figure 48: Velocity vectors in XY plane showing flow acceleration, deceleration and recirculation zones as well as the formation of von Karman vortex street behind the cylinder.

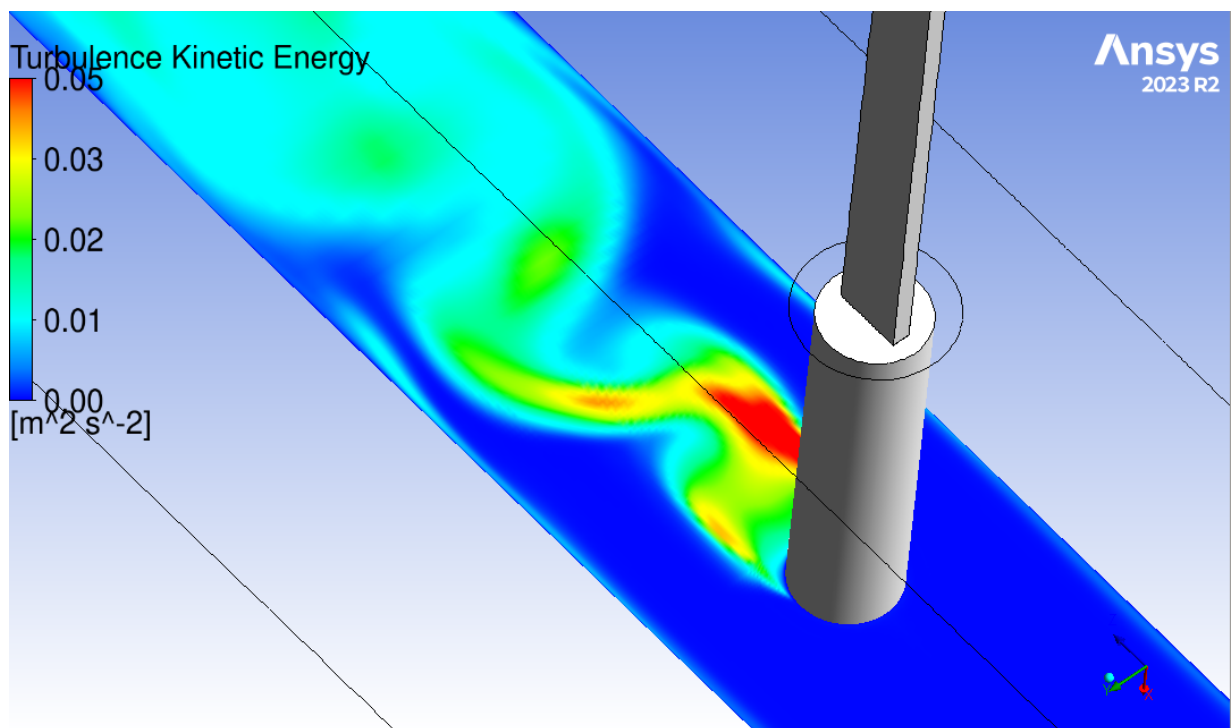


Figure 49: Formation of a von Kármán vortex street behind the AMOVI tube visualized with the turbulence kinetic energy.

2.6.2 Vibration analysis

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 9 s of simulation time. The first simulation second was omitted. This data acquisition times provides sufficient resolution for the FFT analysis. Experimental data of 30 s was also used for the comparison.

Figure 50 presents the calculated and the experimental tube displacements. It can be seen that the vibration displacements in the MOR and SYSC simulations are very similar. The root mean square (RMS) value of the time-dependent displacement was calculated for the available simulation results and the experiment. SYSC predicts an RMS value of 0.065 mm, while MOR gives 0.064 mm. These values are a factor of three smaller than the measured mean value (0.19 mm). Although this deviation is not small, the predicted displacements are still in the same order of magnitude of the measurements.

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 by MOR and SYSC to be 0.16 mm, the measured value is 0.6 mm. This is in-line with the observations of the RMS values: ANSYS simulations underestimate the experimental value.

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 (number of samples per second) was 5000 Hz (MOR and SYSC), 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: MOR and SYSC predict $f_{1,s} = 15.7$ Hz for the vortex-shedding frequency, which is very close to the measured one of 15.2 Hz. The second frequency peak at 22.1 Hz in the experiment results from the reduced first eigenfrequency of the vibrating tube and MOR predicts 23.7 Hz, while SYSC - 23.5 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 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.

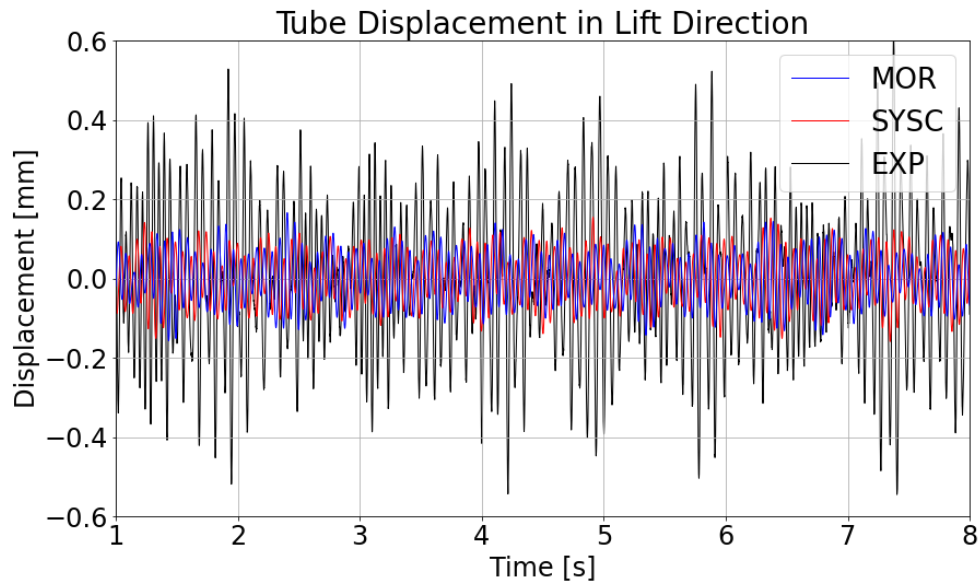


Figure 50: Tube displacements in lift direction for a single vibrating tube exposed to a $Q=2.9$ l/s flow rate.

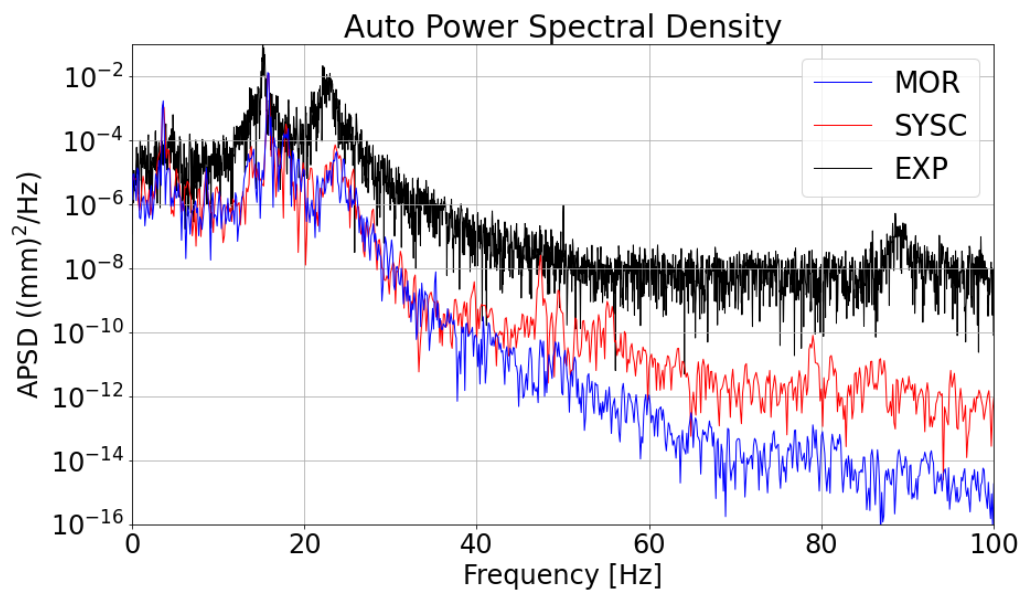


Figure 51: APSD of the tube displacements for a single vibrating tube exposed to a $Q=2.9$ l/s flow rate.

2.6.3 Further investigations to improve the numerical results through the application of a hybrid turbulence model

So far (within WP3), mesh-independence studies were performed on a CFD basis. This means that particular profiles and fluid-related quantities such as velocities were compared. Since we are dealing with FSI here, it was decided to check if the results are also mesh-independent in terms of vibration amplitudes. For this, additional meshes with

approx. 6 and 8 million elements (Figure 52 - Figure 54) were generated with ANSYS Meshing. The mesh was refined mainly around the vibrating tube region in axial and radial directions, while preserving the initial mesh quality. The exact refinement regions were also determined according to the needs of the Zonal Large Eddy Simulation (ZLES) turbulence approach (Menter, 2015), which was used in the next steps.

As it can be seen in Figure 55 and Figure 56, the results in terms of displacement amplitudes and resulting frequency peaks do change, which proves the fact that mesh-independence studies on CFD basis are not sufficient for the FSI analyses. The RMS tube displacements increase by 14% from 0.067 mm (4 million elements) to 0.078 mm (8 million elements), while the measured one is still far away with its 0.196 mm. The vortex-shedding peak decreases from 15.8 Hz (4 million elements) to 15.6 Hz (8 million elements), both being close to the measured 15.0 Hz. The picture of the reduced eigenfrequency is similar: it increases from 23.7 Hz to 24.0 Hz, while CEA measured 23.3 Hz. It can be concluded that the double size mesh slightly improved a bit the results, but only in terms of displacements. The PSD of the displacements calculated with the 6 million elements mesh was omitted in Figure 56, since it is very similar to the curve generated with 8 million elements mesh, and therefore, impedes the comparison between 4 and 8 million mesh results.

It can be concluded that doubling the number of elements improves the displacement amplitudes, but insignificantly when compared to the initial deviation from the measured data.

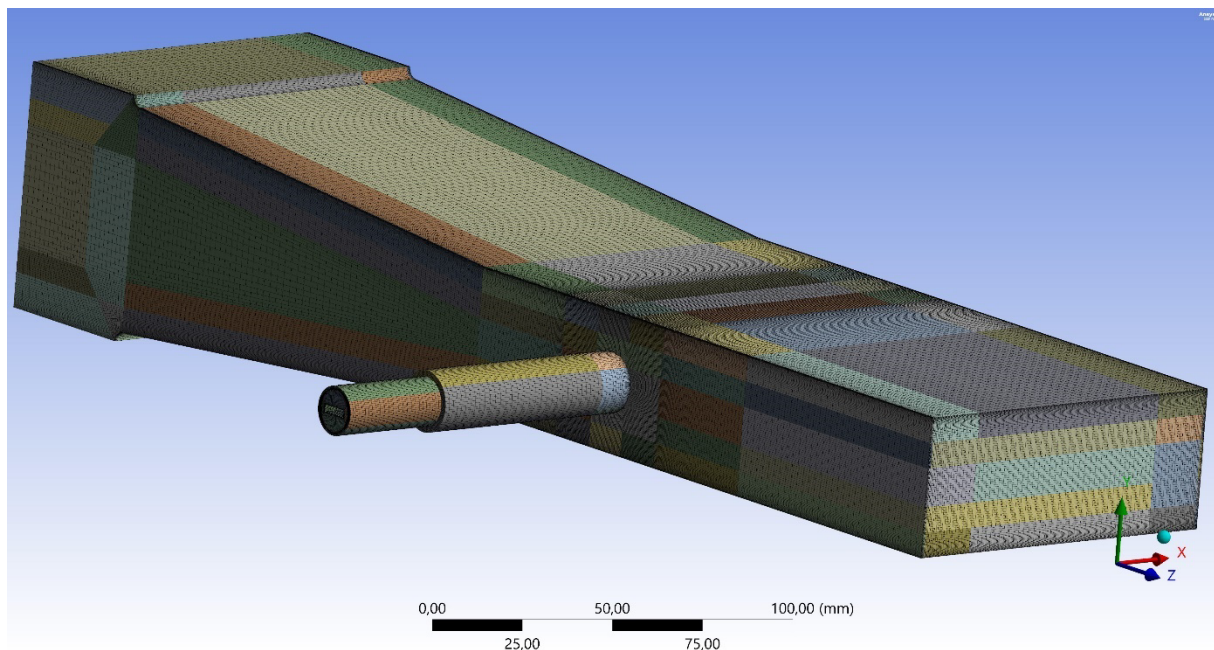


Figure 52: Mesh with 8 million elements.

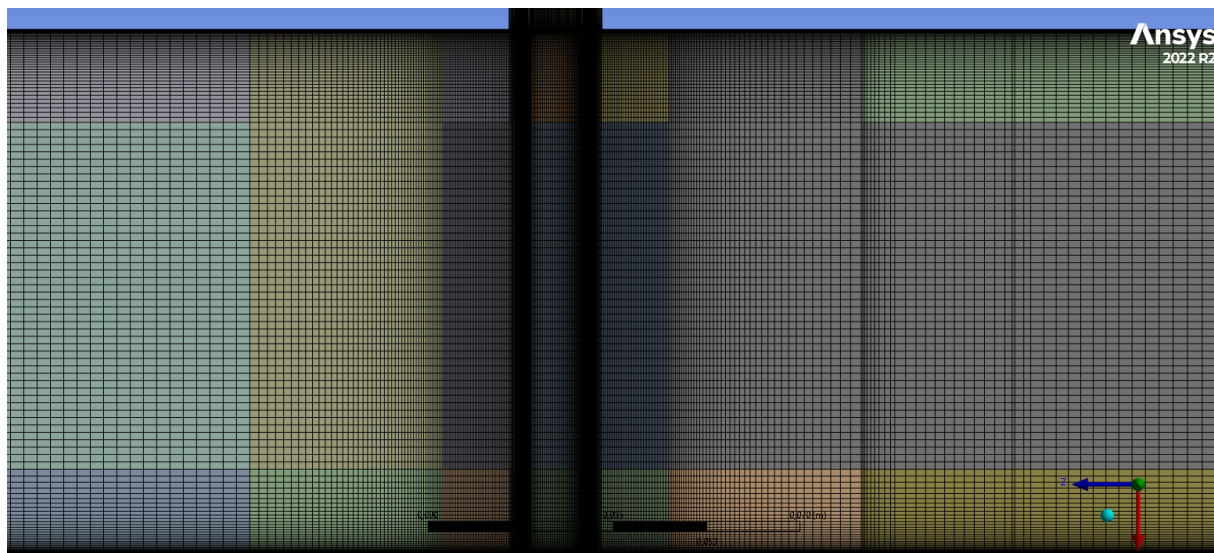


Figure 53: Mesh with 8 million elements (longitudinal view).

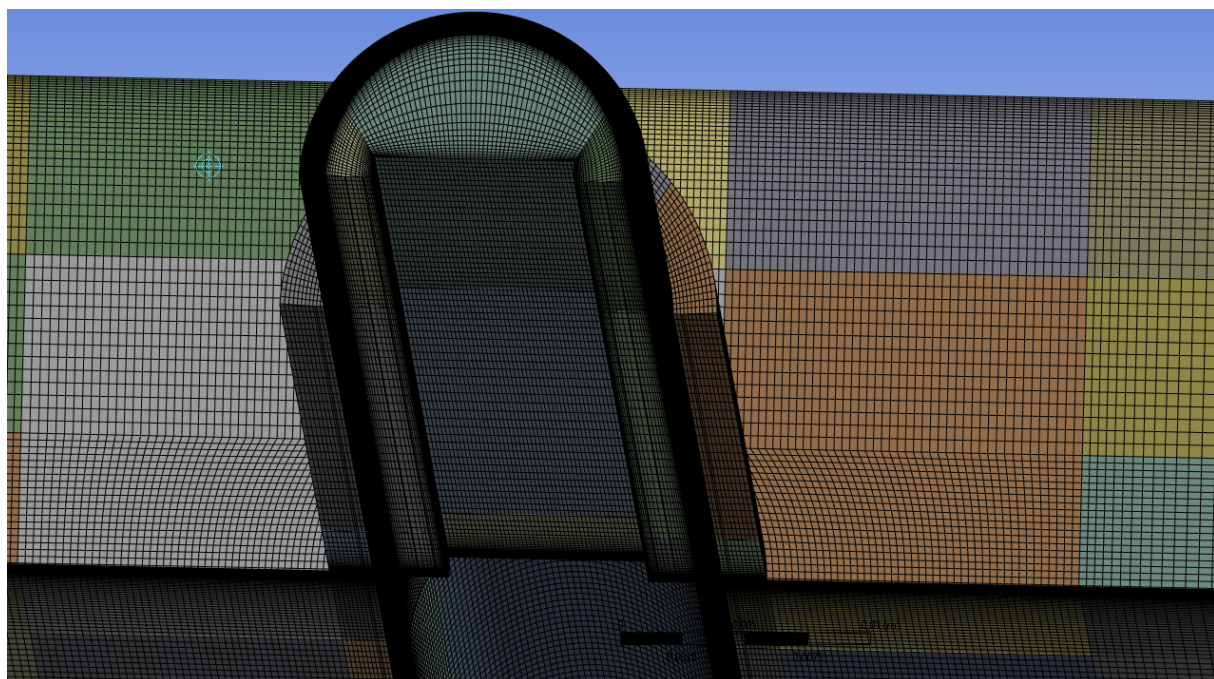


Figure 54: Mesh with 8 million elements (longitudinal view).

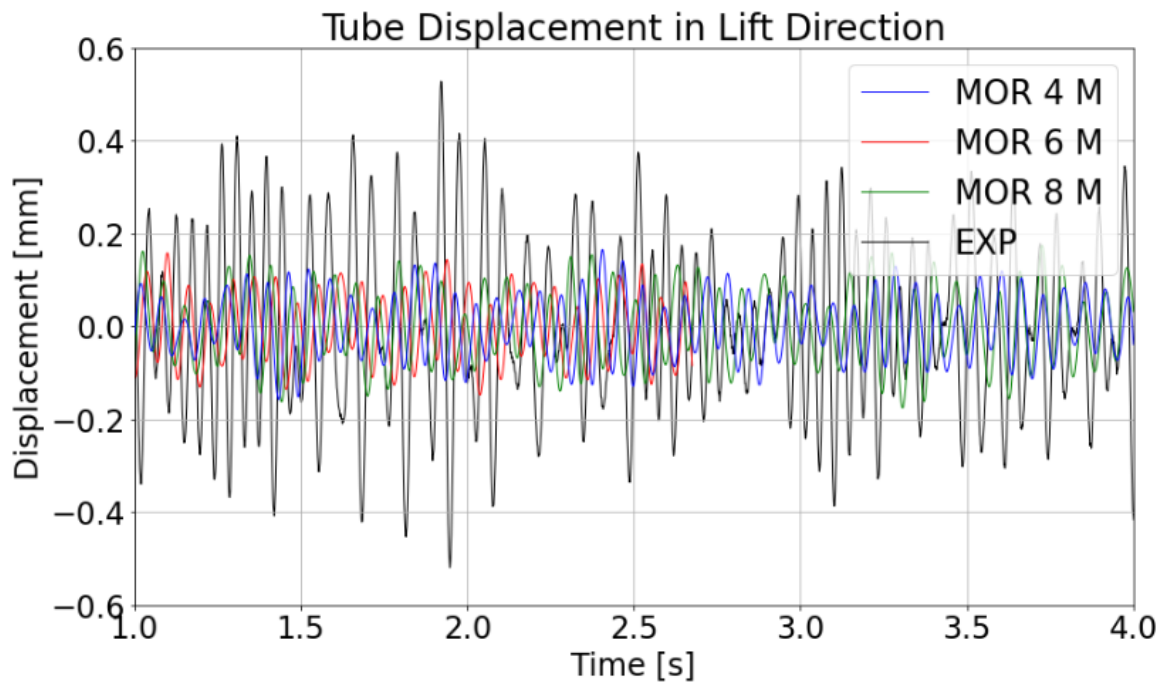


Figure 55: Tube displacements in lift direction.

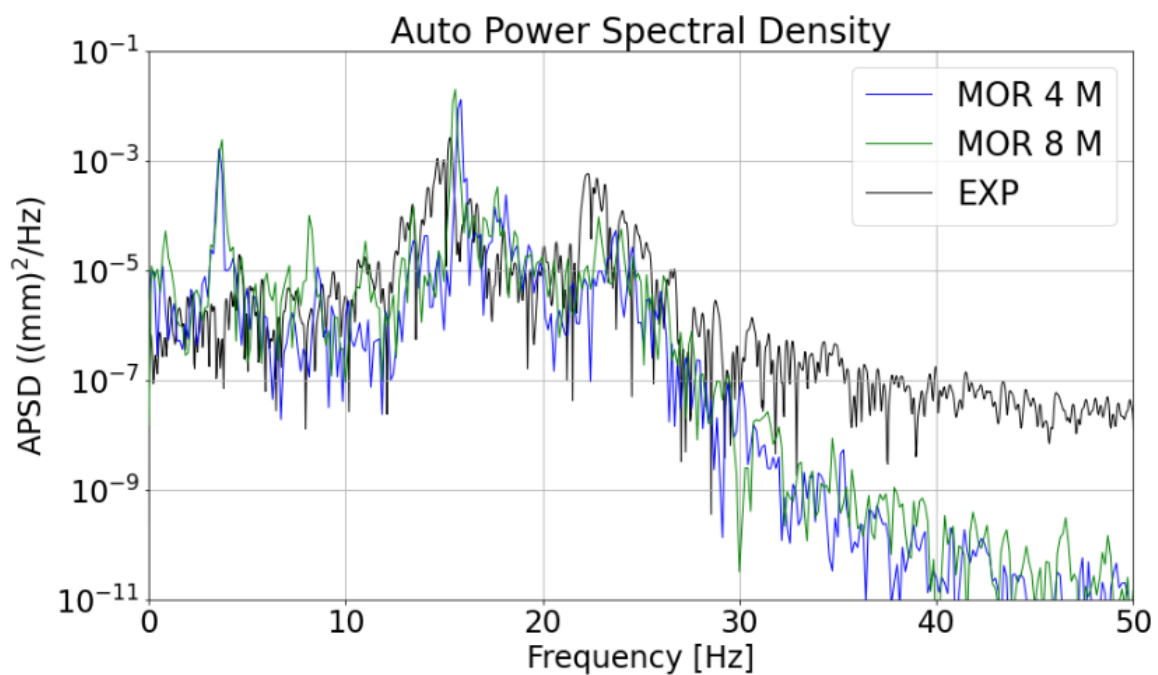


Figure 56: APSD of the tube displacements in lift direction.

It was further decided to investigate the turbulence modelling on the tube vibration behavior. The good experience (Papukchiev, FSI analysis of flow-induced vibrations in a BWR instrumentation tube experiment, 2022), (OECD/NEA, The FSI Benchmark based on OKBM experiments to validate coupled CFD and CSD calculations, 2023) so far with the

ZLES turbulence approach in ANSYS CFX (Menter, 2015) for FIV analyses was the motivation for the further improvement of the results. ZLES is a typical hybrid turbulence approach that combines URANS and LES in a single simulation. The advantage of the hybrid model over the pure LES is the reduced computational effort. The idea behind ZLES is to predefine different zones with different treatments of turbulence in the pre-processing stage. The domain is split into a URANS and LES zones before the simulation. Between the different regions, the turbulence model is switched from URANS (SST in the present analysis) to LES/WMLES (wall-modeled LES). In order to maintain consistency, synthetic turbulence is generated at the LES-URANS interfaces. ZLES is actually not a new model, but an infrastructure that combines existing elements of technology in a zonal fashion. The recommendations for each zone are therefore the same as those applicable to the individual models (Menter, 2015).

Figure 57 shows the separation of the flow domain in pre-defined LES (red) and URANS (blue) zones for the needs of the ZLES turbulence approach. This has to be specified by the user in the ANSYS CFX input using CEL. For the current case, the LES zone was specified approx. one tube diameter before and after the vibrating tube. At the interface between the URANS and the LES zone, the activation of the ZLES model automatically applies a harmonic turbulence generator (Menter, 2015). During a GO-VIKING Progress Meeting, an important question arose: may there be an influence of the synthetic turbulence generator at the URANS/LES interface upstream the tube on its vibration response? To answer this question, additional comparative calculations with significantly enlarged LES zone were performed. They showed very similar results to the ones produced so far - the influence of the position of the URANS/LES in this particular case was negligible.

The mesh used in the next ZLES simulations was further refined from 8.1 to 8.5 million elements and these extra 0.4 million elements were positioned in the LES zone. Further, the time steps size was reduced from 2.0×10^{-4} s down to 1.0×10^{-4} s in order to lower the mean CFL down to 0.6 ($CFL_{max} \sim 12$).

Since the hybrid turbulence model uses LES in the tube region, the resulting y^+ value on the FSI interface was investigated. This is shown in Figure 58. The y^+ value varies between 1 and 3.7 on the tube surface. Significantly higher values up to 17 appear on the top and bottom surfaces of the tube. These high values result from the large flow acceleration in the 1 mm gaps between both tube ends and the channel walls.

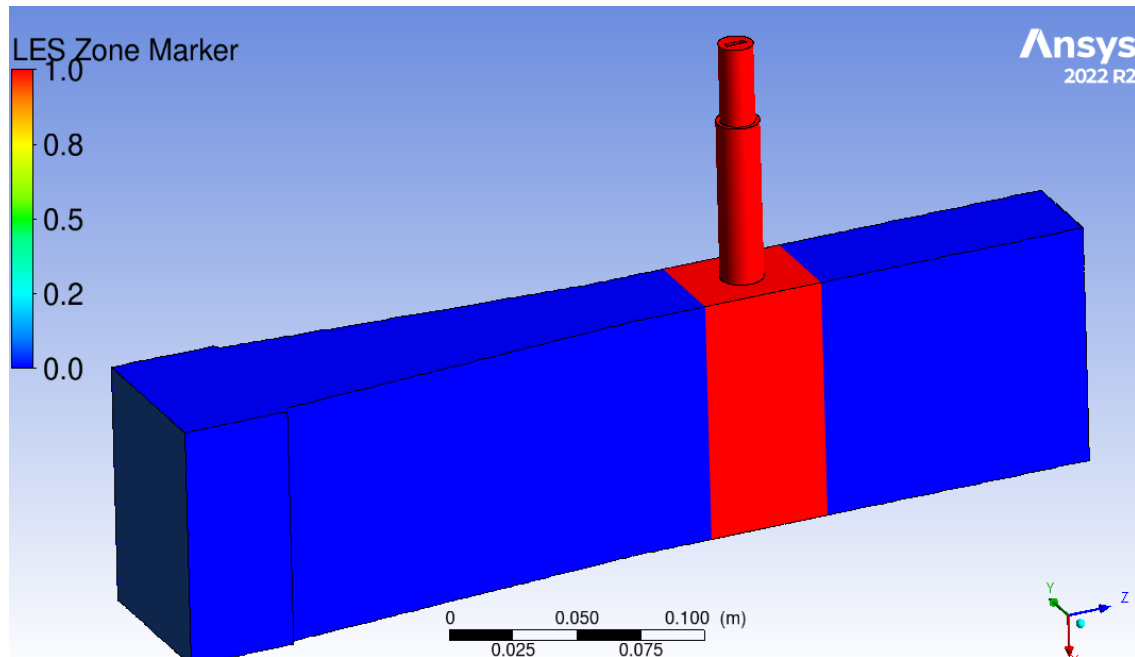


Figure 57: Separation of the flow domain in pre-defined LES (red) and URANS (blue) zones for the needs of the ZLES turbulence approach.

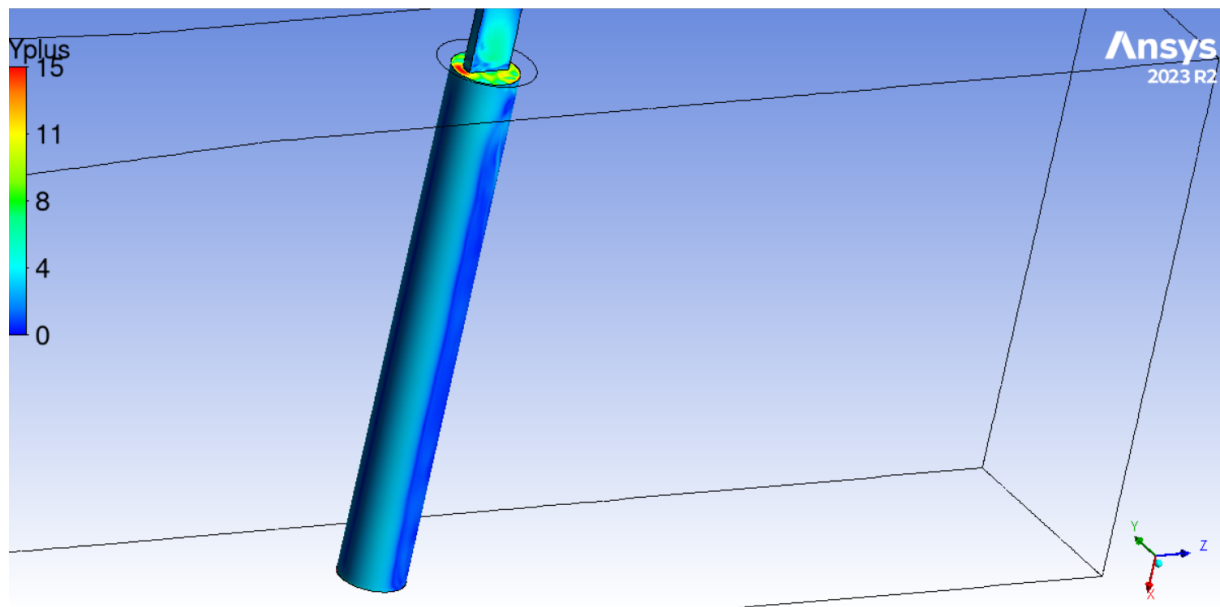


Figure 58: Y+ values on the FSI surface for the single tube case with $Q=2.9$ l/s at the inlet of the test section.

Figure 59 presents the velocity magnitude distribution in longitudinal direction, calculated with ZLES. As expected, the ZLES model resolves more turbulence structures behind the cylinder compared to the SST results in Figure 47. Free stream velocity of 0.74 m/s is found upstream the vibrating AMOVI tube. As observed before, the flow accelerates around the cylinder up to 1.3 m/s and the highest velocity is again found in the gaps between tube ends and channel wall. Negative streamwise velocity down to -1.1 m/s appears behind the

cylinder with recirculation zones, which is typical flow pattern for cylinders exposed to a cross-flow. Also in this simulation, a mass transfer between main and side (blade) channels takes place. The complex flow structure is also visualized with velocity colored streamlines in Figure 60. Figure 61 shows the turbulence kinetic energy in the domain with the vortex-shedding from both sides of the vibrating tube.

Figure 62 presents the tube displacement in lift direction with the SST and the ZLES turbulence models. It is obvious, that the vibration amplitudes, calculated with the more sophisticated hybrid model, are larger, and hence, closer to the experimental value. The RMS displacement of the tube bottom increases by ~80% from 0.078 to 0.14 mm. Although it is still smaller than the measured value, the improvement is noticeable. Concerning the frequency domain (Figure 63), the SST (15.5 Hz) is negligibly closer to the measured value (15.0 Hz) than the ZLES prediction (16.0 Hz). The predicted reduced eigenfrequency by the ZLES is just 0.1 Hz higher than the one predicted by the SST model: 23.3 Hz (exp), 23.9 Hz (MOR), 24.0 Hz (ZLES MOR).

It is important to mention that in a last step, the same numerical setup and mesh were used, but just the turbulence model was changed from ZLES to SST. As expected, the results changed back to the ones observed before with the SST, which proved that the significant improvement in these analyses indeed resulted from the more sophisticated hybrid turbulence approach and not from the smaller time step or a slightly finer mesh.

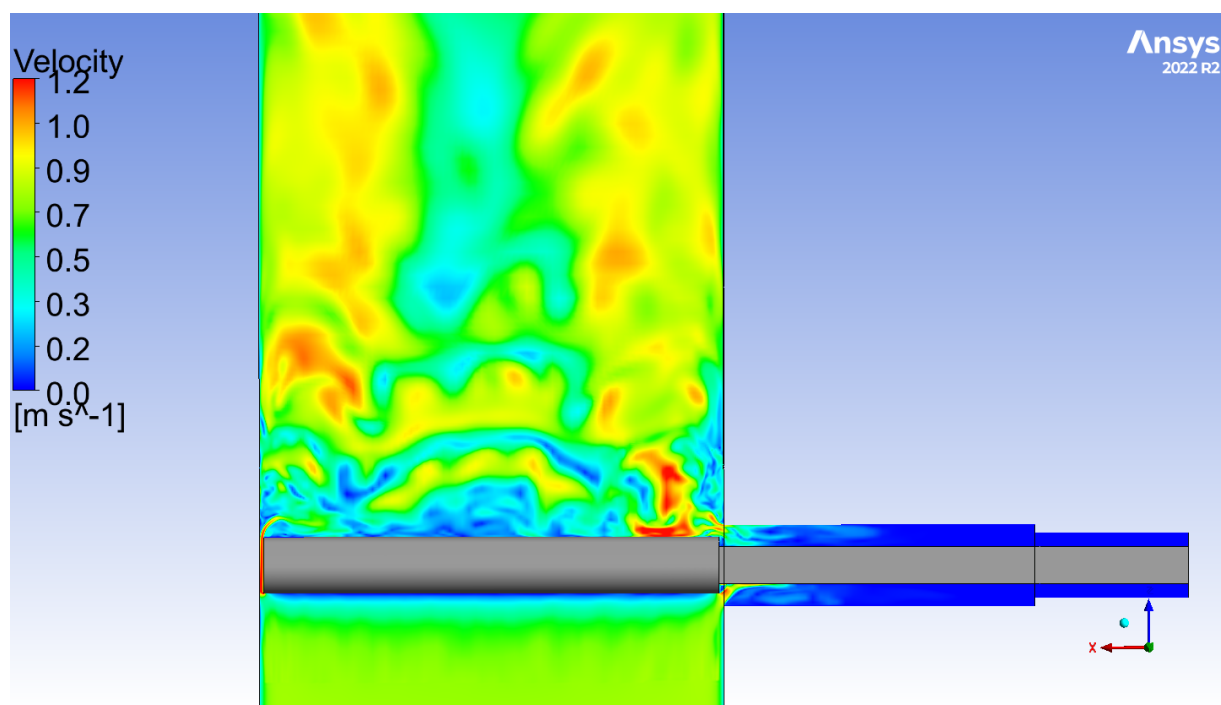


Figure 59: Velocity magnitude in longitudinal direction, calculated with ZLES.

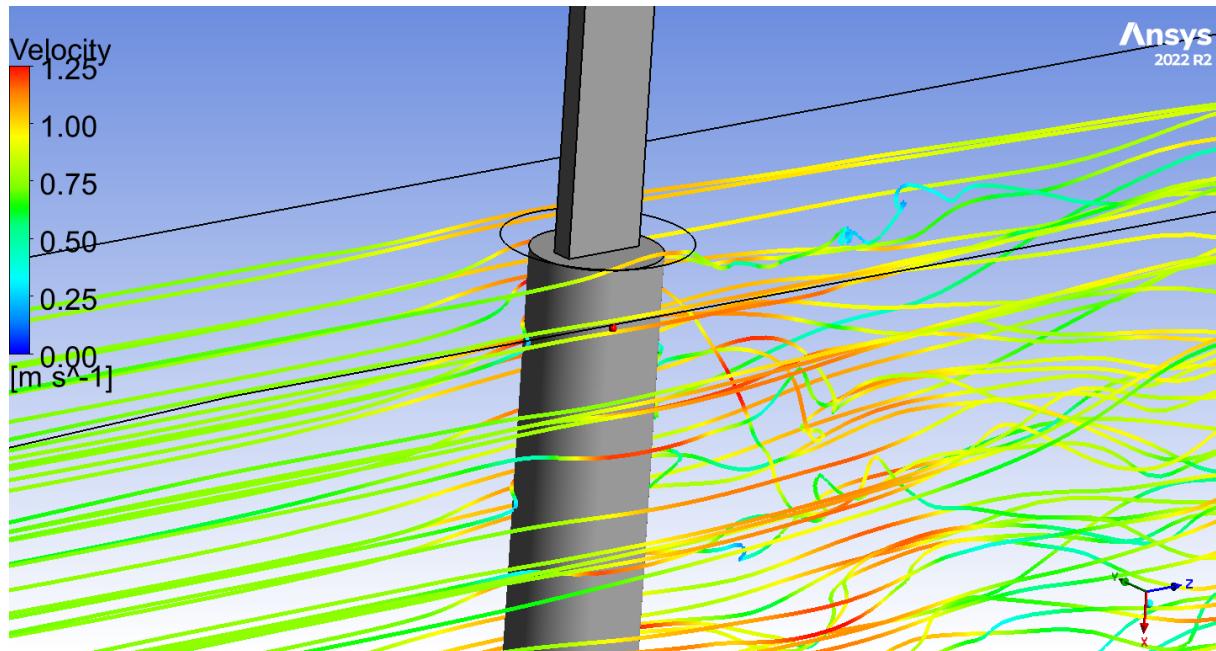


Figure 60: Velocity-colored streamlines in the vicinity of the vibrating tube.

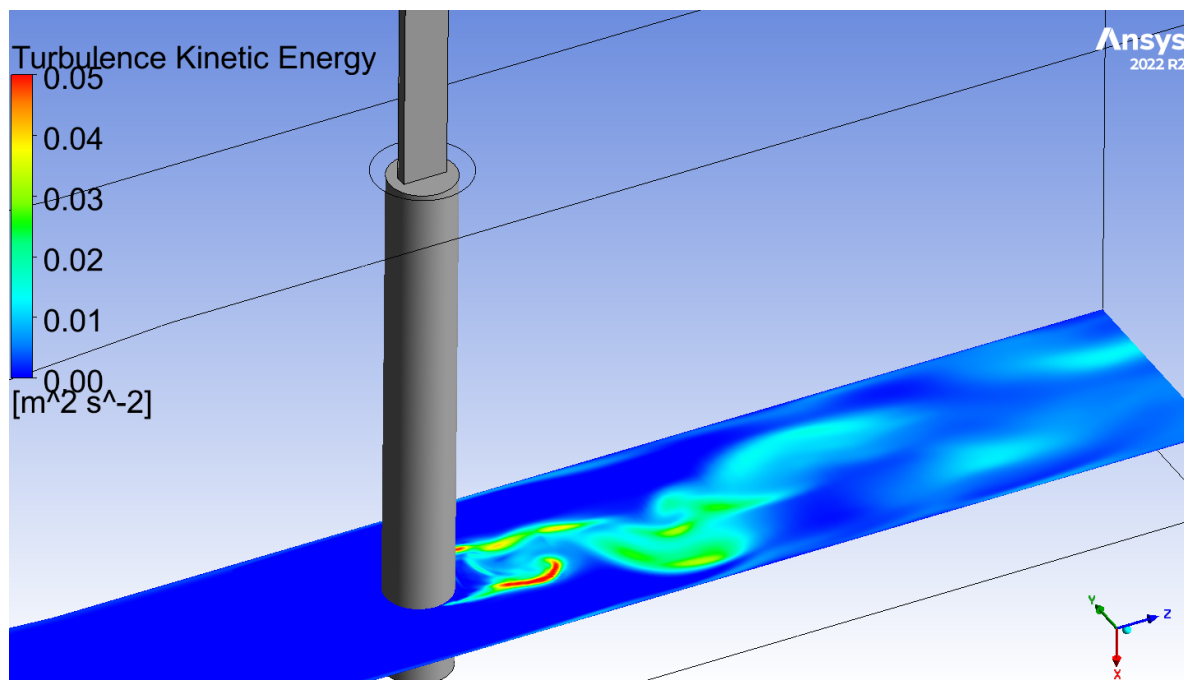


Figure 61: Turbulence kinetic energy in a horizontal cross-section close to the end of the vibrating tube (large displacement).

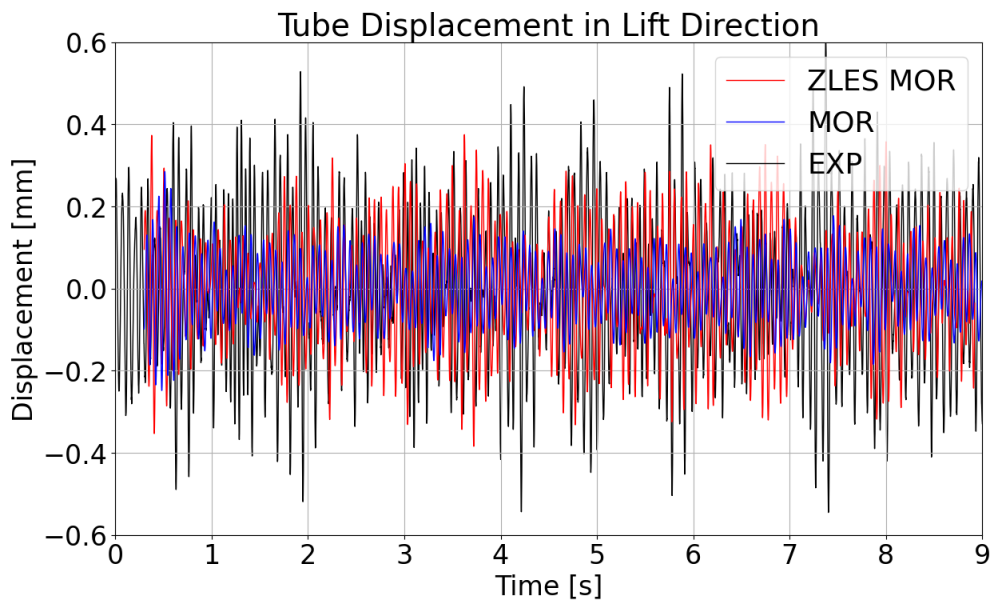


Figure 62: Tube displacement in lift direction with the SST (MOR) and the ZLES (ZLES MOR) turbulence models.

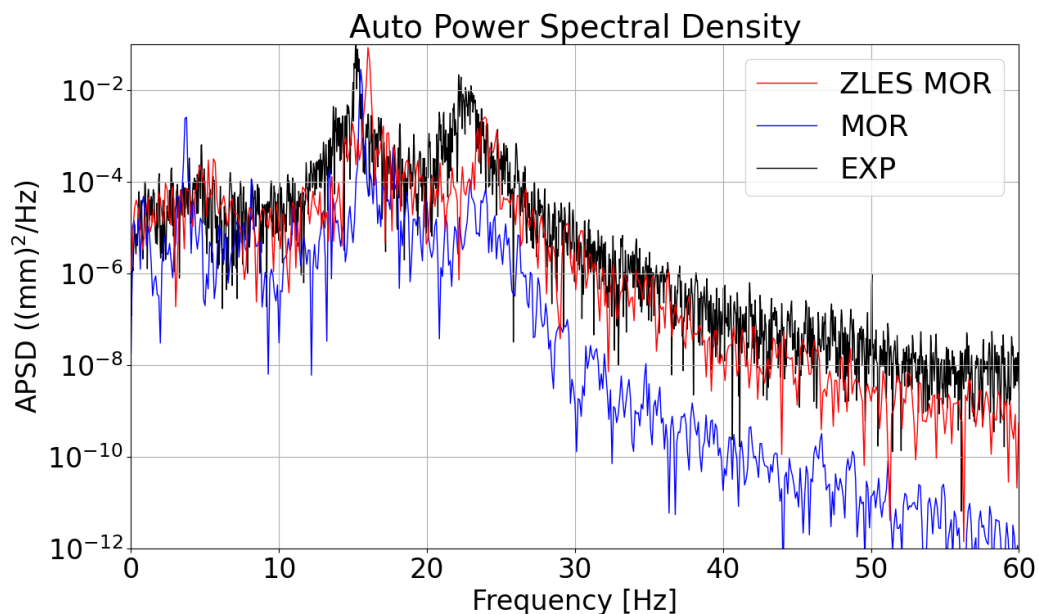


Figure 63: Auto power spectral density of the tube displacements calculated with the SST (MOR) and the ZLES (ZLES MOR) turbulence models.

2.6.4 Further investigations on the number of modes considered in the MOR approach

The presented analyses were performed with a MOR model that accounts for the first six vibrating modes of the AMOVI tube and its blade. The analyses so far proved, as expected, that the tube vibrates primarily in its lift direction. This corresponds to the first mode shape,

as shown in Figure 64. It was interesting to investigate the importance of the higher modes on the vibration response of the structure. Therefore, the input data for the MOR approach for modes 2-6 was removed from the MOR algorithm and only mode 1 was considered. Then the simulation was repeated with only one mode considered. The results for the tube displacements and the frequency response are exposed in Figure 64 and Figure 65. The results look very similar, which leads to the conclusion that indeed mode 1 is the most influential for the vibration amplitudes and frequencies of the AMOVI tube and its blade, while the higher modes play secondary role. Since the MOR approach is fast-running and the main CPU time overhead is generated by the CFD calculation, omitting the higher modes in the FSI simulation would not bring remarkable increase in the overall efficiency of the run. Therefore, the next bundle simulations were also performed with the first six vibration modes.

Table 12 summarizes all results for the single vibrating tube case with 2.9 l/s flow rate at the test section inlet. The MOR and FOM provide very similar results, proving that the MOR is sufficiently accurate and can be used to substitute the FOM in this case to lower the overall CPU time. ZLES showed superior behavior over SST predicting displacements that are much closer to the measured ones. The results in the frequency domain are accurately predicted by both models. Finally, it was proven that the first bending mode is mostly responsible for the size of tube displacements.

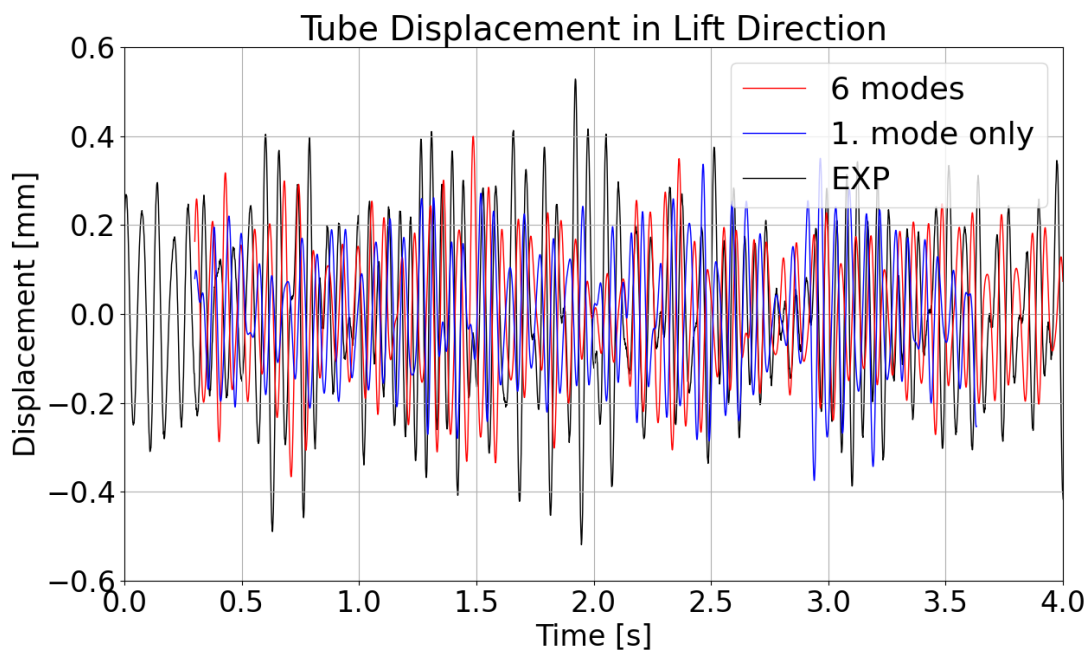


Figure 64: Tube displacements calculated with the consideration of 1 and 6 modes.

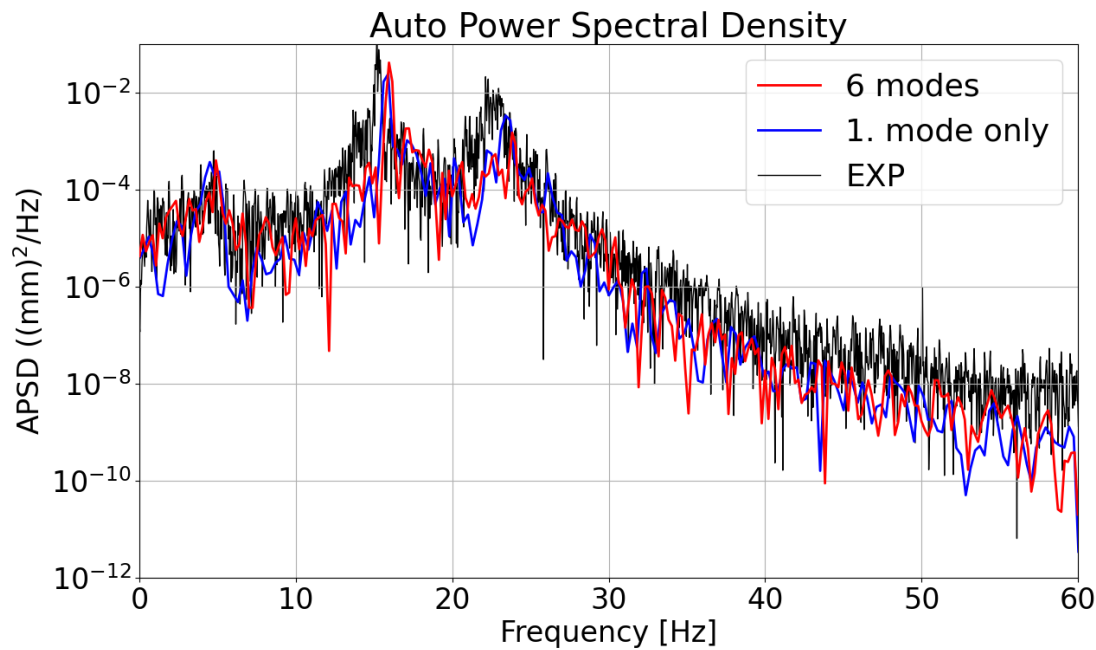


Figure 65: APSD of the tube displacements calculated with the consideration of 1 and 6 modes.

Table 12: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube with 2.9 l/s flow rate

Parameter	MOR SST	SYSC SST	ZLES MOR	ZLES MOR (1. mode only)	Experiment
Displacement RMS, mm	0.064	0.065	0.140	0.133	0.19
Max. displacement, mm	0.16	0.16	0.39	0.39	0.60
Vortex-shedding frequency, Hz	15.7	15.7	16.0	15.9	15.2
Reduced eigenfrequency, Hz	23.7	23.5	24.0	23.8	22.1
Low-frequency peak, Hz	3.8	3.8	5.9	5.8	4.8

2.7 Results for the Q=1.7 l/s experiment

This experiment was simulated only with ANSYS CFX-MOR. The same setup as in the previous case was used. The only thing being changed is the volumetric flow rate at the test section inlet. The objective is to check the predictive capabilities of the FSI code with the same input deck.

2.7.1 Flow pattern

Figure 66 shows the calculated instantaneous velocity with MOR at $t = 3.6$ s. The mean free stream velocity in the channel is 0.44 m/s (water flows bottom-top). As in the previous case, the flow behind the cylinder is decelerated, while the water is accelerated around the tube. The maximum velocity of 0.8 m/s is observed in the 1 mm gap between the left tube end and the channel wall. The flow in the right part of the blade channel, close to the blade fixation point is stagnant. Again, there is mass exchange between both channels: some water leaves the main channel and then enters through a small gap into the blade channel.

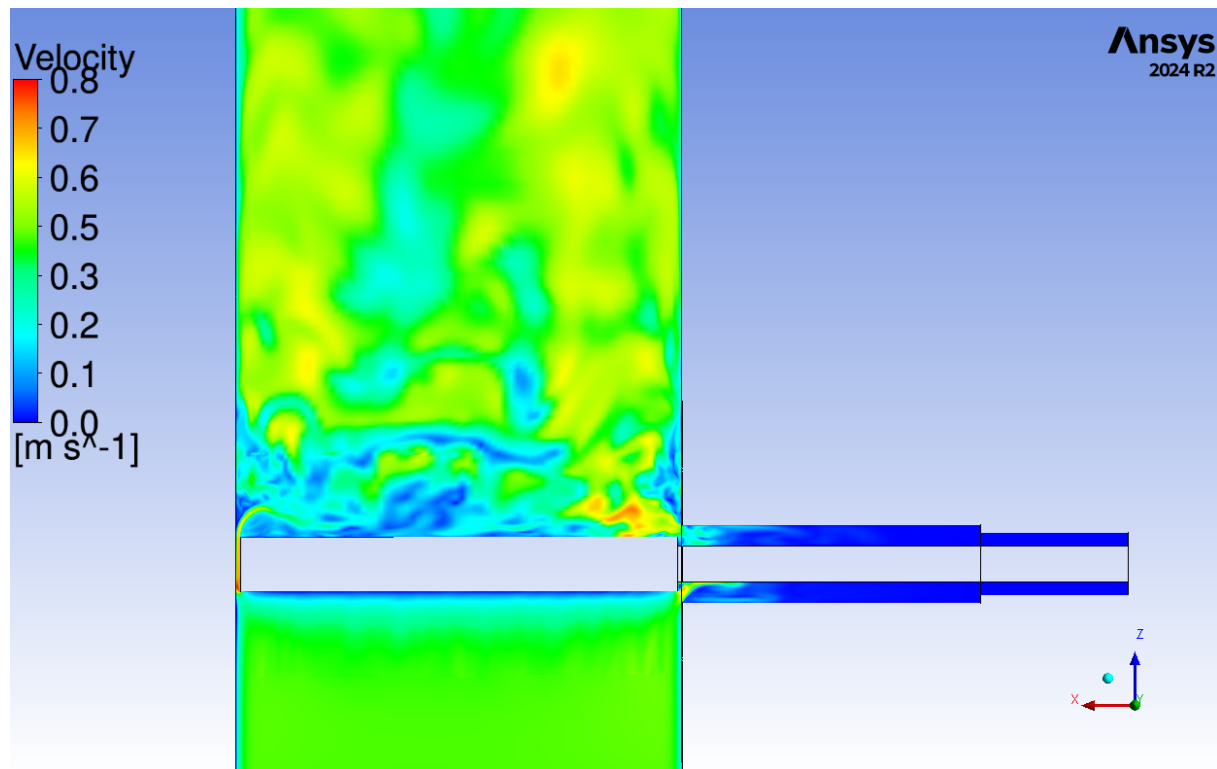


Figure 66: Velocity distribution in a vertical plane through the middle of the test section ($Q=1.7$ l/s).

2.7.2 Vibration analysis

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 4 s of simulation time. The first simulation second was omitted. This data acquisition times provides sufficient resolution for the FFT analysis. Experimental data of 30 s was also used for the comparison.

Figure 67 presents the calculated and the experimental tube displacements. The root mean square (RMS) value of the time-dependent displacement was calculated for the available simulation results and the experiment. ANSYS CFX-MOR predicts an RMS value of 0.024 mm, while the experiment suggests 0.028 mm. The agreement between simulation and data is very

good. The maximum displacement of the tube in lift direction from its normal position is calculated by MOR and SYSC to be 0.061 mm, while the measured value is 0.087 mm.

The APSD was calculated from the displacements using Python scripts. The numerical function `matplotlib.mlab` with Welch's average periodogram was used. The sampling frequency (number of samples per second) was 5000 Hz, the sampling frequency of the experimental data was 1000 Hz. In the FFT analysis, a Hanning windowing was applied. A good agreement between simulation and experiment is observed: MOR predicts $f_{1,s} = 9.1$ Hz for the vortex-shedding frequency, which is very close to the measured one of 9.4 Hz. The second frequency peak (23.5 Hz) in the experiment results from the reduced first eigenfrequency of the vibrating tube and MOR predicts 24.5 Hz. Further, the low frequency peak is visible also in this case, but now it is lower (2.5 Hz). All important results are summarized in Table 13.

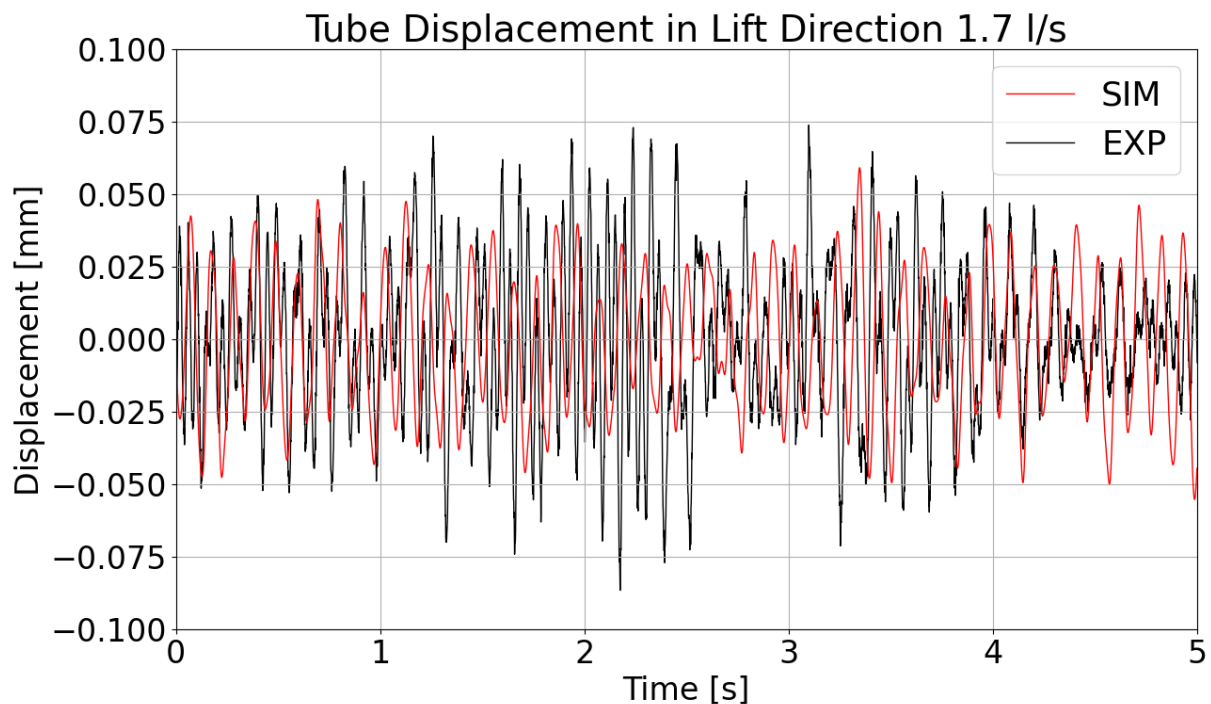


Figure 67: Tube displacement in lift direction with the ZLES turbulence model with $Q=1.7$ l/s at the test section inlet.

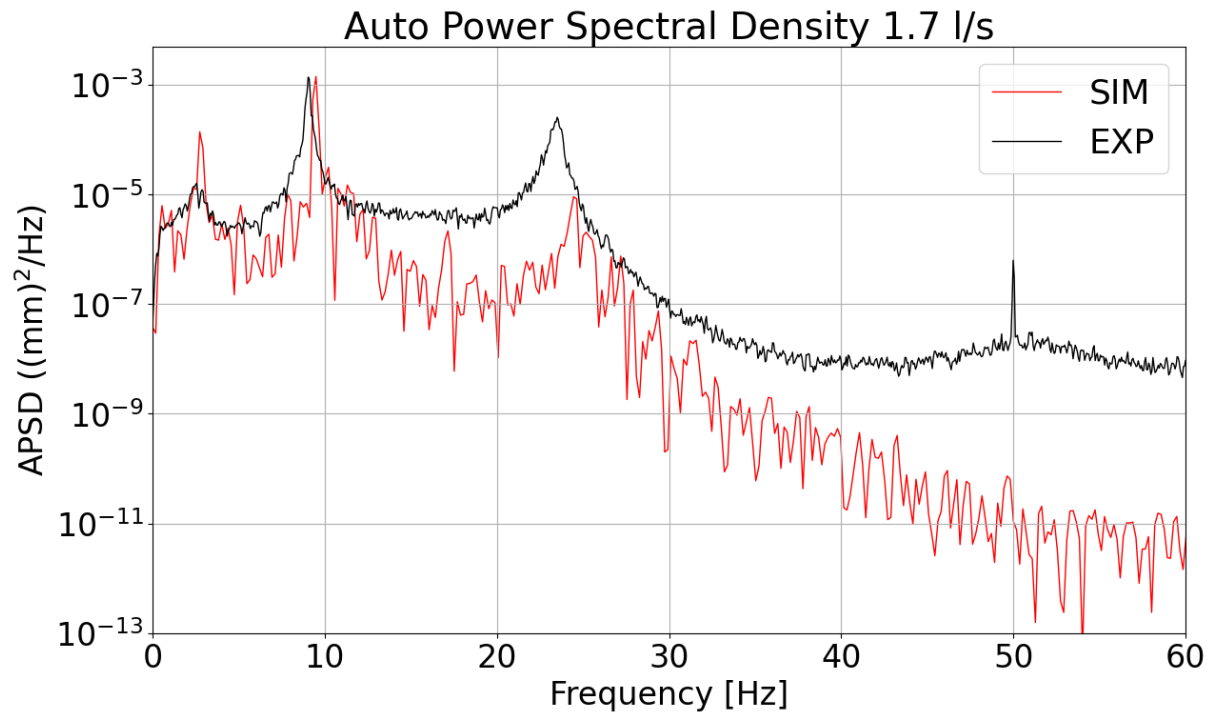


Figure 68: Auto power spectral density of the tube displacements ($Q=1.7$ l/s).

Table 13: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube with 1.7 l/s flow rate

Parameter	ZLES MOR	Exp.
Displacement RMS, mm	0.024	0.028
Max. displacement, mm	0.061	0.087
Vortex-shedding frequency, Hz	9.1	9.4
Reduced eigenfrequency, Hz	23.5	24.5
Low-frequency peak, Hz	2.5	2.5

2.8 Validation of ANSYS CFX-MOR on AMOVI single flexible tube in a rigid tube bundle experiment

The analyses presented here are based on AMOVI test case with single flexible tube in a rigid tube bundle exposed to a volumetric flow rate of 1.6 l/s and 1.95 l/s. For the first one, PIV measurements were provided by CEA and are used here to compare with the experimental data. The second case is very interesting, since it features very large displacements as a result of lock-in or fluid-elastic instability (FEI) phenomenon.

2.8.1 Mesh generation

CAD model of the test section channel with the tube bundle was generated within GO-VIKING WP3 (see also GO-VIKING's D3.2 deliverable) with SpaceClaim from the CEA drawings. The CAD model has been sliced in several sub parts in ANSYS Meshing and a multizone meshing method was applied. Several hexahedral meshes for the test section with tube bundle were generated. The final mesh used in the present ANSYS CFX – MOR simulations with the ZLES model for the tube bundle consisted of 19.6 million elements (Figure 69). The overall mesh quality is good, although for some elements the aspect ratio reaches 135, and this is beyond the best practice suggestions. Among the most important mesh quality parameters are the min. orthogonality angle (13°) and the expansion factor (88). These are outside the limits suggested in (OECD/NEA, Best Practice Guidelines for the use of CFD in Nuclear Reactor Safety Applications - 2024 Update, 2025) for less than 1% of the elements. Furthermore, the CFD code converged within just a few iterations, proving the high overall mesh quality. Mesh independence studies proved that the CFD results are more or less mesh-independent (negligible differences in the evaluated fields).

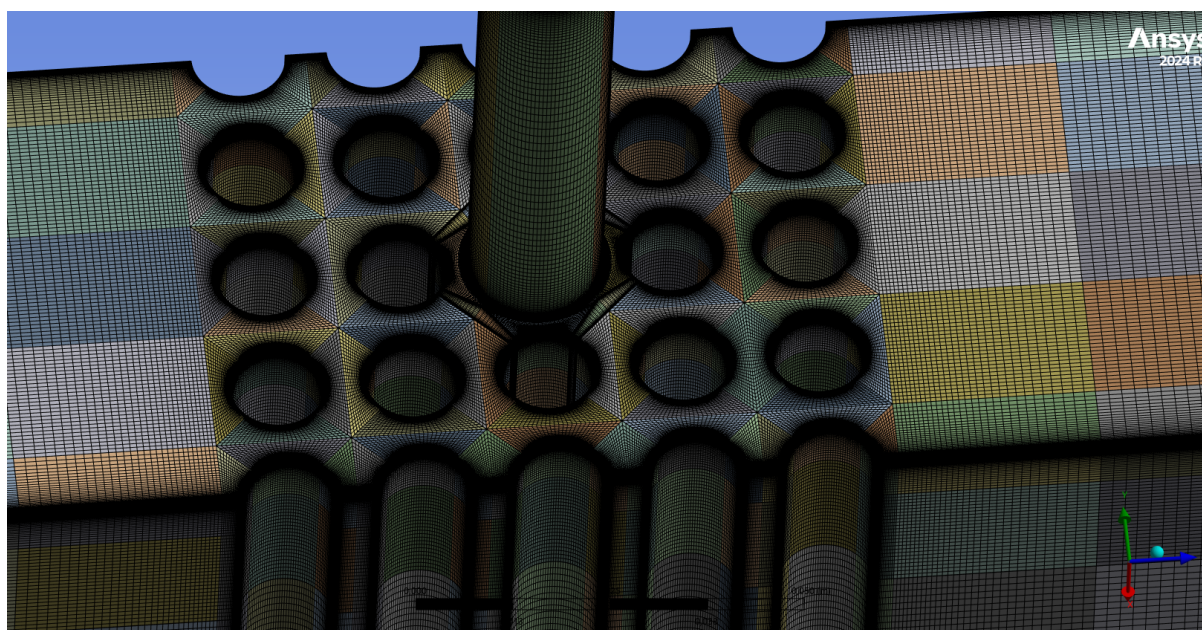


Figure 69: Numerical mesh of the tube bundle with the flexible tube in its center.

2.8.2 Simulation of the flexible tube in a rigid tube bundle experiment with 1.6 l/s inlet flow

2.8.2.1 Simulation setup and boundary conditions

IAPWS IF97 water material data were used from the ANSYS CFX material libraries. The CFD simulations with ANSYS CFX were isothermal, because this AMOVI experiment was performed at constant 17°C fluid temperature. For the present simulation case, 1.6 l/s flow rate and 5 % turbulence intensity were specified at the inlet of the test section inlet

chamber. Since no information on the inlet flow profile was available, a flat velocity profile was imposed. In the current CFD model, this is located 16 tube diameters upstream the vibrating tube, which allows the flow to develop before it reaches the tube surface. Standard gravity is applied in $-z$ direction and reference pressure is set to 1.07 bar. At the outlet patch, the pressure is kept constant.

Since the previous single tube analyses revealed the superior behavior of the ZLES turbulence model, it was selected also for these FSI simulations. The AMOVI domain was split again into a URANS and LES zones before the simulation: the bundle is simulated with LES, while the rest of the computational domain is treated with SST (see Figure 70). In order to maintain consistency, synthetic turbulence is generated at the LES-URANS interfaces. Figure 71 presents the y^+ values on the tube walls being relevant for the LES region. The highest local values reach $y^+ \sim 4$, while for the rest of the tube walls this value is lower.

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. The time step size used in the analyses was 2.0×10^{-4} s, this resulted in a mean Courant number of 0.45. The convergence of the CFD, CSM and the coupled solutions was assured in every time step.

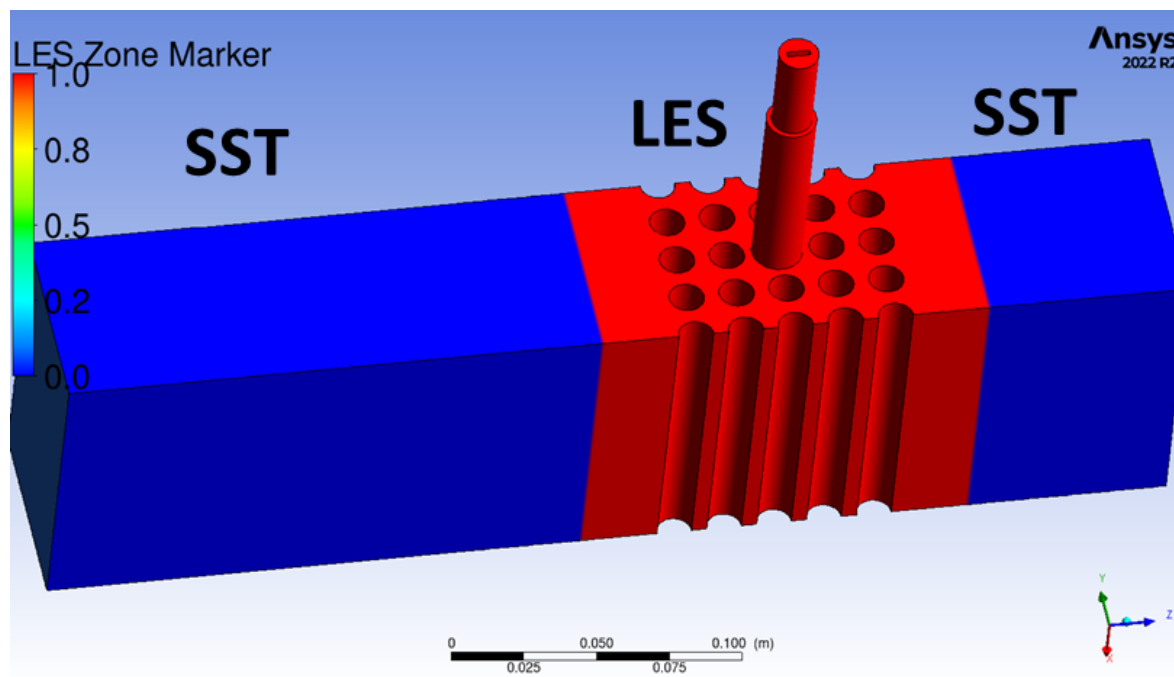


Figure 70: Separation of the AMOVI tube bundle domain in pre-defined LES (red) and URANS (blue) zones for the needs of the ZLES turbulence approach.

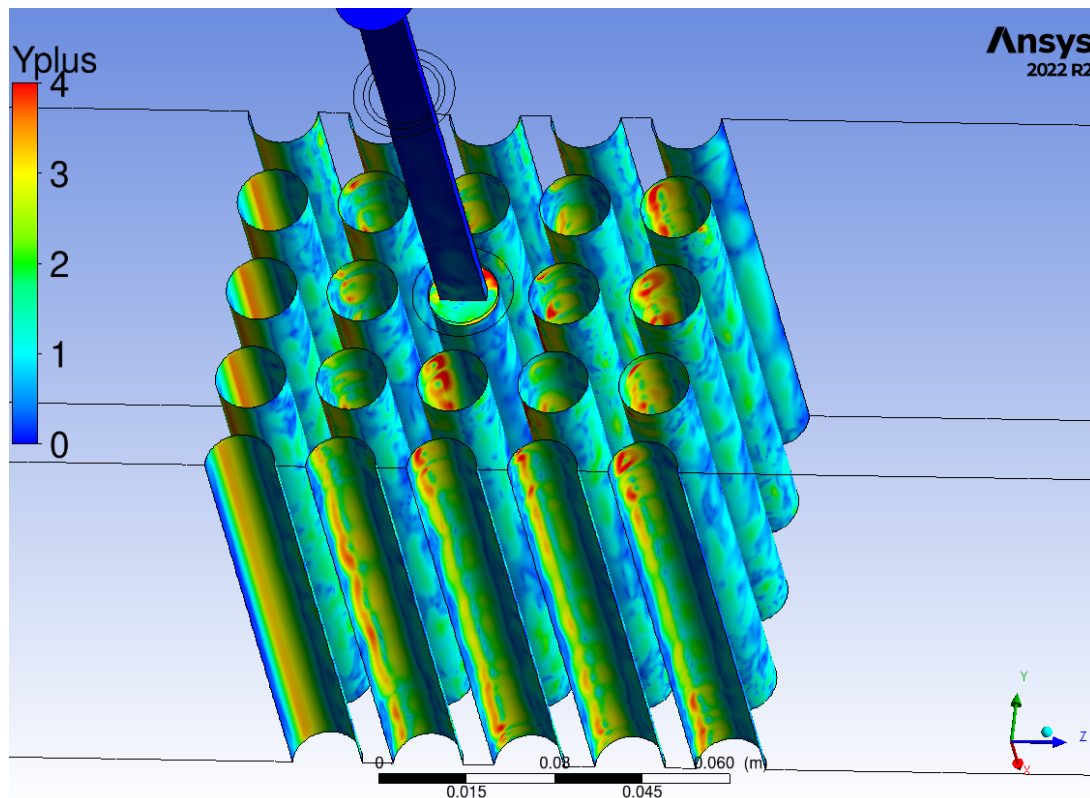


Figure 71: Y+ values on the tube bundle surfaces.

2.8.2.2 Analysis of the Flow Pattern in the AMOVI Test Section

Figure 72 shows the calculated instantaneous velocity distribution in the whole AMOVI test section with ANSYS MOR at $t = 5.5$ s. The turbulent structures, visualized with the Vortex Core Region option in CFX-Post, provide important information about the flow in the channel. It can be seen that in the LES region, smaller turbulence structures are resolved. These can be found in front of the first and behind the last rows of the AMOVI tube bundle, as well as inside the bundle. Figure 73 presents the velocity distribution in the AMOVI test section in a vertical plane, positioned right in the middle of the test channel. Also here, one can recognize the complex flow structure between the tubes with zone with low and high velocities. Figure 74 reveals the velocity distribution in a horizontal plane, positioned in at half tube height. The water flows from left to the right with mean free stream velocity in the channel of 0.24 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 1.1 m/s develop in the AMOVI bundle. These jets are unstable in time and move in flow transverse direction, which drives the vibration of the central, flexible tube primarily in its lift (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. One should keep in mind that Figure 74 represent instantaneous velocity fields, and that 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. These are visualized with the help of velocity-colored vectors in Figure 75 and zoomed in Figure 76. 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. Recirculation zones develop also at the channel walls between the half-tubes. The flow in all of them rotates in counter-clock direction, driven by the high-velocity jets, seen in Figure 74. It is also interesting to observe that sometimes two recirculation zones with swirls rotating in opposite directions develop behind certain tubes. This is seen in Figure 77, which presents the velocity vectors at a different time instance.

In order to compare measured and calculated velocities with the same color map, the experimental data and ANSYS CFX-MOR results were imported in the ParaView software (see Figure 78). The qualitative agreement between experiment and simulation is very good, although the averaging was performed over 7 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-MOR slightly overpredicts the maximal jet velocity and suggests stronger recirculation zones behind the last tube bundle row compared to the experiment.

Figure 73, left, demonstrates that there is mass exchange between the main flow channel and the tube blade channel above it: some water enters the blade channel through a small gap the vertical channel with the flexible tube blade above the bundle, while at the same time, water leaves the blade channel and mixes with the water in the main test section channel behind the vibrating central tube.

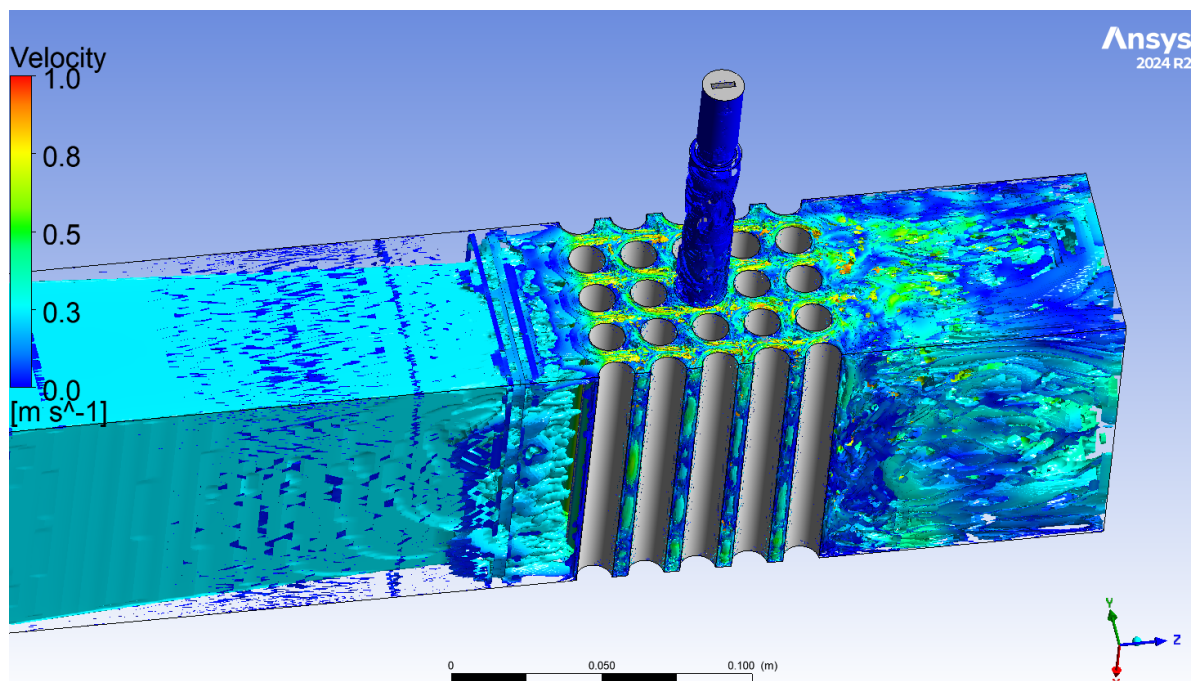


Figure 72: Velocity distribution in the AMOVI test section equipped with a tube bundle ($Q=1.6$ l/s).

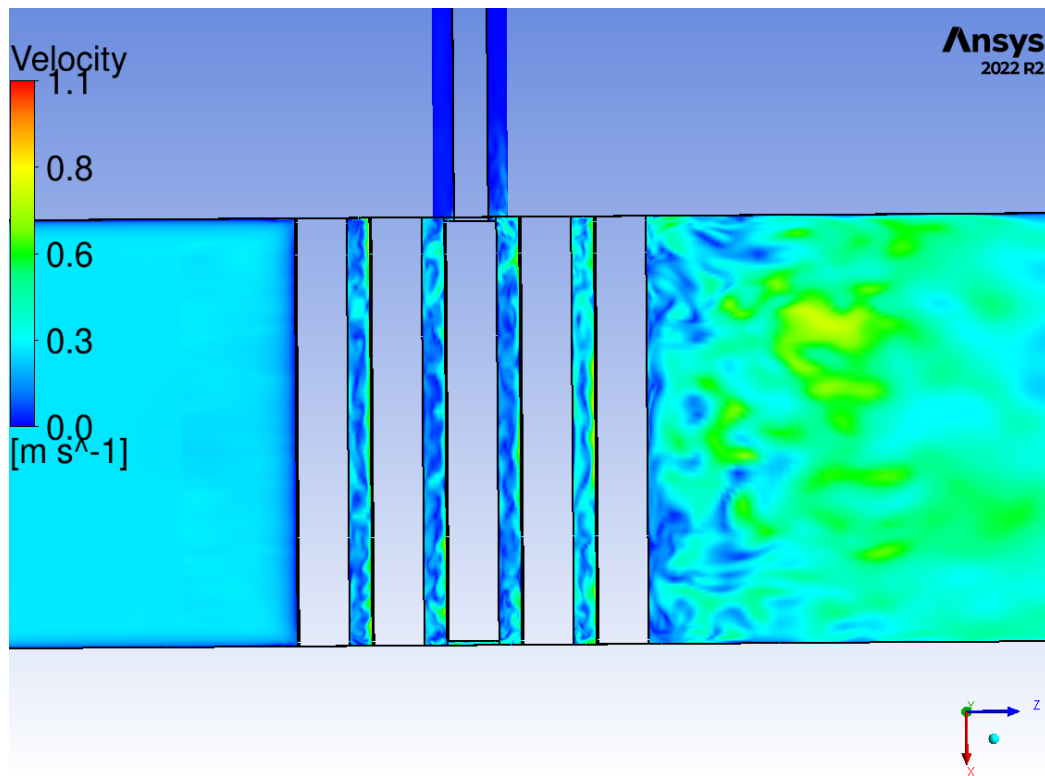


Figure 73: Velocity distribution in the AMOVI test section in a vertical plane ($Q=1.6$ l/s).

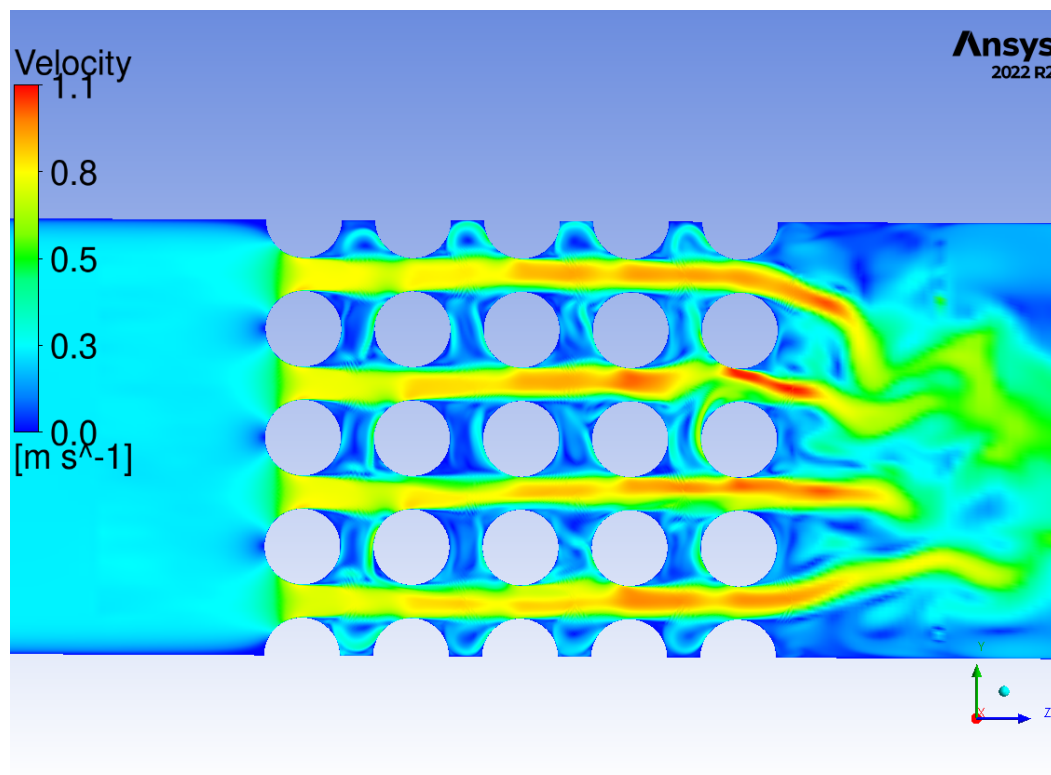


Figure 74: Velocity distribution in the AMOVI test section in a horizontal plane ($Q=1.6$ l/s).

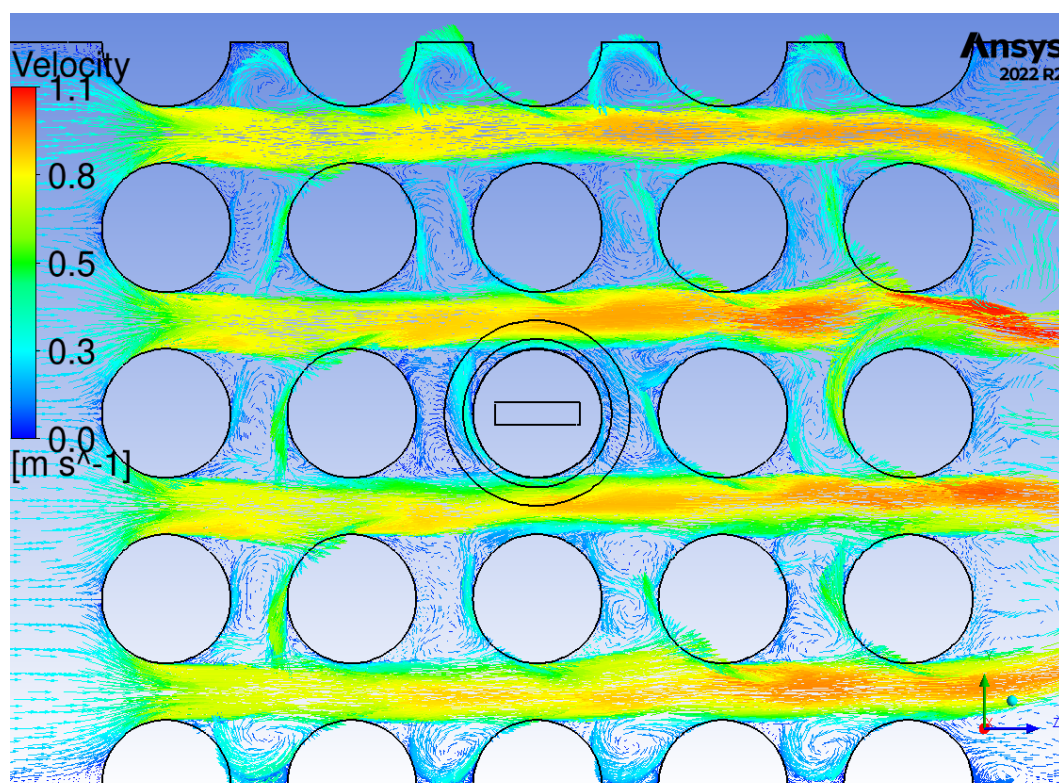


Figure 75: Recirculation zones visualized with velocity-colored vectors ($Q=1.6$ l/s).

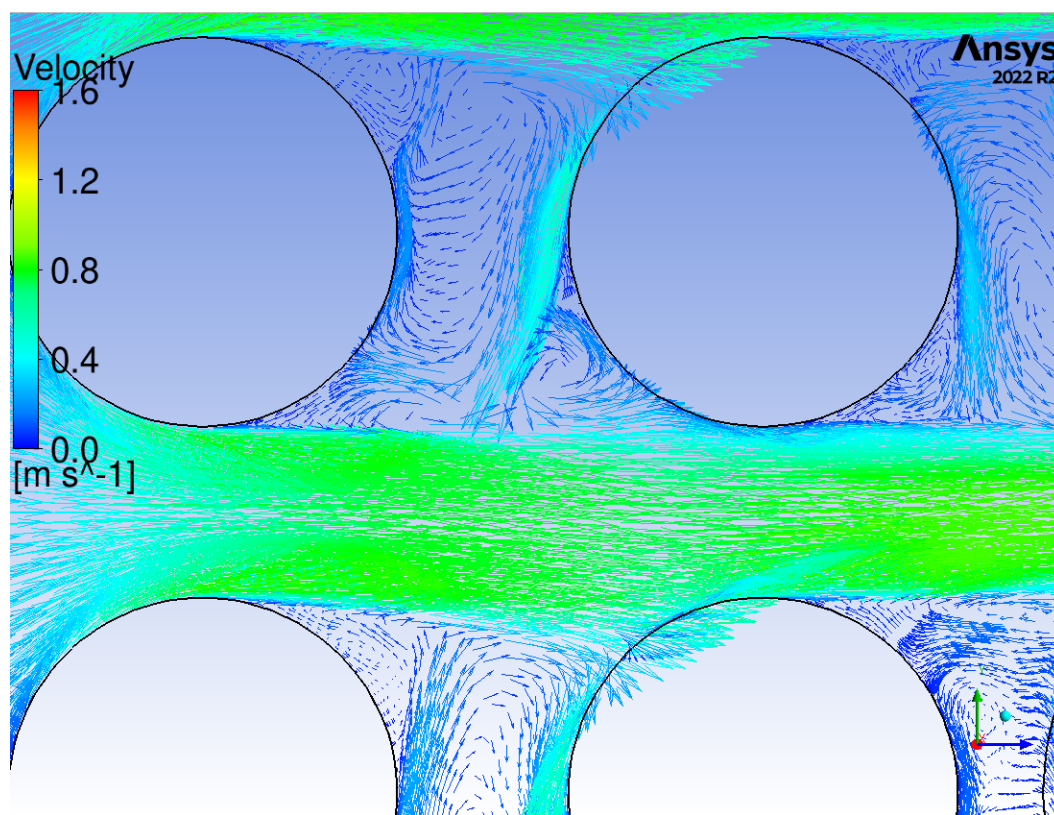


Figure 76: Zoom-in of the top left corner of the picture above ($Q=1.6$ l/s).

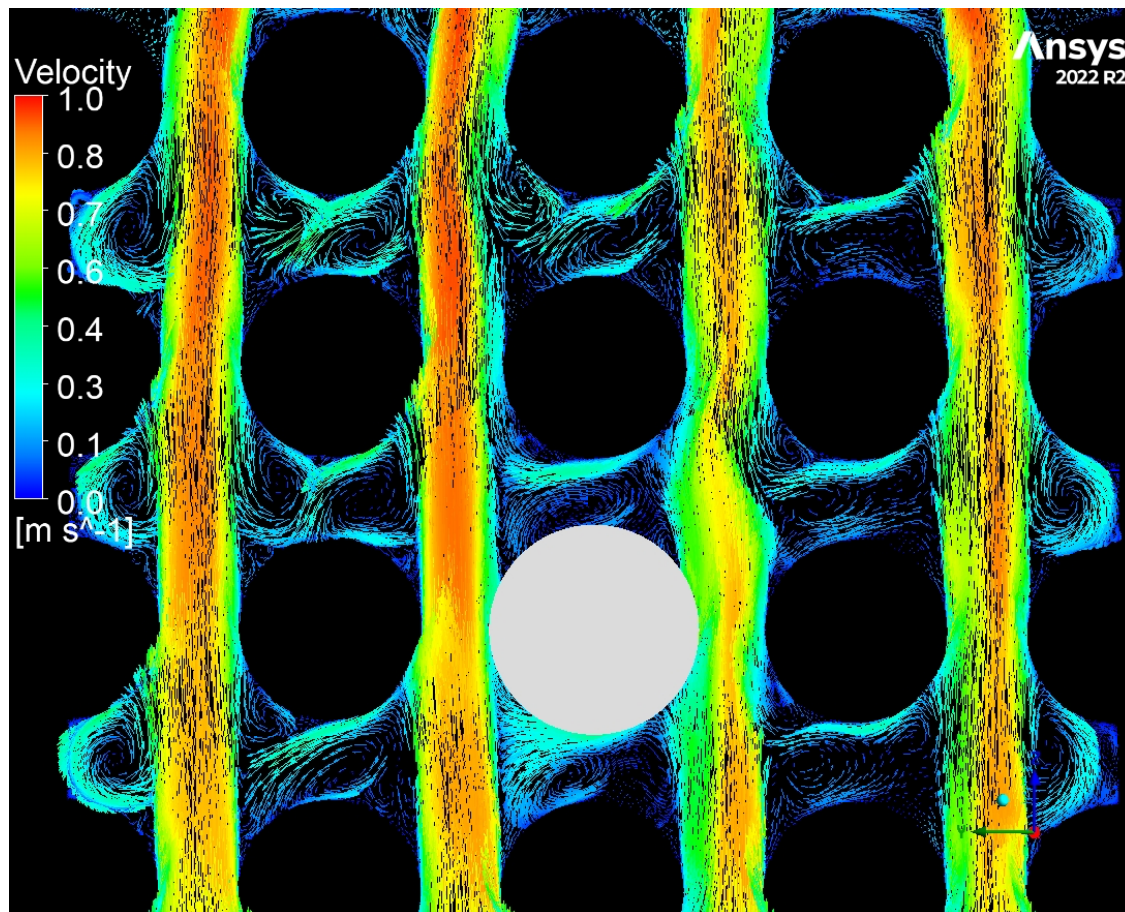


Figure 77: Formation of two recirculation zones behind certain tubes rotating in opposite directions (flow from bottom to top of the picture) ($Q=1.6 \text{ l/s}$).

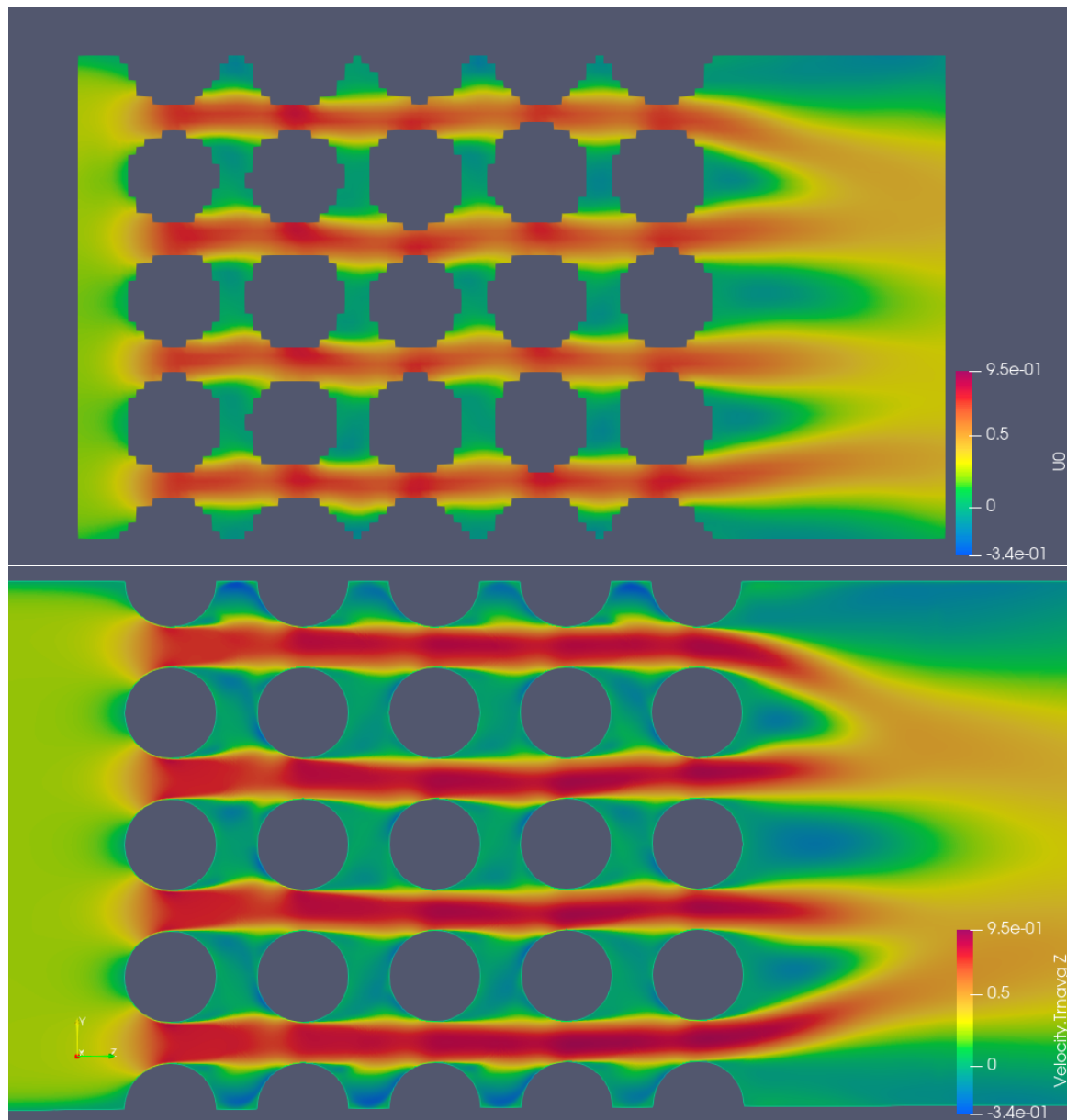


Figure 78: Measured (top) and calculated (bottom) time-averaged streamwise velocity in the tube bundle ($Q=1.6$ l/s).

2.8.2.3 Vibration analysis

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. Figure 79 presents the calculated and the experimental tube displacements over 7 s. Both data vary with time, therefore, the root mean square (RMS) value of the time-dependent displacement was calculated for the available simulation results and the experiment. ANSYS CFX-MOR predicts an RMS value of 0.053 mm and SYSC 0.061 mm for the displacement of the central vibrating tube in the bundle, while the experimental value reaches 0.073 mm. It should be mentioned that the experimental RMS value was calculated over the whole available measurement time of 400 s. Although the FSI simulations

underestimate the measured amplitudes by 27% and 16%, this can be considered as a very good prediction, since it is still in the order of magnitude of the experimental data. 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 0.22 mm (MOR), 0.2 mm (SYSC), while the measured peak amplitude is 0.28 mm. This is in line with the observations of the RMS values: the simulations underestimate the experimental value and is, therefore, not conservative.

The FFT is performed over 6.5 s of simulation time and 400 s of AMOVI measured data. The first 0.2 s from the simulation were discarded from the total simulation time of 6.7 s. These intervals should provide sufficient statistics for the FFT analysis. Figure 80 depicts the calculated APSD and the frequency response of the central tube. The numerical function `matplotlib.mlab` with Welch's average periodogram was used. The sampling frequency (number of samples per second) for the simulation was 5000 Hz and 1000 Hz for the experimental data. In the FFT analysis, a Hanning windowing was applied. The prominent peak displays the reduced eigenfrequency. A very good agreement between simulations and experiment is observed: ANSYS CFX-MOR (MOR) and ANSYS CFX-Mechanical (SYSC) predict $f = 24.4$ Hz, while the experiment provides 23.4 Hz. Table 14 summarizes this data.

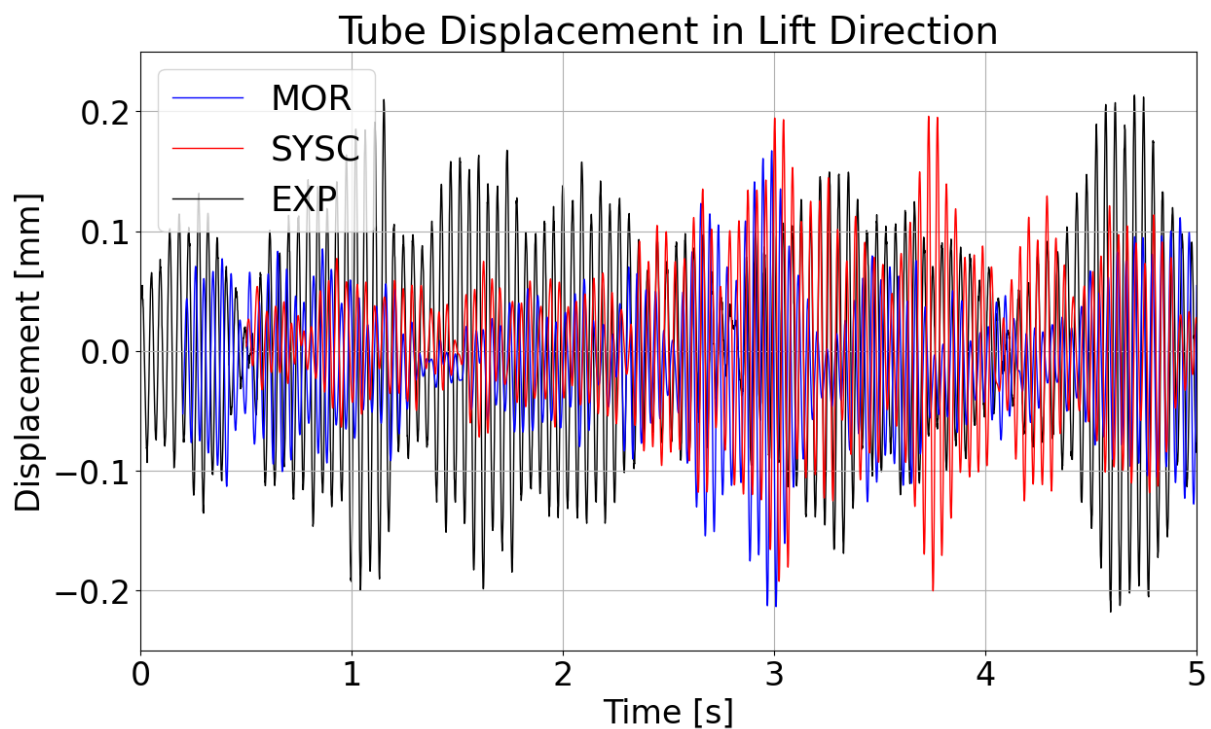


Figure 79: Measured and calculated tube displacements for the 1.6 l/s case.

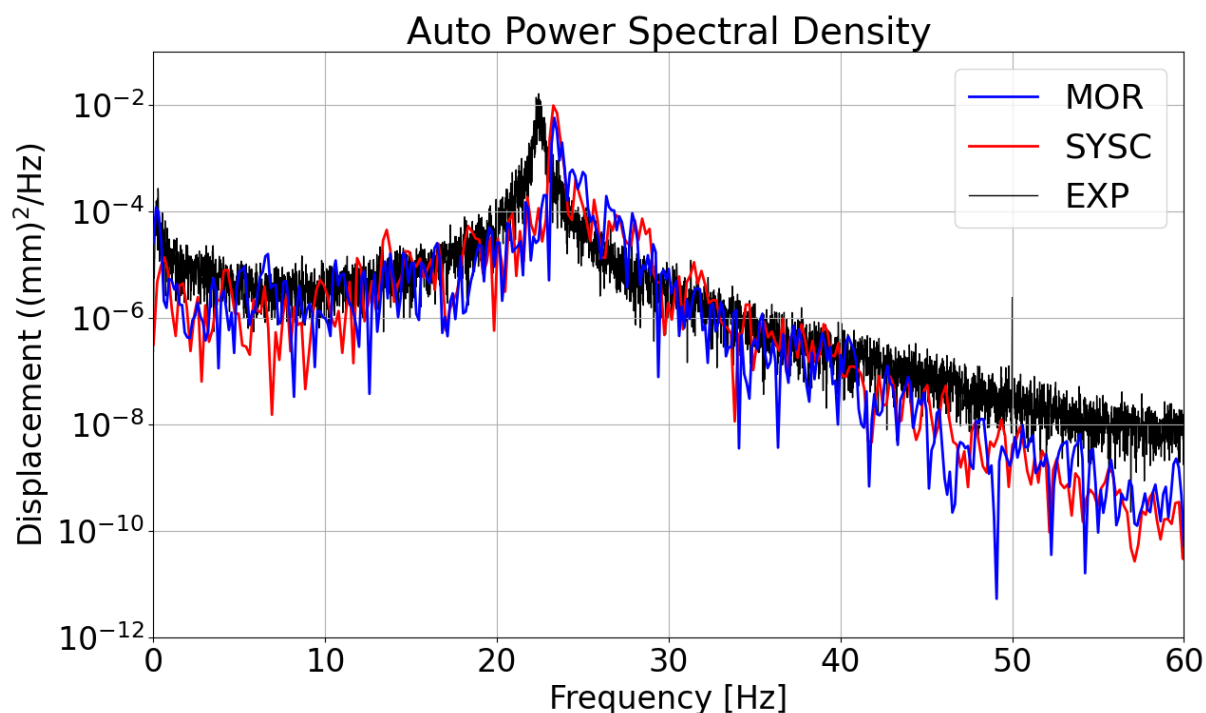


Figure 80: Measured and calculated tube displacements for the 1.6 l/s case.

Table 14: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube in a rigid tube bundle ($Q=1.6$ l/s).

Parameter	SYSC SST	ZLES MOR	Experiment
Displacement RMS, mm	0.053	0.061	0.073
Max. displacement, mm	0.20	0.22	0.28
Vibration frequency, Hz	24.4	24.4	23.4

2.8.3 Simulation of the flexible tube in a rigid tube bundle experiment with 1.95 l/s inlet flow

The available case with the highest volumetric flow rate, provided by CEA, was analyzed with the ANSYS CFX – MOR approach.

2.8.3.1 Simulation setup and boundary conditions

The same mesh and simulation setup was used as in the previous simulation. An inlet turbulent profile was generated, based on the new volumetric flow rate of 1.95 l/s, and applied at the inlet of the test section. The turbulence intensity was specified to be 5%.

The y^+ values increased as a result of the higher volumetric flow rate in this case, but were mostly around 5 (see Figure 81). Again, hot spots with higher values were found at the top and bottom surfaces of the vibrating tube.

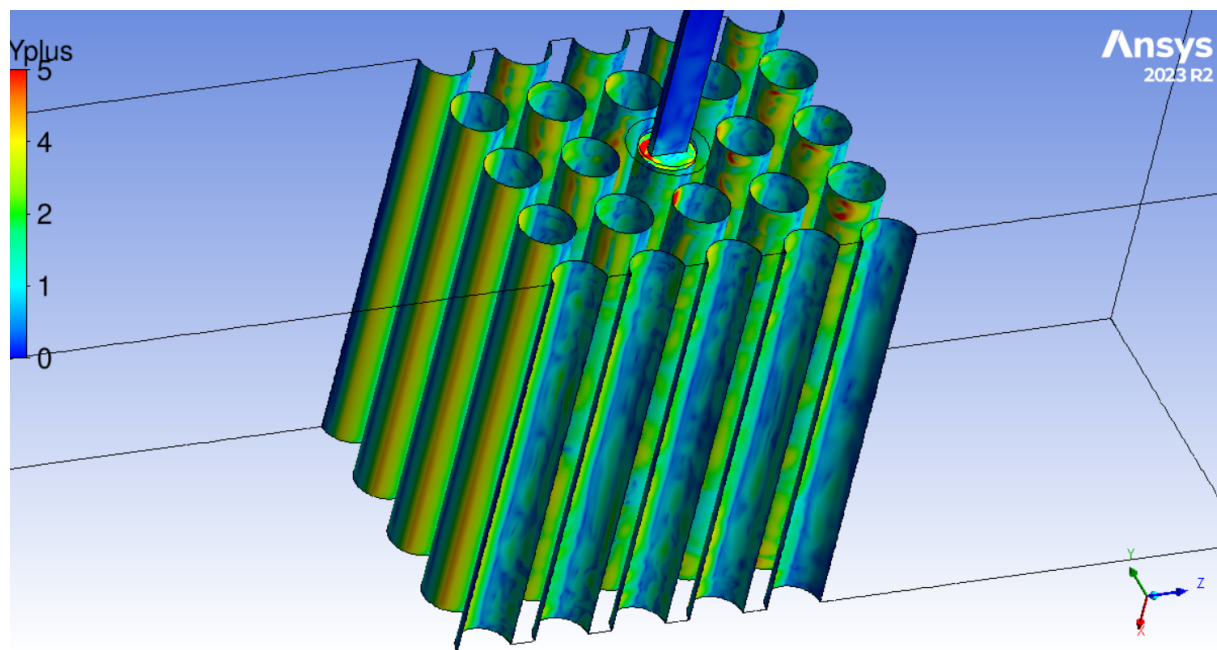


Figure 81: Y^+ value on the walls of the tubes ($Q=1.95$ l/s).

2.8.3.2 Analysis of the Flow Pattern in the AMOVI Test Section

Figure 82 shows the calculated instantaneous velocity distribution in the whole AMOVI test section with ANSYS MOR at $t = 6.2$ s. The turbulent structures, visualized with the Vortex Core Region option in CFX-Post, provide important information about the flow in the channel. As in the previous case, smaller turbulence structures are resolved in the LES region. Figure 83 presents the velocity distribution in the AMOVI test section in a vertical plane, positioned right in the middle of the test channel. It shows similar distribution to the one in the previous case. It is interesting to observe a high velocity stream in red color downstream the tube. This one results from the high-velocity jets (around the tubes in the middle). Either from the left or from the right such velocity jet enters the presented plane and leads to the high velocity stream. Figure 84 reveals the velocity distribution in a horizontal plane, positioned at half tube height. The water flows from left to the right with mean free stream velocity in the channel of 0.28 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. The four higher-velocity jets observed in the previous case are present, but the local maxima reach up to 1.4 m/s. These jets are again instable in time and move in flow transverse direction.

The recirculation zones behind the cylinders and in the corners formed by the half-cylinders and the side walls are visualized with the help of velocity-colored vectors in Figure 85. These do not differ significantly from the ones in the previous case with $Q=1.6$ l/s. Figure 86 shows the time-averaged transverse velocity distribution in the plane. It is interesting to observe that

there are rows with red colors on the left side of the tubes and there are rows with blue colors on the left sides of the tubes. The red color denotes fluid flowing to the top side wall in the picture, while the blue color denotes water flowing to the bottom side wall in this picture. Both high-velocity jets around the tubes try to generate recirculation zones downstream each tube. These rotate in opposite directions. Depending on the jet strength, one or the other swirl prevails and drives the flow towards one of the side walls. This generates the red and blue velocity peaks.

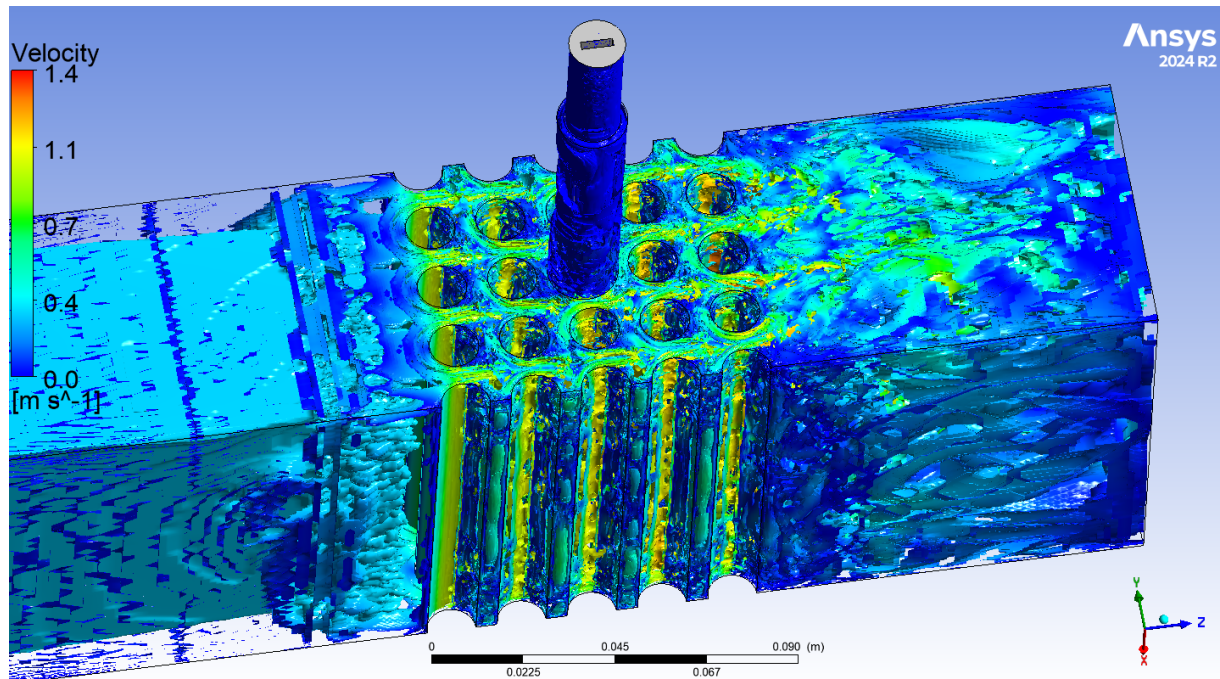


Figure 82: Instantaneous velocity distribution at $t=6.2$ s in the AMOVI test section equipped with a tube bundle ($Q=1.95$ l/s).

2.8.3.3 Vibration Analysis

Figure 87 presents the calculated and the experimental tube displacements. It reveals a large deviation between calculation and experiment (more than factor of 10). In order to better understand this discrepancy, the displacements for both cases we plotted together with the measured values for the $Q=1.6$ l/s case. An increase in the inlet flow rate from 1.6 to 1.95 l/s (22%) leads to an increase in the RMS displacements from 0.053 to 0.081 mm (53%). This is shown in Figure 88 and proves that there is no linear dependency between the free stream velocity and structural displacement. This occurs either in the lock-in region or in the fluid-elastic instability (FEI) region, both characterized by very large structural displacements.

In this particular case, the observed phenomenon is FEI. Since both phenomena are very hard to simulate numerically, it is already a success that ANSYS CFX-MOR predicts this non-linear dependency between the inlet volumetric flow rate and the tube displacement amplitude.

Figure 89 depicts the calculated APSD and the frequency response of the central tube. The numerical function `matplotlib.mlab` with Welch's average periodogram was used. The sampling frequency (number of samples per second) for the simulation was 5000 Hz and 1000

Hz for the experimental data. In the FFT analysis, a Hanning windowing was applied. A very good agreement between simulations and experiment is observed: ANSYS CFX-MOR predicts $f = 24.6$ Hz for the reduced eigenfrequency, while the experiment provides 23.6 Hz. The first experimental frequency peak is significantly higher due to the massively larger displacement amplitudes that have been measured by CEA. In the experimental curve there is also a prominent second peak, which seems to coincide with the second harmonic. Tables Table 12-Table 15 summarizes the results.

2.8.3.4 Further efforts to improve the numerical results

It was decided to perform a VLES (Very Large Eddy Simulation) with the existing mesh. The time step size was kept at $dt=1e-4$ s, but the advection scheme was changed to Central Difference, which is known to be more appropriate for LES than the High-Resolution one. The WALE subgrid-scale turbulence model was used within the LES framework. The idea was to perform a rapid and preliminary LES analysis without large effort to see if it might predict larger amplitudes. The LES simulation was a restart run from the already discussed results. The calculated displacements are shown in Figure 90. It can be seen that significantly larger displacements up to 4 mm develop in the AMOVI test section. These are significantly closer to the ones observed in the experiment (~ 5.5 mm). After reaching ~ 4 mm, the simulation crashes due to poor mesh quality. The large amplitudes deform significantly the mesh, and this leads to very small orthogonality angles ($< 1^\circ$) and extremely high expansion rates and aspect ratios (in the order of thousands).

It is not easy to find out if these large amplitudes in the simulation have physical or rather numerical origin. Different mesh motion options, different meshes, time steps (down to $5.e-5$ s), subgrid-scale turbulence models (WALE and Smagorinsky) were tested, but the results did not change significantly and more than 4 mm displacements were not achieved. Therefore, further analyses are necessary to investigate if these continuously increasing displacements will stabilize at a certain level or continue to increase.

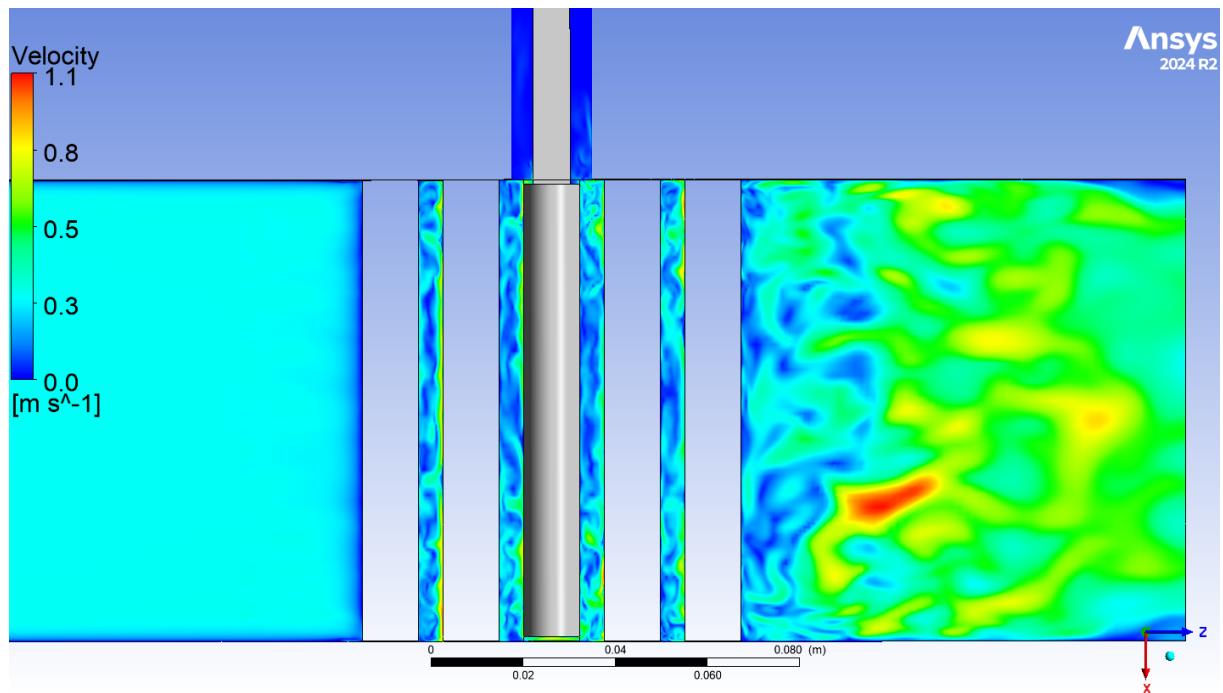


Figure 83: Time-averaged velocity distribution in the AMOVI test section in a vertical plane ($Q=1.95 \text{ l/s}$).

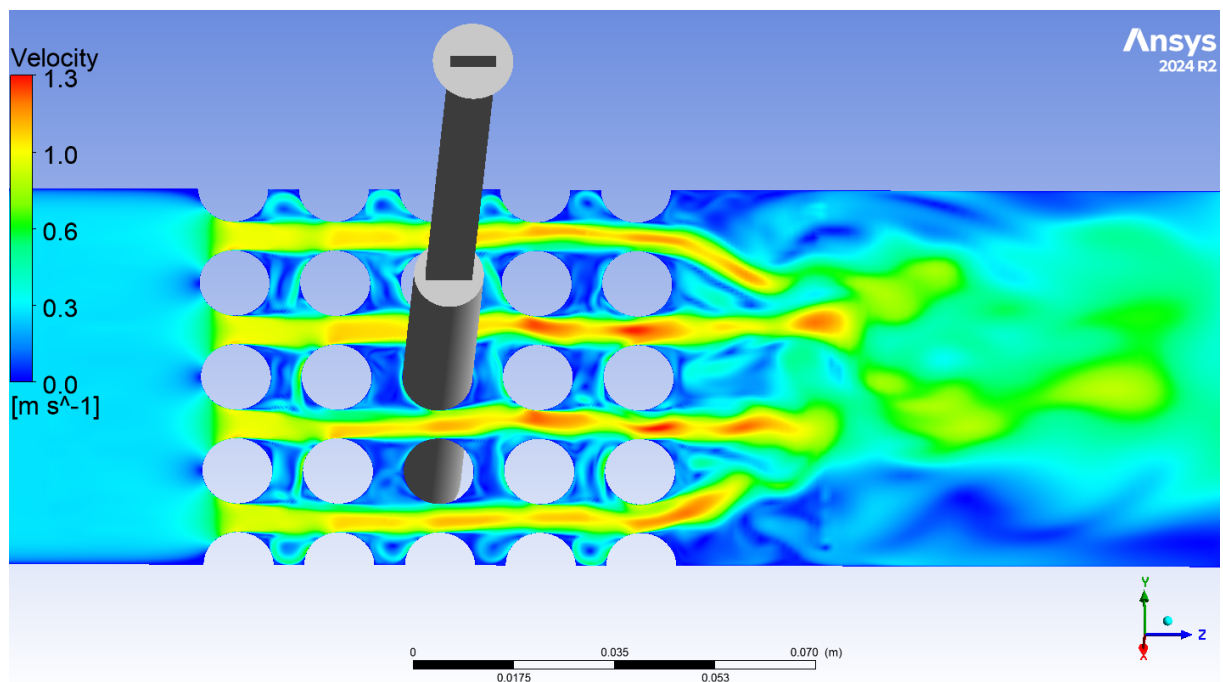


Figure 84: Time-averaged velocity distribution in the AMOVI test section in a horizontal plane ($Q=1.95 \text{ l/s}$).

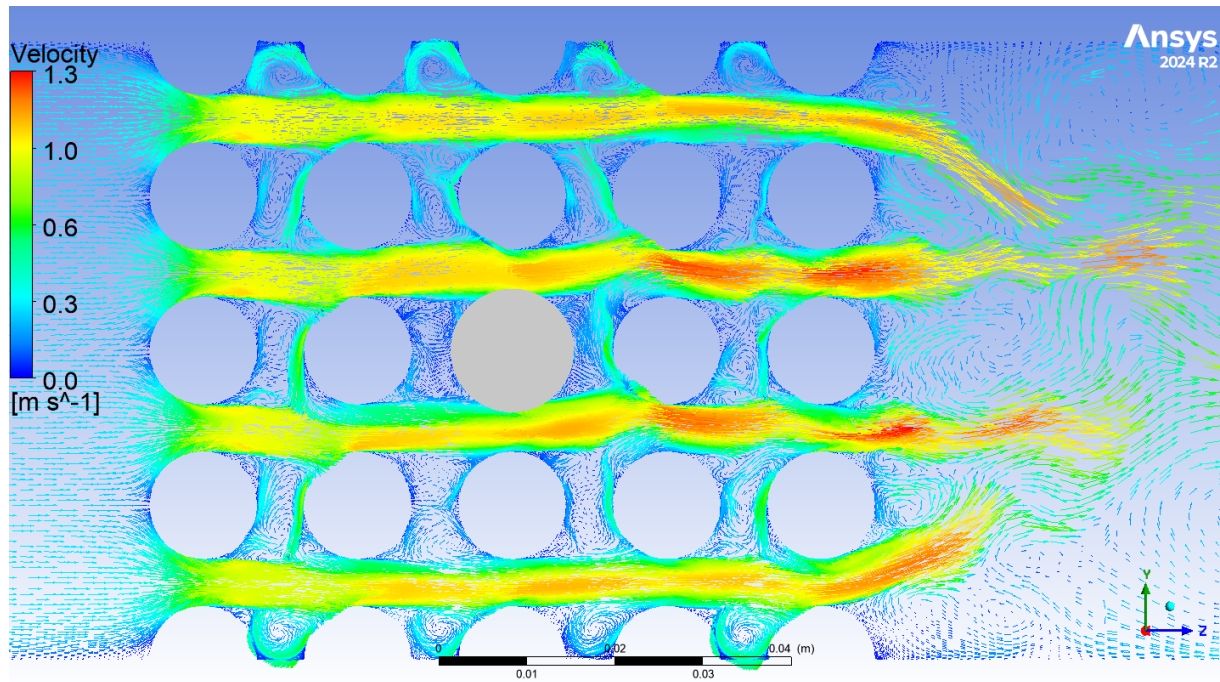


Figure 85: Recirculation zones visualized with velocity-colored vectors at $t=6.2$ s ($Q=1.95$ l/s).

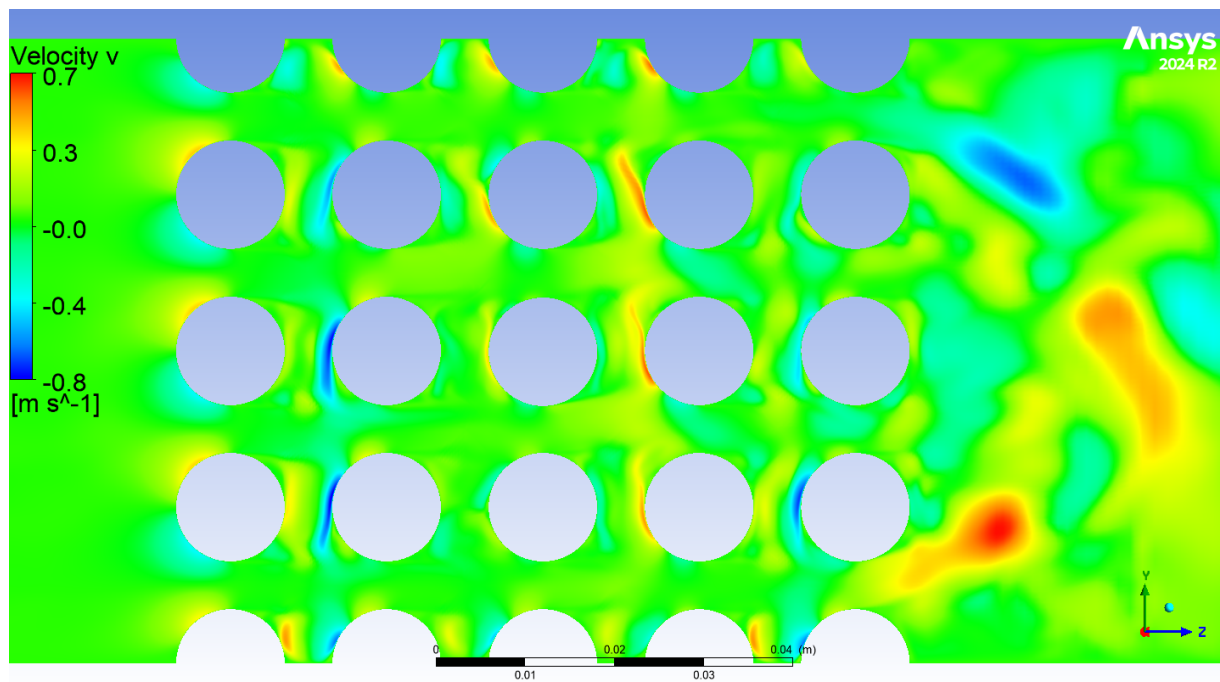


Figure 86: Recirculation zones (time-averaged) visualized with velocity-colored vectors ($Q=1.95$ l/s).

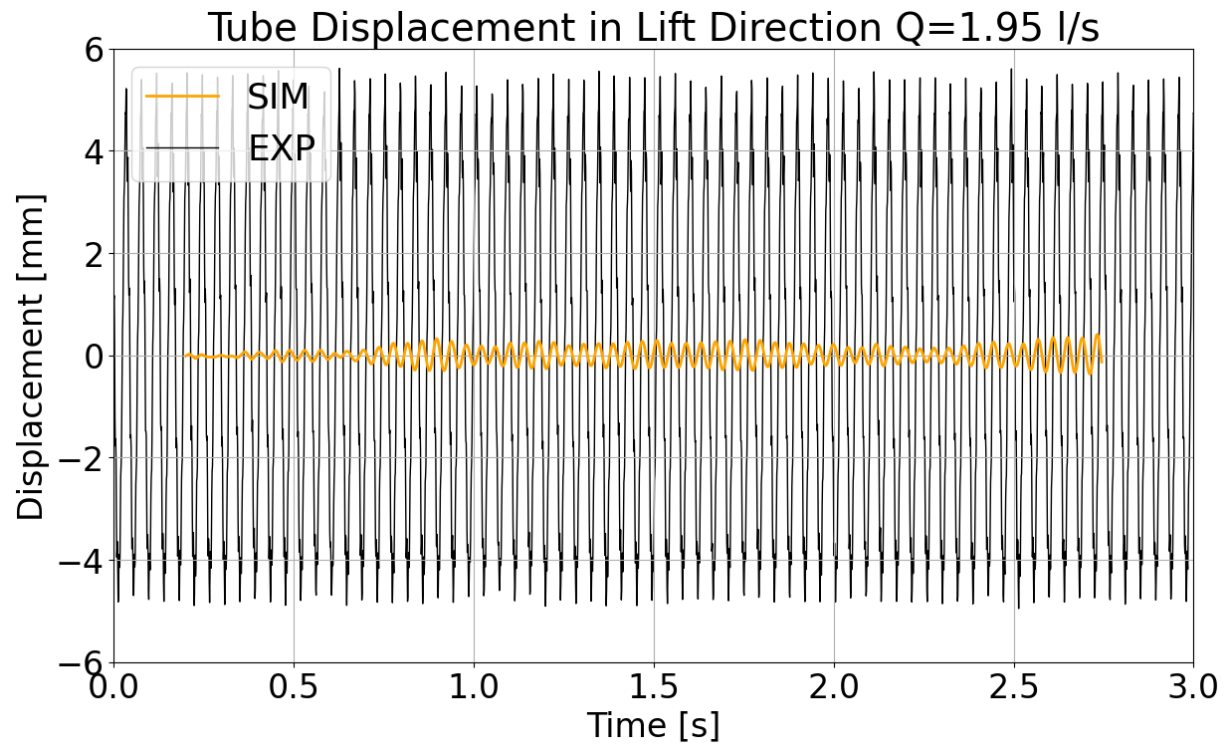


Figure 87: Measured and calculated tube displacements ($Q=1.95$ l/s).

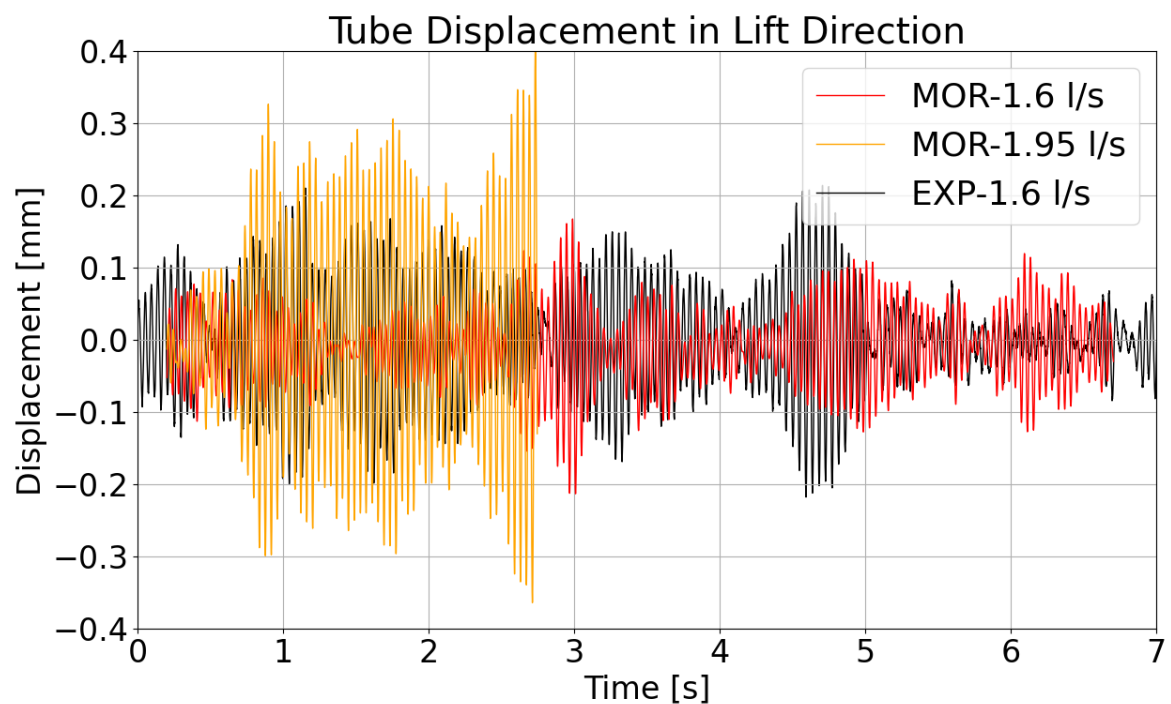


Figure 88: Comparison of the tube displacements in both cases.

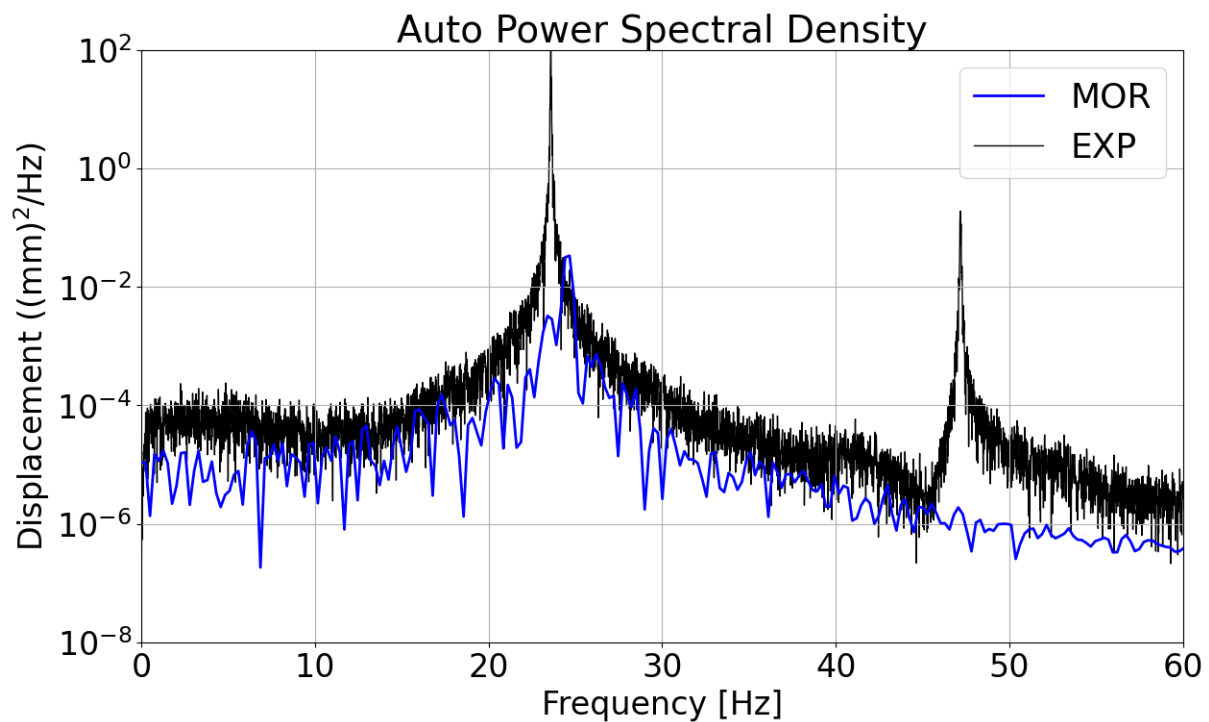


Figure 89: Measured and calculated APSD for the 1.95 l/s case.

Table 15: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube in a rigid tube bundle ($Q=1.95$ l/s).

Parameter	ZLES MOR	Experiment
Displacement RMS, mm	0.08	3.22
Max. displacement, mm	0.40	5.21
Vibration frequency, Hz	24.6	23.6

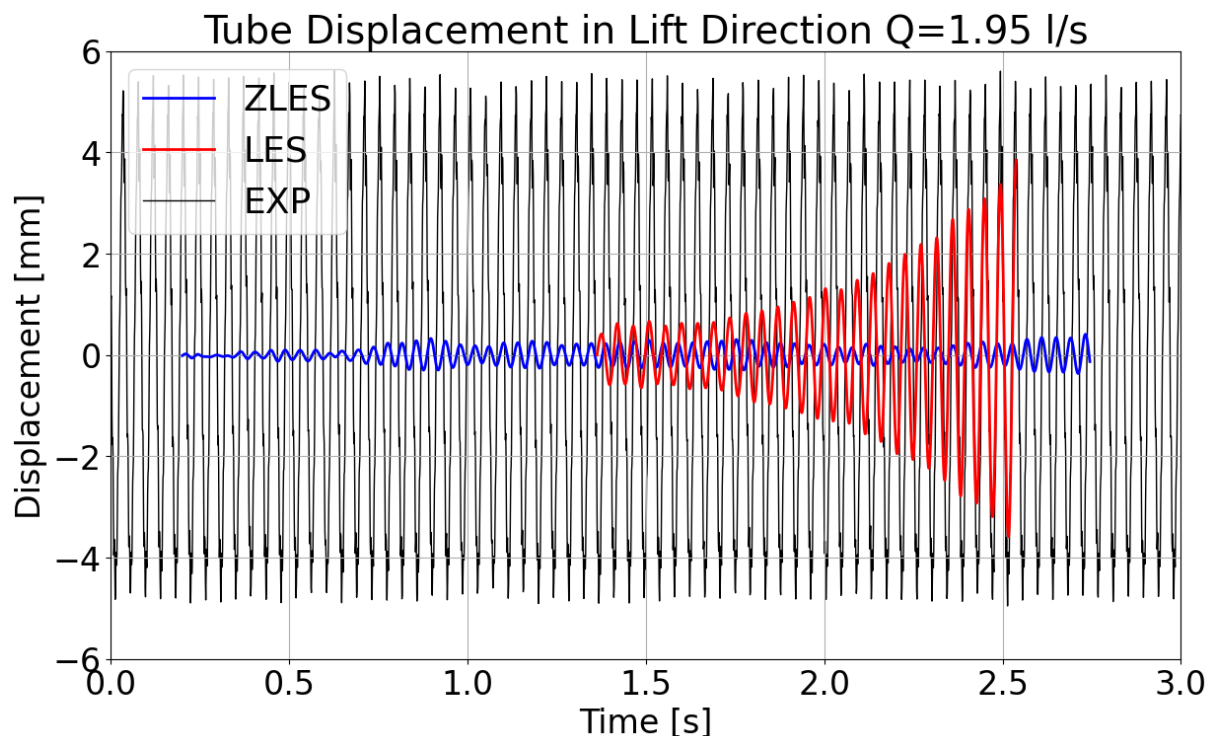


Figure 90: Measured and calculated tube displacements for the 1.95 l/s case.

It was decided to simulate also the other two AMOVI bundle cases with a flexible tube that were shared by the CEA experimentalists: the 1.1 l/s and 0.5 l/s as these results might further help the validation of the FSI codes of ANSYS. The results will be only briefly described below, as the modeling setup is the same like in the presented cases.

2.8.4 Simulation of the flexible tube in a rigid tube bundle experiment with 1.1 l/s inlet flow

This case was of particular interest, since partners within GO-VIKING WP3 performed high-resolution, wall-resolved LES simulations with very large numerical meshes. They found that at this mass flow rate, huge tube displacements develop that eventually lead to the crash of the simulations. Therefore, it was decided to simulate this case with two different approaches ANSYS CFX – MOR and ANSYS CFX – Mechanical.

As mentioned before, the same CFD and CSM setups were used, as in the previous simulations with the ZLES turbulence approach. The new boundary conditions were specified at the inlet of the CFD domain.

Figure 91 presents the instantaneous velocity distribution in a horizontal YZ plane, positioned in the middle of the tube bundle. The high-velocity jets, observed in the previous cases, are also present here, but the magnitude of the velocity is smaller due to the lower inlet flow rate. Again, the velocity profile shape upstream the bundle is destroyed after the first tube row and the recirculation zones are present here as well. The high-velocity jets merge two by two

approximately two tube diameters downstream the last tube row. One should keep in mind that this figure presents an instantaneous and not a time-averaged velocity.

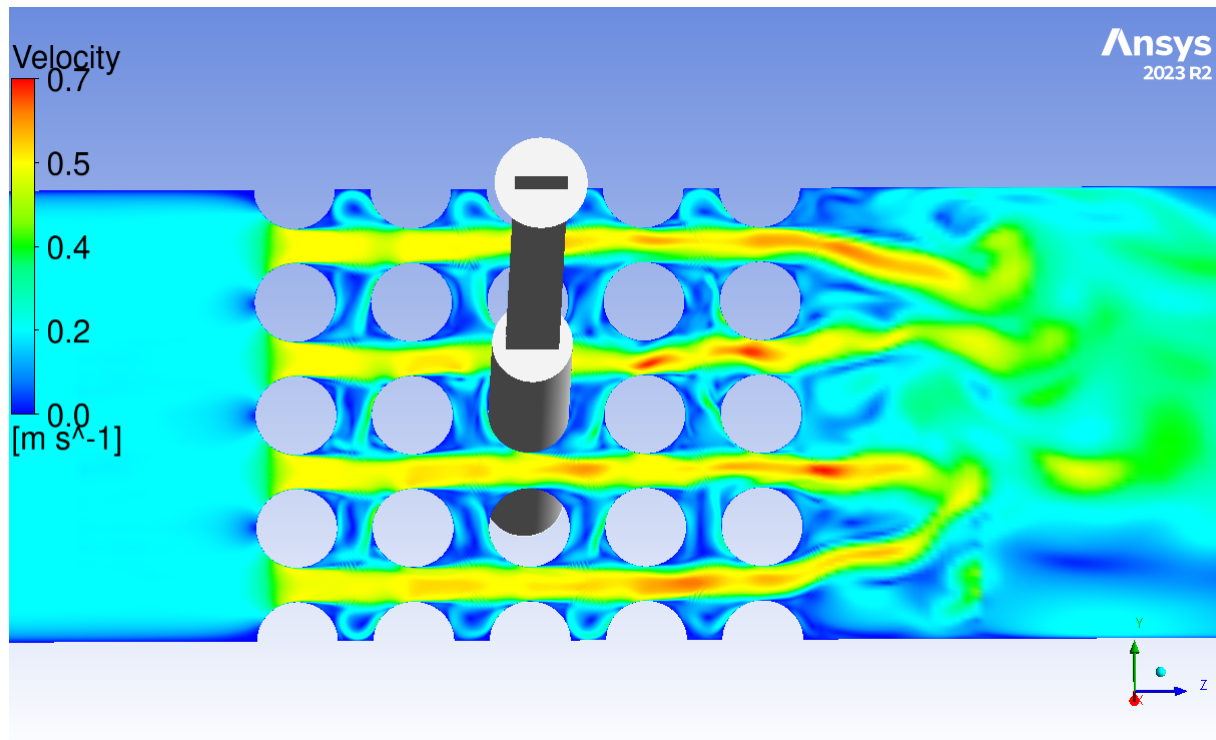


Figure 91: Velocity distribution in the AMOVI test section in a horizontal plane ($Q=1.1$ l/s) at $t=5.06$ s.

Figure 92 presents the calculated and the experimental tube displacements. The ANSYS CFX-MOR results showed excessive tube displacements, as found by EDF in their computations. It is obvious that these are by factors larger than the measured values. The simulations at EDF were performed with the Rigid Body Motion Approach, which is also a simplification of the mechanical system. Although much more sophisticated, the MOR approach used here for the representation of the tube is also a simplified representation of the reality. To exclude the modelling of the structure from the possible causes of these large displacements, a full-order model simulation with ANSYS CFX-Mechanical with the same setup and boundary conditions. Although the initialisation of the fluid domain was different (this led to the different behavior in the first seconds of the computation), it also confirmed the existence of unexpectedly large tube displacement in the range of 0.6 mm. The simulation was not run long due to the prohibitive CPU time. Figure 93 depicts the resulting APSDs from the data. Both simulations deviate by about 1 Hz from the measurement.

Currently, the origin of the large displacements is not very clear. GRS performed simulations with Reynolds Stress turbulence model within GO-VIKING WP3 that produced very similar results. Discussions within project meetings suggested that all codes might predict a lock-in. Also, the time series of the displacements with their constant amplitudes would be in favor of such hypothesis. The experimental data clearly shows that there is no lock-in. Another hypothesis is that these large amplitudes have numerical origin. Three different simulations with three different structural (Rigid Body Motion, Model Order Reduction, full representation

of the vibrating structure) approaches and turbulence models (RSM, hybrid, LES), as well as different numerical schemes, time steps, etc. predict the same behavior. Therefore, this hypothesis is not very solid. It is obvious that further discussions with the experimentalists from CEA are necessary to clearly understand where these discrepancies come from in this particular case.

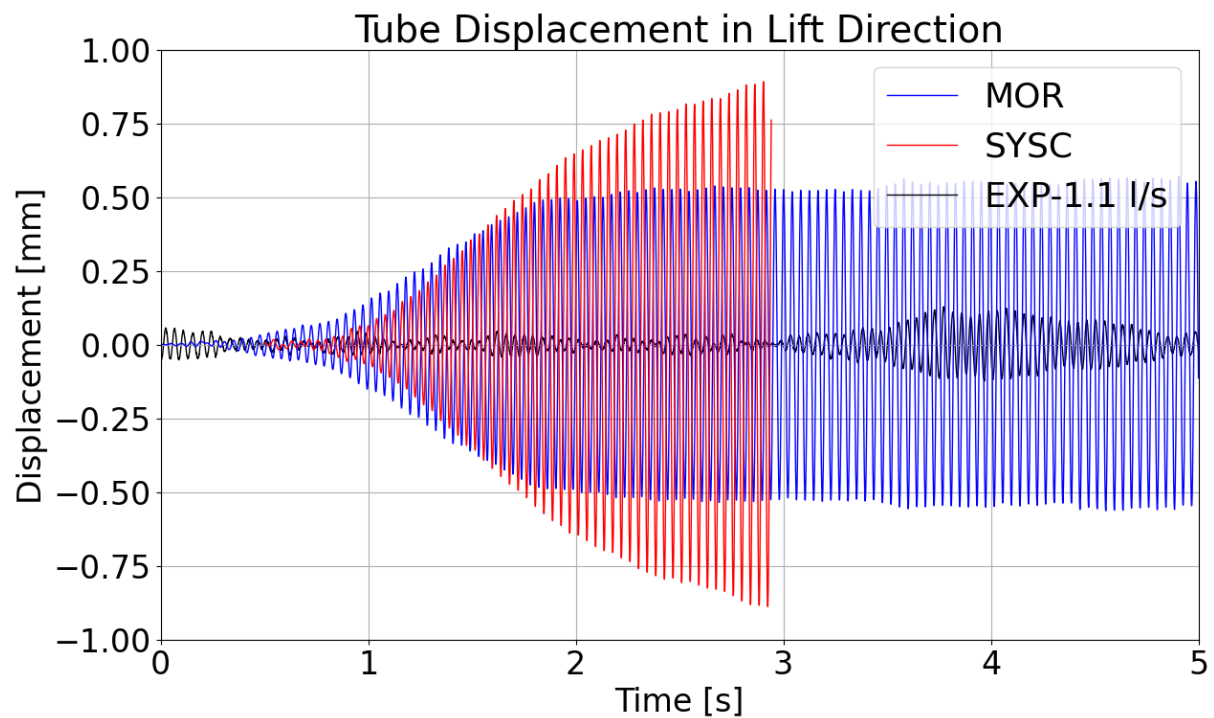


Figure 92 Measured and calculated with ANSYS CFX – MOR and ANSYS CFX – Mechanical tube displacements for the 1.1 l/s case.

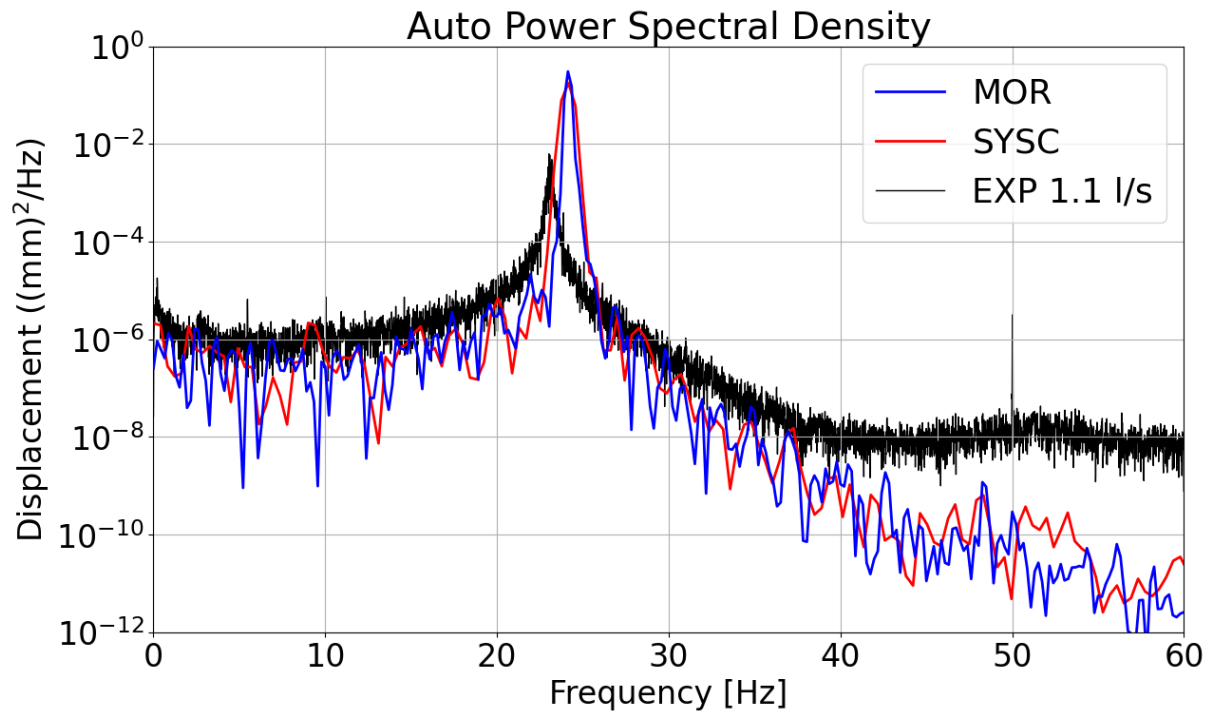


Figure 93 Measured and calculated APSD for the $Q=1.1$ l/s case.

Table 16: Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube in a rigid tube bundle ($Q=1.1$ l/s).

Parameter	ZLES MOR	ZLES SYSC	Experiment
Displacement RMS, mm	0.32	0.39	0.04
Max. displacement, mm	0.55	0.88	0.11
Vibration frequency, Hz	24.1	24.1	23.1

2.8.5 Simulation of the flexible tube in a rigid tube bundle experiment with 0.5 l/s inlet flow

The same CFD and CSM setups were used, as in the previous simulations with the ZLES turbulence approach. The only difference to the previous run was the lower volumetric flow rate of 0.5 l/s at the inlet of the CFD domain.

Figure 94 presents the instantaneous velocity distribution in a horizontal YZ plane, positioned in the middle of the tube bundle. The high-velocity jets, observed in the previous cases, are also present here, but the magnitude of the velocity is smaller due to the lower inlet flow rate. The high-velocity jets merge all together downstream the last tube row and tend to move to one of the side walls, but again, this is an instantaneous and not a time-averaged field.

Figure 95 depicts the tube displacements for the investigated case. The code massively underestimates (by a factor of 10) the measurement (see also Table 17). This is valid for the RMS and for the maximum observed displacement. The resulting APSDs are shown in Figure 96. The frequency peak at 22.8 Hz that results from the tube eigenfrequency is overpredicted by the code by just 0.9 Hz. It is interesting to see that another peak appears in the measured data at 12.5 Hz. Although not so well distinguished in the simulation, it may be found at 10 Hz.

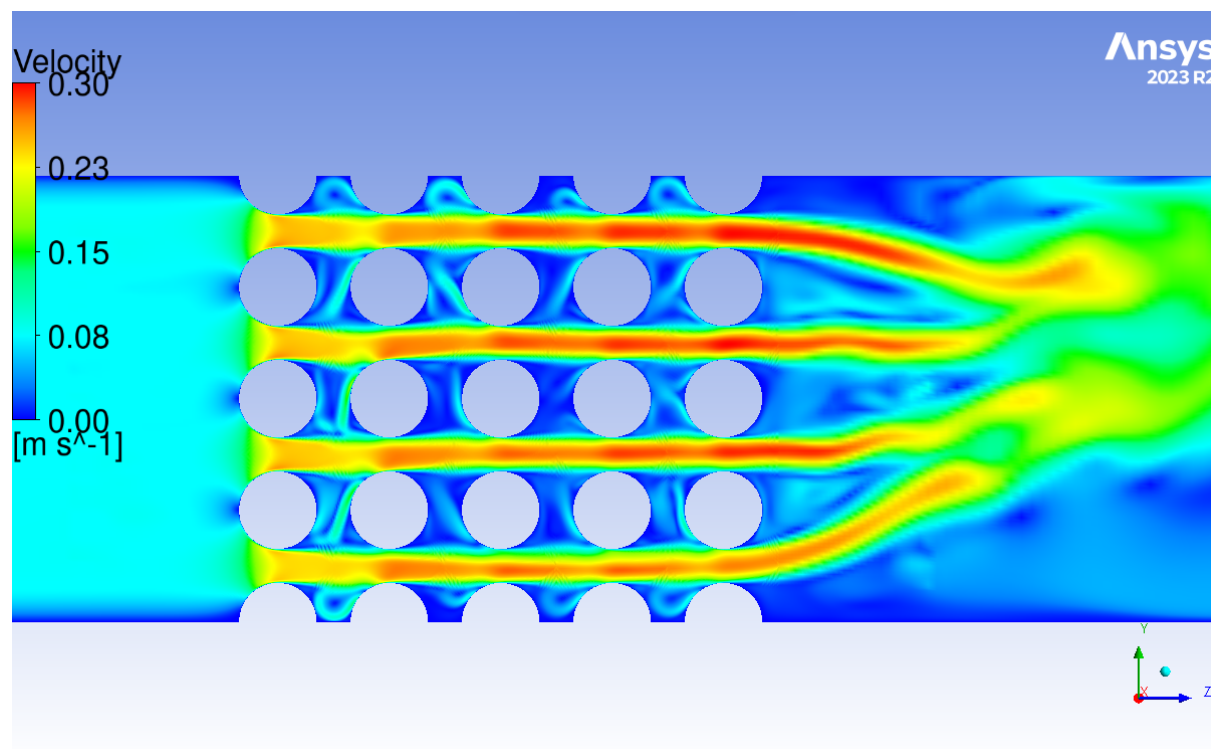


Figure 94 Velocity distribution in the AMOVI test section in a horizontal plane ($Q=0.5$ l/s) at $t=9.27$ s.

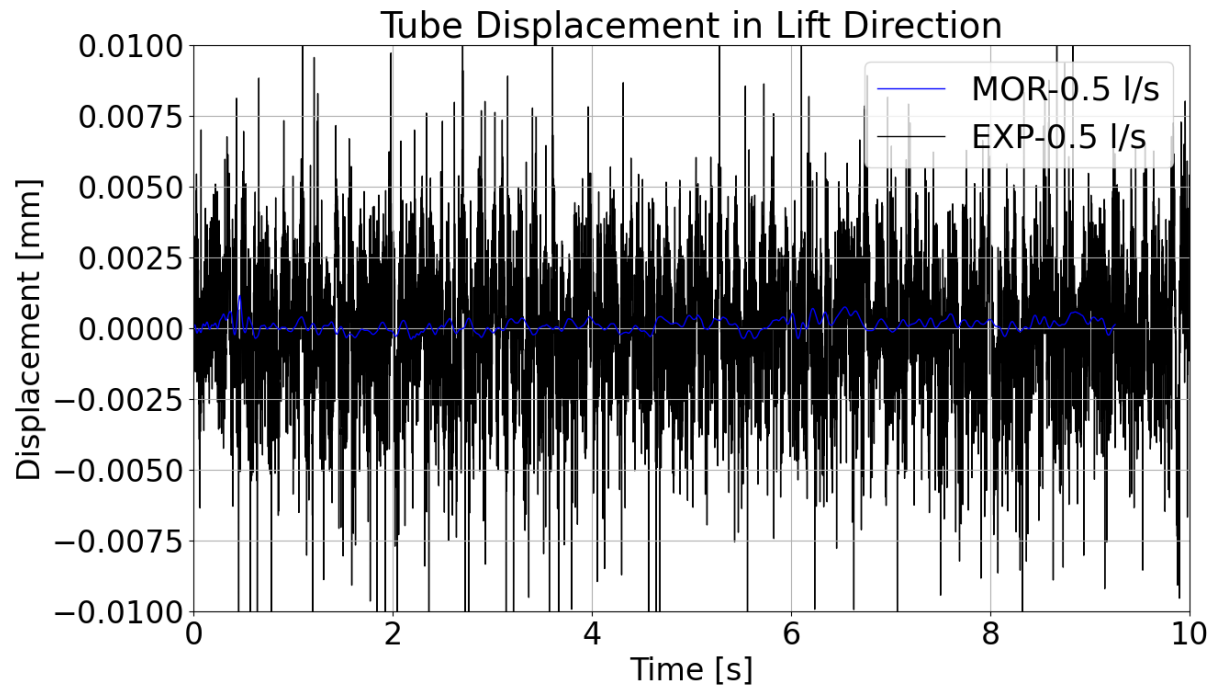


Figure 95 Measured and calculated tube displacements for the 0.5 l/s case.

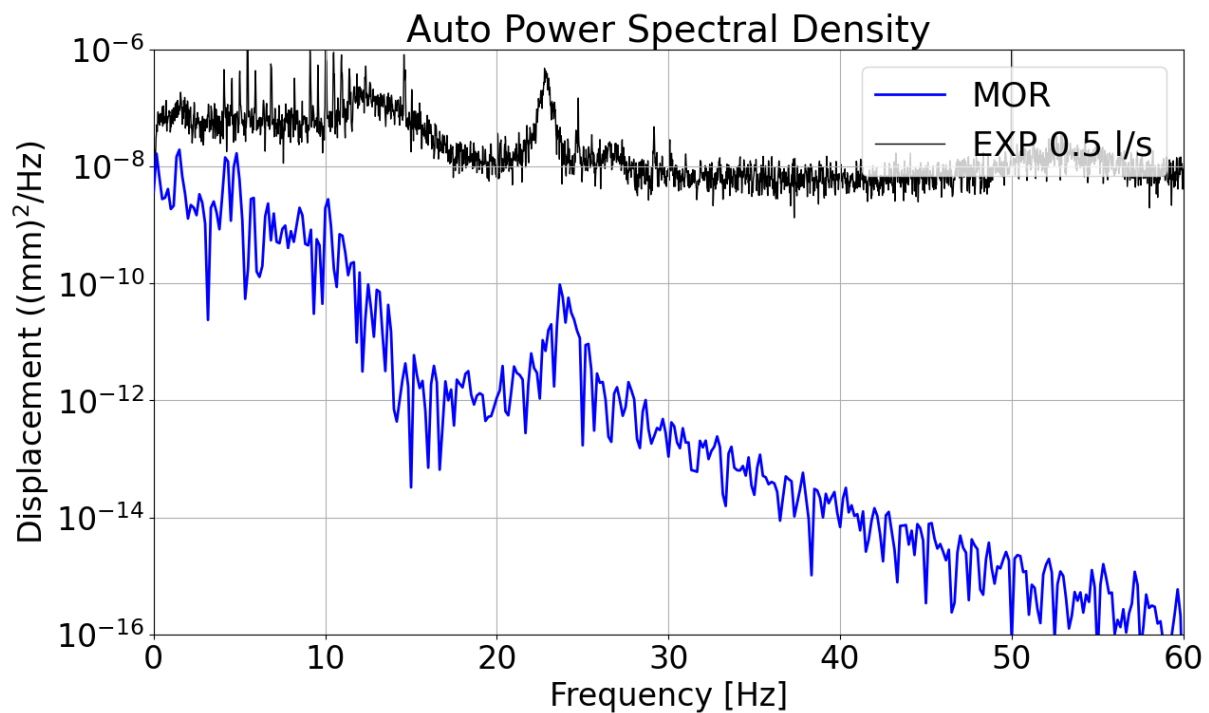


Figure 96 Measured and calculated APSD for the Q=0.5 l/s case.

Table 17 Comparison between numerical simulations and experimental data for the vibration response of the AMOVI single flexible tube in a rigid tube bundle (Q=0.5 l/s).

Parameter	MOR	Experiment
-----------	-----	------------

Displacement RMS, mm	0.0002	0.0025
Max. displacement, mm	0.0012	0.016
Vibration frequency, Hz	23.7	22.8

2.9 Conclusions on the AMOVI validation activities

FSI analyses for two configurations were performed within the validation activities for ANSYS CFX-MOR in WP5: a single flexible tube and a flexible tube in a rigid tube bundle. The single flexible tube cases with a volumetric flow rate of 2.9 l/s and 1.7 l/s were simulated with different FSI and turbulence approaches. All simulations proved that the FSI tools of ANSYS can reliably predict the vibration of the tube and the attached blade. The full-order coupling simulation provided similar results to the one with the reduced order structural model, but at approx. 6 times higher computational cost. This proved that the MOR is in-line with the FOM results. In terms of flow prediction, the hybrid ZLES model outperformed the SST turbulence model, as expected. The main advantage of ZLES lies in the higher accuracy of the tube displacement prediction: larger displacements that are closer to the measured value. In terms of vortex-shedding frequency, both turbulence models performed well.

In a second step, a more complex geometry consisting of a rigid tube bundle and the same flexible tube and its blade in the center of the bundle was investigated. As ZLES proved to deliver the best solution (also in other validation activities on FIV), it was chosen for all bundle simulations. The first case that was simulated was with a volumetric flow rate of $Q=1.6$ l/s. The comparison between experimental and numerical data is very good, not only in terms of vibration characteristics, but also in terms of time-averaged flow field. This case was provided also with PIV data by the experimentalists. In a next step, the bundle case with the highest flow rate of $Q=1.95$ l/s was simulated. This case exhibits very large displacements (~ 5 mm), and according to the experimentalists it represents fluid-elastic instability. Frequencywise, the agreement between simulation and experiment was very good. Displacementwise, an underprediction by ANSYS CFX-MOR was observed. This could be expected, since the simulation of FEI is very hard. Still, the code predicted non-linear increase in the amplitudes when increasing the flow rate from 1.6 to 1.95 l/s, which is already a success.

It was decided to simulate also the cases with $Q=1.1$ l/s and $Q=0.5$ l/s. Both cases showed very good agreement in terms of vibration frequencies. In the first case, the displacements were remarkably overpredicted by the code, but similar behavior was found also by other partners using different turbulence and structural models. Therefore, this case needs to be further discussed with the experimentalists before final conclusions on the core performance are made. The second case (0.5 l/s) revealed massive underprediction of the displacement amplitudes.

It is not easy to draw final conclusions from the above. The simulation of FIV of a single tube or a flexible tube in a bundle using MOR is promising, but further validation is necessary. Attention shall be paid to validation matrices with wider range of boundary conditions and different geometrical configurations (more flexible tubes in a bundle). This is especially

important when the FSI code is intended be validated and then used as a general purpose code, i.e. more or less independent of geometry and boundary conditions.

3 Validation of the Synthetic Turbulence procedure implemented based on Stable Fluid Algorithm for fast-running FSI

This section describes the main approaches that are used by IPP for one-directional and two-directional FSI, considering the reconstruction of the turbulent pulsation field. The main criterion for selecting approaches is their speed and sufficient accuracy for engineering practice. Validation of the developed and described methods is performed using AMOVI Benchmark cases.

3.1 Extending the capabilities of the Stable Fluid Algorithm based on deterministic vorticity source and Synthetic Turbulence for FSI engineering practice

Potentially fast algorithms for approximate flow modeling are methods used in computer graphics. The advantage of such algorithms is that they are based on the original hydrodynamic system of equations for incompressible flow, but the method for calculating the flow evolution over time (transient processes) differs from classical CFD codes. The most known algorithm that laid the foundation for physically-based visualization is the so-called **Stable Fluid Algorithm** (Stam, Stable fluids, 1999). Using the Helmholtz-Hodge decomposition (Bhatia, Norgard, Pascucci, & Bremer, 2013) for a vector field, a fairly simple algorithm for end-to-end calculation of fluid flow can be constructed. Taking into account the computing power of personal computers, variations of this method are also used for so-called *Real Time Fluid Dynamics* in games (Stam, Real-Time Fluid Dynamics for Games, 2003). A distinctive feature of the **Stable Fluid Algorithm** is that the calculated flows are quasi-physical, and the method of calculating convective terms makes this algorithm sufficiently viscous and stable, therefore there is no need to use any closing turbulence model. Stability is achieved through the use of the so-called Lagrangian self-advection (Bowman, Yassaie, & Basu, 2015), which effectively eliminates the nonlinearity of the original hydrodynamic system. Typically, such algorithms are implemented on “soft” or periodic boundary conditions for Cartesian computational meshes using finite differences.

Thus, this algorithm is a good candidate for creating fast methods for calculating hydrodynamic processes taking into account FSI, however it requires additions that will improve the physical quality of calculations and predictions, as well as a clear and logical way to calibrate such models.

Let's consider the system of Reynolds equations for a two-dimensional incompressible isothermal formulation (12). We will decompose the original equations using the expansion of the velocity vector field of the form $U = u - \frac{\Delta t}{\rho} \nabla p$, where the vector U is considered to be a divergent field satisfying (12). Therefore, it is necessary to find a non-divergent pseudo-velocity field u . The harmonic component of the expansion is considered to be equal to zero, since the formulation does not consider sources and

outflows of mass, and we also assume that the flow is vortex, that is there is a “potential” of vortex.

$$\begin{cases} \frac{\partial U}{\partial x} + \frac{\partial V}{\partial y} = 0 \\ \frac{\partial U}{\partial t} + U \frac{\partial U}{\partial x} + V \frac{\partial U}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \eta_{eff} \left(\frac{\partial U^2}{\partial x^2} + \frac{\partial U^2}{\partial y^2} \right) + gn_x + S_U \\ \frac{\partial V}{\partial t} + U \frac{\partial V}{\partial x} + V \frac{\partial V}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \eta_{eff} \left(\frac{\partial V^2}{\partial x^2} + \frac{\partial V^2}{\partial y^2} \right) + gn_y + S_V \end{cases} \quad (12)$$

Where n_x and n_y – projection of gravity onto the coordinate axes, S_U and S_V are auxiliary volumetric sources (synthetic swirlers).

In fact, such a decomposition implies the existence of a certain projection operator L , which allows us to relate the divergent and non-divergent fields of "similar" physical quantities. This is essentially the same as what is performed during the projection step in segregated SIMPLE methods.

Herewith, this operator has two properties. The first is: $\mathbf{U} = \mathbf{L} \cdot \mathbf{u}$, and the second follows from the first: $\mathbf{L} \cdot \nabla p = 0$. Due to the fact that the non-stationary change in pressure in (12) is neglected (the processes are not shock, and the fields are monotonic), then $\partial \nabla p / \partial t = 0$. That is, the change in the non-divergent velocity $\mathbf{u} = \{u, v\}$ over time can be represented as (13), from which it is clear that any change in the field over time is the sum of the actions of each of the factors (force, advection, and diffusion).

$$\begin{cases} \frac{\partial U}{\partial t} = \frac{\partial u}{\partial t} = \alpha u + \beta u_0 + \gamma u_{00} = -\left(u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y}\right) + \eta_{eff} \left(\frac{\partial u^2}{\partial x^2} + \frac{\partial u^2}{\partial y^2} \right) + gn_x + S_U \\ \frac{\partial V}{\partial t} = \frac{\partial v}{\partial t} = \alpha v + \beta v_0 + \gamma v_{00} = -\left(u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y}\right) + \eta_{eff} \left(\frac{\partial v^2}{\partial x^2} + \frac{\partial v^2}{\partial y^2} \right) + gn_y + S_V \end{cases} \quad (13)$$

Where α, β, γ – time derivative discretization coefficients (implicit, second order), which are determined by (14) for the adaptive time step.

$$\alpha = \frac{2\Delta t + \Delta t_0}{\Delta t(\Delta t + \Delta t_0)}; \beta = \frac{\Delta t + \Delta t_0}{\Delta t \Delta t_0}; \gamma = \frac{\Delta t}{\Delta t(\Delta t + \Delta t_0)} \quad (14)$$

The system of equations (13) is solved at time step using a physical decomposition (stepping algorithm). Its essence consists of a stepwise deviation of the velocity change $\mathbf{U}(t) = \{U_0, V_0\}$ from the previous time step, followed by projection onto the divergent field. Solution (12) can then be schematically represented as sequential operations:

$$\mathbf{U}(t) \xrightarrow{\text{source}} \mathbf{u}_1 \xrightarrow{\text{advection}} \mathbf{u}_2 \xrightarrow{\text{diffusion}} \mathbf{u}_3 \xrightarrow{\text{projection}} \mathbf{U}(t + \Delta t) \quad (15)$$

Each of the successive operations (15), except for projection, can be grouped.

If we assume that the advection terms (13) can be represented in the form of Lagrangian self-advection (16) based on the method of characteristics, then the system splits into two independent diffusion equations, which can be solved sequentially.

$$\begin{aligned} u_2(x, y) &= u_1^*(x - u_1\Delta t, y - v_1\Delta t) \cdot BC_{correction} \\ v_2(x, y) &= v_1^*(x - u_1\Delta t, y - v_1\Delta t) \cdot BC_{correction} \end{aligned} \quad (16)$$

Where u_1^* and v_1^* - are interpolation functions, obtained based on the previous step with adding the influence of the source terms (17), $BC_{correction}$ is a special function of BC correction (optional).

$$\begin{aligned} u_1(x, y) &= U_0 + \Delta t(gn_x + S_U) \cdot BC_{correction} \\ v_1(x, y) &= V_0 + \Delta t(gn_y + S_V) \cdot BC_{correction} \end{aligned} \quad (17)$$

The next step is to include the diffusion in the velocity field for each of the components. This is performed implicitly by solving the equations (18), which are independent if the Lagrangian advection is used.

$$\begin{cases} u_3 - u_2(x, y) = \Delta t \cdot \eta_{eff} \left(\frac{\partial^2 u_3^2}{\partial x^2} + \frac{\partial^2 u_3^2}{\partial y^2} \right) \\ v_3 - v_2(x, y) = \Delta t \cdot \eta_{eff} \left(\frac{\partial^2 v_3^2}{\partial x^2} + \frac{\partial^2 v_3^2}{\partial y^2} \right) \end{cases} \quad (18)$$

Once the divergence $\mathbf{u}_3$ is calculated, the Poisson's equation (19) is solved for the pressure field (this follows from applying the ∇ operator, scalar-multiplied by the left and right sides of the *Helmholtz-Hodge* decomposition, excluding the harmonic component).

$$\frac{\rho}{\Delta t} \nabla \cdot \mathbf{u}_3 = \Delta p \quad (19)$$

Thus, by performing relatively simple operations consecutively, the basic parameters of a two-dimensional flow can be obtained. There are a few features to note here. Equation (16) itself is highly diffusive if a classical linear interpolator is applied. This becomes particularly evident on unstructured meshes. The second circumstance is the fact of runout of the points $\{x - u_1\Delta t, y - v_1\Delta t\}$ over the computational domain boundary. Hence, this requires extra control in general cases. The advection estimation by this method makes the algorithm highly stable and gives it an excellent feature: this scheme does not attach any distortion into the flow frequency characteristics.

Relations (18) are solved by one of the well-known methods. In this case, the FVM with central difference is applied for diffusion, with explicit correction for cross-diffusion for non-orthogonal control volumes (the symmetry of the solution has been tested for symmetric BC). The quality of the projection operation (20) largely depends on the quality of the solution (19), and the best approach is to use the FFT, which is most easily implemented for the orthogonal meshes with periodic BC. Since both Dirichlet and Neumann conditions are applied in this formulation, the (19) is also solved by FVM.

$$U(t + \Delta t) = u_3 - \frac{\Delta t}{\rho} \nabla p \cdot BC_{correction} \quad (20)$$

After performing the velocity field correction (20), the flow parameters are considered as known, however, to be precise, they are quasi-physical due to significant diffusion. Consequently, the turbulence model usage is not strictly necessary from the stability standpoint.

In contrast to classic **Stable Fluid Algorithm** the computational model is complemented by the effective viscosity calculation, that based on theoretical one-parameter RNG turbulence model (21), the features of which are discussed in detail in (Avramenko, Basok, & al., 2013) and (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024).

$$\frac{\partial \eta}{\partial t} + \mathbf{U} \nabla \eta = \nabla \left(\frac{\eta}{Pr_k} \nabla \eta \right) + (\eta_{eff} - \eta) \sqrt{C_v^3 (C_{1\varepsilon}^* + C_{2\varepsilon}) S} - \frac{4C_v^3 \eta^2}{S^2 Pr_k} (\nabla S)^2 \quad (21)$$

Function Pr_k depends on coordinates and estimated by using a precomputed interpolation function η/η_{eff} based on relation (22). The considered range of η/η_{eff} changes from 10^{-7} to 10^7 that covers all possible failures during internal iterations. The coefficients values applied in (21) are: $C_{1\varepsilon}^* \approx 0,9571, C_v \approx 0.0724, C_{2\varepsilon} = 1.68$.

$$\left| \frac{Pr_k^{-1} - 1.3929}{0.3929} \right|^{0.6321} \left| \frac{Pr_k^{-1} + 2.3929}{3.3929} \right|^{0.3679} \quad (22)$$

The solution algorithm (21) is no different from that of equations (18), despite the nonlinearity. This is because the cross-diffusion correction is performed iteratively. Therefore, there is no significant difference between solving a linear equation or a nonlinear one.

Essentially, the above-mentioned turbulence model is required for the synthetic turbulence generator only, which is used as the primary improving procedure of fast flow calculation algorithm. However, a significant restriction is imposed on the calculation procedure: it is assumed that turbulence generation and dissipation are in equilibrium ($\varepsilon = \eta_t S^2$).

The synthetic turbulence field generator is generally similar to that presented in (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024), considering some modifications. The fluctuation field is generated at each computational point based on probability values that are uniform across the computational domain. To transfer the velocity fluctuation fields, a modified form of (23) is used compared to (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024) that better corresponds to the correlation property of the generated vortices.

$$\mathbf{u}' = \{u', v'\} = \mathbf{A}[k] \cdot \sqrt{6} \sum_{n=1}^N \sqrt{q_n} \left[\sigma_n \cos \left(k_n \mathbf{d}_n \cdot \mathbf{r} + \varphi_n - \frac{2\pi t}{k_n Le_{ref}} \mathbf{d}_n \cdot \mathbf{U} \right) \right] \quad (23)$$

Where $Le_{ref} = \max(2\pi/k_e)$ is a characteristic linear size of the most energy-carrying vortex, matrix $\mathbf{A}[k]$ is determined by the Cholesky decomposition via the Reynolds stress tensor, that can be optionally computed either the isotropic formulation (via turbulence kinetic energy) or theoretically (via velocity gradients).

The application of (23) is a sort of turbulent content generation (Garnier, Adams, & Sagaut, 2009) in the main flow, which must be correctly considered. In principle, there are three main options (all of them have been implemented). The first involves generation at the interface (e.g., the inlet boundary) followed by using the transport equation without the diffusion term. It is also possible to utilize a disturbance to the inlet flow velocity, but this can lead to «smoothing» or damping pulsation near the moving region. The second approach to consider the turbulent fluctuations in the mean flow is the artificial forcing (Shur M. L., 2014) and (Grickevich & Garbaruk, 2012)) in the form of special volumetric sources S_U and S_V . Another approach related to volumetric sources is so-called Vortex – Generating Device (Moon, Jeong, Park, Ha, & Jeong, 2023), which possesses, perhaps, the best properties in terms of its «noisiness» and practically has no limitations on the «complexity» of the flow.

In case of the coupled oscillations of fluid and solid body, the following characteristics of the interaction between the mean and pulsating flows should be considered. For large oscillation amplitudes, the pulsation field can be examined as a superposition of true turbulent fluctuations and those induced by the moving wall. Thus, accounting for induced velocity and pressure pulsations (which generally inherit the wall's oscillation frequencies) in the mean flow can lead to an artificial increase of hydrodynamic force. To avoid the difficulty of «self-induction» of fluctuations, a variant where the turbulent content generator for the entire computational domain is applied exclusively as an additive for calculating the hydrodynamic force is considered. Due to the substantial viscosity of the computational algorithm, the fluctuations induced by the moving boundary will be sufficiently strongly damped by the mean flow. In fact, a non-rigid connection is realized between the synthetic turbulence and the average velocity field, which is influenced by the moving boundary.

The definition of static pressure pulsations (the main source of unevenness in flow around a body) is based on the well-known Poisson equation (24), and this equation also governs induced pulsations from the moving boundary.

$$\nabla^2 p' = -\rho[\alpha_p 2\nabla \mathbf{U} : (\nabla \mathbf{u}')^T + \beta_p \nabla \cdot \nabla \cdot (\mathbf{u}' \otimes \mathbf{u}' - \mathbf{R}_{URANS})] \quad (24)$$

Where $\mathbf{R}_{URANS}$ – Reynolds stresses, which are determined using (21), ρ – density of the medium, α_p and β_p – control coefficients of components in the pulsation field.

An alternative, less well-founded, but significantly faster, method is the so-called algebraic method of estimating pulsations, which also utilize gradient relations. This method typically requires calibration and constraints, but can be useful as a first approximation due to the lack of boundary conditions as such. Table 18 shows the various groups of relations that are used to estimate pressure pulsations.

Table 18: Algebraic relations for calculating the pressure fluctuation field

Relations	Key (ST)
Algebraic relations ¹	
$p' = C_{st}\rho(u' \cdot u')$	(K10)
$p' = C_{st}\rho \cdot \text{sgn}(u')^T (u')^2$	(K11)
Algebraic relations of gradient type²	
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} \nabla u' : (\nabla u')^T;$ $\max(k_{cut})^{-2} = L_{ref}^2$	(K20)
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} (\nabla u')^2$	(K21)
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} \text{sgn}(\nabla u')^T (\nabla u')^2$	(K22)
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} \nabla U : (\nabla u')^T$	(K30)
$p' = C_{st}\rho \cdot \max(k_{cut})^{-2} \nabla U \cdot \nabla u'$	(K31)
<p>Notes:</p> <p>¹applied to the centers of control volumes adjacent to the wall.</p> <p>²applied to boundary nodes located on the wall.</p> <p>The characteristic size L_{ref}^2 can be selected individually for each CV.</p> <p>C_{st} – the amplification factor, for (K10)-(K11) the typical value is from 0.5 to 1.5. Generally, requires calibration.</p>	

After all components of the average and synthetic fields have been calculated, it is necessary to determine the forces acting on the rod. The general vector form is represented as (25).

$$F_{tot} = \left\{ \int_S [- (p + p') n + \rho(\eta + \eta_t^C) [(\nabla U + \nabla U^T) + (\nabla u' + \nabla u'^T)] \cdot n] dS \right\}_x \quad (25)$$

Where η_t^C is a turbulent viscosity calculated at the center of the CV adjacent to the solid wall (optionally activated).

When implementing (25), the procedure takes into account certain features related to the ability to control the contributors. It is possible to take into account the contribution of shear stress components in determining the hydrodynamic force both from the mean field and from the synthetic turbulence side. It is also possible to control the contribution of the

main flow and the synthetic one to the total force. Moreover, this can be done as a function of the Reynolds number and thus perform calibration based on experimental data.

As mentioned above, each equation type is solved using a modified cell-oriented FVM solver. This modification involves taking into account boundary nodes, for which separate equations are generated. These equations are similar to FDM, accounting for the non-uniformity of the mesh and the computational boundary.

First of all, this allows many calculations (especially synthetic turbulence) to be vectorized, making the computation of cumbersome operations more efficient. The gradient and divergence calculation operators are based on a single-stage algorithm based on the Green-Gauss theorem with linear interpolation onto faces from the centers of the CV. For boundary faces, information is taken from the computational nodes. The gradients of the synthetic quantities $\{u', v'\}$ are calculated using a series similar to (23).

To calculate auxiliary quantities (for example, $BC_{correction}$) equation (26) is solved at each time step and the distance from the solid walls is calculated.

$$\begin{aligned}\nabla^2 \varphi &= -1 \\ d_{wall} &= |\nabla \varphi| + \sqrt{|\nabla \varphi|^2 + 2\varphi}\end{aligned}\quad (26)$$

The rod's displacements within the computational domain are accounted for in two ways. In one case, this is accomplished natively by applying a block generator near the moving region (Figure 97).

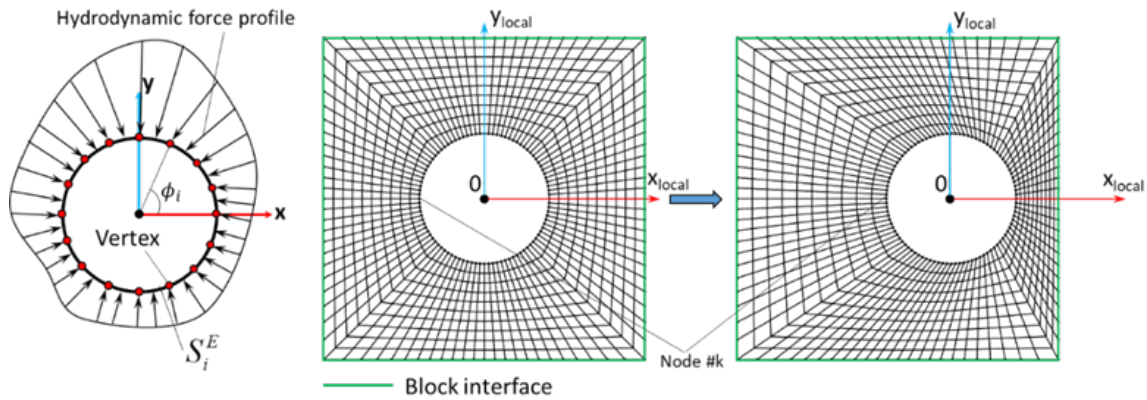


Figure 97: Dynamic Mesh Block

In the second case, a simple model of diffusion deformation of the mesh along a selected direction is used (separately for each coordinate) with a variable diffusion coefficient depending on the selected options (the Laplace equation is solved). The diffusion coefficient can be calculated either as inversely proportional to the local value of the CV, or as inversely proportional to the square of the volume, as well as to the characteristic linear dimension along the direction of displacement.

To interpolate the results from the centers of the CV to the nodes of the computational mesh, the Lagrangian interpolation method is used (Thomas & Xu, 1995).

3.2 Definition of oscillators and forces calculation method

In general, various types of oscillators have been implemented. The report (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024) examines simple systems in the form of a concentrated mass on ideal springs or so-called angular oscillatory systems of the **Rigid Body** type. The solvers based on the Normal Form method for beams in the Euler-Bernoulli and Timoshenko approximations are represented in the report (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025). In general, such methods are well suited for the concept of fast calculations, but have one significant drawback from the point of view of the formulation. The point is that methods based on natural frequencies (plus correction based on the influence of the fluid) are well suited for one-way analysis, but in general for turbulent flows they are in some sense “rigid” or predetermined by the spectral feature. This means that nonlinear interactions (especially vortex structure and separation points) can generally be “suppressed” by the frequency that is embedded in the oscillator. For this reason, it is preferable to remain in differential form and solve the coupled equation. Accordingly, in this case, in addition to the connection through force members, the problem of forming correct (special) BCs on the moving boundary arises.

Within the FDM solver, a two-section (each section is homogeneous) Euler-Bernoulli beam is implemented, which can move along a single selected direction:

$$-\frac{d}{dz^2} \left(EI \frac{d^2 w}{dz^2} \right) + q + \frac{dm}{dz} - \rho A \frac{d^2 w}{dt^2} - D \frac{dw}{dt} = 0 \quad (27)$$

where w – is the transverse beam displacements (in the direction corresponding to the X-axis), m ; q – is a distributed load, N/m; m – is the length distributed moment along the uniform beam segment, (N·m)/m; D – is the damping coefficient, kg/(m·s); I – is the beam cross-section inertia bending moment, m⁴; ρ – is the beam material density, kg/m³; A – is the beam cross section, m².

The beam sections are assumed to be symmetrical about the principal axes of the sections, and axial forces and torsional moments are not considered. For each individual section of the composite beam, only transverse force factors are considered, ignoring damping and distributed externally applied bending moments, which significantly simplifies the formulation.

To obtain the unique (single-valued) solution of the system equations (27), initial and boundary conditions are required, as well as the rigid connection conditions for the two uniform segments of the beam model. The procedure initial conditions are defined as the initial distributions of the displacement's projection and the beam points velocities. Various combinations of the force and/or kinematic boundary conditions can be applied at the beam ends. The rigid fixed and a free beam end were chosen among all available equation pairs to set up the boundary conditions. Mathematical formulation of each applied conditions is represented in Table 19.

Table 19: Mathematical formulation of the combinations of basic boundary conditions used for the beam ends in procedure

Condition type	Variable's equations	
Rigid fixed beam end	1	$w _{z=z_b} = 0$
	2	$\frac{dw}{dz} _{z=z_b} = 0$
Free beam end	1	$EI \frac{d^2w}{dz^2} _{z=z_b} = 0$
	2	$EI \frac{d^3w}{dz^3} _{z=z_b} = 0$

In the junction point of the two uniform beam segments, the equality conditions of the displacements, rotation angles, moments, and forces at the adjoining ends are imposed. In this case, no concentrated force and/or bending moment is applied at the adjacent point. Mathematical formulation of the stated above conditions listed in Table 20.

Table 20: Mathematical formulation of the two ends conjunction of the uniform segments in the beam

Conjunction type	Equation general view
Displacements	$w_1 _{z_1=L_1} = w_2 _{z_2=0}$
Rotation angles	$\frac{dw_1}{dz_1} _{z_1=L_1} = \frac{dw_2}{dz_2} _{z_2=0}$
Force moments	$EI_1 \frac{d^2w_1}{dz_1^2} _{z_1=L_1} = EI_2 \frac{d^2w_2}{dz_2^2} _{z_2=0}$
Forces	$EI_1 \frac{d^3w_1}{dz_1^3} _{z_1=L_1} = EI_2 \frac{d^3w_2}{dz_2^3} _{z_2=0}$

To be more specific, let's consider the AMOVI (see section 2.1) beam. In Table 20, the parameters corresponding to the first segment of the rod, which has a rectangular cross-section, are denoted by the index «1», while to the second one with an annular cross-section are denoted by the index «2». The variables z_1 and z_2 – represent the distances measured from the beginning of the respective uniform segment of the AMOVI beam. The Young's modulus of both segments is assumed to be identical as they are made of the same material. In the AMOVI model, the primary loading factor is a uniformly distributed linear

load applied to the annular cross-section segment, which represents the effect of the fluid flow under experimental conditions. The assumed loading scheme of the tube and its blade is depicted in Figure 98.

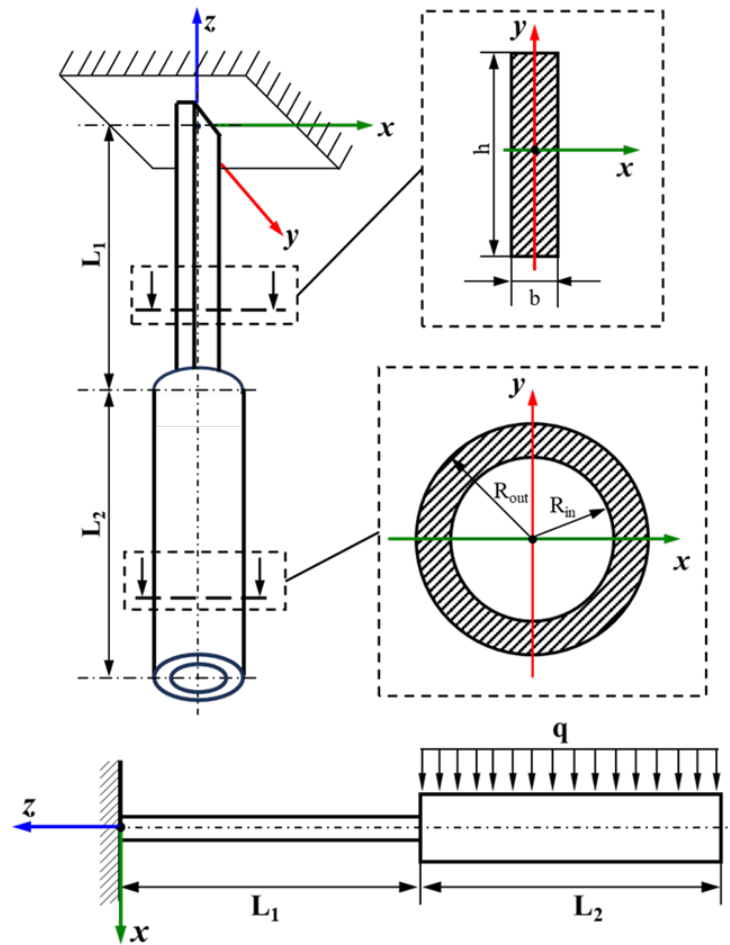


Figure 98: General view of the two-part AMOVI beam

In the procedure, the finite difference method was utilized to numerically solve the equations system (27). A central difference scheme was used to approximate the fourth-order derivative of displacement with respect of the coordinate, while the second-order time derivative was approximated using an implicit, equilibrium second-order scheme based on values from the previous and pre-previous time steps. Despite the simplicity of constructing such a solver, significant difficulty arises, which is associated with the fact that with a sufficiently large integration step over time, significant dissipative properties of the system appear. If the discretization along the length is insufficient, the natural frequencies are distorted (the smaller the discretization in space, the shorter the oscillation period). A strange situation arises when a high-quality solution of the displacements for the current time step requires a comparable amount of time as for solving a one-parameter turbulence model (here it is assumed that the time integration step for the beam is significantly smaller than for (21)).

To overcome such difficulties, time discretization is represented in terms of kinematic quantities (velocity $s = dw/dt$ and acceleration $a = d^2w/dt^2$ using the concept of fractional time steps. Then the discrete analogue for the elementary section of the beam (27), taking into account all the assumptions, is represented as (28), and the increments of displacement and velocity per time step as (29).

$$EI \left(\frac{w_{i-2}}{\Delta z^4} - 4 \frac{w_{i-1}}{\Delta z^4} + 6 \frac{w_i}{\Delta z^4} - 4 \frac{w_{i+1}}{\Delta z^4} + \frac{w_{i+2}}{\Delta z^4} \right) + \rho A a_i = q \quad (28)$$

Where a_i – is the acceleration of beam computational point at the current time step.

$$\begin{cases} w_i = w_i^0 + s_i^0 \Delta t + 0.25 \left[(a_i)^2 + (a_i^0)^2 \right] \\ s_i = s_i^0 + 0.5(a_i + a_i^0) \end{cases} \quad (29)$$

Where the w_i^0 – is the displacement at the previous time step; s_i^0 – is the material point velocity at the current time step; s_i^0 – is the velocity at the previous time step; a_i^0 – is the material point acceleration at the previous time step.

The influence of discretization and time step on the characteristics of free oscillations when using kinematic parameters is shown in Figure 99.

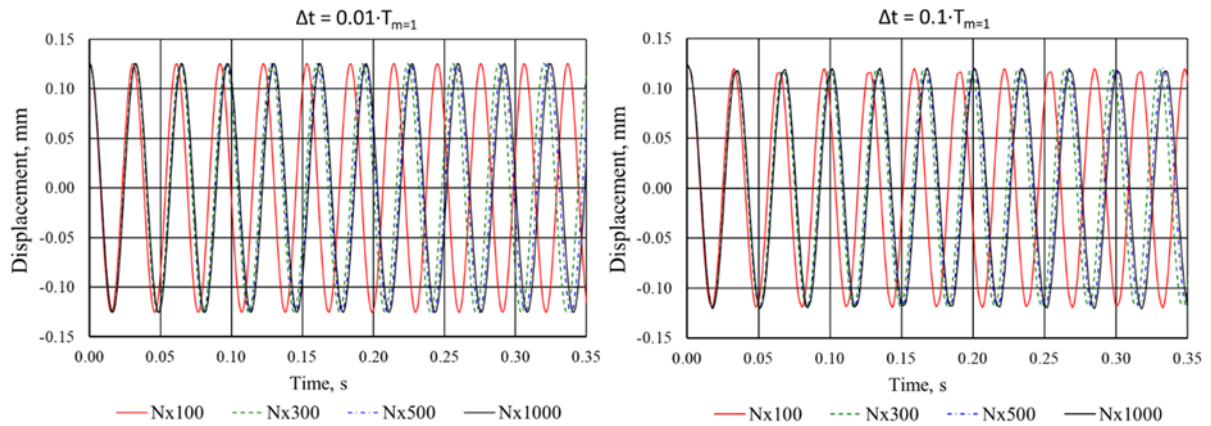


Figure 99: The influence of discretization and time step on the characteristics of free oscillations

Thus, spatial discretization (almost 250 points along the entire beam length) has a decisive influence on the amplitude and frequency characteristics. The numerical damping in the derivative description has been significantly reduced.

In this sense, the use of solvers built on the basis of normal form oscillations has advantages in terms of the required number of calculation points along the length, as well as the speed of calculations. Perhaps the only drawback that appears in special cases is the absence of an asymptotic solution for large time steps (the step is greater than the period of the first mode), as well as the rigidity of the spectrum for fully coupled oscillations.

All of the above is implemented in a special calculation procedure that supports the use of two-dimensional meshes in *.msh format, as well as three-dimensional meshes in *.cfx5 format with subsequent conversion into a 2D structure.

3.3 Organization of calculations and available options

All the above is implemented in a special calculation procedure that supports the use of two-dimensional meshes in *.msh format, as well as three-dimensional meshes in *.cfx5 format with subsequent conversion into a 2D structure. Several versions of internal structured/unstructured mesh generators are also provided. Table 21 summarizes the main options that are implemented in the first version of the procedure for the end-to-end calculation of coupled oscillations. Such a variety of parameters is necessary both for optimizing calculations and for introducing additions in future versions.

Table 21: Main parameters of the calculation procedure

Parameter	Value	Description
TypeGeometry	ROD/RODS	Type of calculation geometry
MeshType	FLUENT/CFX/GENblock	Mesh generation type
dicrOption ¹	ZCA/MCA/OCA/ORA	Type of cross-diffusion correction: Zero correction algorithm (ZCA) Minimum correction algorithm (MCA) Orthogonal correction algorithm (OCA) Over-Relaxation algorithm (ORA)
intOption	CEN/CEN-F	Method of flow interpolation
gradBCzero	True/False	Calculating the gradient at the centers
divBCzero	True/False	Calculating divergence at the centers
BeamType	ebFDMkin/ebNF/tNF	BEAM solver: Euler-Bernoulli FDM – ebFDMkin Euler-Bernoulli (analytics) – ebNF Timoshenko Beam (analytics) - tNF
displFlag	Min/Ave/Max	Reference displacement value for flow
LargeDeflFlag	True/False	Accounting for large beam deformations
FlagLagrAdvect	True/False	Application of Lagrangian advection to U-V velocity variables
FlagLagrAdvectTurb	True/False	Application of Lagrangian advection to the RNG 1-eq turbulence model
LagrAdvType	True/False	Using the internal step

LagrCFL	1.0	Courant number for the internal step
LagrMaxSubStep	10.0	Maximum number of time sub-steps for Lagrangian advection
LagrFlagBC	True/False	Forced correction of the boundary conditions at the stage of Lagrangian advection
VortexSynForce	True/False	Synthetic vortex generator
WDCorrection	True/False	Wall Distance correction BC
dempFmin	0.0	Minimum multiplier on the wall
dempFcoef	5.0	Gradient multiplier of a function
dempFn	1.0	Damping coefficient
WDSTBL	True/False	Damping function at the wall for synthetic turbulence
WDTurbBL	True/False	Damping function at the wall for the turbulence model
TurbViscVelDiff	1.0	The coefficient of influence of turbulent viscosity on velocity diffusion
TurbViscToForce	0.0/1.0	The influence coefficient of turbulent viscosity for the force calculation
ShearForce	1.0	Accounting for the shear stresses when calculating forces
uransForce	1.0	The coefficient of amplification/influence of the average flow field on the force
stForce	1.0	The coefficient of amplification/influence of the synthetic flow field on the force
signPF	-1.0/0.0/1.0	Auxiliary variable for the force from the static pressure field
uransEV	0.0/1.0	Reynolds stress for synthetic turbulence: 0.0 – by kinetic energy. 1.0 – considering the velocity gradient.
MeanPressSourceCor	1.0	Contribution of the average source field in the calculation of pressure pulsations
StartTurbTime	Any (by default 0.0)	Beginning of accounting for the influence of synthetic turbulence, [s]
StartDisplTime	Any (by default 0.0)	Start of accounting of rod movements, [s]

StSource1	1.0	α_p , see (24)
StSource2	1.0	β_p , see (24)
StCoefPress	1.0	Pressure pulsation amplification factor
relaxWall, relaxU, relaxV, relaxUV, relaxP, relaxTurb, relaxMD	0.1	Relaxation coefficients for each variable
visualRANSfields	True/False	Visualization of mean fields at each time step
visualSTfields	True/False	Visualization of synthetic fields at each time step
visualMeshDispl	True/False	Visualization of the deformed mesh at each time step
AutoRange	True/False	Automatic selection of limits for each variable
PrintSolverInfo	True/False	Output of complete information about each solver and local algorithm
¹ – cross-diffusion correction is largely the reason for the presence of internal iterations (usually up to 4). If uniform meshes are used, then the solution for the linear part of the problem can be accelerated. Default values are highlighted in bold in the table.		

Figure 100 shows the boundary conditions (within the AMOVI benchmark) and the typical computational meshes used for calculations. It should be noted that all necessary conditions that applied considering the end-to-end calculation (application of Lagrangian advection) are indicated.

At the inlet, either a uniform velocity and turbulent viscosity profile or a functional dependence on the coordinate can be applied (30).

$$V_{in}(x) = \frac{Q}{S_{in}} coefV_{max} \left(1 - 0.985 \left| \frac{2x}{H} \right| \right)^{1/m} \quad (30)$$

Where Q – is the volumetric flow rate, S_{in} – the cross-sectional area at the channel inlet, H – the channel width.

Coefficient m is taken equal 6.6, and the value $coefV_{max}$ calculated based on the integration of the profile over the channel width ($coefV_{max} \approx 1.152$).

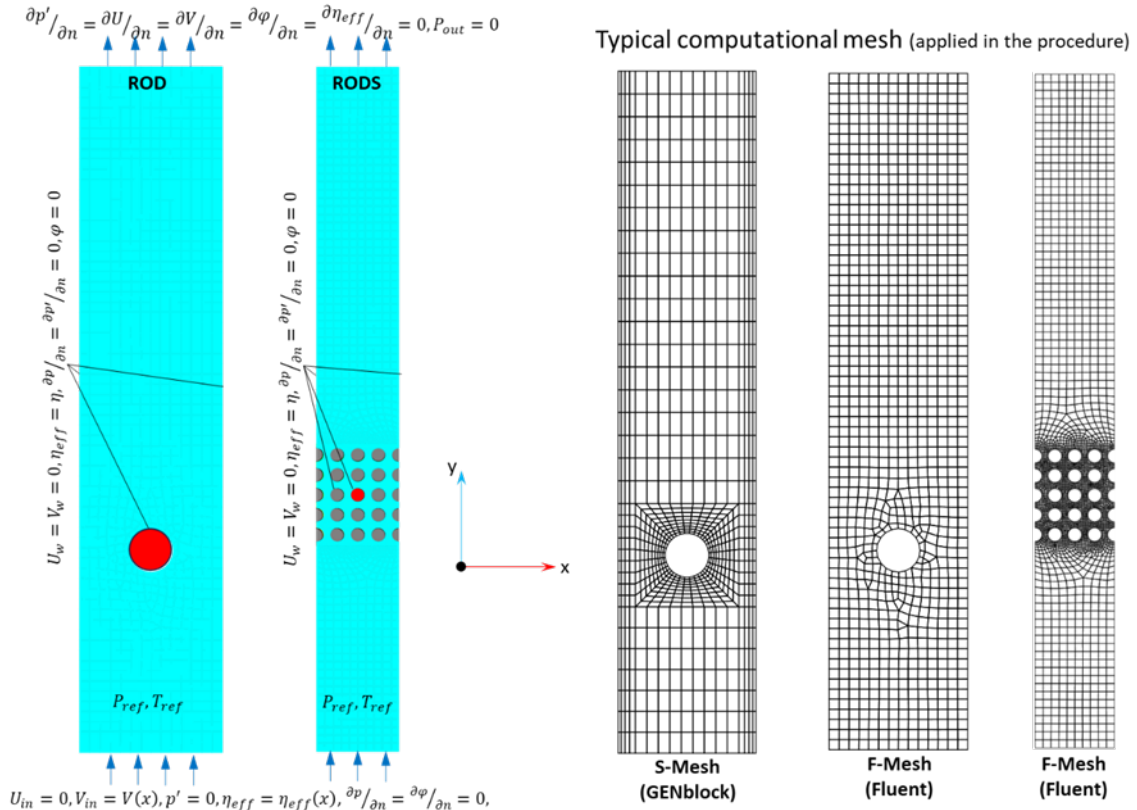


Figure 100: Boundary conditions and typical meshes applied in the calculation procedure

The procedure for calculating the Lagrangian self-advection, as well as the convective term in the turbulence model, can (options **FlagLagrAdvect**, **FlagLagrAdvectTurb**) use its own estimate of the integration time step based on a given Courant number (**LagrCFL**). The transfer calculation is implemented in two ways: based on the transfer from the old to the nearest node (analogous to index interpolation) and also using trilinear parametric interpolation with the replacement of extrapolation with a special “dimensional” function. For this purpose, a virtual rectangular mesh area is created with an internal perimeter in the form of the dimensions of the calculated geometry. This method is generally more accurate (transfer is virtually independent of the time step), but is significantly more diffusive and resource-intensive (requires optimization). The first option is further used.

If the **FlagLagrAdvect** and/or **FlagLagrAdvectTurb** options are not enabled, the turbulence model describes the scalar flow using classical upwind (first- or second-order). In this case, the intermediate velocity fields are calculated by solving system (13). The use of the SOU scheme does not introduce any additional complexity for an unstructured mesh due to the organization of the cross-diffusion correction algorithm (the **dicrOption** option). Structurally, everything remains identical; the only thing is that the calculation speed drops by approximately 5-10% due to operations related to calculating gradients on the edges. The basic block diagram of the calculation procedure is shown in Figure 101.

The calculation procedure is organized in such a way that, depending on the call point of the corresponding solver, it is possible to flexibly configure the coupling. For example, if

the synthetic field is dominant compared to the influence of non-uniformity from the mean field, then the direct URANS can be calculated once, or, for example, on a characteristic turbulent time scale. It is also possible to implement a fundamentally implicit calculation scheme, where synthetic turbulence and displacements are calculated jointly until the convergence criterion is reached at each time step. This configuration is not used in subsequent calculations.

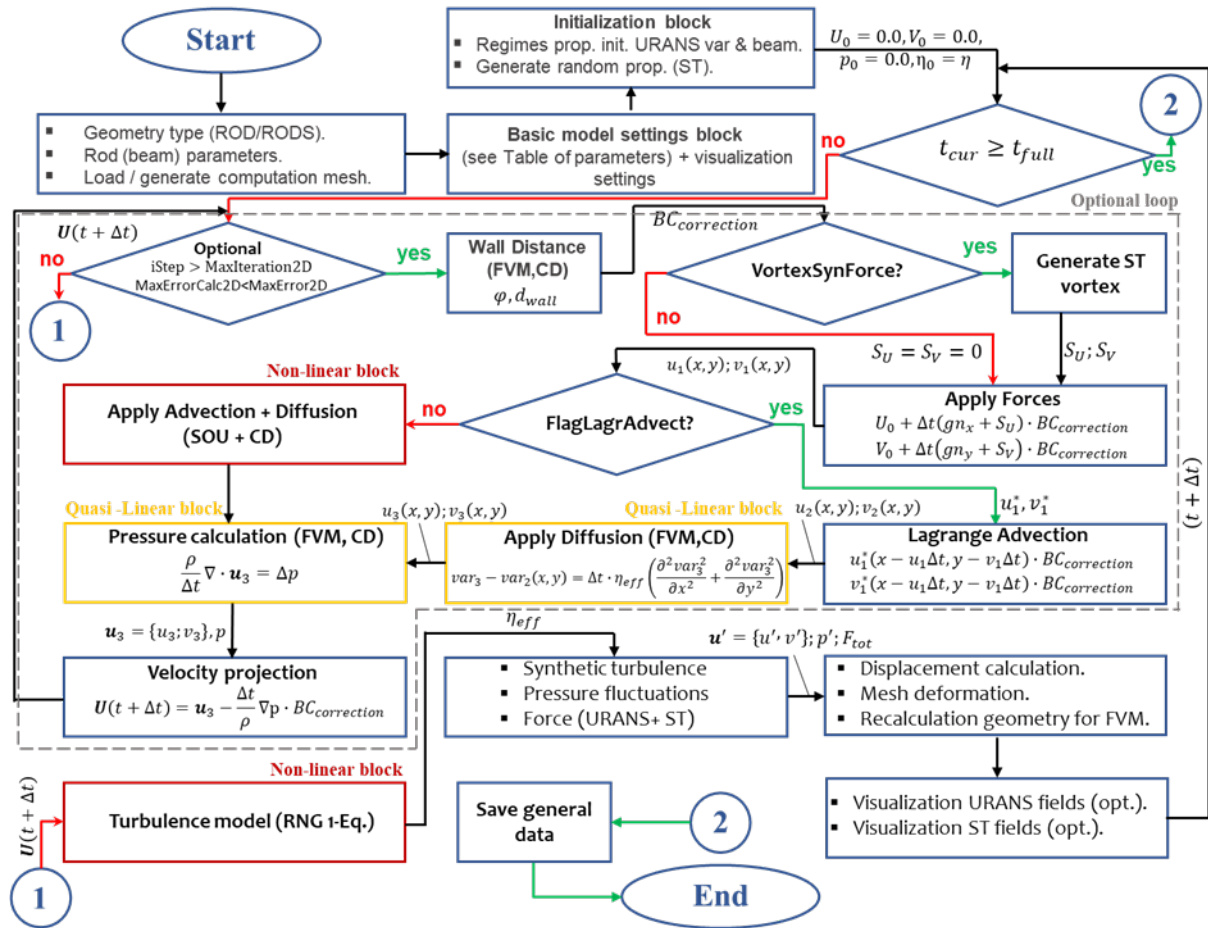


Figure 101: The basic block diagram of the calculation procedure

In addition to the procedure, a user library for the ANSYS CFX calculation code has been implemented, which contains solvers for beams, as well as a procedure for generating synthetic turbulence (3D) with an algebraic estimate of pressure pulsations (Table 18). Each solver is implemented using Junction Box, which allows for flexible interaction with the main ANSYS CFX kernel and, depending on the connection point, the organization of explicit or implicit coupling. The library supports parallel calculations. For ease of calculation configuration, user options are provided, which are accessible by declared names through User Input (see Table 22).

The synthetic force acting on the rod, in contrast to the calculation procedure, takes into account only the contribution of the pulsating pressure, therefore, when using algebraic

non-gradient relationships (see Table 21, K10-K11), a problem arises. The velocity pulsations on the wall are assumed to be zero, while the pressure pulsations must be different from zero. Strictly speaking, the K10-K11 ratios and similar ones are not applicable for calculating the effect of a synthetic field on a wall. For this purpose, the library provides for the calculation of the force around the rod at a distance exceeding the momentum displacement layer, which makes it possible to ignore the attenuation of velocity pulsations near the wall. In the calculation options considered below, this distance is ~ 1.5 mm.

Table 22: User input parameters for Lib ANSYS CFX

Parameter	Value	Description
LRef ¹	0,002 [m]	Thickness of a single control volume in the transverse direction.
NModes	1-10	Maximum number of modes considered for ebNF and tNF solvers
NSubSteps	1-10	Number of time integration sub-steps for the beam solver
NX	100	Number of partitions for half a beam
Res Option	0/1/2	Reference displacement value for flow 0 – Min, 1 – Ave, 2 – Max
Solver	FDMK /ebNF/tNF	BEAM solver: Euler-Bernoulli FDM (kin) –FDMK Euler-Bernoulli (analytics) – ebNF Timoshenko Beam (analytics) - tNF
xRef	0.0 [m]	Absolute coordinates of the center of considered rod
yRef	0.0 [m]	
coefSTpuls ²		see Table 18
pPulsOpt	Key (ST)	see Table 18
signP	1/-1	Additional option for testing
stForce	1.0	Force amplification coefficient from a synthetic field
uransForce	1.0	Force amplification coefficient from a mean field
turbForce	1/0	Synthetic field calculation activator
turbM	w BASE/eBASE	Option of the two-parameter model used: w – omega base; e – epsilon base
nameROD	WALLROD	Name of the flexible rod boundary

nameBL	WALLROD	Name of the flexible rod boundary on which the algebraic pressure pulsation is calculated according to the calculation layer
¹ – ANSYS CFX supports quasi-2D modeling, provided that the computational mesh in the transverse direction has a thickness equal to one control volume. Therefore, this must be taken into account when calculating the total or specific hydrodynamic load. ² – The proportionality coefficient of pressure pulsations, especially for gradient algebraic models, is a grid function (similar to LES) and requires additional testing. Default values are highlighted in bold in the table.		

It is worth noting that synthetic turbulence in the calculation procedure and in the ANSYS CFX user library have differences not only in spatial implementation. Firstly, this version of the library does not have the ability to apply the Poisson equation to pressure pulsations due to the technical difficulty of calculating the diffusion of stresses in the average flow. In this case, the gradient of the pulsation quantities is calculated analytically based on differentiation (23), and the calculation procedure uses the Green-Gauss theorem and the central difference for stress diffusion. Despite this, the fundamental difference is still the fact that in ANSYS CFX the velocity pulsation generator is two-parameter (k and ϵ or k and ω), while in the calculation procedure it is actually based on one parameter – eddy viscosity.

3.4 Validation of the developed Synthetic Turbulence Generator

This subsection presents the main results of the applicability assessment of the calculation procedure based on the Stable Fluid Algorithm with a one-parameter synthetic turbulence generator. The application of synthetic turbulence as a framework for the ANSYS CFX computational code is also discussed. The AMOVI experimental data provided for flexible tube cases (see description in section 2.1) was used for validation. Table 23 summarizes the main parameters of AMOVI test cases provided for configurations with Single Flexible Tube and Flexible Tube in a Rigid Tube Bundle.

Table 23: Overview of available AMOVI test cases with FIV

	Flow Rate (L/s)	Pressure (bar)	Temperature (°C)	PIV/LDV measurements data available
Single flexible tube (lift direction)	0.5	2.19	18	-
	1.7	3.33	21	-
	2.9	3.20	25	-
	4.0	3.53	26	-
	0.5	1.48	17	-

Flexible tube (lift direction) in a rigid tube bundle	1.1	0.9	17	-
	1.6	1.07	17	PIV
	1.95	3.17	23	-

Further validation activities were performed for Single Flexible Tube cases with 1.7 l/s, 2.9 l/s and 4.0 l/s flow rates, as well as for the Flexible Tube in a Rigid Tube Bundle with 1.6 l/s and 1.95 l/s.

3.4.1 Validation of STG on AMOVI Single Flexible Tube Experiment

This section presents the results of simulations of the AMOVI Single Flexible Tube test cases used for validation. For each simulated case, both the displacements of the flexible rods and the appropriate APSD are compared with experimental data. Vibration analysis was performed using either the ebNF (Euler-Bernoulli Normal Form) option based on analytical solutions described in section 4.1.2 of the report (Filonov, et al., 2026) or ebFDM (Euler-Bernoulli FDM) method described in section 3.2. Of greatest interest are the regimes between which a significant increase in the vibrations amplitude occurs. Therefore, cases with the maximum flow rate of the coolant through the test section are necessarily considered.

3.4.1.1 Simulation of the Single Flexible Tube experiment with $Q=4.0$ l/s

As already mentioned, the most interesting regimes are those with maximum flow rates, since in these cases, the interaction between the moving wall and the flow is fundamentally coupled. Therefore, both the physical adaptation of the Stable Fluid Algorithm (SFA) based on synthetic turbulence and the user library in ANSYS CFX with various algebraic expressions (Table 18) for pressure pulsations need to be tested in such regimes.

The SFA+ST simulations used default settings (Table 21), selected based on preliminary testing (Lagrangian advection is used). Note that no calibration coefficients or correction parameters were selected based on the considered experiments, or any other experiments. Random variables were generated individually for each run of the selected case. The upper limit of the integration time step was 5 ms.

For testing ANSYS CFX+ST, a similar calculation mesh (F-mesh) was used, as well as an additionally modified F-Mesh (BL), which additionally utilizes prismatic meshing near the wall. The SST turbulence model, first-order time integration, and the High Resolution advection computational model were applied. Random parameters for the synthetic turbulence generator were predefined and uniform across all cases.

For the current case with $Q=4.0$ l/s, all options of algebraic expressions for the turbulence model (Table 18) were tested with the following proportionality coefficients C_{st} : K10-K11 – 1.0; K20-K22 – 10^{-5} ; K30-K31 – 10^{-2} .

Figure 102 shows the displacement results and the corresponding vibration spectrum for a single flexible tube with a flow rate of 4.0 l/s. No fundamental difference in spectral

characteristics was found when using ebNF and ebFDM (FDMK), only the amplitudes differ (within 10-15% in absolute value, with ebNF being higher than ebFDM).

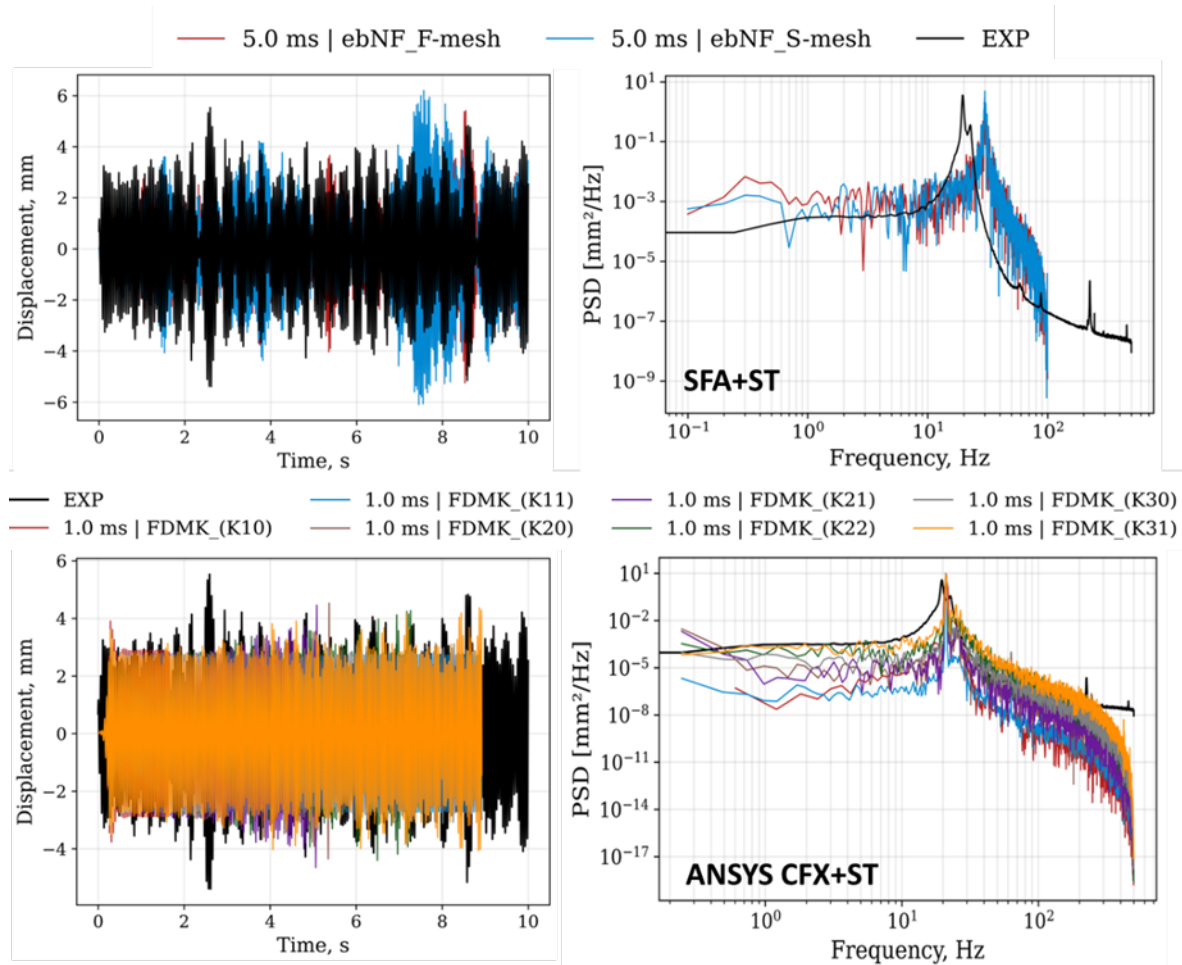


Figure 102: Results of displacements calculation for the single flexible tube with $Q = 4.0$ l/s

The amplitude can be "regulated" by selecting a reference point on the beam (Figure 103) for transmitting displacements to the two-dimensional flow. In current simulations, the geometric center of the active part of the rod was chosen as the reference point for test calculations. Moreover, based on the range of beam displacements, it is clear that the results are quite sensitive to the discretization of the region near the rod, which for such regimes mainly affects the parameters of synthetic turbulence, the intensity of which determines the amplitude of vibrations.

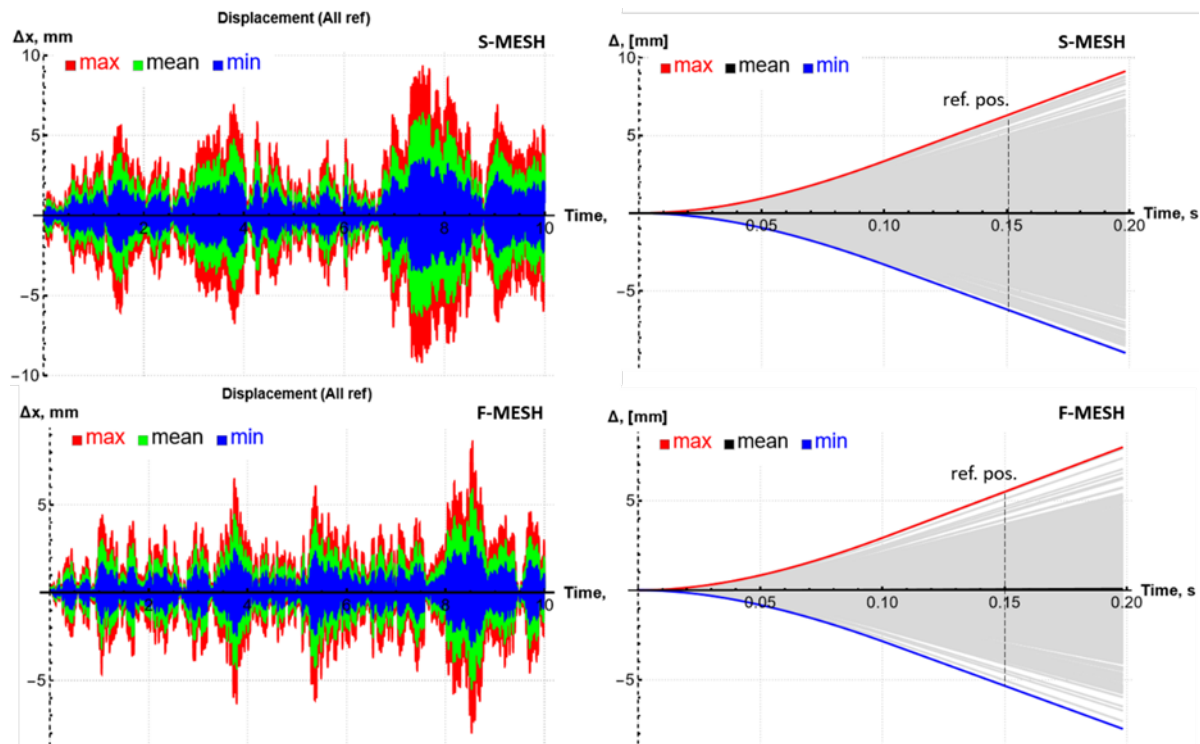


Figure 103: Beam displacements over time at different reference points and the range of deviations obtained for the single flexible tube case with $Q = 4.0$ l/s using SFA+ST

The results obtained using SFA+ST generally satisfactorily predict the amplitude values of the coupled vibrations. The peak in the spectrum is slightly shifted toward higher frequencies compared to the experiment, which is explained by the "effect" of pre-assigned frequencies when using the normal form method (the first mode of vibration corresponds to a frequency of ~ 30.04 Hz for a composite beam). When using Lagrangian advection, the natural frequencies of the system are not distorted. Therefore, it is preferable to use solvers without a pre-assigned frequency spectrum, although this is more computationally intensive.

The results obtained using ANSYS CFX + ST for various pressure pulsation estimation options also indicate satisfactory vibration prediction compared to experiment. Several important features should be highlighted:

- The time integration step was set to no more than 1 ms, which is due to excessive numerical flow diffusion at larger steps (2.5 ms and 5 ms were tested). This means that oscillations were either not excited on meshes similar to those in SFA or attenuated rather quickly.
- The second feature is the algebraic estimation of pressure pulsations. Options related to gradient algebraic expressions (see Table 18, K20-K22; K30-K31) turned out to be the most applicable, although their physical behavior still requires careful interpretation.

In general, in the SFA+ST procedure the flow is predominantly diffusive (Figure 104) due to the algorithm, but the coupling with the turbulence model almost always induces complex

oscillations. In regimes with relatively high flow rates, the force acting on the rod is mainly determined by the synthetic one-parameter field.

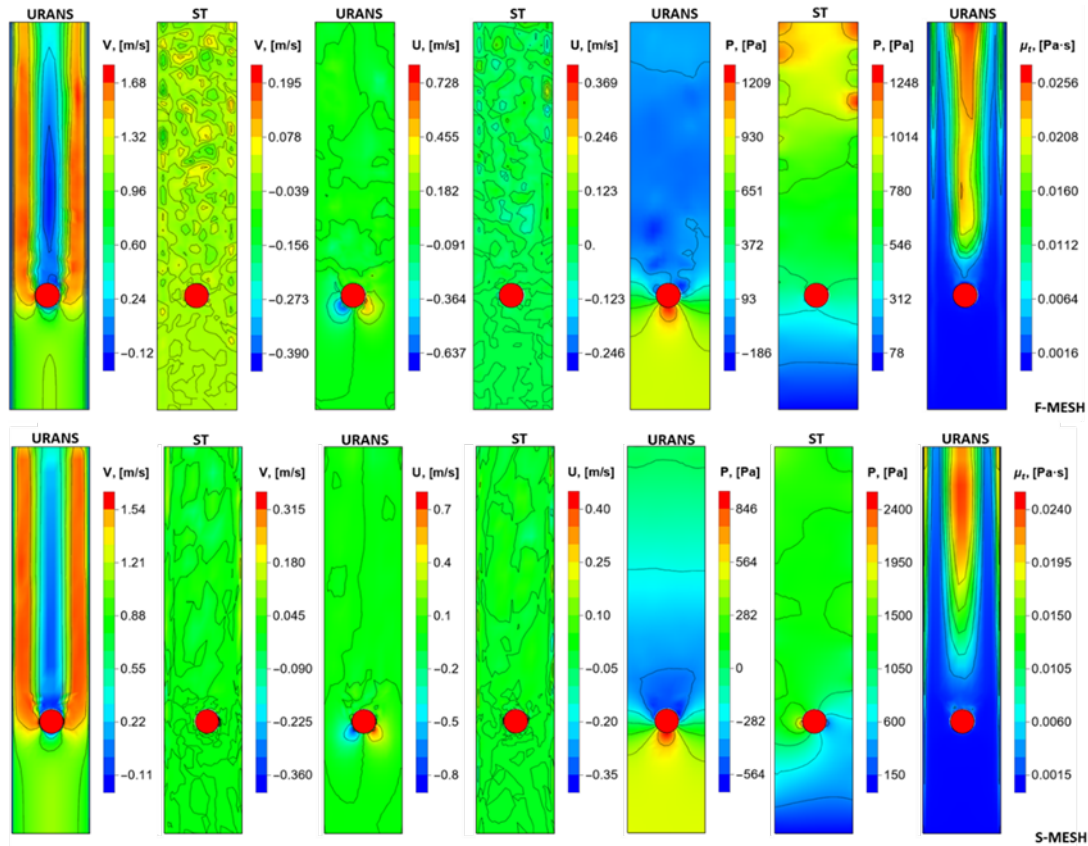


Figure 104: Visualization of the main instantaneous fields calculated using SFA+ST for S-Mesh and F-Mesh meshes (case $Q = 4.0$ l/s, automatic legend)

Using larger time steps in ANSYS CFX+ST (> 1.0 ms) results in a flow that is over-diffusive (Figure 105, a), as displayed by the eddy viscosity profile, which is similar to the one-parameter RNG turbulence model used in SFA+ST. In this case, oscillations are practically not induced. If the integration time step is less than or equal to 1.0 ms, then oscillations are induced and the eddy viscosity profile differs significantly from the calculation procedure (due to the two-parameter turbulence model, in contrast to the procedure where a one-parameter model and Lagrangian advection are implemented). The method for determining pressure pulsations certainly plays a role. Figure 105 shows that the use of the proportionality relations of the signed/unsigned square of the pulsation velocity (Figure 105, a, K10-K11) is fundamentally different from the gradient method for determining pressure pulsations (Figure 105, b, K31). In the second case, the character of p' is local and is caused by some kind of flow obstruction with subsequent relaxation. Overall, this behavior is more plausible in the physics of turbulent flows, but this method of determination is quite sensitive to the computational mesh (essentially possessing the same property as LES at the boundary when applying shear stress conditions). This circumstance requires mandatory calibration of algebraic pressure pulsation models.

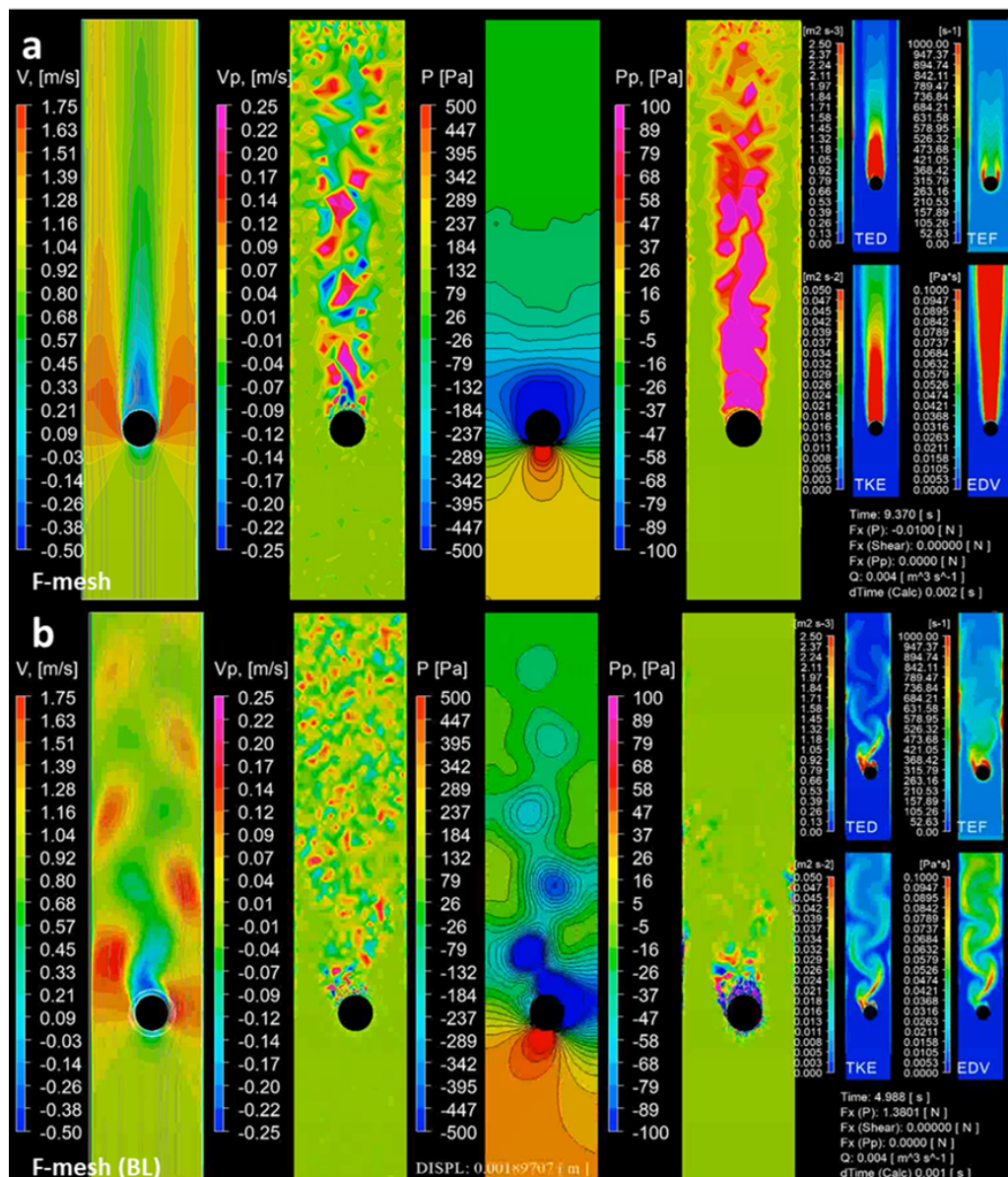


Figure 105: Visualization of the main instantaneous fields calculated using ANSYS CFX + ST for the single flexible tube case with $Q = 4.0$ l/s (a – integration time step 2.5 ms, K10; b – integration time step 1.0 ms, K31)

Table 24 provides summary information of the comparison of experimental results with calculated ones. The estimated displacement characteristics demonstrate that the prediction using ANSYS CFX + ST is somewhat better than that of the SFA+ST computational procedure. However, while testing the ANSYS CFX in a quasi-2D approximation for the same regime without synthetic turbulence, the results were similar (Filonov, et al., 2026). In other words, for this regime, the ST procedure has a significant impact specifically within the SFA procedure framework. For this reason, it is particularly interesting to evaluate the vibrations parameters of regimes with lower flow rates, where resonant properties (Fluid Elastic Instability) are virtually absent.

Table 24: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube (Q=4.0 l/s)

	EXP	SFA+ST (EBNF)		ANSYS CFX+ST (FDMK)						
		F-mesh	S-mesh	K10	K11	K20	K21	K22	K30	K31
Displacement RMS, mm	1.945	1.37	1.89	2.04	1.92	1.97	1.98	1.93	1.93	1.98
Max. displacement, mm	5.4	5.3	6.07	3.91	3.59	4.53	4.46	4.27	3.61	4.37
Peak#1, Hz	19.53	29.9	29.9	21.16	20.99	20.99	20.99	20.99	20.99	20.99
Peak#2, Hz	22.7	-	-	25.1	24.9	25.0	25.0	24.9	24.9	24.9

In further calculations, options for the algebraic determination of pressure pulsations based on velocity gradients (K20-K22, K30, K31) will be applied. However, since the K20-K22 options are similar to each other, the following options are considered further: K11, K22, K30, K31.

3.4.1.2 Simulation of the single flexible tube experiment with Q=2.9 l/s and Q=1.7 l/s

To establish sensitivity to operating parameters (primarily to the inlet velocity), regimes with a flow rate of 2.9 and 1.7 l/s are considered. The calculation settings remain the same, including the proportionality coefficients for determining pressure pulsations.

Figure 106 presents the displacement evaluation results, as well as the spectra for a flow rate of 2.9 l/s. The summary characteristics of the displacements are presented in Table 25. The obtained results, in terms of characteristic frequencies, are generally similar to those for the maximum flow rate, but have a number of distinctive features in amplitude. First, a disproportionate decrease in the RMS oscillation amplitude compared to the overall flow rate is observed using both SFA+ST and ANSYS CFX+ST, which generally indicates sensitivity to fluid-elastic instability. Secondly, specifically for this case, the results obtained using SFA+ST are in better agreement with the experiment, while for the Q = 4.0 l/s regime, better agreement was achieved using ANSYS CFX+ST. This indicates different contributors to the overall force, including due to differences in the number of independent parameters in synthetic turbulence.

Figure 107 shows the displacement evaluation results, as well as the appropriate APSD for the case with Q=1.7 l/s. The summary characteristics of the obtained displacements are given in Table 26. For this case, the reduction in the calculated amplitude is proportional to the reduction in flow rate, which indicates a sufficient distance from the region of possible resonance, although, compared to the experiment, the rod displacements are overestimated.

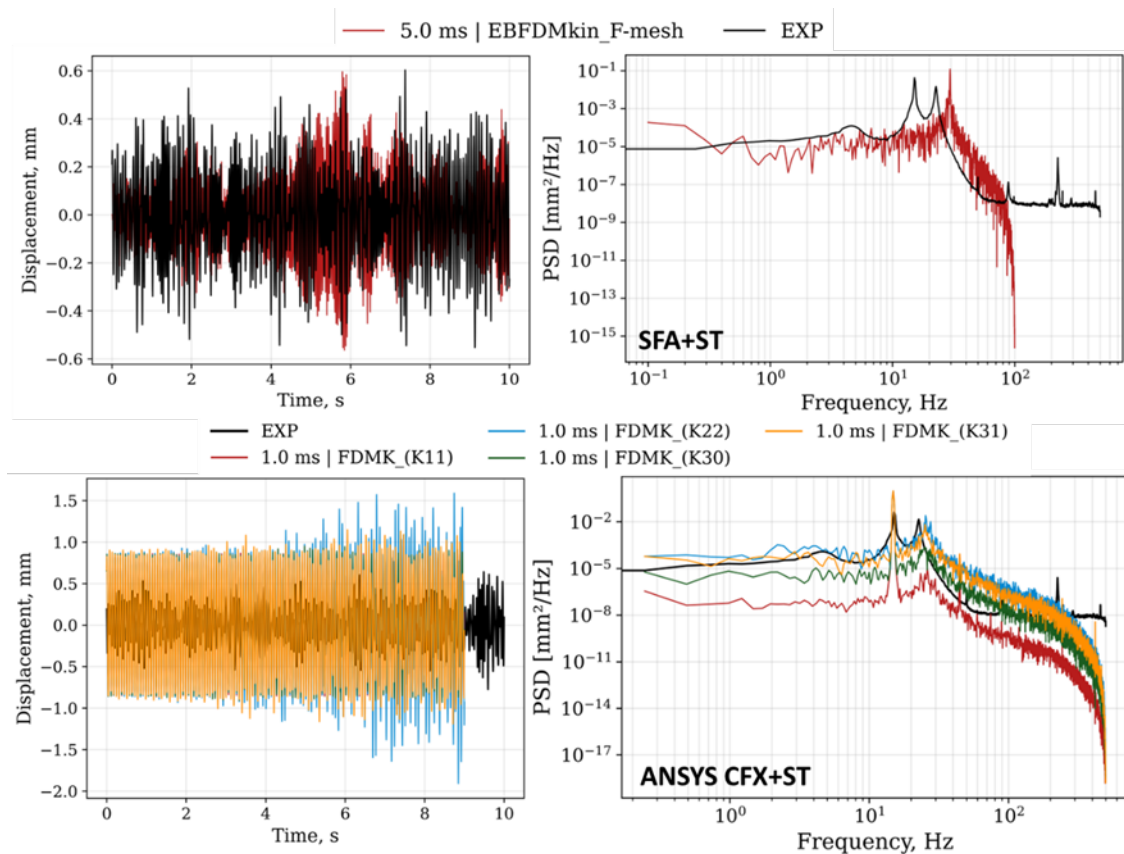


Figure 106: Results of displacements calculation for the single flexible tube with $Q = 2.9$ l/s

Table 25: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube ($Q=2.9$ l/s)

	EXP	SFA+ST	ANSYS CFX+ST (FDMK)			
		FDMK	K11	K22	K30	K31
Displacement RMS, mm	0.205	0.202	0.600	0.623	0.601	0.611
Max. displacement, mm	0.6	0.6	0.86	1.59	0.96	1.15
Peak#1, Hz	15.14	-	14.89			
Peak#2, Hz	22.71	28.1	25.39			

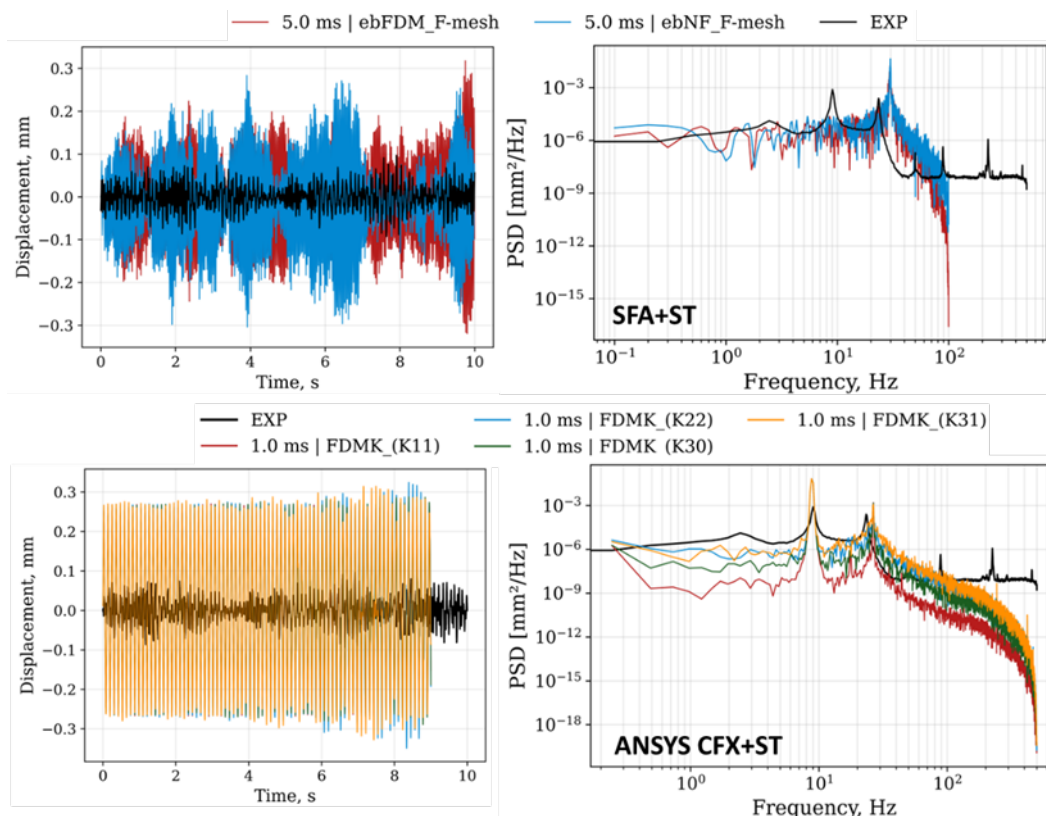


Figure 107: Results of displacements calculation for the single flexible tube with $Q = 1.7$ l/s

Table 26: Comparison of calculated and measured displacement RMS and frequency peaks observed for single flexible tube ($Q=1.7$ l/s)

	EXP	SFA+ST		ANSYS CFX+ST (FDMK)			
		EBNF	FDMK	K11	K22	K30	K31
Displacement RMS, mm	0.028	0.118	0.089	0.165	0.166	0.165	0.166
Max. displacement, mm	0.1	0.29	0.31	0.27	0.32	0.29	0.31
Peak#1, Hz	9.03	-	-	8.79			
Peak#2, Hz	23.44	28.4	27.5	26.61			

3.4.2 Validation of STG on AMOVI Single Flexible Tube in a Rigid Tube Bundle Experiment ($Q=1.95$ l/s and $Q=1.6$ l/s)

This section presents the results of the simulations of AMOVI Single Flexible Tube in a Rigid Tube Bundle test cases selected for validation. Estimating the displacement of a rod located in an in-line tube bundle is a challenging task, primarily because the analyzed moving rod is placed in the flow "wake" of the rigid tubes. From a computational perspective, it is necessary to model the region between the tubes along the flow in sufficient detail, taking into account the anisotropic properties of such flow. In this case,

turbulence is decisive. Below are the results of testing the proposed approaches for two cases with $Q = 1.95$ l/s and $Q=1.6$ l/s, between which a transition to near-resonant oscillations occurs. For each simulated case both the displacements of flexible rod and the appropriate APSD are compared with experimental data.

Figure 108 shows the results of displacement estimation for the $Q = 1.95$ l/s case using SFA+ST and ANSYS CFX+ST. It's worth noting that SFA+ST was able to predict large amplitudes, but sensitivity to the set of random parameters pre-generated for the synthetic turbulence procedure was observed. This influence is manifested in the physical time required to transition to oscillations with large displacements. SFA+ST calculations terminated when a zero or negative control volume was reached.

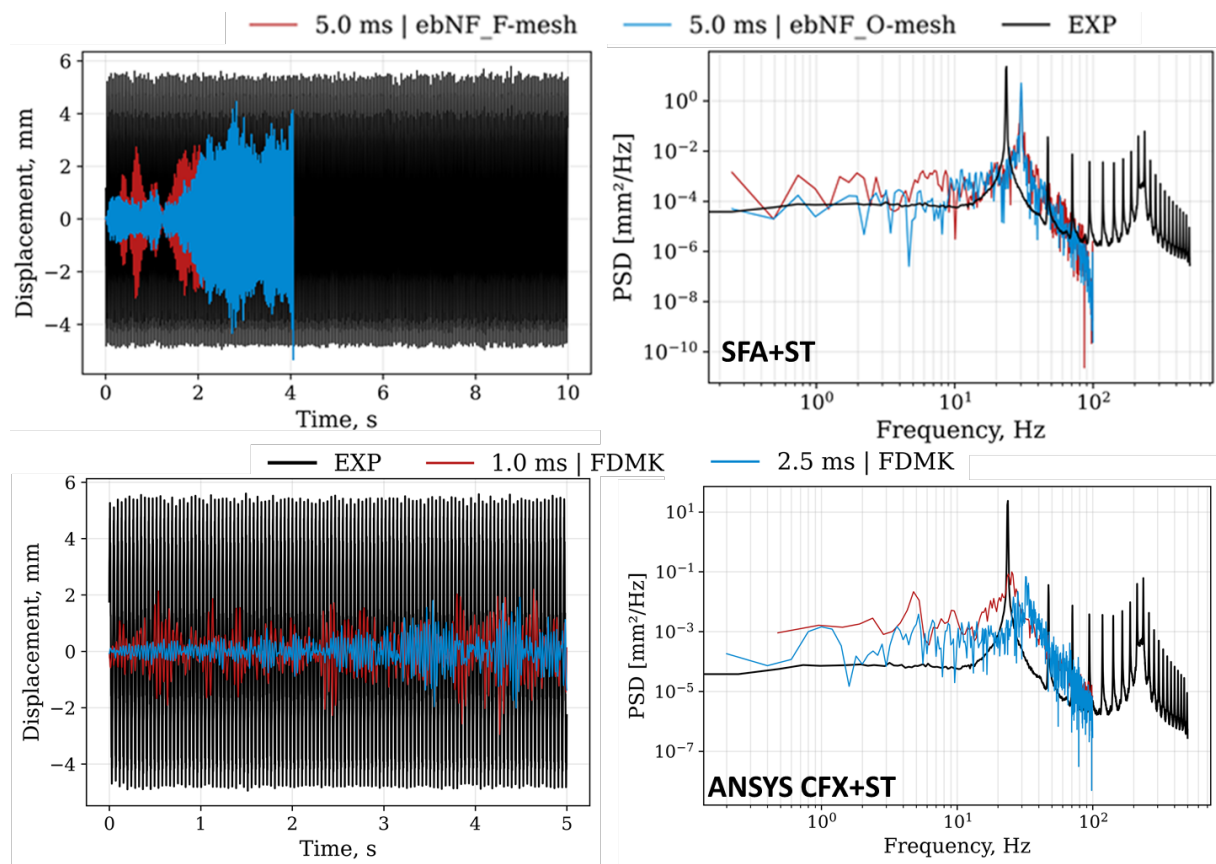


Figure 108: Results of displacements calculation for the case with a flexible tube in a rigid tube bundle and $Q = 1.95$ l/s

When applying ANSYS CFX+ST, vibrations were excited, but with a significantly lower amplitude. Moreover, peak frequency shifts were observed in both approaches relative to the experiment. It should be noted that vibration excitation in ANSYS CFX+ST was performed at a larger time step (2.5 ms) than for a single rod.

As the flow rate through the assembly decreases, the amplitude decreases nonlinearly (Figure 109), indicating sensitivity to resonant coupled oscillations. Although the predicted amplitudes obtained for $Q = 1.6$ l/s are higher than the experimental ones, the ability to

account for fluid-elastic instability is an important property of the SFA+ST calculation procedure. A summary of the simulated cases is presented in Table 27.

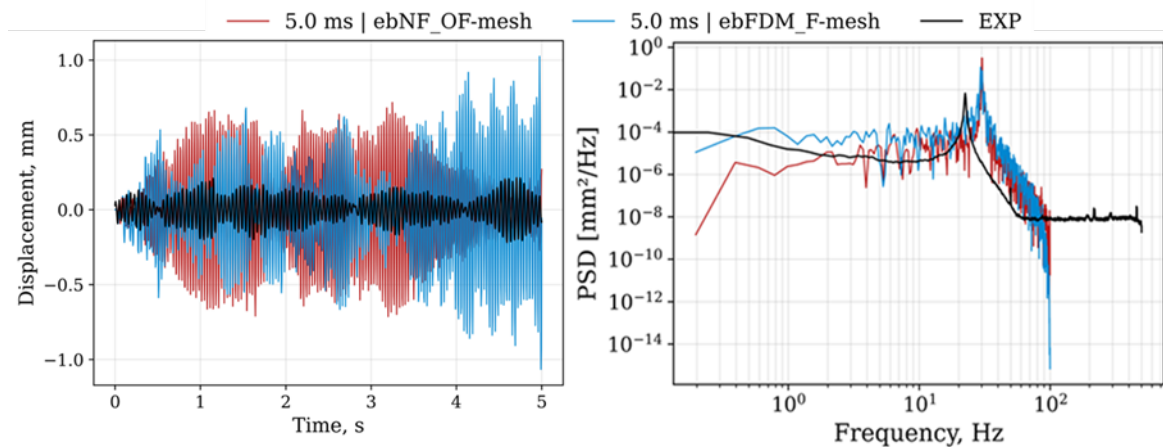


Figure 109: Results of displacements calculation for the case with a flexible tube in a rigid tube bundle and $Q = 1.6$ l/s

The beam displacement profile (amplitudes), as well as the values at the rod's reference geometric points for two selected flow rates are shown in Figure 110. The displacement amplitudes for case $Q=1.95$ l/s are such that central tube can touch the neighboring ones (similar amplitudes are also observed in the experiment).

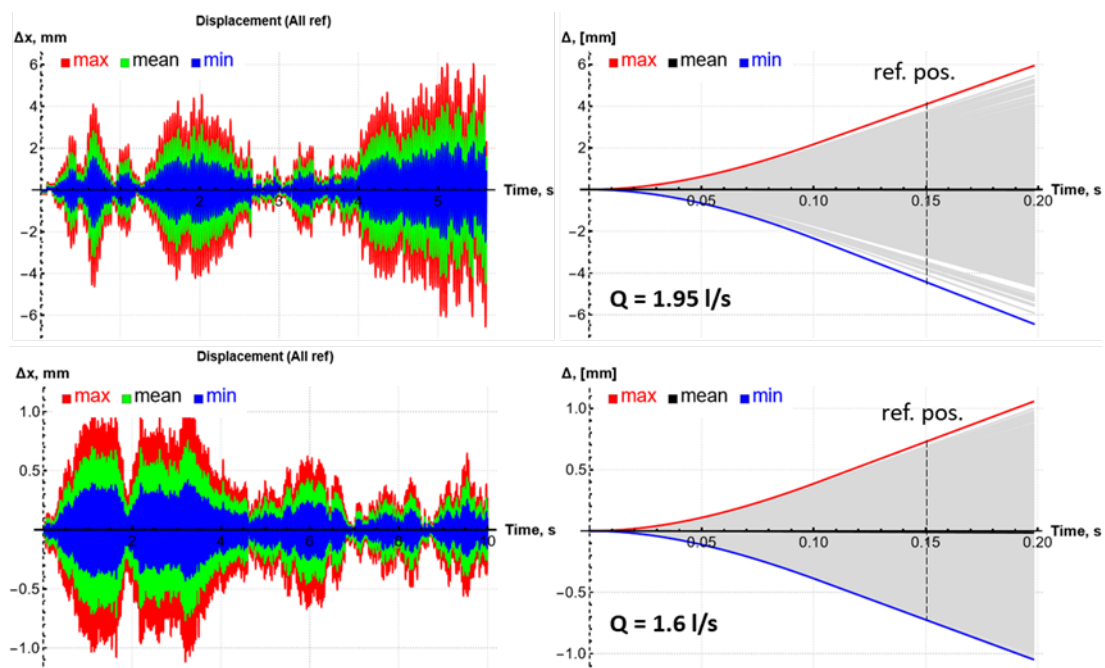


Figure 110: Tube displacements over time at different reference points and the range of deviations obtained for the flexible tube in a rigid tube bundle cases with $Q = 1.95$ l/s and $Q=1.6$ l/s using SFA+ST

Table 27: Comparison of calculated and measured displacement RMS and frequency peaks observed for flexible tube in a rigid tube bundle ($Q=1.95$ l/s and $Q=1.6$ l/s)

	Q=1.95 l/s				Q=1.6 l/s		
	EXP	SFA+ST (EBNF)	ANSYS CFX+ST (FDMK)		EXP	SFA+ST	
			2.5ms	1ms		EBNF	FDMK
Displacement RMS, mm	3.223	1.62	0.5	0.64	0.073	0.376	0.292
Max. displacement, mm	5.3	4.45	2.2	2.2	0.22	0.71	1.1
Peak, Hz	23.68	30.01	31.8	25.1	22.46	30.17	25.17

3.4.3 Overall summary of STG validation on AMOVI cases

The description of the fast-running approaches used by IPP to calculate coupled vibrations is provided in the sections above. To generate turbulent pulsations, which can be taken into account when determining the resultant force, the synthetic turbulence generation (STG) procedure described previously in the report (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024) was improved and applied. Two approaches of application STG were considered: the first is the implementation in an already existing solver (ANSYS CFX), and the second was to implement a simple solver for two-dimensional flow (focusing on end-to-end calculations within a single time step) with a simplified procedure for generating turbulent pulsations under the assumption of equilibrium generation and dissipation of turbulent kinetic energy. The main approaches for organizing the calculation procedure with Stable Fluid Algorithm method adaptation, as well as the method for calculating the resultant force, are outlined in sections 3.1 and 3.2.

The simulation of the AMOVI benchmark was considered as a validation of the proposed methods for "fast-running analysis" of FIV based on the synthetic turbulence generation. Different cases with one flexible rod and flexible rod in a rigid tube bundle were chosen for validation. For each simulated case, the flexible rod displacement in time as well as the appropriate APSD were compared with the experimental data.

When comparing the results obtained for the same experimental AMOVI cases, a fundamental difference is observed with the same discretization of the computational domain, which is the following:

- Oscillations are always excited in the SFA+ST computational procedure (at least within the timestep range of 0.1 to 30 ms and even higher), which is most likely due to the assumption of equilibrium between turbulence generation and dissipation. In other words, high numerical diffusion does not interfere with the excitation and development of oscillations. In ANSYS CFX+ST oscillations at a calculation step greater than 1 ms may not occur or may have a significantly smaller amplitude.
- When coupled oscillations are established, the displacements amplitudes agree quite well for the cases where the flow unevenness (and hence the magnitude of

the force) is determined directly by the moving boundary, and the oscillation modulation is determined by the turbulent nature. In a sense, these regimes are close to resonances and FEI (e.g. Single Flexible Tube AMOVI cases with 2.9 and 4.0 l/s).

- PSD prediction using ANSYS CFX+ST performs better than the SFA+ST procedure, which is likely determined by the advection calculation method (the SFA+ST procedure uses the Lagrangian approximation, which inherently does not distort the system's eigenvalues and possibly does not allow to take into account the interaction of the two systems - the flow path and the rod). The method for determining pressure pulsations may also influence this.
- For the calculation of complex flow configurations (rod bundles), the fields obtained by synthetic turbulence induce significantly greater hydrodynamic forces than the average URANS fields, which determines the sensitivity to fluid - elastic instability.

Figure 111 shows the predictive abilities demonstrated by the developed CFX+ST and SFA+ST methods for simulated AMOVI cases compared with the results obtained using CFX without generation of synthetic turbulence fields (applying for FSI: RB – rigid body solution, EBNF/FDMK – special analytical or FDM beam solvers). This comparison demonstrates much better agreement of displacement RMS values obtained with CFX+ST and especially SFA+ST methods with the experimental data for both analyzed configurations – single flexible tube and flexible tube in a rigid tube bundle.

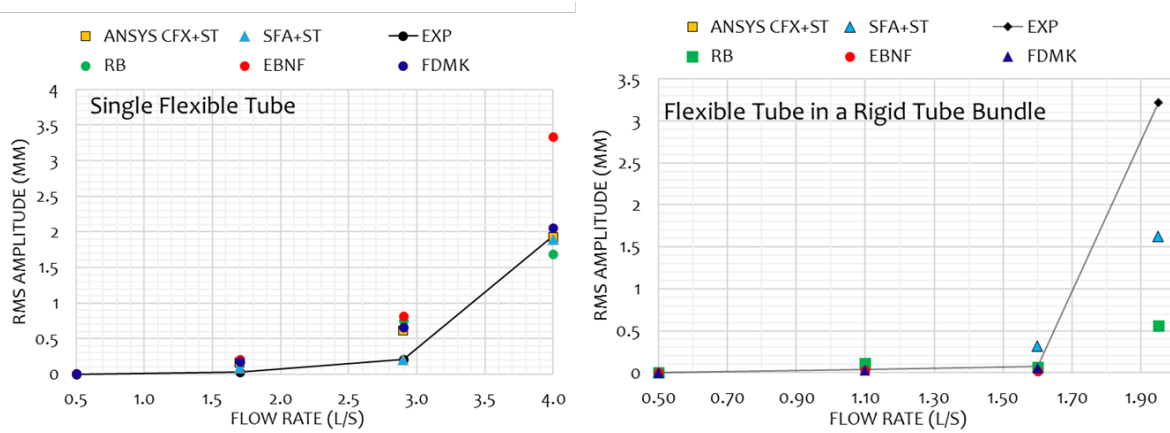


Figure 111: Comparison of simulated and experimental values of displacement RMS as a function of the flow rate

Figure 112 demonstrates the comparison of estimated specific time $T^*(31)$ spent for calculations of analyzed AMOVI cases when applying SFA+ST and CFX+ST with different options.

$$T^* = \frac{\text{TIME spent on calculation}}{(N_{\text{time steps}} \times N_{\text{nodes}} \times N_{\text{cores}})} \quad (31)$$

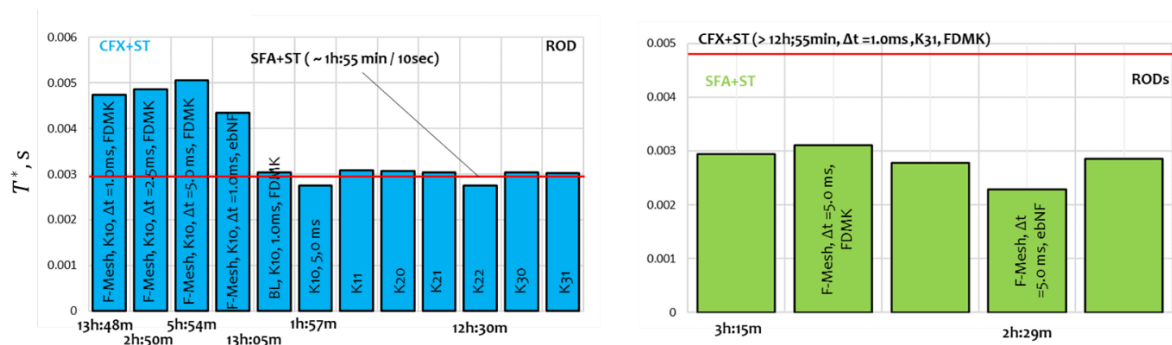


Figure 112: Comparison of the specific time spent for calculations of analyzed AMOVI cases

The provided comparison shows that both methods provide the ability to perform fast-running FIV analysis (the maximum time spent for calculation of 10 seconds transient in CFX+ST was 13h 48m), while SFA+ST calculations are less computationally intensive and could be used as the fastest possible calculation of coupled vibrations. It should be also noted that the computational procedure of SFA+ST is written in an interpreted programming language (essentially it is a prototype), which means it is several times slower than codes written with Fortran/C ++ and, accordingly, has significant potential for speeding it up. Also, the SFA+ST fast calculation procedure can be calibrated on available experimental data, followed by validation on some of them (as it performed on AMOVI cases), and then be applied with greater reliability to practical problems.

4 Validation of the Projection Based Fluid Model for fast running FSI

This chapter presents the fast-running fluid-structure interaction (FSI) model development by TU Delft on construction of a reduced order fluid model using the projection of the fluid response onto the structural modes. The chapter describes the theory of the projection based reduced order model (ROM) as well as some alternatives for reducing the computational effort of FSI simulations required to build the ROM model.

4.1 Introduction

In this section, the validation of the projection based reduced order fluid model is presented. The theory behind the methodology used to obtain the fluid response was presented in Deliverable 5.1. The main idea is repeated here for clarity.

When a turbulence induced fluid-structure interaction problem is considered, this could be modelled with full-order models for the fluid and structure and appropriate coupling at the fluid-structure interface. Turbulent flow could be either (partly) resolved using DNS or LES approaches or could be (mostly) modelled using a RANS approach. In the current methodology it is assumed that a RANS approach is taken. This implies that turbulence (pressure) fluctuations are not resolved and only statistical information about the velocity fluctuations (Reynolds stresses) is known. The RANS model can provide information about the turbulence statistics (e.g. turbulence kinetic energy) and how these statistics change when e.g. the structure deforms, but it does not provide the turbulence pressure fluctuations directly. Since the pressure fluctuations are essential to obtain a proper prediction of the structural vibration amplitude, these need to be taken into account somehow. Here it is assumed that some sort of Pressure Fluctuation Model (PFM) is available that provides a prediction of the pressure fluctuations occurring for a given statistical description of the turbulence. An example of such a PFM is the AniPFM developed by NRG PALLAS.

Schematically, the full-order model is represented in Figure 113.

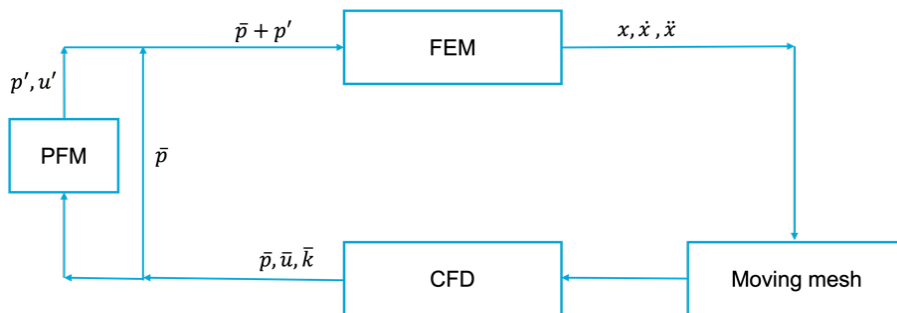


Figure 113: Full order model

The structure is modeled using a Finite Element Method (FEM), which provides the state of the fluid-structure interface in terms of its displacement, velocity and acceleration. A moving mesh approach is used to update the fluid mesh and a URANS (Unsteady RANS) is used to solve for the mean flow properties (including the mean turbulence statistics). The mean pressure (and to be precise also the mean wall shear stress) feed back into the

structure. To represent the instantaneous pressure fluctuations, a PFM can be used. The combined effect of the mean fluid response and turbulence pressure fluctuations, force the structure. In this approach solving for the fluid response is computationally expensive, and even solving the full FEM for the structure can be computationally expensive as well. To reduce the complexity and expense of these parts, a fluid response model is built for the response to a selected number of structural vibration modes. The principle is shown in Figure 114.

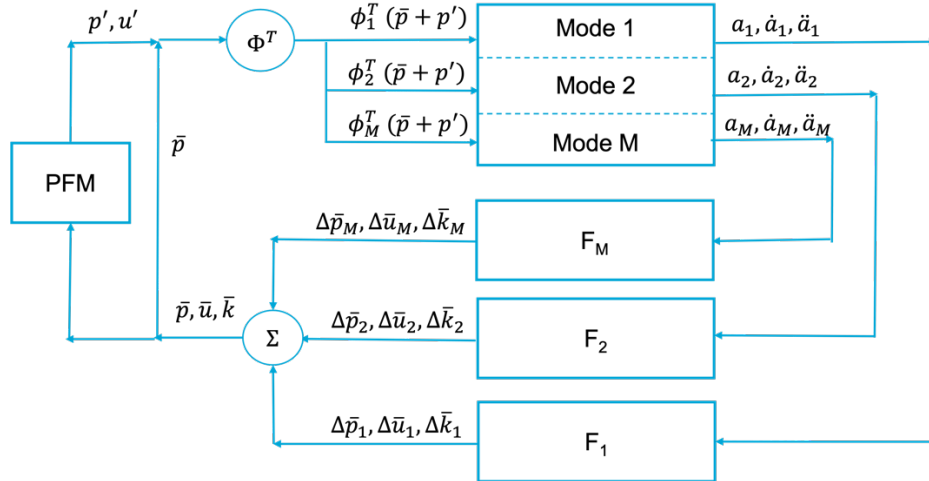


Figure 114: Modal reduced order model

The expensive CFD part that needs to be solved on a moving mesh is replaced by response functions that respond to the kinematics of the fluid-structure interface. At the same time, the structure is modelled by a reduced set of vibration modes. Since the structural response is expected to be dominated by the first few structural modes, only the fluid response to the kinematics of these structural modes is taken into account. Although the methodology provides the fluid response both in volume fields at cell level and as surface fields at the fluid-structure interface, when using only a modal representation of the structure, the only effect that the fluid has on these structural modes can be represented

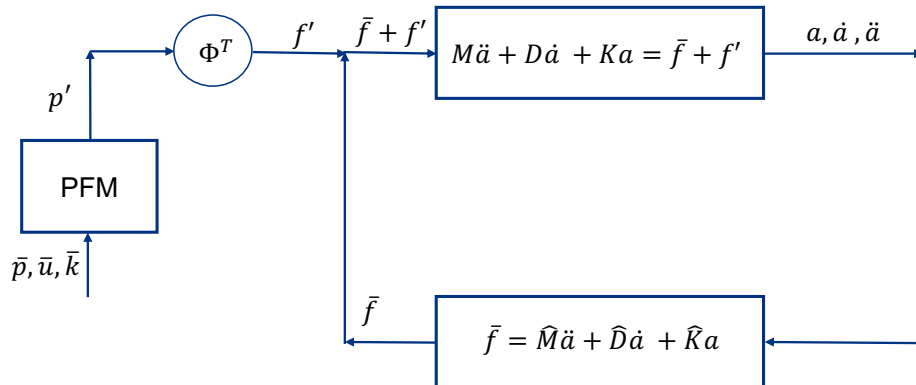


Figure 115: Modal reduction of the fluid-structure interaction system

by the changing load on the fluid-structure interface, projected onto the structural modes, i.e. the modal forcing, see Figure 115. The fluid response at the fluid-structure interface has a direct influence on the forcing of the different structural modes – this creates added

mass, damping and stiffness terms that can be incorporated directly into the structural dynamics, see Figure 116. Note that in the methodology the mean (RANS) fluid response to the interface kinematics is modelled and that the turbulence forcing is still obtained by a PFM.

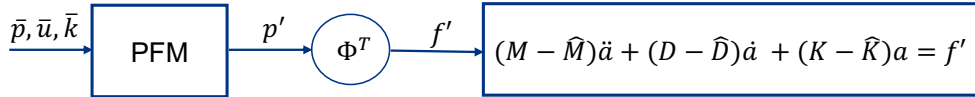


Figure 116: Unified modal reduction for turbulence induced vibration

4.2 Building ROM for cantilever rod case

In this section the methodology is applied to the Cantilever Rod case (as described in deliverable D2.1). The necessary steps to build to reduced order fluid model, as well as the input parameters used, are given in the next sections.

4.2.1 Meshing and material properties

The case is simplified by considering a single (axisymmetric) domain, see Figure 117.

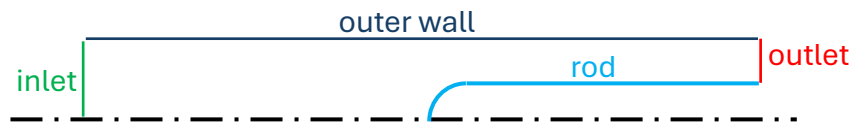


Figure 117: Domain cantilever rod

The tip of the rod is located at the origin and the x-axis is chosen as the symmetry axis. The domain inlet is located 1100 mm upstream of the tip, the outlet 1045 mm downstream of the tip (at the location of the clamped end of the rod). The outer wall has a diameter of 20.86 mm and the rod an outer diameter of 10.01 mm. The fluid mesh is block-structured with the block-topology and number of points along the block edges given in Figure 118 and Table 28. The mesh consists of 631296 hexahedra in total.

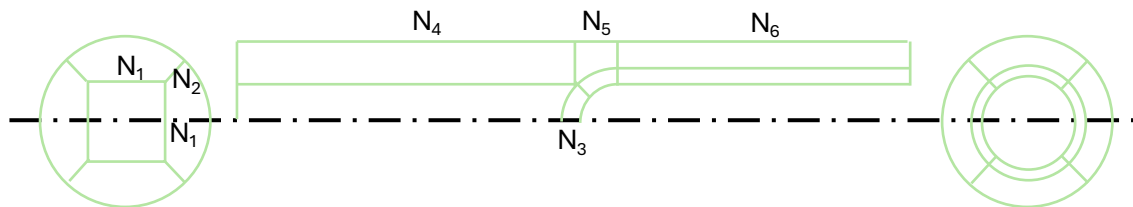


Figure 118: Mesh block structure cantilever rod

Table 28: Mesh block parameters cantilever rod

EDGE	# points	type	size1 [mm]	size2 [mm]
N ₁	9	Uniform	9e-4	-
N ₂	19	Exponential	1e-5	6e-4

N_3	17	Exponential	$1e-5$	$5e-4$
N_4	129	Exponential	$1e-3$	$2.6e-2$
N_5	5	Uniform	$1e-3$	-
N_6	501	Uniform	$1e-3$	-

The structure model consists of a steel tube, with an aluminum tip. The steel tube has a 10.01 mm outer diameter and an 8.83 mm inner diameter. The density of the steel material has been modified to include the effect of the lead shot with which the steel rod is filled, to obtain an effective mass of 588 g/m. The aluminum tip extends 10 mm inside of the steel rod, has a 2mm cylindrical part extending the steel rod and a spherical nose. Since the deformation of the structure is small compared to its overall length, the structure mechanics are assumed linear, and the material behavior is described by the Young's modulus and Poisson ratio. The material properties are given in Table 29.

Table 29: Effective material properties cantilever rod

Material	ρ [KG/m ³]	E [GPa]	ν [-]
high density Steel	33676	193	0.26
ALUMINUM	2740	69	0.33

A representation of the tip only and the full rod is given in Figure 119: Structure model tip (left) and full rod (right). The mesh consists of 893 tetrahedra for the tip and 35936 hexahedra for the steel rod.

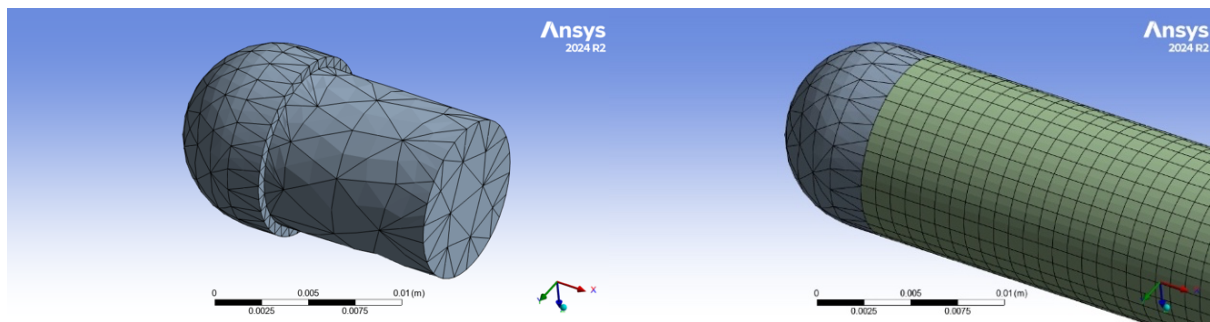


Figure 119: Structure model tip (left) and full rod (right)

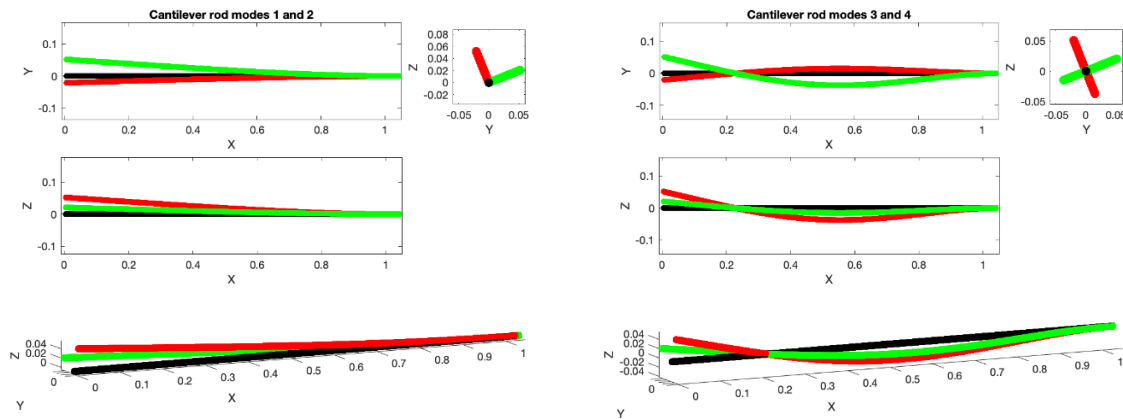
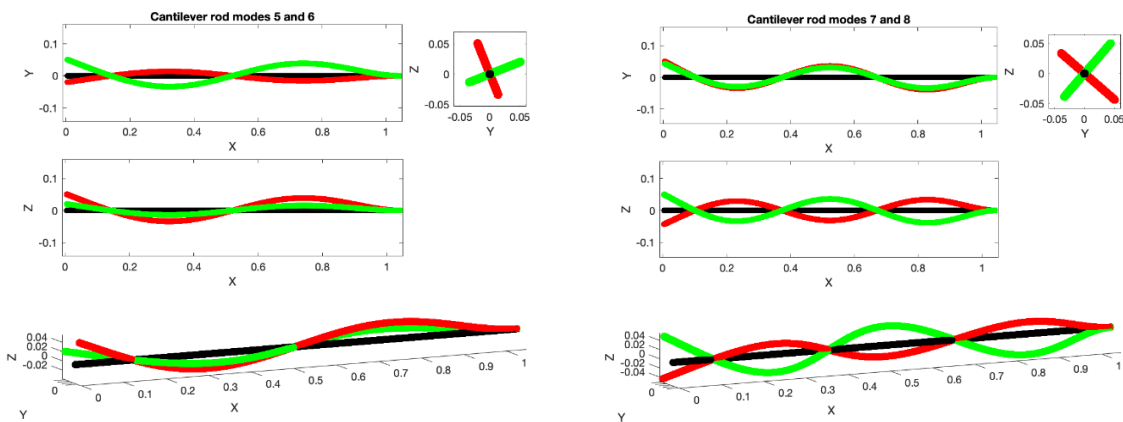
4.2.2 Solution procedure

Construction of the reduced order model consists of a number of simulations, which are set using Ansys Workbench. To obtain the necessary data for the fluid ROM, we require:

1. Modal analysis of the structure to obtain a number of mode shapes
2. Steady state fluid simulation to obtain the appropriate initial / reference state
3. Forced motion simulations to sample the fluid modal response
4. ROM data extraction to build the fluid response model

Step 1: structural modal analysis

A Modal analysis is performed in Ansys Mechanical on the rod which has the mesh and material properties as described in Section 4.2.1. At the clamped end a “Fixed Support” condition is imposed and the first 8 modes are computed. Since the rod is axisymmetric, we obtain pairwise modes that are 90 degrees offset around the axis of symmetry. So, with 8 modes, we obtain effectively the first 4 bending modes of the rod, see Figure 120 and Figure 121.

**Figure 120: Structural bending mode pairs 1 and 2****Figure 121: Structural bending mode pairs 3 and 4**

Since the rod tip is modelled using unstructured tetrahedral elements, the case is not perfectly axisymmetric, and each mode-pair can have slightly different eigenfrequency, see Table 30.

Table 30: Natural frequencies first 8 bending modes

Mode	1	2	3	4	5	6	7	8
freq. [Hz]	4.1116	4.1118	25.752	25.752	72.029	72.029	140.91	140.93

The shape of these modes is exported as an STL-surface; only the nodes on the fluid-structure interface are written (i.e. not the internal structure degrees-of-freedom). The undeformed shape is exported as well. When the STL-surface is read, we denote the surface of mode m , $\vec{x}_m$, as a collection of N_b node locations:

$$\vec{x}_m = \{\vec{x}_i^{STLm}\}_{i=0\dots N_b}.$$

Note that mode $m = 0$ represents the surface coordinates of the undeformed shape. Note as well that the STL surface that is written by Ansys Mechanical has the same triangulation and the same node topology for all (un)deformed rod representations. Therefore, we obtain a discrete representation of the rod's mode-shape deformations as:

$$\vec{\phi}_m = \vec{x}_m - \vec{x}_0. \quad (4.1)$$

Step 2: Steady state simulation

A steady-state simulation is run for the fluid. The fluid is modelled as incompressible, isothermal, using a RSM-BSL turbulence model to capture the anisotropy of the turbulence. Although not explicitly used in this part of the work, the anisotropic information could be used in the aniPFM model. The operating and boundary conditions are given in Table 31.

Table 31: Operating and boundary conditions cantilever rod model

Property	description	Value
medium	Water with constant properties	
	- Density	998.23 [kg m ⁻³]
	- Temperature	25 [°C]
	- Dynamic viscosity	1.002 10 ⁻³ [kg m ⁻¹ s ⁻¹]
Inlet	Subsonic condition with mass flow imposed	
	- Mass flow rate	0.515 [kg s ⁻¹]
	- Turbulence intensity	1%
Outlet	Subsonic outlet with static pressure imposed	
	- Relative static pressure	0 [Pa]
walls	Smooth walls with no-slip condition	

The simulation is run in double precision for 4000 iterations; the convergence of the residuals as well as the momentum on the rod's surface in the lateral directions is shown in Figure 122.

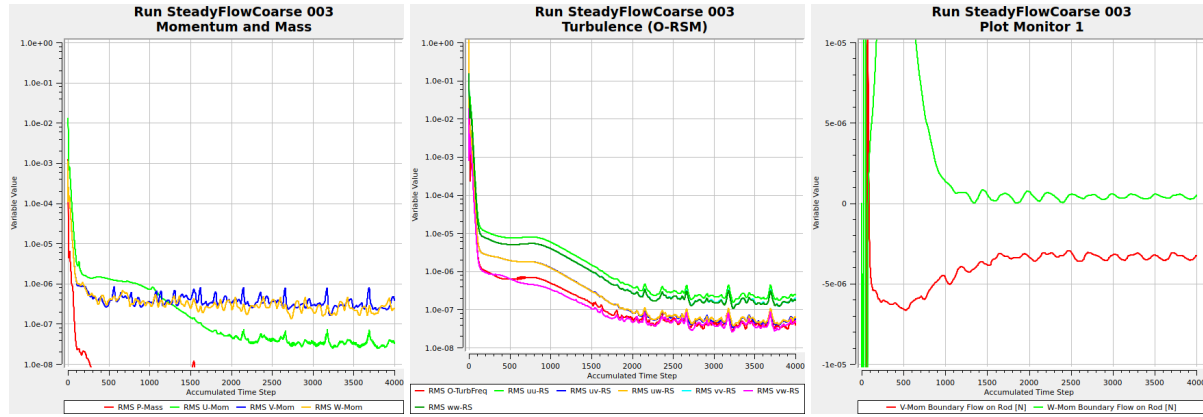


Figure 122: RANS convergence and resultant force cantilever rod

All residuals drop to below 10^{-6} and in the lateral momentum only small fluctuations remain visible and a small offset in the Y-direction of about $4.0 \cdot 10^{-6}$ N. Since the case is axisymmetric about the x-axis, the lateral forces should be zero.

Step 3: forced vibration simulation

We use the method described in Deliverable D5.1 (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024) to obtain the fluid perturbation sensitivities to the modal displacement, velocity and acceleration by imposing a forced motion as defined by the amplitude function $g_\tau(t/\tau)$. To facilitate creating the simulation definitions (.def files), two CFX-Pre macro files are created through a Matlab scripts (see Appendix A and Appendix B). The first macro contains the following main steps:

- Definition of the variables for which the fluid response needs to be sampled: these can be scalar or vector variables. The Reynolds stress tensor components that follow from the RSM-BSL turbulence model, are sampled as individual scalar variables 'uu', 'uv', 'uw', 'vv', 'vw' and 'ww'.
- The script reads the STL modes and defines the mode shapes as defined in equation (4.1). A macro is written to define a user-defined function 'fmode' that will use interpolation data to return the function value. The interpolation data is defined at the node locations $\vec{x}_0$ and the data to interpolate is defined by $\vec{\phi}_m$. For any node on the undeformed rod, the interpolation function has components 'm1dx', 'm1dy', 'm1dz', 'm2dx', etc. to represent the displacement of mode 1 in x, mode 1 in y, etc.
- Next expressions are created that define the kinematics of the imposed motion: amplitude (Am), simulations time (Tsim), number of time steps (Ndt), time step ($dt = Tsim/Ndt$), reduced time ($tred = \text{mod}(\text{Time}/Tsim, 1)$), the amplitude function and its time derivatives ($fA = g_\tau(t/\tau)$, $dfAdt = \dot{g}_\tau(t/\tau)$, $d2fAdt2 = \ddot{g}_\tau(t/\tau)$ the definition of the imposed boundary displacement ($dx = Am * fA * fmode.m1dx(\text{Initial X}, \text{Initial Y}, \text{Initial Z})$, and similar for dy, dz)
- More expressions are created to obtain the fluid perturbation contributions based on the modal displacement, velocity and acceleration (e.g. dpKexpr, dpDexpr, dpMexpr) by multiplying the sampled variables with the non-zero

components of inverse of the projection matrix (c_{11inv} , c_{13inv} , c_{22inv} , c_{33inv}), and to obtain the resultant modal forcing ‘ fm_{1x} ’, ‘ fm_{1y} ’, etc.

- Finally, user defined variables are defined that store the sampled perturbation fields called ‘ $d\{u|p|... \}\{M|D|K\}$ ’ as introduced in Deliverable 5.1. These fields provide the fluid response in the whole domain based on the modal acceleration, velocity and displacement. When this field is sampled on the fluid-structure interface (rod’s surface), the fluid response as experienced by the structure is represented.

The second macro (Appendix B) will export the solver definition (.def) files that are required to run the forced vibration motion, one solver definition file for each mode that is forced.

Step 4: ROM data extraction

Although, in principle, the fluid response is known for the whole (3D) fluid domain, we only sample the fluid response at the fluid-structure interface. For that the ‘ $d\{u|p|... \}\{M|D|K\}$ ’ values are exported to a “.csv” file which includes the surface coordinates and perturbation fields for all fluid variables. This is done by creating a .cse script that can be executed by CFD-Post to read the results for each forced vibration mode, extract data at the rod’s surface and export the data to .csv files. The .cse script is generated through the Matlab script given in Appendix C. Secondly, the time evolution of the “modal forcing” is used to build added mass, damping and stiffness terms in the same manner as reported in deliverable D5.1. These added mass, damping and stiffness matrices $\hat{M}$, $\hat{D}$ and $\hat{K}$ are $N \times N$ matrices, where N is the number of modes that are used to build the reduced order model.

Although the reduced order fluid model could, in principle, be coupled to the full order structure model, in the next verification and validation sections, the structural model is also represented as a modal reduced order model, i.e. for each mode a stiffness k_i and mass m_i are determined such that

$$\frac{k_i}{m_i} = 4\pi^2 f_i^2,$$

where f_i is the natural frequency of the mode as given in Table 30, and k_i is obtained from a full order static structure solve for which a given load $\vec{F}_{FOM}$ is imposed to the structure resulting in a rod (surface) displacement $\vec{d}_{FOM}$. By projecting both the load and displacement onto each of the modes, we can find the modal stiffness k_i using

$$\begin{aligned} a_i &= \vec{\phi}_i^T \vec{d}_{FOM}, \\ f_i &= \vec{\phi}_i^T \vec{F}_{FOM}, \end{aligned}$$

where $\vec{\phi}_i$ is the mode shape, a_i the modal amplitude and f_i the modal forcing. For static deformations the relation $k_i a_i = f_i$ holds, so that the modal stiffness k_i is obtained

$$k_i = \frac{f_i}{a_i} = \frac{\vec{\phi}_i^T \vec{F}_{FOM}}{\vec{\phi}_i^T \vec{d}_{FOM}}.$$

The thus obtained modal mass m_i and modal stiffness k_i parameters represent the structural model in vacuum conditions. Together with the fluid response models $\hat{M}$, $\hat{D}$ and $\hat{K}$, a reduced order model for the response of the rod submerged in water at a Reynolds number of 21,000 is obtained. In order to obtain representative vibration results for turbulence induced vibration, still a model is required to obtain turbulent fluctuations (that can be projected onto the modes to obtain the turbulence forcing).

4.3 Verification

A first verification is made by running the modal model without added mass, damping and stiffness terms and comparing it to the full structural model (in vacuum). For a given tip load of 0.2 N in the positive y direction, the rod should start to oscillate. Since there is no physical damping modelled, this should be an oscillation about some offset in y.

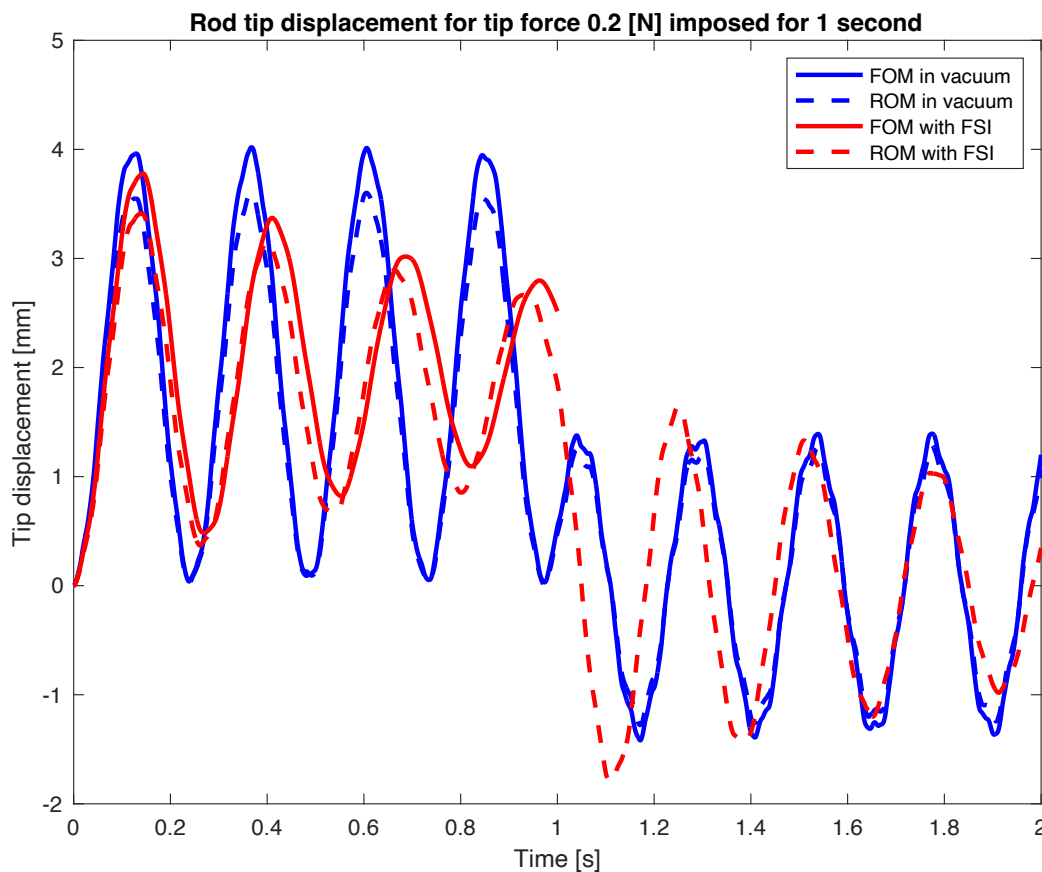


Figure 123: Comparison of full and reduced order model in vacuum and in water

The results in Figure 123 show that the tip displacement is well predicted by the structural modal model, although the full order model (FOM) has slightly higher initial displacements (of about 10%). The dominant frequency of the first bending mode (that are represented by a single mode-pair) can be clearly identified $f_1 \approx 4.1$ Hz. As soon as the tip force is removed, the rod vibrates freely back around its equilibrium state, and in this region, the tip displacement predicted by the FOM and ROM are very much aligned, where there is still

a slightly higher vibration amplitude for the FOM (this time less than 10%). The second verification is running the coupled full order structure and fluid model with a given tip load of 0.2 N in the positive y direction. The vibrational response is now affected by the interaction with the fluid, represented as an added mass, damping and stiffness term in the ROM. For the full order model, we did not manage to make a switch from the forced vibration to a free vibration by switching off the external load, and therefore, there is no free vibration response for the FOM in Figure 123. However, the response of the ROM can still be compared to the FOM for the first second of the simulation in which the force is applied. The effect of the added mass and damping caused by the interaction between the flow and the structure can be clearly identified: the vibration response shows a clear decay in the vibration amplitude for both the FOM and ROM results. As with the vibration in vacuum, the FOM has an about 10% higher initial amplitude, although the minimum displacement is now also higher than the ROM response, indicating that the models predict a very similar amplitude and decay of the vibration, albeit that the FOM oscillates about a larger mean displacement. It is hypothesized that this is due to the truncation of the number of modes that are used to build the ROM and that no structural damping is included: for the vacuum case there is no physical damping present and all modes remain present in the dynamic motion; since the imposed point-load excites all modes, part of the (static) displacement is represented by the truncated modes. Within the vacuum test, the dynamic effect of these modes remains present in the dynamic motion of the rod, since they are not damped. However, for the vibration of the rod within the water flow, there is a fluid induced damping. The static displacement of the truncated modes will remain present in the FOM for as long as the point-load is applied, but the dynamic effect diminishes because of the fluid induced damping, which has a larger effect on the higher (truncated) modes than the lower modes. The added mass effect is in particular visible in the vibration frequency: the coupling of the structure to the fluid ROM predicts a vibration frequency $f_1 \approx 3.75$ Hz and for the FOM this is slightly lower $f_1 \approx 3.65$ Hz, which are very much in line with the data reported for the first vibration mode in deliverable D2.2 (analytical $f_1 = 3.71$ Hz, experimental $f_1 = 3.75$ Hz).

4.4 Validation

For the validation we compare to the results of the Cantilever Rod experiments as discussed in deliverable D2.1. The free vibration (in air) of the rod in the experimental setup is reported to be 4.10 Hz, which is matched well with the frequency of the full and reduced order structural models (in vacuum). Also, the free vibration in still water of the experiment is reported to be 3.72 Hz which is also close to the vibration for the rod in interaction with fluid for both the full and reduced order models. Note that the reduced order model was constructed for the $Re=21,000$ case and not for still water. However, D2.1 also reports that the dominant vibration frequency is insensitive to the flow rate at least for Reynolds numbers based on the mean flow in the annulus ranging between 7,000 – 30,000.

4.4.1 Wall turbulence pressure fluctuation model

Within the fluid ROM there is currently no modelling of the pressure fluctuations due to turbulence. To introduce the pressure fluctuations, an under-resolved DNS simulation is run for a fully developed flow in an axisymmetric domain with an inner wall with a radius matching the rod and an outer radius matching the outer wall. The length of the segment is 0.025 m. The domain uses $100 \times 200 \times 40$ cells in the axial, azimuthal and radial direction respectively. The mesh spacing normal to the walls targets a y^+ of about 1, whereas the mesh spacings in the axial and circumferential direction have an x^+ and z^+ of about 40. The mesh and a visualization of the wall pressure fluctuations and turbulence structures (Q-criterion iso-surface for a value of $2e5$) colored with velocity magnitude is given in Figure 124.

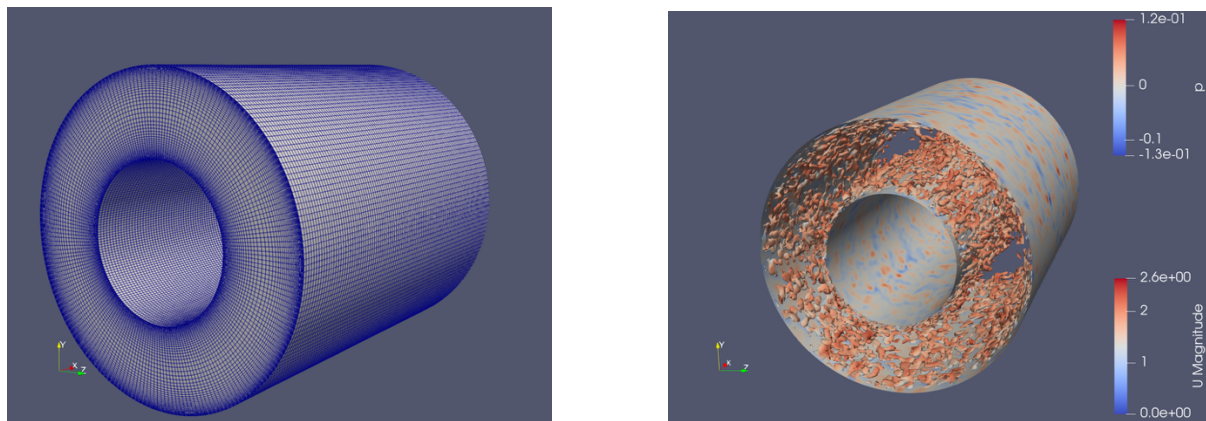


Figure 124: Periodic annular flow mesh (left) and wall pressure and Q-criterion iso-surfaces coloured by velocity magnitude (right)

To obtain turbulent pressure fluctuations that can excite the rod vibration, an under resolved DNS is run on a periodic domain, using a pressure gradient to retain a bulk speed of 1.96 m/s (which corresponds to a mass flow of 0.515 kg s^{-1}). The time step is adjustable to retain a Courant number of 0.9. After the flow has transitioned to turbulent, the time step is fixed to $3.0 \times 10^{-5} \text{ s}$ and the wall pressure fluctuations are sampled every 5 time steps (a rate of $\sim 6 \text{ kHz}$) for a total duration of 3 s. Due to restarting the simulation in OpenFOAM (OpenFOAM-v2412), the pressure fields exhibits an alternating disturbance between odd and even time steps. These disturbances can average out by centering the pressure fields about a mean of zero and averaging between an odd and even time step. An example of the centered pressure fields of an odd and even time step and its average are shown in Figure 125.

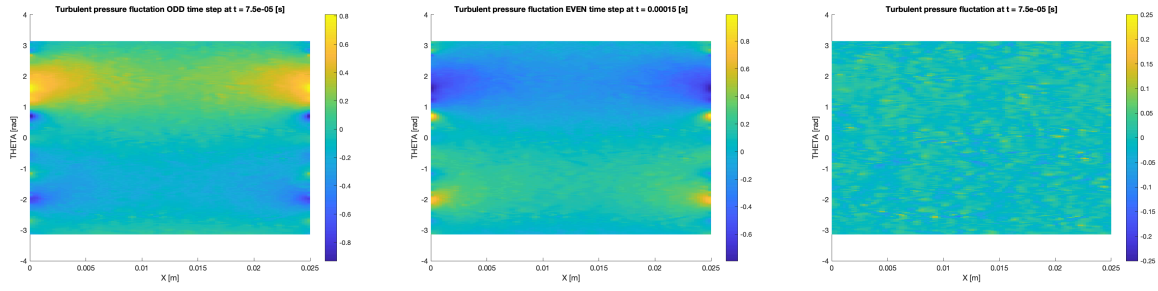


Figure 125: Turbulence wall pressure fluctuations obtained from OpenFOAM periodic annular flow: pressure field at odd time steps (left), at even time steps (middle) and averaged pair (right)

Since the periodic domain spans only a fraction of the full rod length, three different scenarios are made to represent pressure fluctuations over the whole rod:

1. **Full coherence:** the 2.5 cm periodic domain is multiplied about 40 times to span the whole surface area of the rod (which is a little over 1 m long). The pressure fluctuations are imposed on the whole rod (except for the semi-spherical tip). These loads are projected onto the different mode shapes to obtain the modal forcing for the ROM, i.e. evaluating

$$f_i(t_n) = \rho_0 \iint_{\Gamma_{\text{rod}}} \vec{\phi}_i(\vec{x}) \cdot \vec{p}'(\vec{x}, t_n) \vec{n} dS,$$

in which $\rho_0 = 998.23 \text{ kg/m}^3$ is the density of the water to convert the sampled kinematic pressure fluctuations to dynamic pressure fluctuations. This step can be seen as the inner-product between the direction of the modal displacement and the direction in which the pressure force acts. Since the mode shapes have been sampled on a different mesh than the pressure fluctuations, an interpolation (Matlab's `scatteredInterpolant`) is used to obtain the modal displacements at the face-center locations of the fluid mesh.

$$f_i(t_n) = \rho_0 \sum_{s=1}^{\text{nSegments}} \sum_{j=1}^{\text{nFaces}} \vec{\phi}_i(\vec{x}_j - \vec{x}_s) \cdot (\vec{n}S)_j p'(\vec{x}_j, t_n),$$

where $\vec{x}_s$ denotes the starting location of the section. For each 2.5 cm section a projection is defined by

$$W_{ij}^s = \vec{\phi}_i(\vec{x}_j - \vec{x}_s) \cdot (\vec{n}S)_j,$$

so that the modal forcing can be obtained by

$$f_i(t_n) = \rho_0 \sum_{s=1}^{\text{nSegments}} W^s p'(t_n).$$

In this first approach, the pressure fluctuation samples are taken at the same time step for all segments. This introduces a strong correlation along the rod, i.e. if the pressure fluctuations create a resultant force in some direction, the same resultant force is experienced by the rod each 2.5 cm. This may exaggerate the vibration response. Since the pressure fluctuation field is only sampled for 3 seconds, the

same pressure fluctuation sequence repeats every 3 seconds, creating a temporal periodicity in the forcing.

2. **Random coherence:** To remove the strong, instantaneous correlation between the sections, each section will have a random offset in its starting point in the time signal of p' . Once the 3 seconds of time data has been used, new random offsets are determined for each section. Each section will therefore have a spatially and temporally correlated p' based on the sampled DNS data every 3 seconds.
3. **“Frozen” turbulence approach:** In the previous approach all sections are completely uncorrelated. In the “frozen” turbulence approach a specific advection speed U_c for the turbulent pressure fluctuations is assumed. For each section the sampled pressure field is imposed with a time lag equal to the length of the section divided by the advection speed. This also means that this approach becomes more similar to the full coherence method for increasing U_c (and is identical when U_c approaches infinity).

4.4.2 Results

In Figure 126, the response of the rod tip is shown when imposing the turbulence pressure fluctuations using one of the three approaches mentioned in the previous section. The RMS of the tip response is reported in Table 32. The first method with full correlation, shows the largest tip displacement with an RMS of 0.045 mm. This value is lower than what was observed in the experiments, see Figure 127, although not as significant as the uncorrelated result, that underpredict the vibration response by about an order of magnitude. For the sectional correlation based on an assumed advection speed, the vibration response is either highly underestimated (for the lowest $U_c = 1.04\text{m/s}$ by more than an order of magnitude and, therefore, it is not shown in Figure 127), significantly underestimated (for the medium $U_c = 1.67\text{m/s}$ by about an order of magnitude) and underestimated (for the highest $U_c = 4.17\text{m/s}$ by about a factor of 2). Since pressure fluctuation data was sampled for only 3 seconds, all methods that resample the data every three seconds results show some form of periodicity in the tip displacement, although the initial startup from the rod at rest behaves slightly differently. Only the uncorrelated approach, where new random offsets for imposing the pressure data at every segment are used, does not show the periodicity.

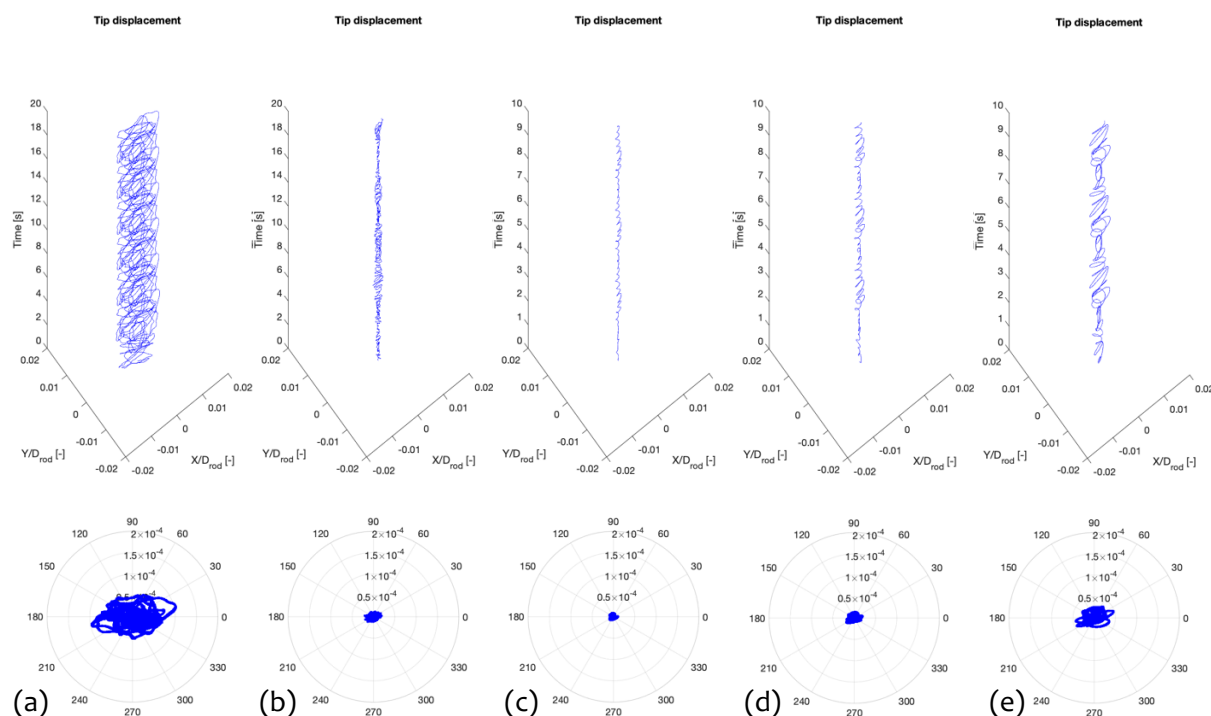


Figure 126: Reduced order fluid and structure model forced by turbulence pressure fluctuations sampled from fully developed annular channel flow; (a) fully correlated, (b), uncorrelated, frozen turbulence with (c) $U_c = 1.04\text{m/s}$, (d) $U_c = 1.67\text{m/s}$, (e) $U_c = 4.17\text{m/s}$.

Table 32: RMS tip displacement for reduced order fluid and structure model forced by turbulence pressure fluctuations, same conditions as Figure

Method	(a)	(b)	(c)	(d)	(e)
RMS tip displacement [mm]	0.0450	0.0088	0.0054	0.0082	0.0212

Using pressure fluctuations sampled from a reduced, periodic domain representing fully developed annular flow to force the reduced order unified fluid-structure interaction system shows that duplicating the sectional data with full coherence, results in the highest prediction of tip displacement. Since all sections along the length of the rod are fully correlated, this means that when the turbulence pressure fluctuations do not balance out over the section, a resultant force is acting on that section. That same resultant force is then acting on each 2.5cm of the 1m rod at the same time, forcing the rod in the same direction. This is beneficial for forcing low bending modes as these have deflections in a single direction, compared to high bending modes that have deflections in both positive and negative direction, thereby partially filtering out the correlation that is acting on the whole rod length (compare e.g. the bending modes of Figure 120 to Figure 121). Although the vibration magnitude is quite well predicted using the fully developed annular flow over a small section, it is uncertain whether this is due to chance, or that it can be used in future analyses. For that one would like to vary the length of the periodic domain, the total sampling time and the annulus Reynolds number within the current framework.

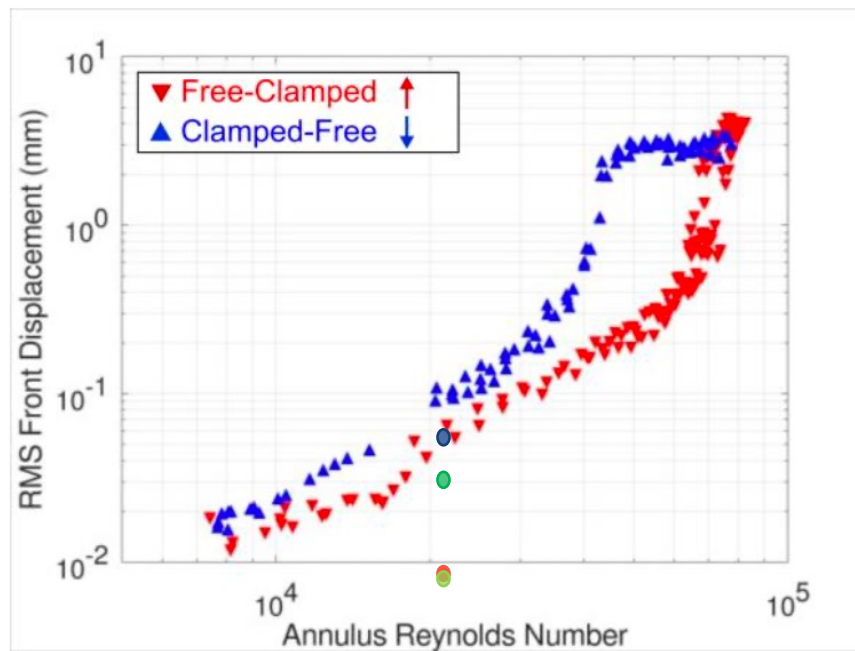


Figure 127: Cantilever front displacement [taken from D2.1] with ROM tip displacement using full correlation (blue), uncorrelated (red) and advective $U_c = 1.67\text{m/s}$ (light green), $U_c = 4.17\text{m/s}$ (dark green)

4.5 Conclusions

The method that was developed to construct a reduced order fluid response model based on a reduced set of structural vibration modes is validated against the Cantilever Rod, turbulence induced vibration case. The fluid modal response is obtained from forced vibration simulations using the imposed forcing kinematics derived in the previous deliverable D5.1. Although the reduced order fluid model can provide the fluid response in the whole (volumetric) domain, when coupled to the same selection of structural modes, only the fluid modal forcing response needs to be included (represented as added mass, damping and stiffness effects) to obtain the coupled fluid-structure interaction response. This model provides a very inexpensive model that represents the fully coupled interaction between the structural vibration and the mean fluid response.

The model is verified for two conditions: 1) the rod vibrating in vacuum, and 2) the rod vibrating in water. The rod is forced at the tip using 1 second and then released. The response of the full order structural model compared to the reduced order modal model that includes the first 8 eigenmodes (4 bending modes) shows that the structure dynamics are well represented by these modes for the vibration in vacuum. When including the added mass, damping and stiffness matrices, the response of the coupled system shows a similar reduction in vibration frequency caused by the added mass effect, also observed in experimental data and the full order model response.

To assess turbulence induced vibration, data is required about the turbulence wall pressure fluctuations. This data may be obtained from an LES/DNS simulation (on a static or reduced domain), experimental data of the spectrum, or alternatively, a synthetic turbulence generator and pressure fluctuation model.

For the validation, the approach to obtain pressure fluctuations was restricted to sampling data from a DNS of a periodic 2.5cm domain representing fully developed annular flow. Wall pressure fluctuations, sampled for 3s @3kHz, are projected onto the vibration mode shapes to obtain the modal forcing due to turbulence. Three approaches were taken to use the 2.5cm sampled data to the full rod length of about 1m: for each 2.5cm section of the rod, the data is (a) duplicated at the same time instance, i.e. creating full coherence between the sections, (b) duplicated at random time instances, i.e. creating a decorrelated signal between the sections and (c) duplicated with a time lag between downstream sections, with the time lag defined by the section length and an assumed advection speed. In this comparison, method (a) with full correlation shows the best tip displacement prediction compared to experiments, with an RMS of 0.045mm versus about 0.080mm for the experiment. When the pressure fluctuations are decorrelated (b), the vibration response is largely underpredicted by an order of magnitude. Using a correlation based on an assumed advection speed (c), shows an underprediction of an order of magnitude for low velocities, and an underprediction of about a factor 4 for the largest tested advection velocity $U_c = 4.17\text{m/s}$.

5 Conclusions

In this work package, fast-running methods are identified that improve computational speed for such cases by reducing computational complexity. Each of the methods described in this report, aims to reduce the computational complexity by simplifying a specific part of the flow induced vibration problem. Two methods target in particular the effect of turbulence and turbulence pressure fluctuations: by introducing a synthetic turbulence model, the goal is to provide an alternative to the use of LES with a certain degree of accuracy, which will allow for engineering analysis of complex equipment configurations with acceptable computational costs. Two other approaches use the decomposition of the structure dynamics into decoupled modes and rely on the use of these modes to build reduced order models for the structure or the fluid response within a coupled FSI simulation.

In Chapter 1, the current AniPFM model was attempted to be validated against the cantilever rod experiment at an annular $Re=21.2k$. A few updates of the model were first presented along with the current challenges that could not be addressed within the timeframe of the project. Ultimately, the consistent observation of a ‘double parabola’ persisted in the p' field. This is known to exist due to a net source in the pressure Poisson equation that, though centered around zero for the whole domain, has local offsets before the rod tip and after the rod tip. This caused the 1-way FSI study to reveal vibration predictions that were, unexpectedly, directional instead of chaotic, and at an offset instead of centered. Furthermore, the vibration amplitudes were grossly overpredicted by ~ 8 times compared to the experiment. In terms of reactor safety and operation, this means that the AniPFM predictions of the vibration amplitude are very conservative. The frequency of vibrations, however, were correctly predicted to be the same for front and side displacements and only slightly higher than the experiment (within 2%). It is concluded that the vibration amplitudes are more affected by the p' generated by AniPFM than the vibration frequencies, which appear to be more influenced by the chosen structural model.

The mean flow and turbulence statistics input to the AniPFM model comes from a pure CFD, which results revealed good predictions of the 1st order statistics, i.e the mean axial and radial components of velocity at several elevations in the vicinity of the rod tip. This was directly an output of the base URANS $k - \omega$ SST. For validation of AniPFM, 2nd order statistics, namely the mean $u'v'$ Reynolds stress tensor component and the mean turbulent kinetic energy were compared. The experimental results coming from PIV were considered mainly to realize trends while the high fidelity data was taken as the standard for the magnitudes. The comparison revealed that the AniPFM predicted significantly different trends when compared on a location basis, but it could also be interpreted as a different evolution of the quantities in a holistic sense moving downstream where ultimately the match becomes decent as the flow becomes purely annular.

These comparisons also revealed the current gap that must be improved with future work. The main limitation was reasoned to come not from the AniPFM methodology but the exclusion of introducing anisotropy either via the Wilcox correction or using a 2-equation model that uses the Boussinesq approximation for the Reynolds stress tensor which the generated synthetic turbulence replicates.

With a 2-way FSI run in mind, vibration tests of the cantilever rod were made in vacuum using standalone Deal II, and in still water using coupled OpenFOAM-preCICE-Deal II FSI solver. The results for the solid and fluid model were calibrated and validated with experimental data as well as high fidelity DNS and LES data reported in (Hussain M. M., Bistany, Norddine, Harris, & Dubyk, 2025).

In chapter 2 the fast-running approaches used by IPP to calculate coupled vibrations are validated. The method is based on generation of turbulent pulsations, which can be taken into account when determining the resultant force. The synthetic turbulence generation (STG) procedure described previously in the report (van Zuijlen, Zwijsen, Filonova, Dubyk, & Papukchiev, 2024) was improved and applied. Two approaches of application STG were considered: the first is the implementation in an already existing solver (ANSYS CFX), and the second was to implement a simple solver for two-dimensional flow (focusing on end-to-end calculations within a single time step) with a simplified procedure for generating turbulent pulsations under the assumption of equilibrium generation and dissipation of turbulent kinetic energy. The main approaches for organizing the calculation procedure with Stable Fluid Algorithm method adaptation, as well as the method for calculating the resultant force, are outlined.

The simulation of the AMOVI benchmark was considered as a validation of the proposed methods for "fast-running analysis" of FIV based on the synthetic turbulence generation. Different cases with one flexible rod and flexible rod in a rigid tube bundle were chosen for validation. For each simulated case, the flexible rod displacement in time as well as the appropriate APSD were compared with the experimental data.

When comparing the results obtained for the same experimental AMOVI cases, a fundamental difference is observed with the same discretization of the computational domain, which is the following:

Oscillations are always excited in the SFA+ST computational procedure (at least within the timestep range of 0.1 to 30 ms and even higher), which is most likely due to the assumption of equilibrium between turbulence generation and dissipation. In other words, high numerical diffusion does not interfere with the excitation and development of oscillations. In ANSYS CFX+ST oscillations at a calculation step greater than 1 ms may not occur or may have a significantly smaller amplitude.

When coupled oscillations are established, the displacements amplitudes agree quite well for the cases where the flow unevenness (and hence the magnitude of the force) is determined directly by the moving boundary, and the oscillation modulation is determined by the turbulent nature. In a sense, these regimes are close to resonances and FEI (e.g. Single Flexible Tube AMOVI cases with 2.9 and 4.0 l/s).

PSD prediction using ANSYS CFX+ST performs better than the SFA+ST procedure, which is likely determined by the advection calculation method (the SFA+ST procedure uses the Lagrangian approximation, which inherently does not distort the system's eigenvalues and possibly does not allow to take into account the interaction of the two systems - the flow path and the rod). The method for determining pressure pulsations may also influence this.

For the calculation of complex flow configurations (rod bundles), the fields obtained by synthetic turbulence induce significantly greater hydrodynamic forces than the average URANS fields, which determines the sensitivity to fluid - elastic instability.

The predictive abilities of the developed CFX+ST and SFA+ST methods for simulated AMOVI cases compared with the results obtained using CFX without generation of synthetic turbulence fields demonstrate much better agreement of displacement RMS values obtained with CFX+ST and especially SFA+ST methods with the experimental data for both analyzed configurations – single flexible tube and flexible tube in a rigid tube bundle.

Comparing computational effectiveness shows that both methods provide the ability to perform fast-running FIV analysis (the maximum time spent for calculation of 10 seconds transient in CFX+ST was 13h 48m), while SFA+ST calculations are less computationally intensive and could be used as the fastest possible calculation of coupled vibrations. It should be also noted that the computational procedure of SFA+ST is written in an interpreted programming language (essentially it is a prototype), which means it is several times slower than codes written with Fortran/C ++ and, accordingly, has significant potential for speeding it up. Also, the SFA+ST fast calculation procedure can be calibrated on available experimental data, followed by validation on some of them (as it performed on AMOVI cases), and then be applied with greater reliability to practical problems.

In Chapter 2, FSI analyses for two configurations were performed within the validation activities for ANSYS CFX-MOR: a single flexible tube and a flexible tube in a rigid tube bundle. The single flexible tube cases with a volumetric flow rate of 2.9 l/s and 1.7 l/s were simulated with different FSI and turbulence approaches. All simulations proved that the FSI tools of ANSYS can reliably predict the vibration of the tube and the attached blade. The full-order coupling simulation provided similar results to the one with the reduced order structural model, but at approx. 6 times higher computational cost. This proved that the MOR is in-line with the FOM results. In terms of flow prediction, the hybrid ZLES model outperformed the SST turbulence model, as expected. The main advantage of ZLES lies in the higher accuracy of the tube displacement prediction: larger displacements that are closer to the measured value. In terms of vortex-shedding frequency, both turbulence models performed well.

In a second step, a more complex geometry consisting of a rigid tube bundle and the same flexible tube and its blade in the center of the bundle was investigated. As ZLES proved to deliver the best solution (also in other validation activities on FIV), it was chosen for all bundle simulations. The first case that was simulated was with a volumetric flow rate of $Q=1.6$ l/s. The comparison between experimental and numerical data is very good, not only in terms of vibration characteristics, but also in terms of time-averaged flow field. This case was provided also with PIV data by the experimentalists. In a next step, the bundle case with the highest flow rate of $Q= 1.95$ l/s was simulated. This case exhibits very large displacements (~ 5 mm), and according to the experimentalists it represents fluid-elastic instability. Frequencywise, the agreement between simulation and experiment was very good. Displacementwise, an underprediction by ANSYS CFX-MOR was observed. This could be expected, since the simulation of FEI is very hard. Still, the code predicted non-linear increase in the amplitudes when increasing the flow rate from 1.6 to 1.95 l/s, which is already a success.

It was decided to simulate also the cases with $Q=1.1$ l/s and $Q=0.5$ l/s. Both cases showed very good agreement in terms of vibration frequencies. In the first case, the displacements were remarkably overpredicted by the code, but similar behavior was found also by other partners using different turbulence and structural models. Therefore, this case needs to be further discussed with the experimentalists before final conclusions on the core performance are made. The second case (0.5 l/s) revealed massive underprediction of the displacement amplitudes.

It is not easy to draw final conclusions from the above. The simulation of FIV of a single tube or a flexible tube in a bundle using MOR is promising, but further validation is necessary. Attention shall be paid to validation matrices with wider range of boundary conditions and different geometrical configurations (more flexible tubes in a bundle). This is especially important when the FSI code is intended to be validated and then used as a general purpose code, i.e. more or less independent of geometry and boundary conditions.

In Chapter 4 the method that was developed to construct a reduced order fluid response model based on a reduced set of structural vibration modes is validated against the Cantilever Rod, turbulence induced vibration case. The fluid modal response is obtained from forced vibration simulations using the imposed forcing kinematics derived in the previous work. Although the reduced order fluid model can provide the fluid response in the whole (volumetric) domain, when coupled to the same selection of structural modes, only the fluid modal forcing response needs to be included (represented as added mass, damping and stiffness effects) to obtain the coupled fluid-structure interaction response. This model provides a very inexpensive model that represents the fully coupled interaction between the structural vibration and the mean fluid response.

The model is verified for two conditions: 1) the rod vibrating in vacuum, and 2) the rod vibrating in water. The rod is forced at the tip using 1 second and then released. The response of the full order structural model compared to the reduced order modal model that includes the first 8 eigenmodes (4 bending modes) shows that the structure dynamics are well represented by these modes for the vibration in vacuum. When including the added mass, damping and stiffness matrices, the response of the coupled system shows a similar reduction in vibration frequency caused by the added mass effect, also observed in experimental data and the full order model response.

To assess turbulence induced vibration, data is required about the turbulence wall pressure fluctuations. This data may be obtained from an LES/DNS simulation (on a static or reduced domain), experimental data of the spectrum, or alternatively, a synthetic turbulence generator and pressure fluctuation model.

For the validation, the approach to obtain pressure fluctuations was restricted to sampling data from a DNS of a periodic 2.5cm domain representing fully developed annular flow. Wall pressure fluctuations, sampled for 3s @3kHz, are projected onto the vibration mode shapes to obtain the modal forcing due to turbulence. Three approaches were taken to use the 2.5cm sampled data to the full rod length of about 1m: for each 2.5cm section of the rod, the data is (a) duplicated at the same time instance, i.e. creating full coherence between the sections, (b) duplicated at random time instances, i.e. creating a decorrelated signal between the sections and (c) duplicated with a time lag between downstream

sections, with the time lag defined by the section length and an assumed advection speed. In this comparison, method (a) with full correlation shows the best tip displacement prediction compared to experiments, with an RMS of 0.045mm versus about 0.080mm for the experiment. When the pressure fluctuations are decorrelated (b), the vibration response is largely underpredicted by an order of magnitude. Using a correlation based on an assumed advection speed (c), shows an underprediction of an order of magnitude for low velocities, and an underprediction of about a factor 4 for the largest tested advection velocity $U_c = 4.17\text{m/s}$.

6 Bibliography

- Cioncolini, A., Zhang, S., Li, H., Nabawy, M., & Iacovides, H. (2022). *Deliverable 2.1 of the GO-VIKING project (101060826) Single-phase cantilever rod experiment and benchmark setup*. The University of Manchester, Department of Mechanical, Aerospace and Civil Engineering, Manchester, UK.
- van Zuijlen, A., Zwijsen, K., Filonova, Y., Dubyk, Y., & Papukchiev, A. (2024). *Deliverable 5.1 of the GO-VIKING project (101060826) Development of fast-running FSI methods*. Delft, The Netherlands.
- Kottapalli, S., Shams, A., van Zuijlen, A. H., & Pourquie, M. J. (2019). Numerical investigation of an advanced U-RANS based pressure fluctuation model to simulate non-linear vibrations of nuclear fuel rods due to turbulent parallel-flow. *Annals of Nuclear Energy*, 128.
- Wilcox, D. C. (1993). *Turbulence Modeling for CFD*. La Cañada, California: DCW Industries.
- Zwijsen, K., van den Bos, N., Frederix, E., Roelofs, F., & van Zuijlen, A. H. (2024). Development of an Anisotropic Pressure Fluctuation Model for the Prediction of Turbulence-Induced Vibrations of Fuel Rods. *Nuclear Engineering and Design*, 425, 113316. doi:10.1016/j.nucengdes.2024.113316
- Hussain, M. M., Bistany, S., Norddine, T., Harris, E., & Dubyk, Y. (2025). *Deliverable 2.2 of the GO-VIKING project (101060826) Simulation of the single-phase Cantilever Rod benchmark*. Petten, The Netherlands.
- Einzinger, J. F. (2014). Bi-directional fluid-structure interaction with model order reduction. ANSYS Germany GmbH.
- Papukchiev, A. (2019). Numerical Analysis of Reactor Relevant Vibrations Using Advanced Multiphysics CFD-CSM Methods. *Nuclear Engineering and Design* 350 (2019) 21–32.: Elsevier.
- Papukchiev, A. (2022). FSI analysis of flow-induced vibrations in a BWR instrumentation tube experiment. Proc. of the NURETH-19 conference, online, March 6-11, 2022.
- OECD/NEA. (2023). The FSI Benchmark based on OKBM experiments to validate coupled CFD and CSD calculations. <https://www.oecd-neo.org/nsd/docs/2023/csni-r2023-2.pdf>.
- Menter, F. (2015). Best practice: scale-resolving simulations in ANSYS CFD”, Version 2.00. https://cfd.grs.de/sites/default/files/downloads/ansys/ANSYS_BPG-SRS-2.01.pdf.
- OECD/NEA. (2025). Best Practice Guidelines for the use of CFD in Nuclear Reactor Safety Applications - 2024 Update. https://www.oecd-neo.org/jcms/pl_102607/best-practice-guidelines-for-the-use-of-cfd-in-nuclear-reactor-safety-applications-2024-update?details=true.
- Lavoie, P., Avallone, G., Gregorio, F. D., Romano, G. P., & Antonia, R. A. (2007). Spatial resolution of PIV for the measurement of turbulence. *Experiments in Fluids*, 43, 39-51. doi:10.1007/s00348-007-0319-x

- Sarno, L., Tai, Y.-C., Carravetta, A., Martino, R., Papa, M. N., & Kuo, C.-Y. (2019). Challenges and improvements in applying a particle image velocimetry (PIV) approach to granular flows. *Journal of Physics: Conference Series*, 1249, 012011. doi:10.1088/1742-6596/1249/1/012011
- Piteau, P., X., D., Panunzio, D., Lagrange, R., & Antunes, J. (2022). Experimental investigation of inflow fluidelastic instability for rotated triangular tube bundles subject to single-phase and two-phase transverse flows. Paris, France: Proc. of the FIV2022 conference.
- Stam, J. (2003). Real-Time Fluid Dynamics for Games.
- Stam, J. (1999). Stable fluids. *Proceedings of the 26th annual conference on Computer graphics and interactive techniques (SIGGRAPH '99)*, 121–128. doi:10.1145/311535.311548
- Shur M. L., S. P. (2014). Synthetic Turbulence Generators for RANS-LES Interfaces in Zonal Simulations of Aerodynamic 230 and Aeroacoustic Problems. *Flow, Turbulence and Combustion*, 1(93), 63-92.
- Panunzio, D., Prévost, L., & Piteau, P. (2023). Single-Phase AMOVI Experiments and Benchmark Setup, GO-VIKING Deliverable D3.1. <https://go-viking.eu/images/resources>.
- Avramenko, A., Basok, B., & al., e. (2013). Renormalization group analysis of turbulence. *National Academy of Sciences of Ukraine*, 300.
- Bhatia, H., Norgard, G., Pascucci, V., & Bremer, P. -T. (2013). The Helmholtz-Hodge Decomposition—A Survey. *IEEE Transactions on Visualization and Computer Graphics*, 8(19), 1386-1404. doi:10.1109/TVCG.2012.316
- Bowman, J., Yassaei, M., & Basu, A. A. (2015). Fully Lagrangian Advection Scheme. *J Sci Comput*(64), 151-177. doi:10.1007/s10915-014-9928-8
- Filonov, V., Filonova, Y., Dubyk, Y., Mistry, H., Papukchiev, A., & Benhamadouche, S. (2026). GO- VIKING deliverable D3.2: AMOVI model preparation and simulation results.
- Garnier, E., Adams, N., & Sagaut, P. (2009). Large Eddy Simulation for Compressible Flows. *Springer*, 276.
- Grickevich, M. S., & Garbaruk, A. V. (2012). Imbedded LES using a volumetric turbulent source. *Physical and mathematical sciences*, 27-35.
- Moon, H., Jeong, J., Park, S., Ha, K., & Jeong, J.-H. (2023). Numerical and experimental validation of vortex generator effect on power performance improvement in MW-class wind turbine blade. *Renewable Energy*(212), 443-454. doi:10.1016/j.renene.2023.04.104
- Thomas, S., & Xu, Y. (1995). On multivariate Lagrange interpolation. *Mathematics of computation*, 1147-1170. doi:10.1090/S0025-5718-1995-1297477-5
- Hussain, M. M., Bistany, S., Norddine, T., Harris, E., Filonov, V., Dubyk, Y., & Filonova, Y. (2025). GO- VIKING deliverable D2.2: Simulation of the single-phase Cantilever Rod benchmark.

Appendix A: Matlab script to create .pre macro to define mode shape interpolations

```
% Create a .pre script to initialize many things:

% VariableList for averaging
turbModel = 'RSM'; % currently just RSM-BSL
if strcmp(turbModel,'RSM')
    avgVarListScalar = { 'Pressure', 'Eddy Viscosity', 'Turbulence Eddy Dissipation', 'Turbulence Eddy Frequency', 'Turbulence Kinetic
Energy', 'Reynolds Stress uu', 'Reynolds Stress uv', 'Reynolds Stress uw', 'Reynolds Stress vv', 'Reynolds Stress vw', 'Reynolds Stress ww'};
    avgVarNotationListScalar = { 'p', 'ev', 'ted', 'tef', 'tke', 'rsuu', 'rsuv', 'rsuw', 'rsvv', 'rsvw', 'rsww'};
    varSunit = { 'Pa', 'Pa s', 'm^2 s^-3', 's^-1', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-
2'};
    avgVarListVector = { 'Velocity' };
    avgVarListVectorComp = { 'uvw' };
    avgVarNotationListVector = { 'V' };
    varVunit = { 'm s^-1' };
else
    error('Unknown turbulence model / quantities');
end
NvarS = length(avgVarListScalar);
NvarV = length(avgVarListVector);
% the user needs to import the mode shape as interpolation functions unless
% we can write a .pre function for that ...
preFile='myCantileverBeam3DCoarse_forceOpt2.pre';

% General initialization
prefixdir = '../2a_ModalAnalysis/';
prefixfile = 'myCantilever';
modeltype = '3D'; % 1D or 3D interpolation data
interpInPre = false; % should always be false, other option extremely slow
Nmodes = 8;
srfname = 'Rod'; % name of surface to sample
modeFamp = [1 1 5 5 25 25 125 125];

% We need to define the simulation time which is used for the S-shape
% imposed motion, the number of time steps, and the modal amplitude
Tper = 0.25; % simulation time of 1 period [s]
NdtP = 1000; % number of timesteps per period [-]
NP = 1; % number of periods [-]
Am = 1e-2; % modal amplitude scaling [m]
if (NP ~= 1)
    warning('Automatic scaling of reduced order model may only work for single period simulations!');
end

% Some constants
% inverse of C - system matrix to derive added mass, damping, stiffness
c11inv = 1.144063107029098e-3;
c13inv = 7.452753954361003e-2;
c22inv = 4.216689737335880e-2;
c33inv = 7.601809033448315e0;
% Coefficients for S-shape curve with 0 value, first and second derivative
% at its extremes and integrated zero values.
c4 = 1230.1875;
c5 = -7381.125;
c6 = 17222.625;
c7 = -19683;
c8 = 11071.6875;
c9 = -2460.375;

%% Open the .pre file for continuous writing
fp = fopen(preFile,'wt');
disp(['Opened file ' preFile ' for writing...']);
% write header
txt{1}='COMMAND FILE: ';
txt{2}=' CFX Pre Version = 24.2';
txt{3}='END';
for j=1:3
    fprintf(fp,'%s\n',txt{j});
end

%% This will create the mode shape interpolation functions
dimtxt = 'xyz';
DIMtxt = 'XYZ';
% At some point we may add the option for 1D interpolation
% We can include the interpolation as a table into the .pre scripts, or we
% can write .csv Interpolation Data files
abspath = cd(cd(prefixdir));
abspath = [abspath '\'];
% use absolute path for referencing csv file
csvfile = [abspath prefixfile '_3D_Nmodes' int2str(Nmodes) '.csv'];
% 1D/3D interpolation dependent
Xmode0 = stlread([prefixdir 'mode0.stl']);
Xmode0 = Xmode0.Points;
N = length(Xmode0(:,1));
CVSdata = zeros(N,(Nmodes+1)*3);
CVSdata(:,1:3) = Xmode0;
txt{1} = 'LIBRARY: ';
txt{2} = ' CEL: ';
for j=1:2
    fprintf(fp,'%s\n',txt{j});
end
if ~interpInPre
    fprintf(fp,'%s\n', ' &replace FUNCTION: fmode');
end
```



```

fprintf(fpre,'%s\n', 'Argument Units = [m], [m], [m]');
fprintf(fpre,'%s\n', 'Option = Profile Data');
fprintf(fpre,'%s\n', 'Profile Function = On');
fprintf(fpre,'%s\n', 'Spatial Fields = X, Y, Z');
fprintf(fpre,'%s\n', 'Visibility = False');
end

for imode=1:Nmodes
% we also create a csv file for interpolations data
disp(['Processing mode ' int2str(imode) ' of ' int2str(Nmodes)]);
% we can read the STL directly
Dmode = stlread([prefixdir 'mode' int2str(imode) '.stl']);
Dmode = Dmode.Points;
Dmode = Dmode - Xmode0;
CVSdata(:,imode*3+(1:3)) = Dmode;
% Write the initial information
for idim=1:3
if interpInPre
txt{1} = ['&replace FUNCTION: fmode' int2str(imode) dimtxt(idim)];
txt{2} = 'Argument Units = [m,m,m]';
txt{3} = 'FMUI Function = Off';
txt{4} = 'GT SUITE Function = Off';
txt{5} = 'GT SUITE Turboshift = Off';
txt{6} = 'Option = Interpolation';
txt{7} = 'Profile Function = Off';
txt{8} = 'Result Units = []';
txt{9} = 'Table Function = Off';
txt{10} = 'INTERPOLATION DATA: ';
txt{11} = 'Data = \';
for j=1:11
fprintf(fpre,'%s\n',txt{j});
end
fprintf(fpre,'%15e,%15e,%15e,%15e,%15e,\n\n',[Xmode0(1:N-1,:) Dmode(1:N-1,idim)]);
fprintf(fpre,'%15e,%15e,%15e,%15e,%15e,%15e\n',[Xmode0(N,:) Dmode(N,idim)]);
% txt{1} = 'Extend Max = On';
% txt{2} = 'Extend Min = On';
txt{3} = 'Option = Three Dimensional';
txt{4} = 'END # INTERPOLATION DATA: ';
txt{5} = ['END # FUNCTION:fmode' int2str(imode) dimtxt(idim)];
for j=3:5
fprintf(fpre,'%s\n',txt{j});
end
else
fprintf(fpre,'%s\n',['DATA FIELD: m' int2str(imode) 'd' dimtxt(idim)]);
fprintf(fpre,'%s\n',['Field Name = m' int2str(imode) 'd' dimtxt(idim)]);
fprintf(fpre,'%s\n',['Result Units = []']);
fprintf(fpre,'%s\n',['END # DATA FIELD:m' int2str(imode) 'd' dimtxt(idim)]];
end
end
end
if ~interpInPre
csvfile_ = strrep(csvfile,'\','/'); % we always use forward slash in .pre file
fprintf(fpre,'%s\n', 'DATA SOURCE:');
fprintf(fpre,'%s\n',['File Name = ' csvfile_]);
fprintf(fpre,'%s\n', 'Option = From File');
fprintf(fpre,'%s\n', 'END # DATA SOURCE:');
fprintf(fpre,'%s\n', 'END #&replace FUNCTION:fmode');
end
txt{1} = 'END # CEL: ';
txt{2} = 'END # LIBRARY: ';
txt{3} = '> update';
for j=1:3
fprintf(fpre,'%s\n',txt{j});
end
% we write the csv file well
fout = fopen(csvfile,'wt');
fprintf(fout,'%s\n',[Name]);
fprintf(fout,'%s\n','fmode');
fprintf(fout,'%s\n',[Spatial Fields]);
fprintf(fout,'%s\n','X [ m ], Y [ m ], Z [ m ]');
fprintf(fout,'%s\n',[Data]);
txt{1} = 'X [ m ], Y [ m ], Z [ m ]';
fmt = '%.15e,%.15e,%.15e';
for i=1:Nmodes
txt{1} = [txt{1} ' ', m' int2str(i) 'dx [ ] ', m' int2str(i) 'dy [ ] ', m' int2str(i) 'dz [ ]'];
fmt = [fmt ' ,%.15e,%.15e,%.15e'];
end
fmt = [fmt '\n'];
fprintf(fout,'%s\n',txt{1});
fprintf(fout,fmt,CVSdata');
fclose(fout);

%% This will create expressions
txt{1} = 'LIBRARY: ';
txt{2} = 'CEL: ';
txt{3} = 'EXPRESSIONS: ';
for j=1:3
fprintf(fpre,'%s\n',txt{j});
end
Ndt = NP*NdtP;
Tsim = NP*Tper;
% Am
fprintf(fpre,'%s\n',['Am = ' num2str(Am) '[m]']);
% Tsim
fprintf(fpre,'%s\n',['Tsim = ' num2str(Tsim) '[s]']);
% Ndt
fprintf(fpre,'%s\n',['Ndt = ' int2str(Ndt)]];
% dt

```



```

fprintf(fp, 'dt = Tsim / Ndt');
% tred
fprintf(fp, 'tred = mod(Time / Tsim, 1)');
% c11inv
fprintf(fp, 'c11inv = num2str(c11inv,16) * Tsim^4 / (Am^2 * Ndt)');
% c13inv
fprintf(fp, 'c13inv = num2str(c13inv,16) * Tsim^2 / (Am^2 * Ndt)');
% c22inv
fprintf(fp, 'c22inv = num2str(c22inv,16) * Tsim^2 / (Am^2 * Ndt)');
% c33inv
fprintf(fp, 'c33inv = num2str(c33inv,16) / (Am^2 * Ndt)');
if interpInPre
    % dx % 1D/3D interpolation dependent
    fprintf(fp, 'dx = Am * fA * fmode1x(Initial X, Initial Y, Initial Z)');
    % dy % 1D/3D interpolation dependent
    fprintf(fp, 'dy = Am * fA * fmode1y(Initial X, Initial Y, Initial Z)');
    % dz % 1D/3D interpolation dependent
    fprintf(fp, 'dz = Am * fA * fmode1z(Initial X, Initial Y, Initial Z)');
else
    % dx % 1D/3D interpolation dependent
    fprintf(fp, 'dx = Am * fA * fmode.m1dx(Initial X, Initial Y, Initial Z)');
    % dy % 1D/3D interpolation dependent
    fprintf(fp, 'dy = Am * fA * fmode.m1dy(Initial X, Initial Y, Initial Z)');
    % dz % 1D/3D interpolation dependent
    fprintf(fp, 'dz = Am * fA * fmode.m1dz(Initial X, Initial Y, Initial Z)');
end
% fA
fprintf(fp, 'fA = num2str(c4,16) * tred^4 + ...
            num2str(c5,16) * tred^5 + ...
            num2str(c6,16) * tred^6 + ...
            num2str(c7,16) * tred^7 + ...
            num2str(c8,16) * tred^8 + ...
            num2str(c9,16) * tred^9 ');
% fdAdt
fprintf(fp, 'fdAdt = num2str(4*c4,16) * tred^3 + ...
            num2str(5*c5,16) * tred^4 + ...
            num2str(6*c6,16) * tred^5 + ...
            num2str(7*c7,16) * tred^6 + ...
            num2str(8*c8,16) * tred^7 + ...
            num2str(9*c9,16) * tred^8 ');
% fdAdt2
fprintf(fp, 'fdAdt2 = num2str(4*3*c4,16) * tred^2 + ...
            num2str(5*4*c5,16) * tred^3 + ...
            num2str(6*5*c6,16) * tred^4 + ...
            num2str(7*6*c7,16) * tred^5 + ...
            num2str(8*7*c8,16) * tred^6 + ...
            num2str(9*8*c9,16) * tred^7 ');
% pexpr
fprintf(fp, 'pexpr = Normal Y [Pa]');
txt{1} = 'pexpr = (';
for imode=1:Nmodes
    for idim=1:3
        txt{1} = [txt{1} ' mode' int2str(imode) dimtxt(idim) ' * Normal ' DIMtxt(idim) ' * ' num2str(modeFamp(imode)) ' +'];
    end
end
txt{1}(end) = ' ';
txt{1} = [txt{1} ') * 1 [Pa]'];
fprintf(fp, 'pexpr = ');
% mode0
fprintf(fp, 'mode0x = 1');
fprintf(fp, 'mode0y = 1');
fprintf(fp, 'mode0z = 1');
% mode1 .. modeNmodes
for imode=1:Nmodes
    for idim=1:3
        fprintf(fp, 'mode' int2str(imode) dimtxt(idim) ' = fmode.m' int2str(imode) 'd' dimtxt(idim) '(Initial X, Initial Y, Initial
Z)');
    end
end
% variables in matrix (p,u,...), (M,D,K)
% scalar vars
for iVar=1:NvarS
    varName = avgVarNotationListScalar(iVar);
    fprintf(fp, 'd' varName 'Mexpr = Ndt * (c11inv * varName 'fieldM.Trnavg' + c13inv * varName 'fieldK.Trnavg)');
    fprintf(fp, 'd' varName 'Dexpr = Ndt * c22inv * varName 'fieldD.Trnavg');
    fprintf(fp, 'd' varName 'Kexpr = Ndt * (c13inv * varName 'fieldM.Trnavg' + c33inv * varName 'fieldK.Trnavg)');
end
% vector vars
for iVar=1:NvarV
    varName = avgVarNotationListVector(iVar);
    for idim=1:3
        fprintf(fp, 'd' varName 'Mexpr' DIMtxt(idim) ' = Ndt * (c11inv * varName 'fieldM' DIMtxt(idim) '.Trnavg' + c13inv * varName
'fieldK' DIMtxt(idim) '.Trnavg)');
        fprintf(fp, 'd' varName 'Dexpr' DIMtxt(idim) ' = Ndt * c22inv * varName 'fieldD' DIMtxt(idim) '.Trnavg');
        fprintf(fp, 'd' varName 'Kexpr' DIMtxt(idim) ' = Ndt * (c13inv * varName 'fieldM' DIMtxt(idim) '.Trnavg' + c33inv * varName
'fieldK' DIMtxt(idim) '.Trnavg)');
    end
end
%modal forcing component wise and total fm{i}{x,y,z} => fm{i}
for imode=0:Nmodes
    fprintf(fp, 'fm' int2str(imode) ' = fm' int2str(imode) 'x + fm' int2str(imode) 'y + fm' int2str(imode) 'z');
    fprintf(fp, 'fm' int2str(imode) 'ref = areaInt(Fmode' int2str(imode) 'ref)'@' srfname);
    for idim=1:3
        fprintf(fp, 'fm' int2str(imode) dimtxt(idim) ' = areaInt(Fmode' int2str(imode) ' ' DIMtxt(idim) '@' srfname);
    end
end
txt{1} = ' END';
txt{2} = ' END';

```

```

txt{3} = 'END';
for j=1:3
    fprintf(fpre,'%s\n',txt{j});
end
if ~interpInPre
    fprintf(fpre,'%s\n','>initialiseProfileData Function Name=fmode, Generate CCL=Off');
end
fprintf(fpre,'%s\n','> update');

%% This will declare additional variables
fprintf(fpre,'%s\n','LIBRARY:');
%Fmode{i}, unspecified vector, [Pa]
for imode = 0:Nmodes
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: Fmode' int2str(imode)]);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = VECTOR');
    fprintf(fpre,'%s \n','Units = [ Pa ]');
    fprintf(fpre,'%s \n','Variable Type = Unspecified');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:Fmode' int2str(imode)]);
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: Fmode' int2str(imode) 'ref']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = SCALAR');
    fprintf(fpre,'%s \n','Units = [ Pa ]');
    fprintf(fpre,'%s \n','Variable Type = Specific');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:Fmode' int2str(imode) 'ref']);
end
%modal pressure field
fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: pfieldref']);
fprintf(fpre,'%s \n','Option = Definition');
fprintf(fpre,'%s \n','Tensor Type = SCALAR');
fprintf(fpre,'%s \n','Units = [ Pa ]');
fprintf(fpre,'%s \n','Variable Type = Specific');
fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:pfieldref']);
%{u,p,...}field{M,D,K}, specific scalar, [varSunit]*[m/s^2,m/s,m]
for iVar=1:NvarS
    varName = avgVarNotationListScalar{iVar};
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: ' varName 'fieldM']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = SCALAR');
    fprintf(fpre,'%s \n',['Units = [ ' varSunit{iVar} ' m s^-2 ]']);
    fprintf(fpre,'%s \n','Variable Type = Specific');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldM']);
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: ' varName 'fieldD']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = SCALAR');
    fprintf(fpre,'%s \n',['Units = [ ' varSunit{iVar} ' m s^-1 ]']);
    fprintf(fpre,'%s \n','Variable Type = Specific');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldD']);
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: ' varName 'fieldK']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = SCALAR');
    fprintf(fpre,'%s \n',['Units = [ ' varSunit{iVar} ' m ]']);
    fprintf(fpre,'%s \n','Variable Type = Specific');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldK']);
end
%{V,...}field{M,D,K}, specific vector, [varVunit]*[m/s^2,m/s,m]
for iVar=1:NvarV
    varName = avgVarNotationListVector{iVar};
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: ' varName 'fieldM']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = VECTOR');
    fprintf(fpre,'%s \n',['Units = [ ' varVunit{iVar} ' m s^-2 ]']);
    fprintf(fpre,'%s \n','Variable Type = Specific');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldM']);
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: ' varName 'fieldD']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = VECTOR');
    fprintf(fpre,'%s \n',['Units = [ ' varVunit{iVar} ' m s^-1 ]']);
    fprintf(fpre,'%s \n','Variable Type = Specific');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldD']);
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: ' varName 'fieldK']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = VECTOR');
    fprintf(fpre,'%s \n',['Units = [ ' varVunit{iVar} ' m ]']);
    fprintf(fpre,'%s \n','Variable Type = Specific');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldK']);
end
%{d,u,p,...}{M,D,K}, unspecified scalar, [varSunit]/[m/s^2,m/s,m]
for iVar=1:NvarS
    varName = avgVarNotationListScalar{iVar};
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: d' varName 'M']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = SCALAR');
    fprintf(fpre,'%s \n',['Units = [ ' varSunit{iVar} ' m^-1 s^2 ]']);
    fprintf(fpre,'%s \n','Variable Type = Unspecified');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'M']);
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: d' varName 'D']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = SCALAR');
    fprintf(fpre,'%s \n',['Units = [ ' varSunit{iVar} ' m^-1 s ]']);
    fprintf(fpre,'%s \n','Variable Type = Unspecified');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'D']);
    fprintf(fpre,'%s \n',['ADDITIONAL VARIABLE: d' varName 'K']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = SCALAR');
    fprintf(fpre,'%s \n',['Units = [ ' varSunit{iVar} ' m^-1 ]']);
    fprintf(fpre,'%s \n','Variable Type = Unspecified');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'K']);
end

```

```

end
%{d(V,...){M,D,K}, unspecified vector, [varVunit]/[m/s^2,m/s,m]}
for iVar=1:NvarV
    varName = avgVarNotationListVector{iVar};
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: d' varName 'M']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = VECTOR');
    fprintf(fpre,'%s \n',['Units = [ ' varVunit{iVar} ' m^-1 s^2 ]']);
    fprintf(fpre,'%s \n','Variable Type = Unspecified');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'M']);
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: d' varName 'D']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = VECTOR');
    fprintf(fpre,'%s \n',['Units = [ ' varVunit{iVar} ' m^-1 s ]']);
    fprintf(fpre,'%s \n','Variable Type = Unspecified');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'D']);
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: d' varName 'K']);
    fprintf(fpre,'%s \n','Option = Definition');
    fprintf(fpre,'%s \n','Tensor Type = VECTOR');
    fprintf(fpre,'%s \n',['Units = [ ' varVunit{iVar} ' m^-1 ]']);
    fprintf(fpre,'%s \n','Variable Type = Unspecified');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'K']);
end
fprintf(fpre,'%s\n','END # LIBRARY:');
fprintf(fpre,'%s\n','> update');

%% This will define additional variables
fprintf(fpre,'%s\n','FLOW: Flow Analysis 1');
fprintf(fpre,'%s\n',' DOMAIN: Default Domain');
fprintf(fpre,'%s\n',' FLUID MODELS:');

%Mmode{i}, Vector Algebraic Equation
for imode = 0:Nmodes
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: Fmode' int2str(imode)]];
    fprintf(fpre,'%s \n','Option = Vector Algebraic Equation');
    fprintf(fpre,'%s \n',['Vector xValue = (Wall Shear X - Pressure * Normal X)*mode' int2str(imode) 'x']]);
    fprintf(fpre,'%s \n',['Vector yValue = (Wall Shear Y - Pressure * Normal Y)*mode' int2str(imode) 'y']]);
    fprintf(fpre,'%s \n',['Vector zValue = (Wall Shear Z - Pressure * Normal Z)*mode' int2str(imode) 'z']]);
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:Fmode' int2str(imode)]];
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: Fmode' int2str(imode) 'ref']]);
    fprintf(fpre,'%s \n',['Additional Variable Value = pexpr * (Normal X*mode' int2str(imode) 'x + Normal Y*mode' int2str(imode) 'y + Normal Z*mode' int2str(imode) 'z )']);
    fprintf(fpre,'%s \n','Option = Algebraic Equation');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:Fmode' int2str(imode) 'ref']]);
end
%modal pressure field
fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: pfieldref']);
fprintf(fpre,'%s \n',['Additional Variable Value = pexpr']);
fprintf(fpre,'%s \n','Option = Algebraic Equation');
fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:pfieldref']);
%{u,p,...}field{M,D,K}, Algebraic Equation
for iVar=1:NvarS
    varName = avgVarNotationListScalar{iVar};
    varNameCFX = avgVarListScalar{iVar};
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: ' varName 'fieldM']);
    fprintf(fpre,'%s \n',['Additional Variable Value = ' varNameCFX '*fd2Adt2*Am/Tsim^2']);
    fprintf(fpre,'%s \n','Option = Algebraic Equation');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldM']);
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: ' varName 'fieldD']);
    fprintf(fpre,'%s \n',['Additional Variable Value = ' varNameCFX '*fdAdt*Am/Tsim']);
    fprintf(fpre,'%s \n','Option = Algebraic Equation');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldD']);
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: ' varName 'fieldK']);
    fprintf(fpre,'%s \n',['Additional Variable Value = ' varNameCFX '*fA*Am']);
    fprintf(fpre,'%s \n','Option = Algebraic Equation');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldK']);
end
%{V,...}field{M,D,K}, Vector Algebraic Equation
for iVar=1:NvarV
    varName = avgVarNotationListVector{iVar};
    varNameCFX = avgVarListVector{iVar};
    txtc = avgVarListVectorComp{iVar};
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: ' varName 'fieldM']);
    fprintf(fpre,'%s \n','Option = Vector Algebraic Equation');
    fprintf(fpre,'%s \n',['Vector xValue = ' varNameCFX ' ' txtc(1) '*fd2Adt2*Am/Tsim^2']);
    fprintf(fpre,'%s \n',['Vector yValue = ' varNameCFX ' ' txtc(2) '*fd2Adt2*Am/Tsim^2']);
    fprintf(fpre,'%s \n',['Vector zValue = ' varNameCFX ' ' txtc(3) '*fd2Adt2*Am/Tsim^2']);
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldM']);
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: ' varName 'fieldD']);
    fprintf(fpre,'%s \n','Option = Vector Algebraic Equation');
    fprintf(fpre,'%s \n',['Vector xValue = ' varNameCFX ' ' txtc(1) '*fdAdt*Am/Tsim']);
    fprintf(fpre,'%s \n',['Vector yValue = ' varNameCFX ' ' txtc(2) '*fdAdt*Am/Tsim']);
    fprintf(fpre,'%s \n',['Vector zValue = ' varNameCFX ' ' txtc(3) '*fdAdt*Am/Tsim']);
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldD']);
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: ' varName 'fieldK']);
    fprintf(fpre,'%s \n','Option = Vector Algebraic Equation');
    fprintf(fpre,'%s \n',['Vector xValue = ' varNameCFX ' ' txtc(1) '*fA*Am']);
    fprintf(fpre,'%s \n',['Vector yValue = ' varNameCFX ' ' txtc(2) '*fA*Am']);
    fprintf(fpre,'%s \n',['Vector zValue = ' varNameCFX ' ' txtc(3) '*fA*Am']);
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE: ' varName 'fieldK']);
end
%{u,p,...}{M,D,K}, Algebraic Equation
for iVar=1:NvarS
    varName = avgVarNotationListScalar{iVar};
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: d' varName 'M']);
    fprintf(fpre,'%s \n',['Additional Variable Value = ' d' varName 'Mexpr']);
    fprintf(fpre,'%s \n','Option = Algebraic Equation');
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'M']);

```



```

fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: d' varName 'D']);
fprintf(fpre,'%s \n',['Additional Variable Value = ' 'd' varName 'Dexpr']);
fprintf(fpre,'%s \n',['Option = Algebraic Equation']);
fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'D']);
fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: d' varName 'K']);
fprintf(fpre,'%s \n',['Additional Variable Value = ' 'd' varName 'Kexpr']);
fprintf(fpre,'%s \n',['Option = Algebraic Equation']);
fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'K']);
end
%{V,...}{M,D,K}, unspecified vector, [varVunit]/[m/s^2,m/s,m]
for iVar=1:NvarV
    varName = avgVarNotationListVector{iVar};
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: d' varName 'M']);
    fprintf(fpre,'%s \n',['Option = Vector Algebraic Equation']);
    fprintf(fpre,'%s \n',['Vector xValue = d' varName 'Mexpr' DIMtxt(1) ]);
    fprintf(fpre,'%s \n',['Vector yValue = d' varName 'Mexpr' DIMtxt(2) ]);
    fprintf(fpre,'%s \n',['Vector zValue = d' varName 'Mexpr' DIMtxt(3) ]);
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'M']);
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: d' varName 'D']);
    fprintf(fpre,'%s \n',['Option = Vector Algebraic Equation']);
    fprintf(fpre,'%s \n',['Vector xValue = d' varName 'Dexpr' DIMtxt(1) ]);
    fprintf(fpre,'%s \n',['Vector yValue = d' varName 'Dexpr' DIMtxt(2) ]);
    fprintf(fpre,'%s \n',['Vector zValue = d' varName 'Dexpr' DIMtxt(3) ]);
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'D']);
    fprintf(fpre,'%s \n',['&replace ADDITIONAL VARIABLE: d' varName 'K']);
    fprintf(fpre,'%s \n',['Option = Vector Algebraic Equation']);
    fprintf(fpre,'%s \n',['Vector xValue = d' varName 'Kexpr' DIMtxt(1) ]);
    fprintf(fpre,'%s \n',['Vector yValue = d' varName 'Kexpr' DIMtxt(2) ]);
    fprintf(fpre,'%s \n',['Vector zValue = d' varName 'Kexpr' DIMtxt(3) ]);
    fprintf(fpre,'%s \n',['END # ADDITIONAL VARIABLE:d' varName 'K']);
end
fprintf(fpre,'%s\n',' END # FLUID MODELS:');
fprintf(fpre,'%s\n',' END # DOMAIN:Default Domain');
fprintf(fpre,'%s\n','END # FLOW:Flow Analysis 1');
fprintf(fpre,'%s\n','> update');

%% This will define Transient statistics & Monitoring points
fprintf(fpre,'%s\n','FLOW: Flow Analysis 1');
fprintf(fpre,'%s\n',' OUTPUT CONTROL:');

% monitoring
fprintf(fpre,'%s\n',' MONITOR OBJECTS:');
for imode=1:Nmodes
    fprintf(fpre,' %s\n',['&replace MONITOR POINT: Monitor fm' int2str(imode)]];
    fprintf(fpre,' %s\n','Coord Frame = Coord 0');
    fprintf(fpre,' %s\n',['Expression Value = fm' int2str(imode)]];
    fprintf(fpre,' %s\n','Option = Expression');
    fprintf(fpre,' %s\n',['END # MONITOR POINT: Monitor fm' int2str(imode)]];
end
for imode=1:Nmodes
    fprintf(fpre,' %s\n',['&replace MONITOR POINT: Monitor reffm' int2str(imode)]];
    fprintf(fpre,' %s\n','Coord Frame = Coord 0');
    fprintf(fpre,' %s\n',['Expression Value = fm' int2str(imode) 'ref']]);
    fprintf(fpre,' %s\n','Option = Expression');
    fprintf(fpre,' %s\n',['END # MONITOR POINT: Monitor reffm' int2str(imode)]];
end
fprintf(fpre,'%s\n',' END # MONITOR OBJECTS:');

% transient
for iVar=1:NvarS
    varName = avgVarNotationListScalar{iVar};
    fprintf(fpre,' %s\n',['TRANSIENT STATISTICS: avg' varName 'M']);
    fprintf(fpre,' %s\n','Option = Arithmetic Average');
    fprintf(fpre,' %s\n',['Output Variables List = ' varName 'fieldM']);
    fprintf(fpre,' %s\n',['END # TRANSIENT STATISTICS:avg' varName 'M']);
    fprintf(fpre,' %s\n',['TRANSIENT STATISTICS: avg' varName 'D']);
    fprintf(fpre,' %s\n','Option = Arithmetic Average');
    fprintf(fpre,' %s\n',['Output Variables List = ' varName 'fieldD']);
    fprintf(fpre,' %s\n',['END # TRANSIENT STATISTICS:avg' varName 'D']);
    fprintf(fpre,' %s\n',['TRANSIENT STATISTICS: avg' varName 'K']);
    fprintf(fpre,' %s\n','Option = Arithmetic Average');
    fprintf(fpre,' %s\n',['Output Variables List = ' varName 'fieldK']);
    fprintf(fpre,' %s\n',['END # TRANSIENT STATISTICS:avg' varName 'K']);
end
for iVar=1:NvarV
    varName = avgVarNotationListVector{iVar};
    fprintf(fpre,' %s\n',['TRANSIENT STATISTICS: avg' varName 'M']);
    fprintf(fpre,' %s\n','Option = Arithmetic Average');
    fprintf(fpre,' %s\n',['Output Variables List = ' varName 'fieldM']);
    fprintf(fpre,' %s\n',['END # TRANSIENT STATISTICS:avg' varName 'M']);
    fprintf(fpre,' %s\n',['TRANSIENT STATISTICS: avg' varName 'D']);
    fprintf(fpre,' %s\n','Option = Arithmetic Average');
    fprintf(fpre,' %s\n',['Output Variables List = ' varName 'fieldD']);
    fprintf(fpre,' %s\n',['END # TRANSIENT STATISTICS:avg' varName 'D']);
    fprintf(fpre,' %s\n',['TRANSIENT STATISTICS: avg' varName 'K']);
    fprintf(fpre,' %s\n','Option = Arithmetic Average');
    fprintf(fpre,' %s\n',['Output Variables List = ' varName 'fieldK']);
    fprintf(fpre,' %s\n',['END # TRANSIENT STATISTICS:avg' varName 'K']);
end
fprintf(fpre,'%s\n',' END # OUTPUT CONTROL:');
fprintf(fpre,'%s\n','END # FLOW:Flow Analysis 1');
fprintf(fpre,'%s\n','> update');

fclose(fpre);
disp(['Done writing to ' preFile ', file closed.']);

```

Appendix B: Matlab script to create .pre macro to define .def simulation definition files for each mode

```
% Create a .pre script modify settings and save .def files:
% We assume that we have a fully functional .cfx file as input
% Variables to change:
%   location of .csv file for mode interpolation (done once)
%   mode to impose

%baseName = 'CantileverRSM-BSL_base.cfx';
defPrefix = 'CantileverExtremeCoarseRSM-BSL_tau0_1_mode';
preFile = 'createCantileverExtremeCoarseDef_tau0_1.pre';
csvFile = 'myCantilever_3D_Nmodes8.csv';
Nmodes = 8;

dimtxt = 'xyz';
fpre = fopen(preFile,'wt');
% write header
txt{1}='COMMAND FILE: ';
txt{2}=' CFX Pre Version = 24.2';
txt{3}='END';
for j=1:3
    fprintf(fpre,'%s\n',txt{j});
end

% first we change the interpolation file to local location
txt{1} = 'LIBRARY: ';
txt{2} = ' CEL: ';
for j=1:2
    fprintf(fpre,'%s\n',txt{j});
end
fprintf(fpre,'%s\n', &replace FUNCTION: fmode');
fprintf(fpre,'%s\n', Argument Units = [m], [m], [m]');
fprintf(fpre,'%s\n', Option = Profile Data');
fprintf(fpre,'%s\n', Profile Function = On');
fprintf(fpre,'%s\n', Spatial Fields = X, Y, Z');
fprintf(fpre,'%s\n', Visibility = False');

for imode=1:Nmodes
    for idim=1:3
        fprintf(fpre,'%s\n',[' DATA FIELD: m' int2str(imode) 'd' dimtxt(idim)']);
        fprintf(fpre,'%s\n',[' Field Name = m' int2str(imode) 'd' dimtxt(idim)']);
        fprintf(fpre,'%s\n', Result Units = []');
        fprintf(fpre,'%s\n',[' END # DATA FIELD:m' int2str(imode) 'd' dimtxt(idim)']);
    end
end
csvfile_ = strrep(csvFile,'\','/'); % we always use forward slash in .pre file
fprintf(fpre,'%s\n', DATA SOURCE:');
fprintf(fpre,'%s\n',[' File Name = ' csvfile_]);
fprintf(fpre,'%s\n', Option = From File');
fprintf(fpre,'%s\n', END # DATA SOURCE:');
fprintf(fpre,'%s\n', END # &replace FUNCTION:fmode');
txt{1} = ' END # CEL: ';
txt{2} = 'END # LIBRARY: ';
txt{3} = '> update';
for j=1:3
    fprintf(fpre,'%s\n',txt{j});
end
fprintf(fpre,'%s\n',>initialiseProfileData Function Name=fmode, Generate CCL=Off');

% start writing the .def files
for imode=1:Nmodes
    defFile = [defPrefix int2str(imode) '.def'];

    txt{1} = 'LIBRARY: ';
    txt{2} = ' CEL: ';
    txt{3} = ' EXPRESSIONS: ';
    for j=1:3
        fprintf(fpre,'%s\n',txt{j});
    end

    % dx % 1D/3D interpolation dependent
    fprintf(fpre,'%s\n',['dx = Am * fA * fmode.m' int2str(imode) 'dx(Initial X, Initial Y, Initial Z)']);
    % dy % 1D/3D interpolation dependent
    fprintf(fpre,'%s\n',['dy = Am * fA * fmode.m' int2str(imode) 'dy(Initial X, Initial Y, Initial Z)']);
    % dz % 1D/3D interpolation dependent
    fprintf(fpre,'%s\n',['dz = Am * fA * fmode.m' int2str(imode) 'dz(Initial X, Initial Y, Initial Z)']);

    txt{1} = ' END';
    txt{2} = ' END';
    txt{3} = ' END';
    for j=1:3
        fprintf(fpre,'%s\n',txt{j});
    end
    fprintf(fpre,'%s\n',> update');

    fprintf(fpre,'%s\n',['>writeCaseFile filename=' defFile ',operation=write def file']);
    fprintf(fpre,'%s\n',> update');
end

fclose(fpre);
```

Appendix C: Matlab script to create .cse macro for CFD-Post to export fluid surface response modes

```
% VariableList for averaging
turbModel = 'RSM'; % currently just RSM-BSL
if strcmp(turbModel,'RSM')
    avgVarListScalar = { 'Pressure', 'Eddy Viscosity', 'Turbulence Eddy Dissipation', 'Turbulence Eddy Frequency', 'Turbulence Kinetic
Energy', 'Reynolds Stress uu', 'Reynolds Stress uv', 'Reynolds Stress uw', 'Reynolds Stress vv', 'Reynolds Stress vw', 'Reynolds Stress ww'};
    avgVarNotationListScalar = { 'p', 'ev', 'ted', 'tef', 'tke', 'rsuu', 'rsuv', 'rsuw', 'rsvv', 'rsvw', 'rsww'};
    varSunit = { 'Pa', 'Pa s', 'm^2 s^-3', 's^-1', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-2', 'm^2 s^-2' };
2};
    avgVarListVector = { 'Velocity' };
    avgVarListVectorComp = { 'uvw' };
    avgVarNotationListVector = { 'V' };
    varVunit = { 'm s^-1' };
else
    error('Unknown turbulence model / quantities');
end
NvarS = length(avgVarListScalar);
NvarV = length(avgVarListVector);
DIMtxt = 'XYZ';
VELtxt = 'uvw';

csvDir='../ROMsolver/';
csvBaseFile='CantileverCoarseRSM-BSL_tau0_1';
postDir='../ROMsolver/';
postBaseFile='CantileverCoarseRSM-BSL_tau0_1.cse';
resultDir='../7_ForcedMotion/';
resultBaseFile='CantileverCoarseRSM-BSL_tau0_1';
steadyDir='../2b_SteadyFlow/';
steadyBaseFile='SteadyFlowCoarse_003';

Nmodes = 8;
srfname = 'Rod'; % name of surface to sample

%% Write a post file to export field data to .csv files
abspath = cd(cd(postDir));
abspath = [abspath '\'];
% use absolute path for referencing csv file
postFile = [abspath postBaseFile];
fpost = fopen(postFile,'wt');
disp(['Opened file ' postFile ' for writing...']);
% forced motions
fprintf(fpost,'%s\n','COMMAND FILE:');
fprintf(fpost,'%s\n',' CFX Post Version = 24.2');
fprintf(fpost,'%s\n','END');
fprintf(fpost,'%s\n','');
fprintf(fpost,'%s\n','DATA READER:');
fprintf(fpost,'%s\n',' Clear All Objects = false');
fprintf(fpost,'%s\n',' Append Results = false');
fprintf(fpost,'%s\n',' Edit Case Names = false');
fprintf(fpost,'%s\n',' Multi Configuration File Load Option = Single Case');
fprintf(fpost,'%s\n',' Open in New View = true');
fprintf(fpost,'%s\n',' Keep Camera Position = true');
fprintf(fpost,'%s\n',' Load Particle Tracks = true');
fprintf(fpost,'%s\n',' Multi Configuration File Load Option = Single Case');
fprintf(fpost,'%s\n',' Construct Variables From Fourier Coefficients = true');
fprintf(fpost,'%s\n',' Open to Compare = false');
fprintf(fpost,'%s\n',' Files to Compare =');
fprintf(fpost,'%s\n','END');

% Loop over the number of modes
for imode=1:Nmodes
    abspath = cd(cd(csvDir));
    abspath = [abspath '\'];
    % use absolute path for referencing csv file
    csvfile = [abspath csvBaseFile '_mode' int2str(imode) '.csv'];
    csvfile_ = strrep(csvfile,'\','/'); % we always use forward slash in .cse file

    % loading the case
    abspath = cd(cd(resultDir));
    resultFile = [abspath '\ resultBaseFile '_mode' int2str(imode) '_001.res'];
    resultFile_ = strrep(resultFile,'\','/'); % we always use forward slash in .cse file
    fprintf(fpost,'%s\n',['>load filename= ' resultFile_ ', force_reload=true']);
    fprintf(fpost,'%s\n','> update');
    fprintf(fpost,'%s\n','EXPORT:');
    fprintf(fpost,'%s\n',' ANSYS Export Data = Element Heat Flux');
    fprintf(fpost,'%s\n',' ANSYS File Format = ANSYS');
    fprintf(fpost,'%s\n',' ANSYS Reference Temperature = 0.0 [K]');
    fprintf(fpost,'%s\n',' ANSYS Specify Reference Temperature = Off');
    fprintf(fpost,'%s\n',' ANSYS Supplemental HTC = 0.0 [W m^-2 K^-1]');
    fprintf(fpost,'%s\n',' Additional Variable List =');
    fprintf(fpost,'%s\n',' BC Profile Type = Inlet Velocity');
    fprintf(fpost,'%s\n',' CSV Type = CSV');
    fprintf(fpost,'%s\n',[' Case Name = Case ' strrep(resultBaseFile,'-', ' ') '_mode' int2str(imode) '_001']);
    fprintf(fpost,'%s\n',' Export Connectivity = On');
    fprintf(fpost,'%s\n',' Export Coord Frame = Global');
    fprintf(fpost,'%s\n',[' Export File = ' csvfile_]);
    fprintf(fpost,'%s\n',' Export Geometry = On');
    fprintf(fpost,'%s\n',' Export Location Aliases =');
    fprintf(fpost,'%s\n',' Export Node Numbers = Off');
    fprintf(fpost,'%s\n',' Export Null Data = On');
    fprintf(fpost,'%s\n',' Export Type = Generic');
```

```

fprintf(fpost,'%s\n',' Export Units System = Current');
fprintf(fpost,'%s\n',' Export Variable Type = Current');
fprintf(fpost,'%s\n',' External Export Data = None');
fprintf(fpost,'%s\n',' Include File Information = Off');
fprintf(fpost,'%s\n',' Include Header = On');
% Location = Inlet
fprintf(fpost,'%s\n',[' Location List = ' srfname]);
fprintf(fpost,'%s\n',' Null Token = null');
fprintf(fpost,'%s\n',' Overwrite = On');
fprintf(fpost,'%s\n',' Precision = 8');
fprintf(fpost,'%s\n',' Separator = ", "');
fprintf(fpost,'%s\n',' Spatial Variables = X,Y,Z');
varlist='';
for iVar=1:NvarS
%   varName = [avrgVarNotationListScalar{iVar} 'fieldM.Trnavg'];
%   varName = ['d' avrgVarNotationListScalar{iVar} 'M'];
%   varlist = [varlist varName ','];
%   varName = [avrgVarNotationListScalar{iVar} 'fieldD.Trnavg'];
%   varName = ['d' avrgVarNotationListScalar{iVar} 'D'];
%   varlist = [varlist varName ','];
%   varName = [avrgVarNotationListScalar{iVar} 'fieldK.Trnavg'];
%   varName = ['d' avrgVarNotationListScalar{iVar} 'K'];
%   varlist = [varlist varName ','];
end
for iVar=1:NvarV
%   varName = [avrgVarNotationListVector{iVar} 'fieldM.Trnavg'];
%   varName = ['d' avrgVarNotationListVector{iVar} 'M'];
%   varlist = [varlist varName ','];
for idim=1:3
%   varName = [avrgVarNotationListVector{iVar} 'fieldM ' DIMtxt(idim) '.Trnavg'];
%   varName = ['d' avrgVarNotationListVector{iVar} 'M ' DIMtxt(idim)];
%   varlist = [varlist varName ','];
end
end
varlist(end) = [];
fprintf(fpost,'%s\n',[' Variable List = ' varlist]);
fprintf(fpost,'%s\n',' Vector Brackets = ( )');
fprintf(fpost,'%s\n',' Vector Display = Scalar');
fprintf(fpost,'%s\n','END');
fprintf(fpost,'%s\n','>export');
fprintf(fpost,'%s\n','>close');
end

% steady state
abspath = cd(cd(csvDir));
abspath = [abspath '\'];
% use absolute path for referencing csv file
csvfile = [abspath csvBaseFile '_mode0.csv'];
csvfile_ = strrep(csvfile,'\','/'); % we always use forward slash in .cse file

% loading the case
abspath = cd(cd(steadyDir));
steadyFile = [abspath '\ steadyBaseFile '.res'];
steadyFile_ = strrep(steadyFile,'\','/'); % we always use forward slash in .cse file
fprintf(fpost,'%s\n',['>load filename=' steadyFile_ ', force_reload=true']);
fprintf(fpost,'%s\n','> update');
fprintf(fpost,'%s\n','EXPORT:');
fprintf(fpost,'%s\n',' ANSYS Export Data = Element Heat Flux');
fprintf(fpost,'%s\n',' ANSYS File Format = ANSYS');
fprintf(fpost,'%s\n',' ANSYS Reference Temperature = 0.0 [K]');
fprintf(fpost,'%s\n',' ANSYS Specify Reference Temperature = Off');
fprintf(fpost,'%s\n',' ANSYS Supplemental HTC = 0.0 [W m^-2 K^-1]');
fprintf(fpost,'%s\n',' Additional Variable List =');
fprintf(fpost,'%s\n',' BC Profile Type = Inlet Velocity');
fprintf(fpost,'%s\n',' CSV Type = CSV');
fprintf(fpost,'%s\n',[' Case Name = Case ' strrep(steadyBaseFile,'-', ' ')]);
fprintf(fpost,'%s\n',' Export Connectivity = On');
fprintf(fpost,'%s\n',' Export Coord Frame = Global');
fprintf(fpost,'%s\n',[' Export File = ' csvfile_]);
fprintf(fpost,'%s\n',' Export Geometry = On');
fprintf(fpost,'%s\n',' Export Location Aliases =');
fprintf(fpost,'%s\n',' Export Node Numbers = Off');
fprintf(fpost,'%s\n',' Export Null Data = On');
fprintf(fpost,'%s\n',' Export Type = Generic');
fprintf(fpost,'%s\n',' Export Units System = Current');
fprintf(fpost,'%s\n',' Export Variable Type = Current');
fprintf(fpost,'%s\n',' External Export Data = None');
fprintf(fpost,'%s\n',' Include File Information = Off');
fprintf(fpost,'%s\n',' Include Header = On');
fprintf(fpost,'%s\n',[' Location List = ' srfname]);
fprintf(fpost,'%s\n',' Null Token = null');
fprintf(fpost,'%s\n',' Overwrite = On');
fprintf(fpost,'%s\n',' Precision = 8');
fprintf(fpost,'%s\n',' Separator = ", "');
fprintf(fpost,'%s\n',' Spatial Variables = X,Y,Z');
varlist='';
for iVar=1:NvarS
%   varName = [avrgVarListScalar{iVar}];
%   varlist = [varlist varName ','];
end
for iVar=1:NvarV
%   varName = [avrgVarListVector{iVar}];
%   varlist = [varlist varName ','];
for idim=1:3
%   varName = [avrgVarListVector{iVar} ' ' VELtxt(idim) ];
%   varlist = [varlist varName ','];
end
end
end

```

```
varlist(end) = [];  
fprintf(fpost,'%s\n',[' Variable List = ' varlist]);  
fprintf(fpost,'%s\n',' Vector Brackets = ( )');  
fprintf(fpost,'%s\n',' Vector Display = Scalar');  
fprintf(fpost,'%s\n','END');  
fprintf(fpost,'%s\n','>export');  
fprintf(fpost,'%s\n','>close');  
fclose(fpost);  
disp(['Done writing to ' postFile ', file closed.']);
```

