



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



Simulations of the single-phase Cantilever rod benchmark

Authors : Kevin ZWIJSEN (NRG), Mohammed Muaaz M. D. Hussain (NRG PALLAS), Sean Bistany (PSU), Thomas Norddine (EDF), Ewan Harris (EDF-Energy), Yaroslav Dubyk (IPP)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	Simulations of the single-phase Cantilever rod benchmark
Author(s)	Kevin ZWIJSEN, Mohammed Muaaz M. D. Hussain (NRG PALLAS), Sean Bistany (PSU), Thomas Norddine (EDF), Ewan Harris (EDF-Energy), Yaroslav Dubyk (IPP)
Number of pages	228
Document type	Deliverable
Work Package	WP2
Document number	D2.2
Issued by	NRG
Date of completion	2025-12-03 16:57:29
Dissemination level	Public

Summary

Description of the results from the numerical single-phase Cantilever Rod simulations.

Approval

Date	By
2025-12-03 17:17:46	Hidajet HADZIC (Framatome GmbH)
2025-12-04 09:24:26	Papukchiev ANGEL (GRS)



GO-VIKING deliverable D2.2: Simulation of the single-phase Cantilever Rod benchmark

Month 36

Version 1.0
December / 2025

Authors: Mohammed Muaaz M. D. Hussain (NRG PALLAS), Sean Bistany (PSU), Thomas Norddine (EDF), Ewan Harris (EDF-Energy), Yaroslav Dubyk (IPP)

EU DuC = 0E001



Funded by
the European Union

Disclaimer

The content of this report reflects only the author's view. The European Commission is not responsible for any use that may be made of the information it contains.

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



Document information

Grant Agreement / Proposal ID	101060826
Project Title	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
Project Acronym	GO-VIKING
Scientific Coordinators	Angel Papukchiev , GRS & Kevin Zwijsen , NRG PALLAS
Project starting date (duration)	1 st June 2022 – 31 st May 2026 (48 Months)
Related Work Package	WP2
Related Task(s)	Task 2.1 Cantilever Rod Benchmark
Lead Organisation	NRG PALLAS
Contributing Partner(s)	NRG PALLAS, EDF, EDF-E, PSU, IPP
Due Date	31.05.2025
Submission Date	03.12.2025
Dissemination level	PU
NRG PALLAS Reference	2.6206.20/25.329577/CS/MH/ES

History

Date	Version	Submitted by	Reviewed by	Comments
3.12.2025	1.0	NRG PALLAS	Kevin Zwijsen, Angel Papukchiev and Hidajet Hadžić	First submitted version



Table of contents

1.	Purpose	29
1.1	GO-VIKING Research Objectives	29
2.	Introduction	30
3.	Problem/experiment description	30
3.1	Flow Loop and Test Section.....	30
3.2	Rod Configurations.....	32
3.3	Available Measurement Data.....	32
4.	Benchmark	35
4.1	Introduction.....	35
4.2	Numerical simulations.....	35
4.2.1	Geometry & Mesh	36
4.2.1.1	Solid Domain	36
4.2.1.2	Fluid Domain	36
4.2.2	Material Properties	37
4.2.2.1	Solid Properties	37
4.2.2.2	Fluid Properties	38
4.2.3	Boundary Conditions	38
4.2.3.1	Solid Domain	38
4.2.3.2	Fluid Domain	39
4.2.4	Numerical Settings	39
4.2.5	Computational Resources	40
4.2.6	Simulation Strategy	41
4.2.6.1	PSU	41
4.2.6.2	EDF.....	42
4.2.6.3	NRG PALLAS.....	42
4.2.6.4	EDF-Energy	42
4.2.6.5	IPP.....	42
4.3	Obtained Results	43
4.3.1	Flow Field Data Comparison.....	43
4.3.2	Vibration Data Comparison	60
4.4	Benchmark summary and conclusions.....	64
5.	Simulations performed by PSU	66
5.1	Introduction.....	66
5.2	Methodology	66



5.2.1	Configurations	66
5.2.1.1	University of Manchester Experiments.....	67
5.2.1.2	Simulation Setups.....	67
5.2.2	Numerical Simulation Approach	68
5.2.2.1	Equations & Turbulence Analyses	68
5.2.2.2	NekRS & Computational Resources	70
5.2.2.3	Numerical Mesh	71
5.2.3	Verification & Validation	73
5.2.3.1	Pipe Flow	73
5.2.3.2	Comparisons with Experiment	74
5.3	Simulation Results	92
5.3.1	Rounded-End	92
5.3.1.1	Axial Cross-Sectional Fields	92
5.3.1.2	Q-Criterion.....	96
5.3.1.3	Turbulence Anisotropy	97
5.3.1.4	Turbulence Statistic Plots	99
5.3.2	Blunt-end	102
5.3.2.1	Axial Cross-Sectional Fields	102
5.3.2.2	Turbulence Anisotropy	105
5.3.2.3	Turbulence Statistic Plots	107
5.3.3	Wall Shear Stress Comparison	109
5.4	Conclusions and Future Work	109
6.	Simulations performed by EDF	111
6.1	Introduction.....	111
6.1.1	Executive Summary of the Current Study	111
6.2	Physics and Numerical Modelling	111
6.2.1	Euler-Bernoulli beam equation theory	111
6.2.2	Numerical model	112
6.2.2.1	Fluid model.....	112
6.2.2.2	Solid model.....	116
6.2.2.3	Coupling methodology	117
6.2.2.4	High Performance Computing.....	117
6.2.2.5	Post-treatments	118
6.2.3	Calibration of the mechanical damping with the simulations of vibration tests in still water.....	118
6.2.4	Assessment of the inlet turbulent conditions.....	122
6.3	Numerical FSI simulations of the cantilevered beam experiment at $Re=21200$...	124

6.3.1	Precursor calculation alone.....	124
6.3.2	Calculation with a fixed rod.....	125
6.3.2.1	Simulation.....	125
6.3.2.2	Numerical results	125
6.3.3	Calculation with FSI	127
6.3.3.1	Analysis of the flow	127
6.3.3.2	Surface forces and stresses exerted over the rod.....	136
6.3.3.3	FIV analysis and beam motion	138
6.4	Conclusions.....	139
7.	Simulations performed by EDF-Energy	140
7.1	Introduction.....	140
7.1.1	Scope of Current Work.....	142
7.2	Methodology	143
7.2.1	Configurations and boundary conditions.....	143
7.2.1.1	Solid Domain	143
7.2.1.2	Fluid Domain	143
7.2.2	Fluid – Structure Interaction Scheme.....	144
7.2.3	Mesh Deformation	144
7.2.4	Hemispherical/round tip case	144
7.2.5	Blunt tip cases	147
7.2.5.1	$Re = 16,000$ Blunt tip case	147
7.2.5.2	$Re = 46,000$ Blunt tip case	148
7.3	Results and discussion.....	149
7.4	Conclusions.....	156
8.	Simulations performed by NRG PALLAS	158
8.1	Introduction.....	158
8.2	Hemispherical Tip – Centered Rod.....	158
8.2.1	Solid Domain	159
8.2.2	Fluid Domain	162
8.2.3	Forced Flow FSI Study	164
8.2.3.1	Results: $Re=21.2k$	166
8.2.3.2	Results: $Re=40k$	173
8.3	Hemispherical Tip - Off-centered Rod.....	181
8.3.1	Solid Domain	182
8.3.2	Fluid Domain	182

8.3.3	Forced Flow FSI Study:	182
8.3.3.1	Results: $Re=21.2k$	183
8.4	Blunt Tip	190
8.4.1	Solid Domain	190
8.4.2	Fluid Domain	196
8.4.3	Forced Flow FSI Study	197
8.4.3.1	Results: $Re=23.33k$	198
8.5	Conclusions.....	206
9.	Simulations performed by IPP	207
9.1	Physics and numerical modelling	207
9.1.1	Fluid modelling	207
9.1.2	Mechanical modeling	208
9.1.2.1	Governing equations	208
9.1.2.2	Natural frequencies and mode shapes	210
9.1.2.3	Free vibrations.....	213
9.1.2.4	Dynamic response	215
9.1.3	Coupling details	216
9.2	Results of the numerical FSI simulations of the cantilevered beam experiment at $Re=21200$	218
9.2.1	Analysis of the flow	218
9.2.2	FIV and rod displacement analysis	222
9.3	Conclusions.....	223
10.	Overall Conclusions	225
11.	Bibliography.....	227

List of figures

Figure 1 Schematic of the two flow configurations in the test rig (Cioncolini, et al., 2022) ...	30
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 (Cioncolini, et al., 2022)	31
Figure 3 Schematic and dimension of end-pieces for cantilever rod (Hatching represents the rod pipe) (Cioncolini, et al., 2022).....	32
Figure 4 Motion tracking set-up arrangement (Cioncolini, et al., 2022)	33
Figure 5 RMS displacement and frequency for rod with different tip shapes against Reynolds number for the clamped-free configuration. (a) Resultant radial displacement. (b) Displacements from the front, X, and side, Y directions. The displacement is scaled using the rod outer diameter ($D_{rod} = 10.01$ mm). (Cioncolini, et al., 2022)	33
Figure 6 PIV set-up arrangement (Cioncolini, et al., 2022)	34
Figure 7 Time-averaged contours of the normalised axial mean velocity, U , radial mean velocity, V , normal turbulent stresses in the axial and radial directions, $u'u'$ and $v'v'$ respectively and turbulent shear stresses, $u'v'$, (U_{ref} is the mean velocity), for up flow at a Reynolds number of 40k. (Cioncolini, et al., 2022)	34
Figure 8. Time-Averaged Non-dimensional Axial Velocity 10D (Expt., EDF, NRG PALLAS, IPP) and 4.76D (EDF-Energy) ahead of Rod Tip.....	43
Figure 9. Time-Averaged Non-dimensional Radial Velocity 10D (Expt., EDF, NRG PALLAS, IPP) and 4.76D (EDF-Energy) ahead of Rod Tip.....	44
Figure 10. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) 10D ahead of Rod Tip	45
Figure 11. Comparison of Experimental (Cioncolini, et al., 2022) (top left) and Numerical Results for Non-dimensionalized Axial Component of Velocity	46
Figure 12. Comparison of Experimental (Cioncolini, et al., 2022) (top left) and Numerical Results for Non-dimensionalized Radial Component of Velocity	47
Figure 13. Comparison of Experimental (Cioncolini, et al., 2022) (top) and Numerical Results for Non-dimensionalized Stress Components	48
Figure 14. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 15.63786 mm Upstream of the Rod Tip	50
Figure 15. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 3.786006 mm Upstream of the Rod Tip	50

Figure 16. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip	51
Figure 17. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619 mm Downstream of the Rod Tip.....	51
Figure 18. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip.....	51
Figure 19. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip	52
Figure 20. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 15.63786 mm Upstream of the Rod Tip	53
Figure 21. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 3.786006 mm Upstream of the Rod Tip	53
Figure 22. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 0.493836 mm Upstream of the Rod Tip	53
Figure 23. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 0.164619 mm Downstream of the Rod Tip.....	54
Figure 24. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 6.74898 mm Downstream of the Rod Tip.....	54
Figure 25. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 29.135799 mm Downstream of the Rod Tip....	54
Figure 26. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles. Top: at Locations +5.583627 mm (Experiment, PSU, NRG PALLAS, EDF-Energy, IPP) and +5.60465 mm (EDF); Bottom: at Locations -5.609793 mm (Experiment, NRG PALLAS, EDF- Energy, IPP)	56
Figure 27. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles. Top: at Locations +6.242061 mm (Experiment, PSU, NRG PALLAS, EDF-Energy, IPP) and +6.27455 mm (EDF); Bottom: at Locations -6.926661 mm (Experiment, NRG PALLAS, EDF- Energy, IPP)	57
Figure 28. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles. Top: at Locations +7.558929 mm (Experiment, PSU, NRG PALLAS, EDF-Energy, IPP) and +7.56725 mm (EDF); Bottom: at Locations -7.585095 mm (Experiment, NRG PALLAS, EDF- Energy, IPP)	58
Figure 29. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles at Location +5.583627 mm (Experiment, PSU) and +5.60465 mm (EDF).....	59

Figure 30. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles at Locations +6.242061 mm (Experiment, PSU) and +6.27455 mm (EDF).....	59
Figure 31. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles at Locations +7.558929 mm (Experiment, PSU) and +7.56725 mm (EDF).....	60
Figure 32. Time History of the Non-dimensional Rod Tip Front (left) and Side (right) Displacements.....	60
Figure 33. 3D Time History of Rod Tip Displacements	61
Figure 34. PSD of the Tip Displacement in the Front (x-) Direction	63
Figure 35. PSD of the Tip Displacement in the Side (y-) Direction.....	64
Figure 36. UoM Experimental Setup.	67
Figure 37. Barycentric triangle representation.	69
Figure 38. Rounded-end configuration mesh element structure: inlet (blue), walls (white), outlet (red), & fluid (orange).	72
Figure 39. Comparison of Kolmogorov length scale to the distance between GLL points.	73
Figure 40. Fully developed turbulent pipe flow. Mean axial velocity profile (left) and wall-normal RMS of axial velocity fluctuations (right). Comparisons with: (Cioncolini, et al., 2023), (Toonder & Nieuwstadt, 1997) , & (Khouri, et al., 2013).	74
Figure 41. Comparison of pipe wall-normal TKE budget terms normalized by $u\tau_4/v$ compared with (Khouri, et al., 2013). Pressure diffusion, Dp , omitted for clarity.....	74
Figure 42. Time-averaged dimensionless velocity contours for Rounded-end. From left to right: UoM radial, Present radial, UoM axial, & Present axial.	75
Figure 43. Rounded-end axial velocities corresponding to locations in Table 20.	77
Figure 44. Rounded-end radial velocities corresponding to locations in Table 20.....	78
Figure 45. Rounded-end axial normal stresses corresponding to locations in Table 20.	79
Figure 46. Rounded-end radial normal stresses corresponding to locations in Table 20.	80
Figure 47. Rounded-end radial (wall-normal) shear stresses corresponding to locations in Table 20.....	81
Figure 48. Blunt-end axial velocities corresponding to locations in Table 21.	82
Figure 49. Blunt-end radial velocities corresponding to locations in Table 21.....	83

Figure 50. Blunt-end axial normal stresses corresponding to locations in Table 21.	84
Figure 51. Blunt-end radial normal stresses corresponding to locations in Table 21.	85
Figure 52. Blunt-end radial (wall-normal) shear stresses corresponding to locations in Table 21.....	86
Figure 53. Rounded-end axial velocities corresponding to locations in Table 22.	87
Figure 54. Rounded-end radial velocities corresponding to locations in Table 22.....	87
Figure 55. Rounded-end axial normal stresses corresponding to locations in Table 22.	88
Figure 56. Rounded-end radial normal stresses corresponding to locations in Table 22.	88
Figure 57. Rounded-end radial (wall-normal) shear stresses corresponding to locations in Table 22.....	89
Figure 58. Blunt-end axial velocities corresponding to locations in Table 23.	89
Figure 59. Blunt-end radial velocities corresponding to locations in Table 23.....	90
Figure 60. Blunt-end axial normal stresses corresponding to locations in Table 23.	90
Figure 61. Blunt-end radial normal stresses corresponding to locations in Table 23.	91
Figure 62. Blunt-end radial (wall-normal) shear stresses corresponding to locations in Table 23.....	91
Figure 63. Rounded-end axial velocity: instantaneous (top) & time averaged (bottom).....	92
Figure 64. Rounded-end radial velocity: instantaneous (top) & time averaged (bottom).	93
Figure 65. Rounded-end pressure: instantaneous (top) & time averaged (bottom).	93
Figure 66. Rounded-end TKE.	93
Figure 67. Rounded-end Reynolds stresses.	94
Figure 68. Rounded-end turbulence anisotropy.	95
Figure 69. Rounded-end TKE budget terms. Negative production region indicated in black.	95
Figure 70. Q-criterion revealing coherent structures.	96
Figure 71. Comparison of Q-criterion isosurfaces at four different times, T, tracking the stretching and decay of a turbulent structure in the contraction region. T1 (top left), T2 (top right), T3 (bottom left), & T4 (bottom right).....	96
Figure 72. Axial evolution of rounded-end turbulence anisotropy using Barycentric color mapping.	97

Figure 73. Rounded-end Barycentric triangle tracing at axial locations (black is wall). Red arrow is rod wall (left column) & green arrow is pipe wall (right column).	98
Figure 74. Axial evolution of Rounded-end velocity & pressure profiles.	99
Figure 75. Radial evolution of rounded-end velocity & pressure profiles.	100
Figure 76. Axial evolution of rounded-end Reynolds stresses & TKE.	100
Figure 77. Radial evolution of rounded-end Reynolds stresses & TKE.	101
Figure 78. Axial evolution of rounded-end TKE budget terms.....	101
Figure 79. Blunt-end axial velocity: instantaneous (top) & time averaged (bottom).....	102
Figure 80. Blunt-end radial velocity: instantaneous (top) & time averaged (bottom).....	102
Figure 81. Blunt-end pressure: instantaneous (top) & time averaged (bottom).....	103
Figure 82. Blunt-end TKE.	103
Figure 83. Blunt-end Reynolds stresses.	104
Figure 84. Blunt-end turbulence anisotropy.	104
Figure 85. Axial evolution of blunt-end turbulence anisotropy using Barycentric color mapping.	105
Figure 86. Blunt-end Barycentric triangle tracing at axial locations (black is wall). Red arrow is rod wall (left column) & green arrow is pipe wall (right column).	106
Figure 87. Axial evolution of Blunt-end velocity & pressure profiles.	107
Figure 88. Radial evolution of blunt-end velocity & pressure profiles.	107
Figure 89. Axial evolution of blunt-end Reynolds stresses & TKE.....	108
Figure 90. Radial evolution of blunt-end Reynolds stresses & TKE.	108
Figure 91. Dimensionless wall shear stress along entire rod wall (left) and zoomed in near free end (right). Axial position zero is rod tip without rounded-end.....	109
Figure 92. Surface mesh of the curved end-piece of the rod.	113
Figure 93. Cut view of the fluid mesh at the tip of the rod (left) and zoom on the end-piece (right).....	113
Figure 94 : Cut view of the fluid computational domain. Instantaneous velocity magnitude.	113

Figure 95. Sketch of the precursor calculation strategy. Instantaneous velocity magnitude.	114
Figure 96. Histogram of the instantaneous y +values in the fluid domain.....	114
Figure 97. Cut view of the instantaneous y +values in the fluid domain.	115
Figure 98. Histogram of the CFL number.	115
Figure 99. CFL number at the vicinity of the curved end-piece.	116
Figure 100. Zoom in the zones with highest CFL numbers. Cut view of the CFL number field above the mesh grid.	116
Figure 101 : Cut view of the mesh viscosity field in the fluid domain for the ALE methodology.	117
Figure 102. Time record of the displacement of the tip (left) and damping ratio estimated with the logarithmic decrement method. Free vibration simulations with mesh #1 and initial displacement $x_0 = 0.0001 \text{ m}$. Comparisons with several μm coefficients.	119
Figure 103. Time record of the displacement of the tip (left) and damping ratio estimated with the logarithmic decrement method. Free vibration simulations with mesh #2 and initial displacement $x_0 = 0.0001 \text{ m}$. Comparisons with several μm coefficients.	120
Figure 104. Time record of the displacement of the tip (left) and damping ratio estimated with the logarithmic decrement method. Free vibration simulations with mesh #3 and initial displacement $x_0 = 0.0001 \text{ m}$. Comparisons with several μm coefficients.	120
Figure 105. Time record of the displacement of the tip (left) and damping ratio estimated with the logarithmic decrement method. Free vibration simulations with mesh #4 and initial displacement $x_0 = 0.0001 \text{ m}$. Comparisons with several μm coefficients.	120
Figure 106. Mesh sensitivity study of the damping ratio coefficient with $\mu m = 0 \text{ Kg} \cdot \text{s} - 1$ (left) and $\mu m = 0.1 \text{ kg} \cdot \text{s} - 1$ (right).	121
Figure 107. Damping ratio sensitivity to the initial displacement x_0 with $\mu m = 0 \text{ kg} \cdot \text{s} - 1$	121
Figure 108. Tip displacement analysis of the free vibration tests with mesh #4. PSD with several μm coefficients (top left). Attractor of the displacement in the two span-wise directions, $\mu m = 0 \text{ kg} \cdot \text{s} - 1$ (bottom left). 3D representation of the displacement in the span-wise directions with respect to time, $\mu m = 0 \text{ kg} \cdot \text{s} - 1$ (right).	122
Figure 109. Comparison of the experimental upstream velocity profile and the one obtained in the LES precursor domain.	123
Figure 110. Comparison of the experimental upstream Reynolds stress profile and the one obtained in the LES precursor domain.	123

Figure 111. Cut view of the velocity field in the precursor domain.....	125
Figure 112. Cut view of the time averaged (top) and instantaneous (bottom) velocity fields close to the tip of the rod. CFD simulations with a fixed beam in central position.	125
Figure 113. Cut view of the turbulent kinetic energy field close to the tip of the rod. CFD simulations with a fixed beam in central position.....	126
Figure 114. Cut views of the 6 components of the resolved part of the Reynolds stresses close to the tip of the rod. CFD simulations with a fixed beam in central position.	127
Figure 115 : Turbulent profiles $1.56 \times 10^{-2}m$ upstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (Rzz : middle left, Ryy : middle right, Ryz : bottom).....	128
Figure 116 : Turbulent profiles $3.79 \times 10^{-3}m$ upstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (Rzz : middle left, Ryy : middle right, Ryz : bottom).....	129
Figure 117 : Turbulent profiles $4.94 \times 10^{-4}m$ upstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (Rzz : middle left, Ryy : middle right, Ryz : bottom).....	130
Figure 118 : Turbulent profiles $1.64 \times 10^{-4}m$ downstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (Rzz : middle left, Ryy : middle right, Ryz : bottom).....	131
Figure 119 : Turbulent profiles $6.75 \times 10^{-3}m$ downstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (Rzz : middle left, Ryy : middle right, Ryz : bottom).....	132
Figure 120 : Turbulent profiles $2.91 \times 10^{-2}m$ downstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (Rzz : middle left, Ryy : middle right, Ryz : bottom).....	133
Figure 121. Radial profiles Pr,i . Time-averaged axial velocity (top left), time-averaged spanwise velocity (top right), Turbulence Kinetic Energy (bottom left) and Reynolds stress profile Ryz (bottom right).....	134
Figure 122. Axial profiles Pz,i . Time-averaged axial velocity (top left), time-averaged spanwise velocity (top right), Turbulence Kinetic Energy (bottom left) and Reynolds stress profile Ryz (bottom right).....	135

Figure 123 : Cut views of the instantaneous shear stress magnitude (top) and normal stress (bottom).....	136
Figure 124 : Time average and standard deviation of the stress exerted over the rod with respect to position.	136
Figure 125. Axial component (along ez) of the axisymmetric stress on the rounded end-piece. Time-averaged values and standard deviation.....	137
Figure 126. Axial component (along ez) of the axisymmetric stress on the rod. Time-averaged values and standard deviation.....	137
Figure 127 : Comparisons of the experimental and CFD tip displacements in the transverse ex (top left) and ey (bottom left) directions. 2D displacement of the tip obtained with CFD (right).....	138
Figure 128. PSD of the tip displacement in the x-direction	139
Figure 129: Schematic of the cantilever rod experiment (Cioncolini, et al., 2018)	141
Figure 130: Flow configurations of Free-clamped (left) and clamped-free(right)	141
Figure 131: Mesh for the Hemisphere end for $Re=21000$ case: solid domain and section view of fluid domain in rod tip region. The tip region is included in both solid (left) and section (right) views.	146
Figure 132: Close - up view of the structured mesh used for the fluid domain for the Hemisphere end $Re=21000$ case.	146
Figure 133: Mesh Structure for the Blunt 16k case: fluid domain in rod tip region (left), full fluid domain with tip region highlighted (centre), both fluid and solid domains in rod tip region	148
Figure 134: Blunt Tip $Re = 16,000$ case. Displacement of the free-end vibrating beam in the x and y directions.	150
Figure 135: Rounded Tip $Re = 21,000$ case. Displacement of the free-end vibrating beam in the x and y directions.....	151
Figure 136: Blunt Tip $Re = 46,000$ case. Displacement of the free-end vibrating beam in the x and y directions.	151
Figure 137: Plot of Power Spectral Density function of Displacement in X and Y axes for Blunt $Re = 16,000$ Case.	151
Figure 138: Plot of Power Spectral Density function of Displacement in X and Y axes for Round $Re = 21,000$ Case.	152
Figure 139: Plot of Power Spectral Density function of Displacement in X and Y axes for Blunt $Re = 46,000$ Case	152

Figure 140: Contour plots of normalised axial (left) and radial (right) velocities for Blunt $Re = 16,000$ case.	153
Figure 141: Contour plots of normalised axial (left) and radial (right) velocities for Round $Re = 21,000$ case.	153
Figure 142: Contour plots of normalised axial (left) and radial(right) velocities for Blunt $Re = 46,000$ case.	154
Figure 143: Instantaneous turbulent kinetic energy contour plots for annular Reynolds numbers of $Re = 16,000$ case (left), $Re = 21,000$ case (centre) and $Re = 46,000$ case (right)	155
Figure 144: Plots of Horizontal Velocity profiles at 6.75mm downstream of the rod tip for streamwise (left column) and cross stream (right column). The plots are arranged in rows; Blunt $Re = 16,000$ (top row), Round $Re = 21,000$ (middle row), Blunt $Re = 46,000$ (bottom row).	156
Figure 145. Time History of Vibrations for the Experiment (Cioncolini, et al., 2022) (left) and Mesh H212A1 (right).....	161
Figure 146. Solid Mesh H212A1: a) Closeup of the Rod Tip, b) Sectional Front View	162
Figure 147. Time History of Vibrations for the Experiment (Cioncolini, et al., 2022) (left) and Mesh F2_ID (right)	163
Figure 148. Fluid Mesh	164
Figure 149. Time-Averaged Non-dimensional Axial Velocity 10D ahead of Rod Tip	165
Figure 150. Comparison of Numerical (left) and Experimental (Cioncolini, et al., 2022) (right) Results at $Re\ 21.2k$ for (a) Non-dimensionalized In-plane Component of Velocity and (b) Non-dimensionalized Axial Component of Velocity	166
Figure 151. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 15.63786 mm Upstream of the Rod Tip	167
Figure 152. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 3.786006 mm Upstream of the Rod Tip	167
Figure 153. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip	167
Figure 154. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619 mm Downstream of the Rod Tip	168
Figure 155. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip	168

Figure 156. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip	168
Figure 157. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.609793 mm, -6.926661 mm and -7.858095 mm	169
Figure 158. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations ± 5.609793 mm, ± 6.926661 mm and ± 7.858095 mm	169
Figure 159. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.1 mm, -6.1 mm, -7.1 mm, -8.1 mm, -9.1 mm and -10.1 mm	170
Figure 160. Non-dimensional Time History of the Non-dimensional Rod Tip Front (left top) and Side (left bottom) Displacements. 3D Time History of Rod Tip Displacements (right) ..	171
Figure 161. PSD of the Tip Displacement in the Front (x-) Direction	172
Figure 162. PSD of the Tip Displacement in the Side (y-) Direction	173
Figure 163. Comparison of Numerical (left) and Experimental (Cioncolini, et al., 2022) (right) Results at Re 40k for (a) Non-dimensionalized In-plane Component of Velocity and (b) Non-dimensionalized Axial Component of Velocity	173
Figure 164. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 15.63786 mm Upstream of the Rod Tip	174
Figure 165. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 3.786006 mm Upstream of the Rod Tip	174
Figure 166. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip	175
Figure 167. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619 mm Downstream of the Rod Tip	175
Figure 168. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip	175
Figure 169. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip	176
Figure 170. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip	176
Figure 171. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip	176
Figure 172. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.609793 mm, -6.926661 mm and -7.858095 mm	177

Figure 173. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations ± 5.609793 mm, ± 6.926661 mm and ± 7.858095 mm	177
Figure 174. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.1 mm, -6.1 mm, -7.1 mm, -8.1 mm, -9.1 mm and -10.1 mm for $Re=21.2k$ (Dashed Lines) and $Re=40k$ (Solid Lines)	178
Figure 175. Non-dimensional Time History of the Non-dimensional Rod Tip Front (left top) and Side (left bottom) Displacements. 3D Time History of Rod Tip Displacements (right) ..	179
Figure 176. PSD of the Tip Displacement in the Front (x-) Direction	180
Figure 177. PSD of the Tip Displacement in the Side (y-) Direction	181
Figure 178. Schematic of the Rod Misalignment	181
Figure 179. Time-Averaged Non-dimensional Axial Velocity 10D ahead of Rod Tip	183
Figure 180. Comparison of Numerical Off-centered (left), Centered (middle) and Experimental (Cioncolini, et al., 2022) (right) Results at $Re\ 21.2k$ for Non-dimensionalized In-plane Component of Velocity (top) and Non-dimensionalized Axial Component of Velocity (bottom)	184
Figure 181. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 15.63786 mm Upstream of the Rod Tip	185
Figure 182. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 3.786006 mm Upstream of the Rod Tip	185
Figure 183. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip	185
Figure 184. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619 mm Downstream of the Rod Tip	186
Figure 185. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip	186
Figure 186. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip	186
Figure 187. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.609793 mm, -6.926661 mm and -7.858095 mm	187
Figure 188. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.1 mm, -6.1 mm, -7.1 mm, -8.1 mm, -9.1 mm and -10.1 mm for the Centered (solid lines) and Off-Centered (dashed lines) Cases	187
Figure 189. Non-dimensional Time History of the Non-dimensional Rod Tip Front (left top) and Side (left bottom) Displacements. 3D Time History of Rod Tip Displacements (right) ..	188



Figure 190. PSD of the Tip Displacement in the Front (x-) Direction	189
Figure 191. PSD of the Tip Displacement in the Side (y-) Direction	190
Figure 192. Time History of Vibrations for the Mesh H212A10-41	193
Figure 193. Solid Mesh H212A10-41: Closeup of the Rod Tip (left), View of the End Plug (middle) and Sectional Front View (right)	195
Figure 194. Time History of Vibrations for the Mesh F2_ID	197
Figure 195. Fluid Mesh	198
Figure 196. Comparison of Instantaneous Numerical ($t=12.5FP$, left), Time-averaged Numerical (middle) and Instantaneous Experimental (Cioncolini, et al., 2022) (right) Results at $Re\ 23.33k$ for Non-dimensionalized Axial Component of Velocity	199
Figure 197. Comparison of Instantaneous Numerical ($t=12.5FP$, left), Time-averaged Numerical (middle) and Instantaneous Experimental (Cioncolini, et al., 2022) (right) Results at $Re\ 23.33k$ for Non-dimensionalized In-plane Component of Velocity	199
Figure 198. Axial (left) and Transverse (right) Velocity Profiles 45.20711503385 mm Upstream of the Rod Tip	200
Figure 199. Axial (left) and Transverse (right) Velocity Profiles 15.927518269985 mm Upstream of the Rod Tip	200
Figure 200. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 3.949501409525 mm Upstream of the Rod Tip	201
Figure 201. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.40045937981 mm Upstream of the Rod Tip	201
Figure 202. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619 mm Downstream of the Rod Tip	201
Figure 203. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip	202
Figure 204. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip	202
Figure 205. Instantaneous Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations $\pm 5.609793\ mm$, $\pm 6.926661\ mm$ and $\pm 7.858095\ mm$	202
Figure 206. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations $-5.1\ mm$, $-6.1\ mm$, $-7.1\ mm$, $-8.1\ mm$, $-9.1\ mm$ and $-10.1\ mm$	203

Figure 207. Non-dimensional Time History of the Non-dimensional Rod Tip Front (left top) and Side (left bottom) Displacements. 3D Time History of Rod Tip Displacements (right) ..	204
Figure 208. PSD of the Tip Displacement in the Front (x-) Direction	205
Figure 209. PSD of the Tip Displacement in the Side (y-) Direction	206
Figure 210. General view of the CFD model (a) according to the schematic representation of the Cantilever Rod test section (b) from benchmark setup (Deliverable 2.1).....	207
Figure 211. General view of the computational meshes	208
Figure 212. Mode shapes of cantilever rod (Euler-Bernoulli)	212
Figure 213. Influence of axial force to the first natural frequency	213
Figure 214. Free vibration in air, damping ratio 0.001	214
Figure 215. Free vibration in water, damping ratio 0.015	214
Figure 216. Arbitrary force loading	216
Figure 217. General scheme of the coupling	217
Figure 218. Cut view of the time-averaged axial velocity field (bottom) compared with averaged experimental results for Re=21200 (top).....	218
Figure 219. Cut view of the time-averaged transverse velocity field (bottom) compared with averaged experimental results for Re=21200 (top).....	219
Figure 220. Time-averaged axial and transverse velocity profiles 15.63786 mm upstream of the tip compared with experimental results	219
Figure 221. Time-averaged axial and transverse velocity profiles 3.786006 mm upstream of the tip compared with experimental results	220
Figure 222. Time-averaged axial and transverse velocity profiles 0.1646190 mm downstream of the tip compared with experimental results	220
Figure 223. Time-averaged axial and transverse velocity profiles 6.74898 mm downstream of the tip compared with experimental results	220
Figure 224. Time-averaged axial and transverse velocity profiles 29.13580 mm downstream of the tip compared with experimental results	221
Figure 225. Axial profiles – Time-averaged axial U_z velocity (left) and time-averaged transverse U_y velocity (right).....	221
Figure 226. Comparison of the experimental and CFD displacements of the rod tip in the transverse e_x direction	222

Figure 227. Comparison of the experimental and CFD displacements of the rod tip in the transverse e_y direction	222
Figure 228. PSD of the Cantilever Rod Tip displacements in X (top) and Y (bottom) directions	223

List of tables

Table 1. Key Information of the Submissions.....	35
Table 2. Tested Cases	36
Table 3. Solid Domain Details.....	36
Table 4. Fluid Domain Details.....	37
Table 5. Solid Properties.....	38
Table 6. Fluid Properties.....	38
Table 7. Solid Boundary Conditions	39
Table 8. Fluid Boundary Conditions	39
Table 9. Numerical Settings.....	40
Table 10. Computational Resources.....	41
Table 11. Vertical Profile Data Collection.....	55
Table 12. Observed Rod Tip Offset.....	62
Table 13. Obtained Displacement Results:	62
Table 14. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically.....	63
Table 15. Frequencies of the Numerical and Experimental Side Displacement PSD Peak #1 Compared to the Beam Natural Frequencies Obtained Analytically.....	64
Table 16. Geometric details of simulations.....	67
Table 17. Radial line plot locations as a function of axial distance along rod. Origin is rod tip without rounded-end.....	70
Table 18. Vertical line plot locations as a function of radial distance normal to rod wall, without the addition of the rod radius (e.g. $R9 = Di2 + 5 \times 10 - 3$). Origin is rod tip without rounded-end.	70
Table 19. Grid details of fluid meshes.....	73
Table 20. Horizontal rounded-end line plot locations. Origin is tip of rounded-end. Positive indicates downstream (up the rod) of origin.....	76
Table 21. Horizontal blunt-end line plot locations. Origin is tip of blunt-end. Positive indicates downstream of origin.....	76

Table 22. Vertical rounded-end line plot locations. Origin is tip of rounded-end. Positive indicates right of confining pipe axis. Taken from 1Dh upstream of tip and 5Dh downstream.....	76
Table 23. Vertical blunt-end line plot locations. Origin is tip of blunt-end. Positive indicates right of confining pipe axis. Taken from 1Dh upstream of tip and 5Dh downstream. .	76
Table 24. Fluid and solid meshes used for the damping ratio sensitivity study	119
Table 25. Estimation of the damping ratio (mesh sensitivity) with $x_o = 0.0001$ m.....	121
Table 26. Axial positions of the <i>Pr, i</i> radial profiles (The origin is taken at the tip of the curved end-piece)	134
Table 27. Radial positions of the <i>Pz, i</i> axial profiles (The origin is taken at the tip of the curved end-piece)	135
Table 28. Frequencies of the numerical and experimental PSD peaks compared to the beam natural frequencies obtained analytically	139
Table 29: Characteristics of the three test cases	142
Table 30: Summary table of fluid mesh properties for Round <i>Re</i> = 21,000 case.....	145
Table 31: Summary table of Mesh Parameters for Blunt <i>Re</i> = 16,000 Case	147
Table 32: Summary table of Mesh Parameters for Blunt <i>Re</i> = 46,000 Case	149
Table 33: Summary table of reduced RMS displacement amplitudes for each case compared to experimental values.	150
Table 34: Table of peak frequencies extracted for the PSD plots of each case.....	150
Table 35. Solid Material Properties.....	159
Table 36. Solid Mesh Candidates	160
Table 37. Obtained Results: (Experiment (Cioncolini, et al., 2022): <i>f_{n,a}</i> = 4.1Hz , <i>ζ_{n,a}</i> = 0.9×10^{-3}).....	161
Table 38. Fluid Material Properties.....	162
Table 39. Fluid Mesh Candidates	162
Table 40. Obtained Results: (Experiment (Cioncolini, et al., 2022): <i>f_{n,a}</i> = 3.72Hz , <i>ζ_{n,a}</i> = 15×10^{-3})	164
Table 41. Obtained Displacement Results:	171

Table 42. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically.....	172
Table 43. Frequencies of the Numerical and Experimental Side Displacement PSD Peak #1 Compared to the Beam Natural Frequencies Obtained Analytically.....	172
Table 44. Obtained Displacement Results:	179
Table 45. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically.....	180
Table 46. Frequencies of the Numerical and Experimental Side Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically.....	180
Table 47. Obtained Displacement Results:	189
Table 48. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically.....	189
Table 49. Frequencies of the Numerical and Experimental Side Displacement PSD Peak #1 Compared to the Beam Natural Frequencies Obtained Analytically.....	190
Table 50. Solid Material Properties.....	191
Table 51. Solid Mesh Candidates (A=Axial, C=Circumferential, R=Radial, CC=Cell Count)....	191
Table 52. Obtained Results: (Assumed Experimental: $f_{n,a} = 4.10\text{Hz}$, $\zeta_{n,a} = 0.9 \times 10^{-3}$)	194
Table 53. Fluid Material Properties.....	196
Table 54. Fluid Mesh Candidates	196
Table 55. Obtained Results: (Experiment (Cioncolini, et al., 2022): $f_{n,a} = 3.72\text{Hz}$, $\zeta_{n,a} = 15 \times 10^{-3}$)	197
Table 56. Obtained Displacement Results:	205
Table 57. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically.....	205
Table 58. Frequencies of the Numerical and Experimental Side Displacement PSD Peak #1 Compared to the Beam Natural Frequencies Obtained Analytically.....	206
Table 59. Four beam theories	208
Table 60. Governing equations for the most widely used theories.....	209
Table 61. Krylov or ‘beam’ functions	210
Table 62. Boundary conditions formulation	210



Table 63. Natural frequencies in water 211

Abbreviations and acronyms

Acronym	Description
CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
CFL	Courant–Friedrichs–Lewy number
CSM	Computational Solid Mechanics / Computational Structural Mechanics
D2.2	Deliverable 2.2
DNS	Direct Numerical Simulation
DOF	Degrees of Freedom
EDF	Électricité de France
EDF-Energy	EDF Energy (UK Partner)
FEA	Finite Element Analysis
FEM	Finite Element Method
FFT	Fast Fourier Transform
FSI	Fluid-Structure Interaction
FIV	Fluid-Induced Vibrations
GO-VIKING	Gathering expertise On Vibration Impact In Nuclear power Generation
GRS	Gesellschaft für Anlagen- und Reaktorsicherheit
ID	Inner Diameter
IPP	Max Planck Institute for Plasma Physics
LES	Large Eddy Simulation
NRG PALLAS	Nuclear Research and Consultancy Group PALLAS
OD	Outer Diameter
PIMPLE	PISO and SIMPLE (Combined OpenFOAM Solver)
PISO	Pressure-Implicit Split-Operator

PSD	Power Spectral Density
RANS	Reynolds-Averaged Navier–Stokes
Re	Reynolds Number
RMS	Root Mean Square
SIMPLE	Semi-Implicit Method for Pressure-Linked Equations
SSG RSM	Speziale, Sarkar, and Gatski Reynolds Stress Model
SST	Shear Stress Transport
T2.1	Task 2.1 (Cantilever Rod Benchmark)
TKE	Turbulent Kinetic Energy
URANS	Unsteady Reynolds-Averaged Navier–Stokes
VIV	Vortex-Induced Vibration
WP2	Work Package 2
WR-LES	Wall-Resolved Large Eddy Simulation

Summary

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

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

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

Keywords

FIV, FSI, CFD, Cantilever Rod, Benchmark, Numerical Simulations

1. Purpose

In the context of nuclear engineering, turbulence induced vibration of fuel rods can lead to mechanical wear, which can be responsible for safety issues and significant maintenance costs in Nuclear Power Plants (NPPs). The “Gathering expertise On Vibration ImpaKt In Nuclear power Generation” (GO-VIKING) European project aims at developing Fluid Structure Interaction (FSI) methods on simpler geometries. Both experimental and reference numerical data are used to develop engineering tools and assess industrial numerical models. In the context of WP2, a benchmark has been set up on a cantilever beam configuration, in a confined tube and hydraulic diameter representative of a fuel rod. The main objective is to compare the results of high fidelity and medium resolution approaches with the available experimental data. A moderate Reynolds number experiment at $Re=21.2k$ is chosen for the benchmark, such that the high fidelity simulations are affordable.

1.1 GO-VIKING Research Objectives

The overall objective of GO-VIKING is to increase the expertise and improve the tools and skills of the European nuclear stakeholders in the analysis of complex FIV phenomena. This will maintain and further enhance nuclear plant operation and safety.

This will be accomplished by:

- Generation of high-resolution numerical and experimental data of FIV in single and multi-phase flows
- Development and validation of high- and medium-resolution, as well as fast-running practical tools for the FIV analysis
- Implementation of efficient methods for uncertainty propagation in the FIV analysis results
- Synthesis of best practice guidelines for FIV analysis in accordance with the needs of the stakeholders
- Targeted training of graduates and young experts as well as practitioners from stakeholders in FIV analysis and modelling techniques

The GO-VIKING project will contribute to the improvement of the safety of contemporary reactors and the design evaluation of new concepts by making available new experimental results and improved numerical approaches for the evaluation of FIV.

2. Introduction

In this report, the description of the single-phase experiment, benchmark setup, and obtained numerical results are provided.

The report is structured as follows: Chapter 3 provides the Cantilever Rod problem description with all available experimental tests. Chapter 4 presents the selected benchmark case along with a comparison of obtained numerical results against the experiment. Chapters 5-9 present all numerical efforts performed by the institutions PSU, EDF, EDF-Energy, NRG PALLAS and IPP respectively. The overall summary and conclusions are provided in Chapter 10.

3. Problem/experiment description

The experiment of interest is the FIV of a cantilevered rod subjected to axial flow. This was performed at the University of Manchester, UK. This chapter provides key details of the experiment and available data. Readers interested in further details of the same are referred to the D2.1 report (Cioncolini, et al., 2022).

3.1 Flow Loop and Test Section

The test rig consists of an open loop system where a pump drives water through the test section. Both flow directions are possible and are labelled as 1) Free-Clamped setup where the water flows up the vertical test section moving from the free end of the rod to the clamped end of the rod and 2) Clamped-Free setup where the water flows down the vertical test section moving from the clamped end of the rod to the free end of the rod. These are schematically shown in Figure 1.

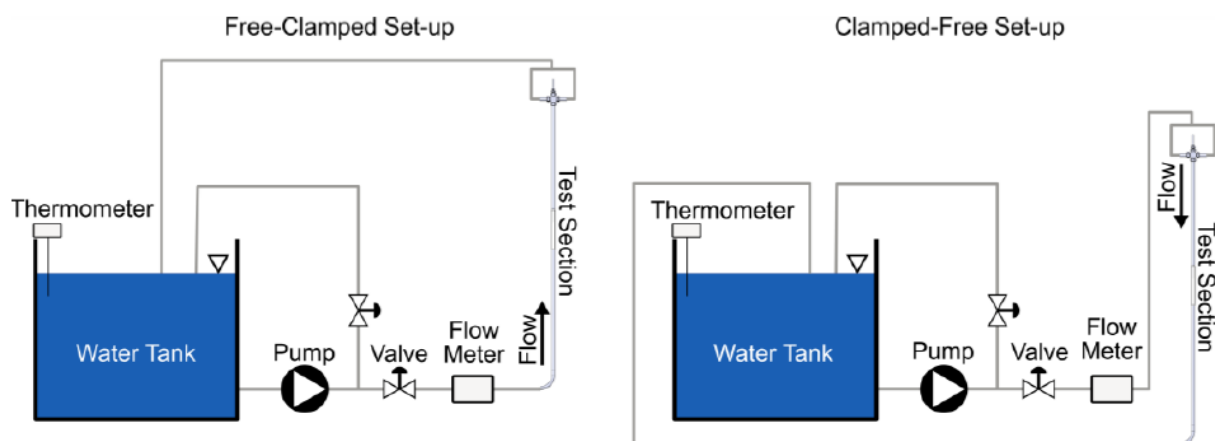


Figure 1 Schematic of the two flow configurations in the test rig (Cioncolini, et al., 2022)

The test section has a vertical vibrating rod that is clamped at the top within a union cross and free at the bottom tip, where vibration takes place, inside a confining vertical tube. A detailed schematic of the same is provided in Figure 2. Both of the aforementioned setup configurations can be implemented on the test section by simply swapping the inlet and outlet connections between the flow loop and the test section. The confining pipe has 20.86 ± 0.05 mm inner diameter (D_1) and 25.06 ± 0.01 mm outer diameter. The pipe is made of AISI 316 stainless steel and is rigidly connected to a fixed frame, attached to the rig.

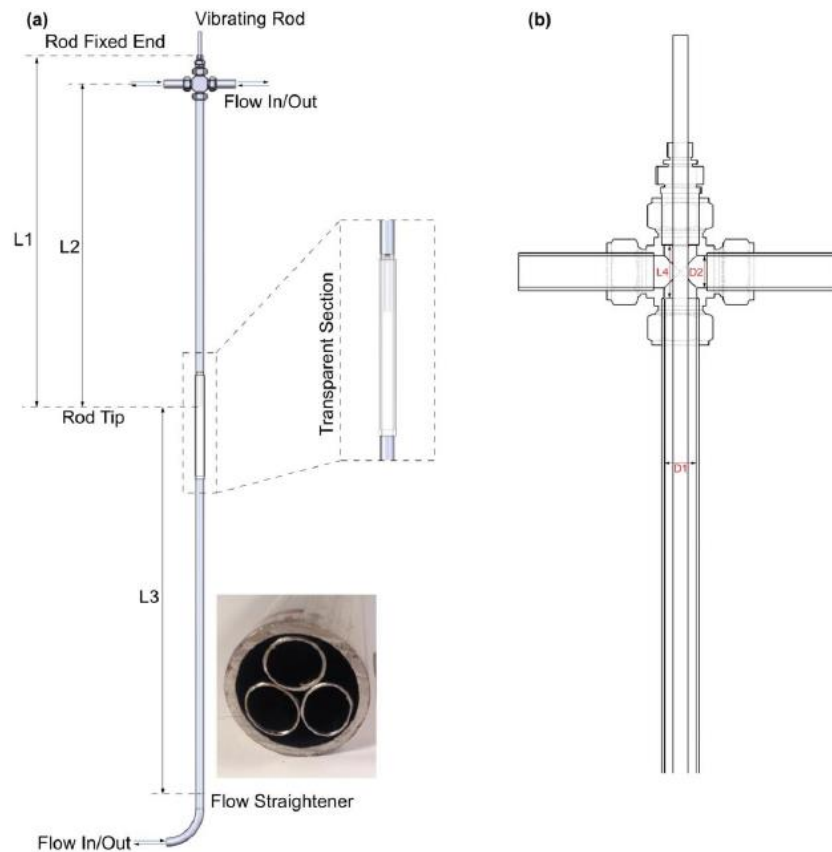


Figure 2 Experimental test section: (a) full test piece with a closeup showing the internal structure of the flow straightener; (b) union cross internal configuration (Cioncolini, et al., 2022)

A matching diameter Perspex transparent section is inserted near the rod tip for optical detection of the rod motion and flow visualization measurement. Because the flow is confined within a cylindrical tube, a surrounding rectangular box with flat faces was needed to reduce optical distortion via index matching. This surrounding structure has a square cross-section and is made of Perspex. The box structure was filled with water during experiments, which proved sufficient to remove the majority of the induced lensing effect of visualizing the flow through the cylindrical pipe. Another viewing window was also present approximately $10D_1$ upstream the rod tip to capture the flow in order to assist the numerical simulations with providing inflow turbulence.

A flow straightener is located at the bottom of the confining tube which acts to remove swirl and secondary flows with limited pressure drop for the Free-Clamped configuration. At the top, the test piece is connected to two symmetrically routed outlet pipes (same size as the test section confining tube) via a union cross which also holds the vibrating rod in place.

As shown in Figure 2, the free length of the rod from the clamping point to the free end (L_1) is 1045 ± 1 mm for the hemispherical end and 1043 ± 1 mm for the blunt end. The distance between the centre of the symmetric pipes and the free end of the rod (L_2) is 944 ± 1 mm for the hemispherical end and 942 ± 1 mm for the blunt end. Inside the union cross, the internal cross pipe section has diameter (D_2) of 21.8 mm and length (L_4) of 35.7 mm. The distance between the top of the flow straightener and the vibrating rod free end (L_3) is 1100 mm for the hemispherical end and 1102 mm for the blunt end, corresponding to a flow recovery length of ~ 53 confining pipe diameters which

is proven to be sufficient for the flow to approach fully-developed conditions upstream of the vibrating rod free-end.

3.2 Rod Configurations

The vibrating rod is made of an 8.83 ± 0.05 mm/ 10.01 ± 0.06 mm inner/outer diameter AISI 316 stainless steel circular tube (surface roughness = $2\text{--}3$ μm). The hydraulic diameter of the annular gap between the confining tube and the vibrating rod is 10.85 mm (20.86 mm – 10.01 mm). Different rod tip configurations are available including hemi-spherical, blunt and 3 cone tip ($h/d = 0.5, 1.0$ and 2.0) configurations. The different cone end-pieces were 3D printed using hard resin. For each experimental run, these cone-shaped end-pieces were firmly affixed onto the flat surface of the blunt end-piece using a secure adhesive. Figure 3 shows these configurations and dimensions of the rod end-pieces.

To minimize any potential local mass effects of the end-piece on the rod dynamics, the end-pieces were made of lightweight aluminum (density = 2.74 g.cm $^{-3}$) and have a mass of approximately 2.8 g (surface roughness = $0.4\text{--}0.5$ μm) for the curved end and approximately 2.1 g for the blunt end so that it does not appreciably alter the linear mass density of the vibrating rod. The vibrating rod was filled with lead shots (density: 11.34 g.cm $^{-3}$; diameters: $0.3\text{--}1.6$ mm; source: manufacturer data) to mimic the loading of fuel pellets in nuclear fuel bundles. The measured mass density of the rod, accounting for rod material and lead shot filling, was 588 g.m $^{-1}$.

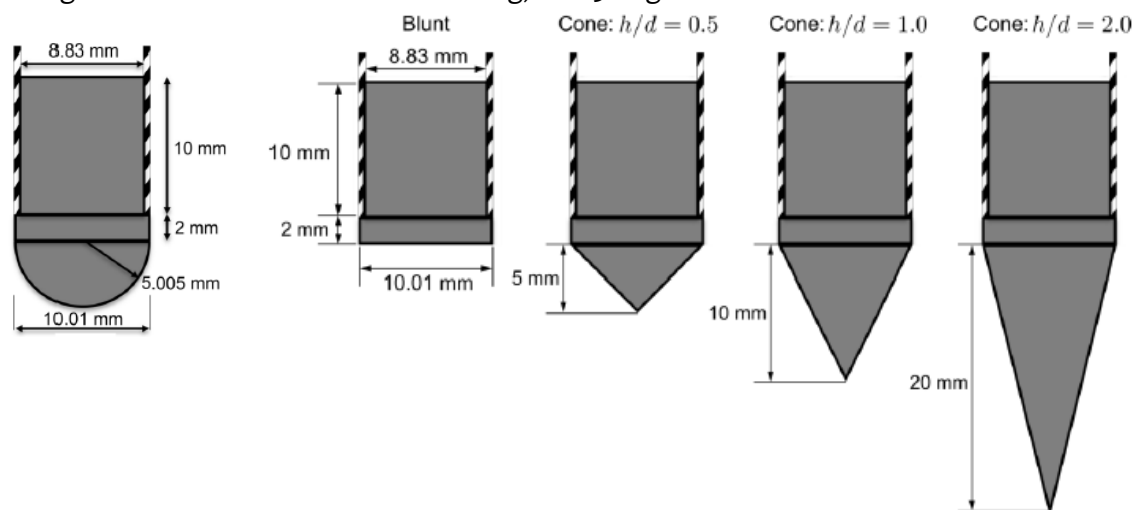


Figure 3 Schematic and dimension of end-pieces for cantilever rod (Hatching represents the rod pipe) (Cioncolini, et al., 2022)

3.3 Available Measurement Data

Broadly, there are two sets of measurement done: 1) Tracking the flow-induced motion of the free-end of the vibrating rod and 2) Flow field measurement upstream, around and downstream of the rod tip. The rod tip motion was captured using a fast-imaging camera (Panasonic Lumix DMC-FZ200; image resolution: 1280×720 pixels; recording frequency: 100 fps; recording duration: 60 s) positioned facing the test piece. The tip motion of the vibrating rod has been recorded from two sides, as a mirror placed at 45 degrees has been used so that both the front and side motions are captured as shown in Figure 4.

The experimental setup was validated with preliminary free vibration experiments in still water which gave the free vibration frequency of the rod in still water (3.72 Hz) and the damping ratio of the rod in still water (0.015 ± 0.001). Similar tests in air gave corresponding values of 4.10 Hz and 0.0009.

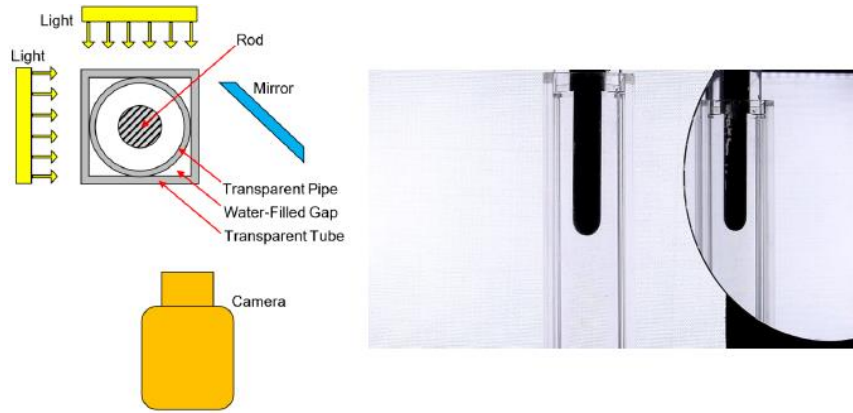


Figure 4 Motion tracking set-up arrangement (Cioncolini, et al., 2022)

The experiments are performed in a large range of Reynolds numbers from 10^4 up to 10^5 for all configurations. Key vibration results are extracted from the rod tip displacement (both in-plane directions) time history. These are the RMS rod vibration amplitude and the vibrational frequency. Select results are shown in Figure 5 for all but the curved end rod configuration.

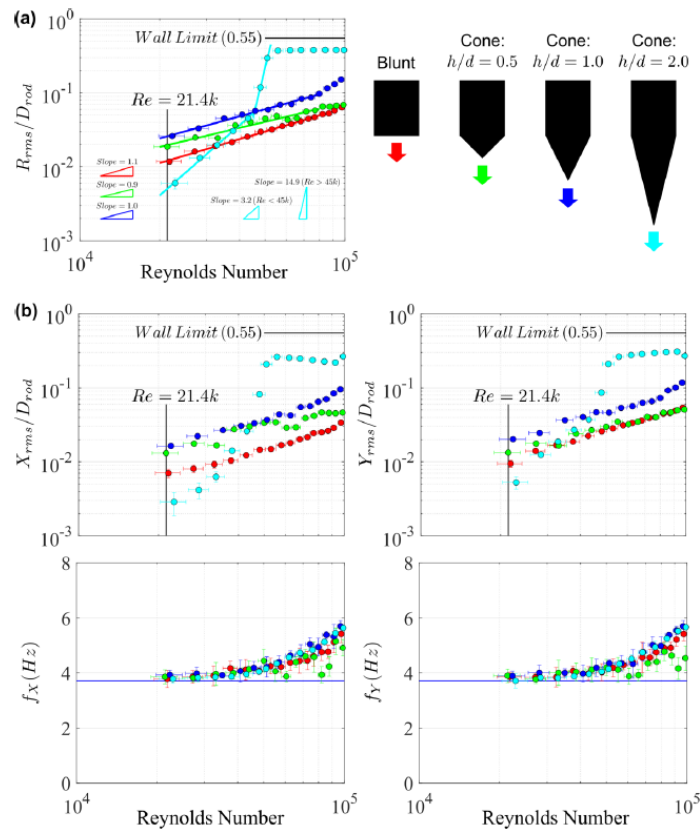


Figure 5 RMS displacement and frequency for rod with different tip shapes against Reynolds number for the clamped-free configuration. (a) Resultant radial displacement. (b) Displacements from the front, X, and side, Y directions. The displacement is scaled using the rod outer diameter ($D_{rod} = 10.01$ mm). (Cioncolini, et al., 2022)

At selected flow rates, PIV measurements were carried out to visualize the complex flow field around the vibrating rod tip. A Dantec Dynamics PIV system was used in these measurements, consisting of a double cavity Nd:YLF laser, that provides maximum output energy of 15 mJ at a wavelength of 527nm. Care was taken to avoid the shadow of the rod obscuring part of the flow visualization region. The PIV setup is shown in Figure 6. To provide upstream conditions for up flows, another viewing window has been selected at approximately $10D$ (D : Diameter of the pipe) below the tip of the rod so as to ascertain the attainment of fully-developed pipe flow conditions upstream of the rod. Key results also include measurement of horizontal and vertical profiles of the axial and horizontal components of velocity. Contour plots are available for these velocity components, Reynolds Stress Tensor components and vorticity. Select contour plots are presented in Figure 7 for the curved end for the free-clamped configuration.

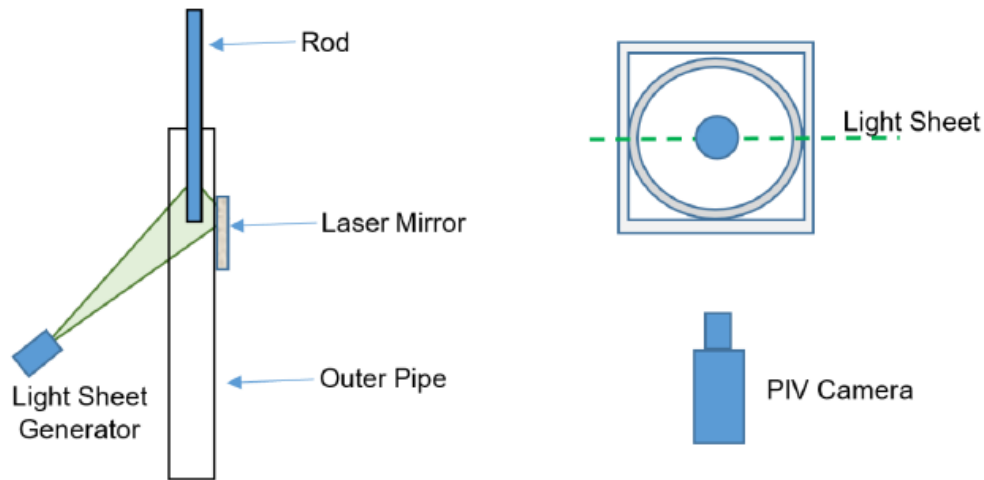


Figure 6 PIV set-up arrangement (Cioncolini, et al., 2022)

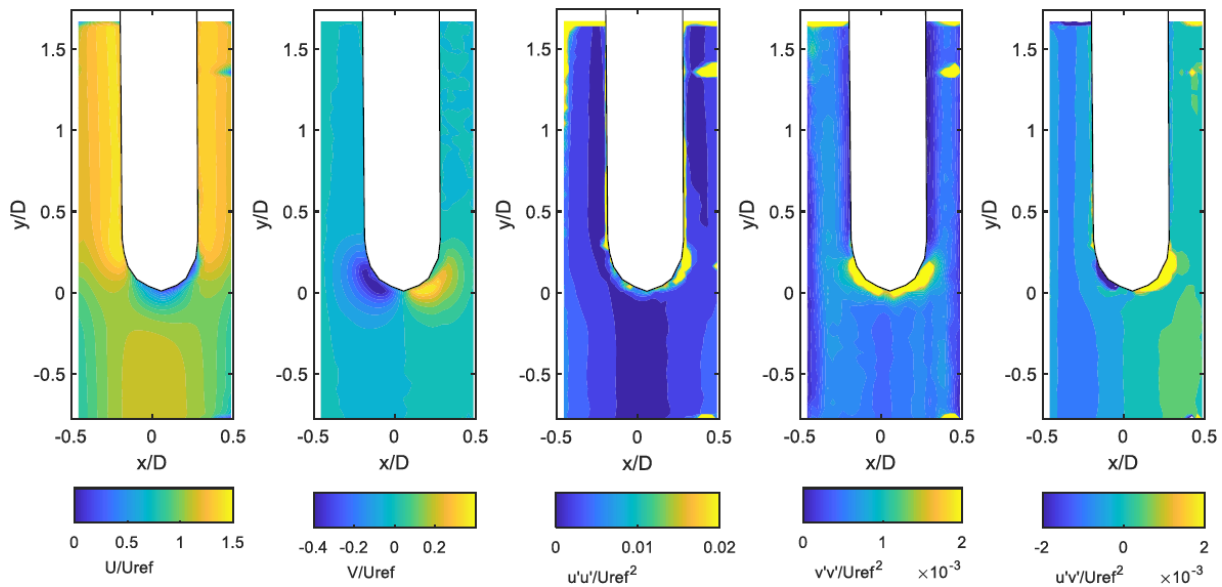


Figure 7 Time-averaged contours of the normalised axial mean velocity, U , radial mean velocity, V , normal turbulent stresses in the axial and radial directions, $u'u'$ and $v'v'$ respectively and turbulent shear stresses, $u'v'$, (U_{ref} is the mean velocity), for up flow at a Reynolds number of 40k. (Cioncolini, et al., 2022)

4. Benchmark

4.1 Introduction

The free-clamped flow at $Re=21.2k$ for the rod with a hemispherical tip was the chosen configuration for the benchmark. The curved rod tip provides a numerically challenging case since the generation of vortices and flow separation would be inhibited. Thus, the vibrations are expected to be predominantly turbulence-induced, which is typical for axial FIV. The lower Re is chosen so that high fidelity methods such as LES would still be computationally feasible in an FSI context.

A total of 5 institutions have participated in this benchmark: PSU, EDF, EDF-Energy, NRG PALLAS, and IPP. Key details of the numerical simulations of the participants are provided in Section 4.2. The key results compared to the experiment are segregated based on flow fields and based on FIV rod response as presented in Section 4.3. The benchmark findings are concluded with a summary provided in Section 4.4.

4.2 Numerical simulations

Table 1 summarizes the participants' individual submissions. PSU performed pure Computational Fluid Dynamics (CFD) simulations while all others performed two-way FSI calculations. PSU and EDF performed high fidelity simulations while NRG PALLAS, EDF-Energy and IPP performed medium resolution URANS simulations. Although the focus of this chapter will be on the benchmark case of $Re=21.2k$ up-flow for the hemispherical rod tip, additional simulations were performed by the institutions and have been summarized in Table 2. Details on all the results achieved by each participating institution can be found in the dedicated chapters 5 through 9.

Table 1. Key Information of the Submissions

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



Table 2. Tested Cases

Institution	Cases Performed
PSU	Hemispherical End: Re=11.077k, 21.2k Blunt End: Re=12.19k
EDF	Hemispherical End: Re=21.2k
NRG PALLAS	Hemispherical End: Re=21.2k, 40k Hemispherical End with Offset: Re=21.2k Blunt End: Re=23.33k
EDF-Energy	Hemispherical End: Re=21k Blunt End: 16k, 46k
IPP	Hemispherical End: Re=21.2k

4.2.1 Geometry & Mesh

4.2.1.1 Solid Domain

The rod in the experiment was a composite one with a stainless steel hollow cylinder, an aluminum end plug and was filled with lead shots. The chosen geometries for the solid domain by the participants performing FSI simulations are summarized in Table 3. NRG PALLAS has excluded only the lead shots, while all other participants performing FSI calculations have adopted a homogenous solid cylinder, which is represented by a beam model by EDF and IPP while an FEM CSM is performed by EDF-Energy. NRG PALLAS and IPP use a rod diameter of 10.01 mm (matching the experiment) while EDF and EDF-Energy use a rod diameter of 10 mm. PSU uses a rod diameter of 476.19 mm owing to the scaling of other parameters including inflow velocity, density and confining pipe diameter to unity.

Table 3. Solid Domain Details

Institution	Geometry Model	Mesh Type	Mesh Count
EDF	Homogenous Rod with one Material	1D Elements	3.8k Elements
NRG PALLAS	Composite Rod with modified SS Hollow Cylinder + Aluminum Plug	Structured Hexahedral for the Hollow Cylinders, Tetrahedral for the Aluminum Plug	178.4k Cells
EDF-Energy	Homogenous Rod with one Material	Structured Hexahedral for the Cylinders, Tetrahedral for the Hemispherical End	62.5k Cells
IPP	Homogenous Rod with one Material	1D Elements	0.1k Points

4.2.1.2 Fluid Domain

In the experiment, the entire test loop can be considered the fluid domain inclusive of the tank and pumps. However, owing to computational constraints, this cannot be adopted. The chosen geometries for the fluid domain by the participants are summarized in Table 4.

NRG PALLAS and IPP have considered the full confining pipe length and the union cross with two outlets while the other participants have assumed a single outlet configuration considering only a portion of the confining pipe length upstream of the rod tip. NRG PALLAS and IPP use a confining pipe diameter of 20.86 mm (matching experiment) while EDF and EDF-Energy use a confining pipe diameter of 21 mm. PSU uses a scaled confining pipe diameter of 1 m.

Table 4. Fluid Domain Details

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

4.2.2 Material Properties

4.2.2.1 Solid Properties

The adopted solid properties are listed in Table 5 for all participants excluding PSU (since there are no solids). Single values are presented for all participants except NRG PALLAS who has used a composite solid. Roughness is only included by NRG PALLAS. Inherent structural damping is included by EDF and NRG PALLAS.

Regarding the solid properties, they are chosen in a way that respects the total mass of the rod obtained from the linear mass density of 588 g/m quoted in the experiment. Therefore, those who use a 1D model (EDF, IPP) or those who have explicitly used a hollow cylinder (NRG PALLAS) end up with a much higher density than those who have a solid cylinder (EDF-Energy). Correspondingly, the Youngs modulus is selected either from standard properties (EDF, NRG PALLAS, IPP) or are modified to respect the flexural rigidity of the rod based on structural tests in vacuum and water (EDF-Energy).

Table 5. Solid Properties

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

4.2.2.2 Fluid Properties

The adopted fluid properties are listed in Table 6 for all participants. With the exception of PSU who has modified fluid properties, all participants adopt water properties at similar temperatures. Concerning the derivation of dynamic viscosity by PSU, the reader is referred to section 5.2.1.2.

Table 6. Fluid Properties

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

4.2.3 Boundary Conditions

4.2.3.1 Solid Domain

The adopted solid boundary conditions are provided in Table 7 for all participants who perform FSI calculations.

Table 7. Solid Boundary Conditions

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

4.2.3.2 Fluid Domain

The adopted fluid boundary conditions are listed in Table 8 for all participants. PSU uses modified inflow velocity of 1 m/s. The mass flow rate value used by IPP can be translated to an inflow velocity of 1.51 m/s, similar to the chosen value by NRG PALLAS. EDF and EDF-Energy have similar inflow velocities. The slight difference between the pairs can be attributed to the slight difference in rod and confining pipe diameters.

Table 8. Fluid Boundary Conditions

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

4.2.4 Numerical Settings



Each participant has made a choice for the numerical settings. This information is presented in Table 9.

Table 9. Numerical Settings

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

4.2.5 Computational Resources

The computational resources consumed by each participant in the endeavor of the benchmark are presented in Table 10.

Table 10. Computational Resources

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

4.2.6 Simulation Strategy

Each participant has taken a unique approach to perform the simulation of the benchmark case. This is presented for each institution:

4.2.6.1 PSU

The following strategy is adopted:

- The inlet recycling length is equivalent to four pipe diameters and serves as the precursor domain. At first, the simulation is run with a lower Re , and the turbulence is fully developed.
- The simulations with the cantilevered rod is initialized with this fully developed turbulent pipe flow mapped to the pipe inlet.
- The Reynolds number is gradually increased for stability of the solver.

Once the target Re is achieved, to flush out any flow development effects, the simulation was run for five to ten flow passes before averaging, collecting turbulence statistics, and collecting time

history data. The latter activities are carried out for the aforementioned ~8.59s (or approx. 20 flow throughs).

4.2.6.2 EDF

The following strategy is adopted:

- First, only the precursor domain is simulated to generate inflow boundary conditions. This is done for 10 s simulation time.
- Then, this is supplied to the inflow of the actual domain with the rod remaining fixed and is thus a pure CFD calculation to flood the domain with turbulent inflow and develop forces on the rod. This is done for a simulation of 12 s, i.e from 10 s to 22 s.
- Finally, the rod is released and the oscillations are captured. This goes on for another 60s, i.e from 22 s to 82 s, which marks the rest of the simulation.

To compare the results, the last 50 s is used to carry out the statistical analysis that is used to compare against the experimental data.

4.2.6.3 NRG PALLAS

The following strategy is adopted:

- First, a pure CFD RANS calculation is done starting the flow from rest and applying inflow boundary conditions. This is done for 2000 iterations which was observed to be enough to generate the flow field around the rod.
- Then, the solver is switched to URANS and the FSI simulation is carried out for a duration of 19.643 s or 14.28 flow passes (FP) where 1 FP = 1.375551606 s.

To perform the statistics in order to compare results, the first 3.5FP is ignored for the vibration while the first 4.5FP is ignored for the velocity fields.

4.2.6.4 EDF-Energy

The following strategy is adopted:

- First, a pure CFD URANS calculation is performed, with the solid rod domain fixed completely and uncoupled from the fluid domain for 1.0 seconds.
- Then, the FSI coupling is enabled at 1.0 s until the end of the simulation at 11 s. The FSI traction is ramped linearly from 1.00 s to 1.01 s.

To perform the statistics in order to compare results, the first 2.02 seconds of the time series are removed. The remaining was then used to perform the PSD calculation and calculate the characteristic frequencies of the system.

4.2.6.5 IPP

The following strategy is adopted:

- First, a pure CFD URANS calculation is done starting the flow from rest and applying inflow boundary conditions with the rod fixed. This is done for 10 s.
- Then, the rod is allowed to move and the FSI simulation is carried out for a duration of 30 s, i.e from 10 s to 40 s.

To perform the statistics in order to compare results, the entire 30 s of the FSI part of the calculation is used.

4.3 Obtained Results

The data obtained from the participants are consolidated into appropriate plots and are presented in this section. Comparison to the PIV experimental data are made in section 4.3.1 while those for the vibration data are provided in section 4.3.2.

4.3.1 Flow Field Data Comparison

Before presenting the results, a first look is taken at the verification taken by participants 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. However, only NRG PALLAS and IPP have a simulation domain that extends to and beyond this location. EDF-Energy has provided data at 4.76D upstream of the rod tip while EDF has provided data at a location inside the precursor domain which implies perfectly developed pipe flow without any influence of the downstream rod. PSU did not submit any upstream data.

The available data is plotted and displayed in Figure 8, Figure 9 and Figure 10 for the non-dimensional axial velocity, non-dimensional radial velocity and non-dimensional Reynolds stress component $u'v'$ respectively. Focusing on Figure 8, 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 high and medium fidelity solutions is at the centre of the confining pipe where the maximum non-dimensional velocity is captured. The maximum variation of $\sim 7\%$ is observed between the solution of EDF and NRG PALLAS for the maximum velocity. Despite these differences in the solution amongst the participants, all of them can still be considered to be a good match against the experiment.

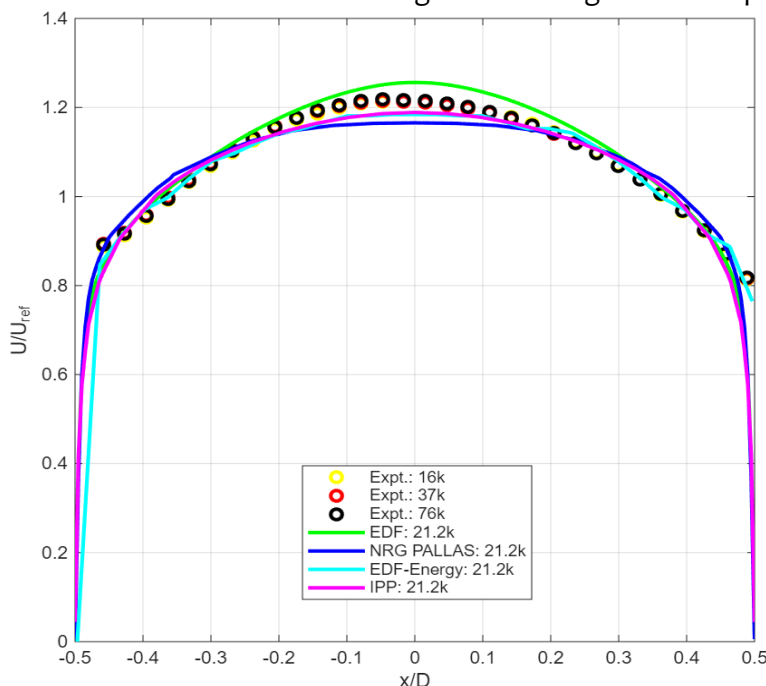


Figure 8. Time-Averaged Non-dimensional Axial Velocity 10D (Expt., EDF, NRG PALLAS, IPP) and 4.76D (EDF-Energy) ahead of Rod Tip

From Figure 9, 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 numerical results predict a similar order of magnitude. In particular, the solutions by NRG PALLAS and IPP show a very smooth and almost symmetric curve matching the trend expected from the experiment (excluding scattered data). The solution by EDF predicts mostly a positive radial velocity except for a distance of $0.1D$ at the left pipe wall. This can be explained from the fact that the EDF solution is taken from the precursor domain where the situation is turbulent pipe flow with no influence of the cantilever rod. Finally, the solution of EDF-Energy predicts a higher magnitude of radial velocity, but this is explained from the fact that the data is collected $4.76D$ ahead of the rod tip instead of $10D$.

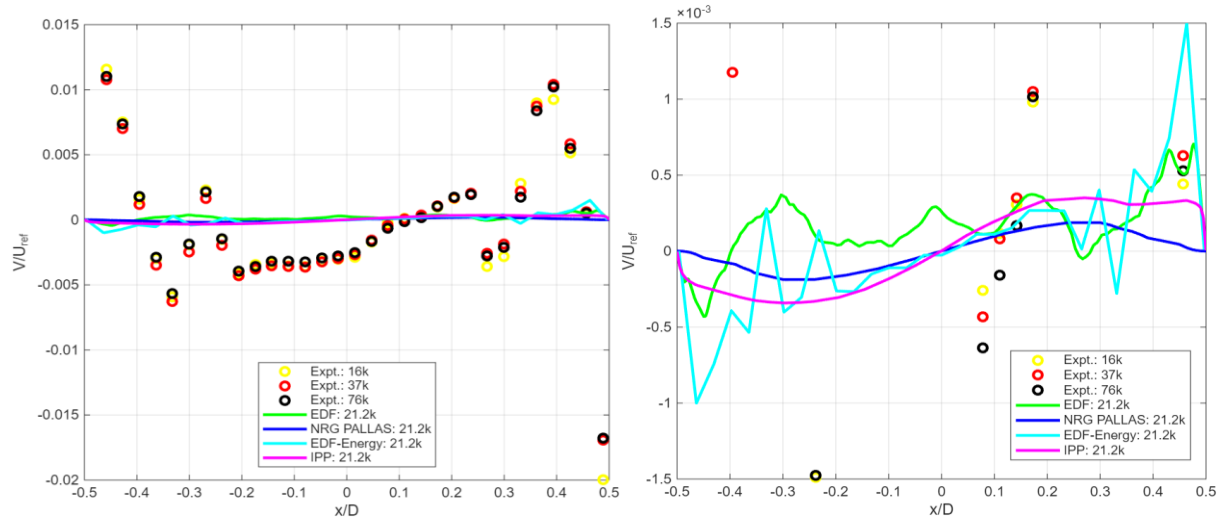


Figure 9. Time-Averaged Non-dimensional Radial Velocity $10D$ (Expt., EDF, NRG PALLAS, IPP) and $4.76D$ (EDF-Energy) ahead of Rod Tip

Figure 10 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 numerical solution of EDF predicts a trend similar to the experiment but overpredicts the magnitude compared to the experiment. But this is expected, because PIV is generally not suitable for measuring second order statistics because 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, et al., 2007) (Sarno, et al., 2019).

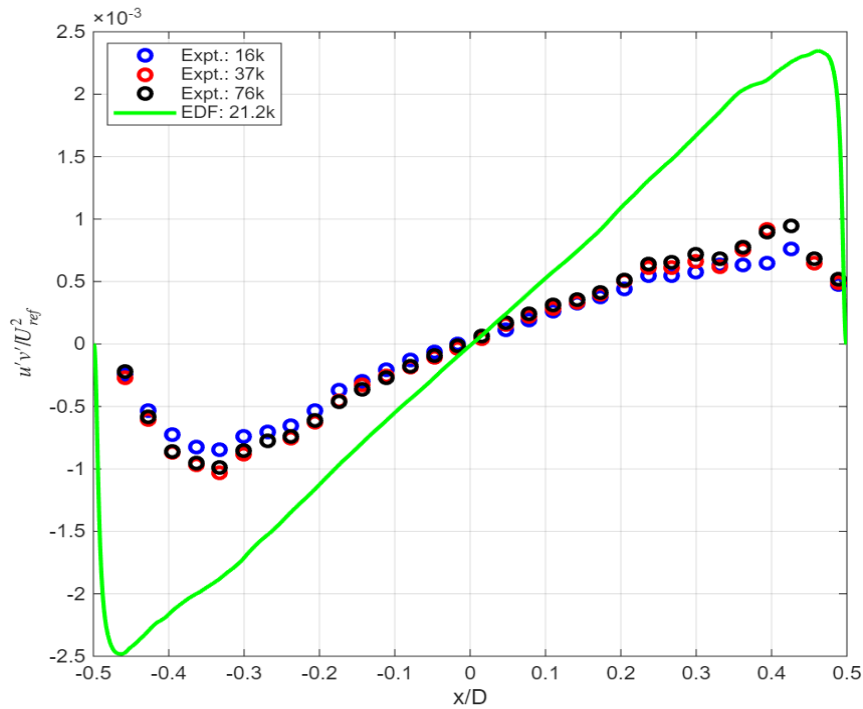


Figure 10. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) 10D ahead of Rod Tip

The time-averaged non-dimensional velocity contour plots are shown in Figure 11 and Figure 12. 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 numerical results are found to be in good agreement with the experimental results.

The time-averaged non-dimensional stress component contours are shown in Figure 13 for the participants who performed high fidelity simulations. The numerical results are again found to be in good agreement with the experimental results and with each other.

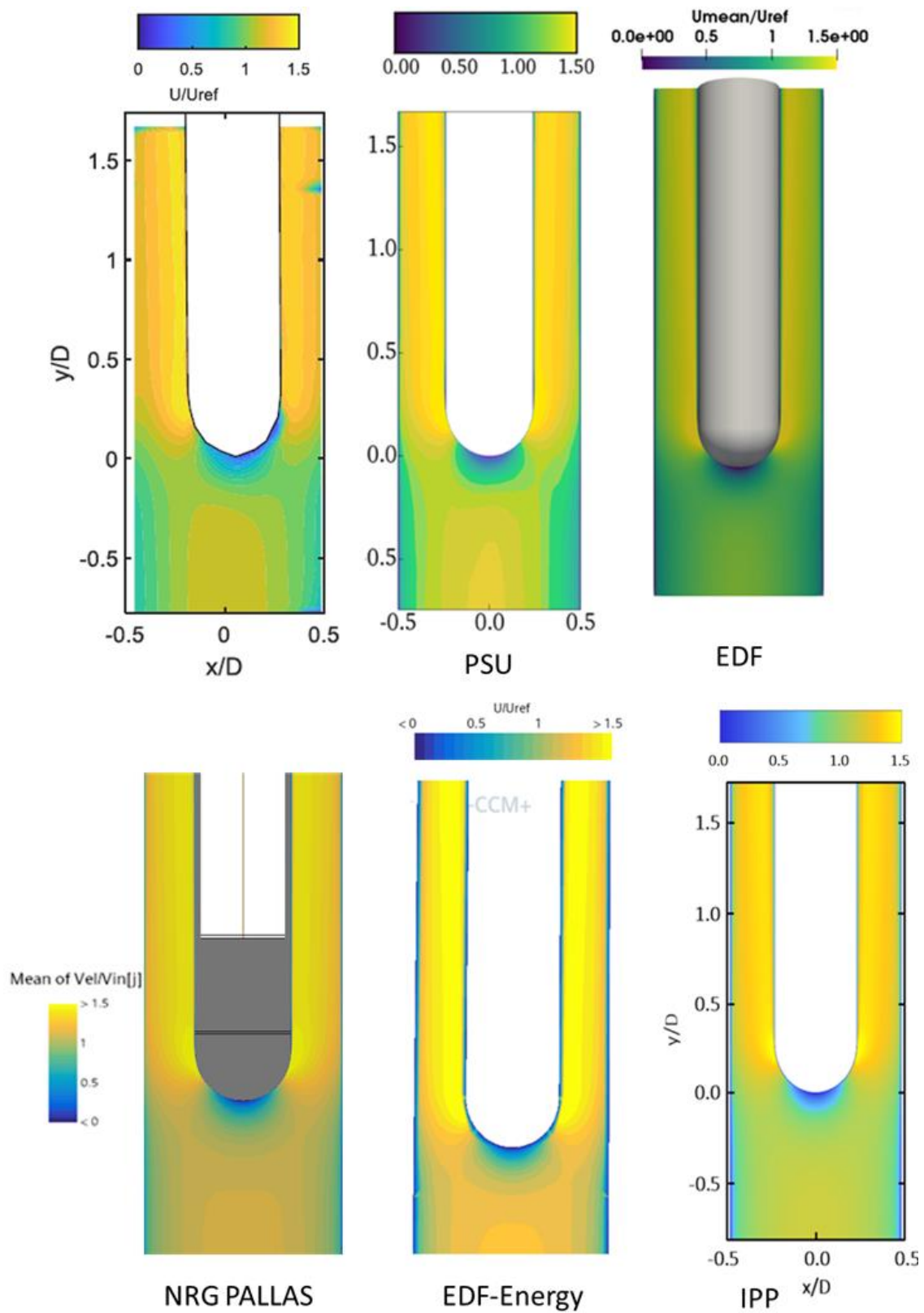


Figure 11. Comparison of Experimental (Cioncolini, et al., 2022) (top left) and Numerical Results for Non-dimensionalized Axial Component of Velocity

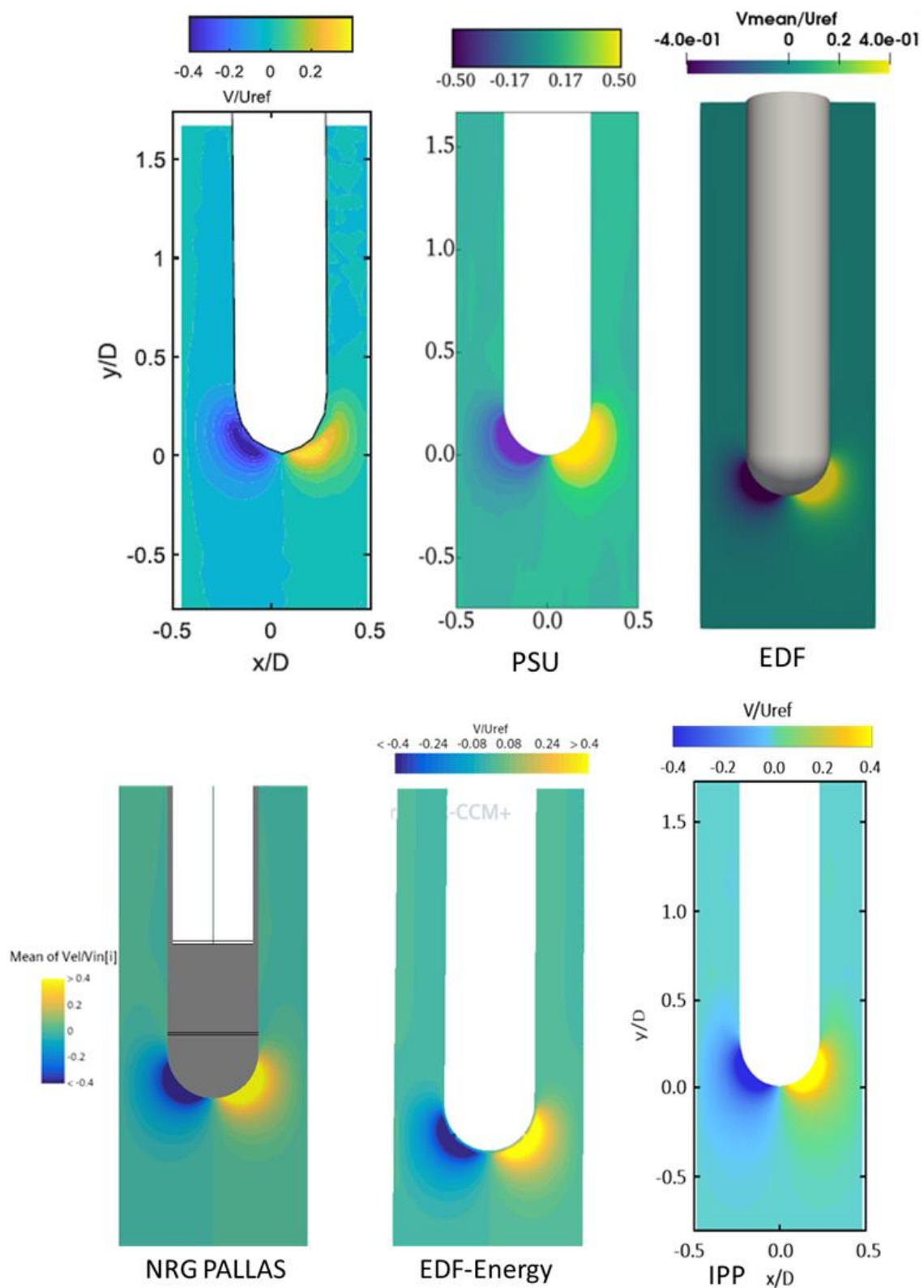


Figure 12. Comparison of Experimental (Cioncolini, et al., 2022) (top left) and Numerical Results for Non-dimensionalized Radial Component of Velocity

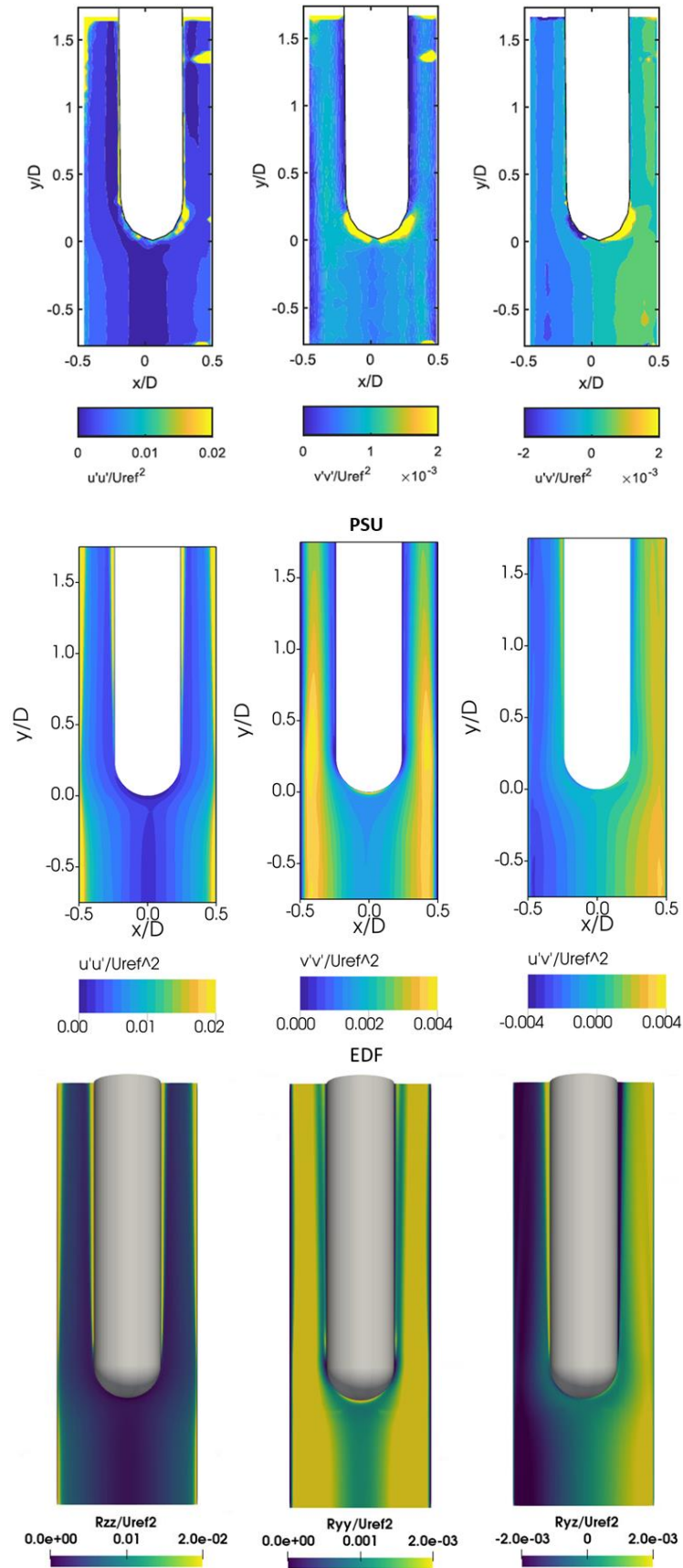


Figure 13. Comparison of Experimental (Cioncolini, et al., 2022) (top) and Numerical Results for Non-dimensionalized Stress Components

Several radial profiles at the PIV measurement locations are provided in Figure 14 to Figure 19. The PIV data inside of the beam are discarded owing to metallic reflections. The time-averaged axial and transverse velocity profiles are drawn. 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. This is reasoned by the potential offset in the vibrating rod tip from the axis of the confining pipe. This is later also confirmed with the rod vibration analysis in section 4.3.2.

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 9) 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 18 and Figure 19, this is reasoned to be from a potential measurement error.

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. The high fidelity approaches of PSU (DNS) and EDF (WR-LES) lead to results that are well in agreement with each other for all locations and especially with the experiment at a location 15.6 mm upstream of the rod (Figure 14) where the effect of the rod offset is not observed. The closest of the medium resolution approaches to these high fidelity solutions at the locations of 15.6 mm (Figure 14) and 3.8 mm (Figure 15) are the results of EDF-Energy (SSG RSM) followed by results of similar accuracy by NRG PALLAS (K-Epsilon) and IPP (K-Omega SST).

Just before the rod tip (Figure 16), all numerical results agree with each other except at the center, where EDF-Energy predicts a higher minimum velocity compared to the other participants. At locations further downstream (Figure 17-Figure 19), all numerical results are also well in agreement with each other, with results in order of (slightly) decreasing agreement with high fidelity results being NRG PALLAS, EDF-Energy and IPP.

Observing the radial component of velocity, the observations are slightly different. Far away from the rod tip (Figure 14), the high fidelity solutions show different trends where EDF predicts a positive albeit small magnitude of radial velocity (flow moving rightward) while PSU predicts negative values in the left half and positive values in the right half. The medium resolution results all show a smooth and symmetric profile with similar low magnitude predictions. Between them, NRG PALLAS and IPP (eddy viscosity models) are closer to each other compared to EDF-Energy (SSG RSM). Moving further downstream in the vicinity of the rod tip (Figure 15-Figure 18), the high fidelity solutions show high agreement with the EDF solution being a bit more wiggly than the PSU solution. The trends of the medium resolution results are similar to the high fidelity solutions but they increasingly underpredict the peak magnitudes moving downstream, with the exception of the location 6.7 mm downstream of the rod tip (Figure 18). This observation can thus be expected to hold in the vicinity of the rod tip, where the flow actively re-orient itself from pipe flow to annular flow. As observed earlier, the solutions of NRG PALLAS and IPP (eddy viscosity models) are closer to each other compared to EDF-Energy (SSG RSM), the latter also being more wiggly in comparison to the former two.

Moving even further away from the rod tip (Figure 19), all numerical results predict low magnitudes of radial velocity. However, the trend predicted by PSU is different compared to the other numerical results. This could be because of the fact that the effect of rod vibrations are not accounted for by PSU. The results of IPP and EDF-Energy are observed to be well in agreement with each other as well as with the high fidelity FSI approach solution of EDF. NRG PALLAS shows a similar trend but underpredicts compared to the EDF results at the location of peaks.

Compared to the experiment, all numerical results for the radial velocity component are observed to overpredict compared to the experimental results in the vicinity of the rod tip (Figure 15-Figure 17) while underpredicting further downstream (only the right half profile of Figure 18). In combination to what was observed for the axial flow component, it could be postulated that the numerical results predict a faster (meaning requiring lesser length) establishment of annular flow from pipe flow compared to the experiment.

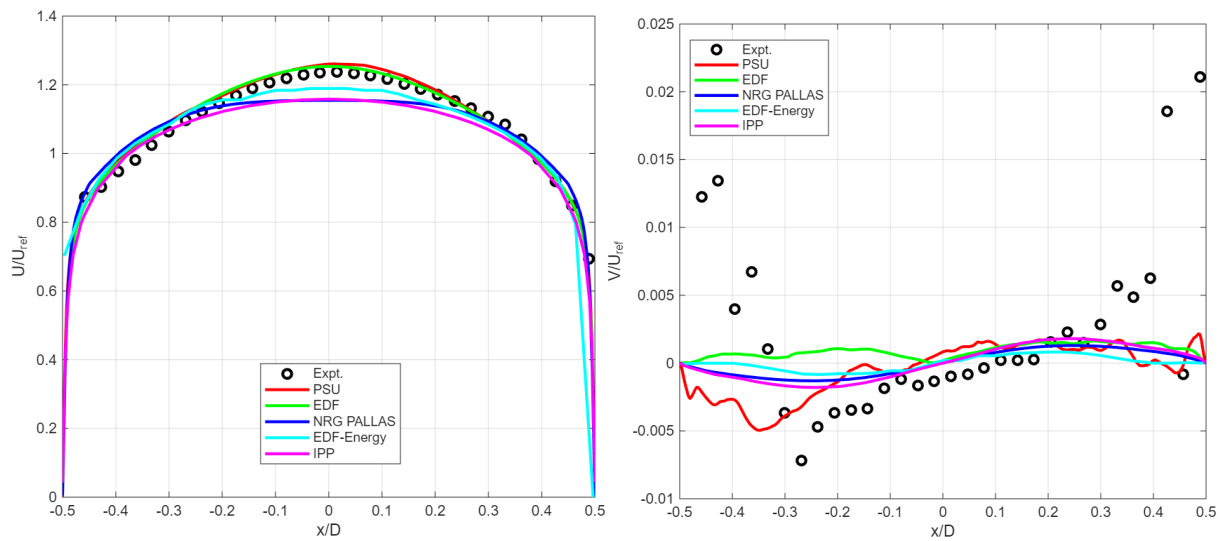


Figure 14. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 15.63786 mm Upstream of the Rod Tip

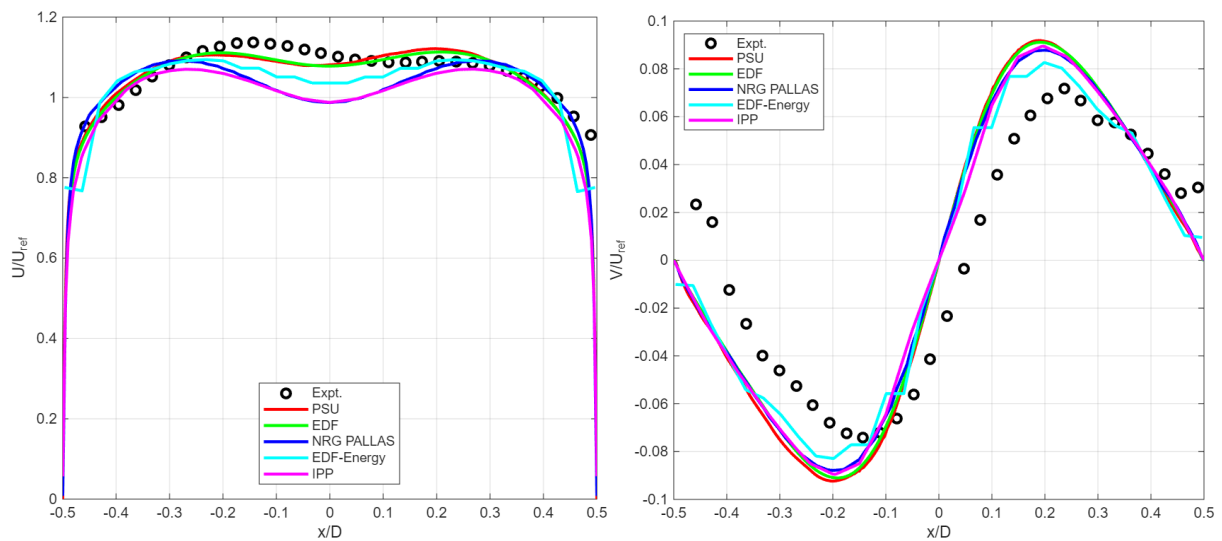


Figure 15. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 3.786006 mm Upstream of the Rod Tip

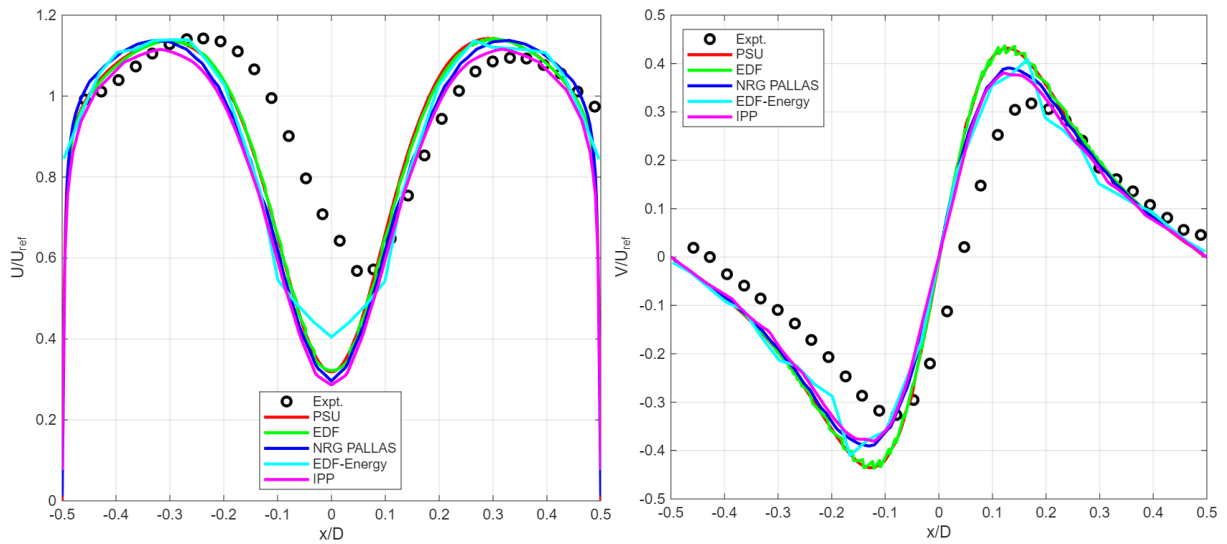


Figure 16. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip

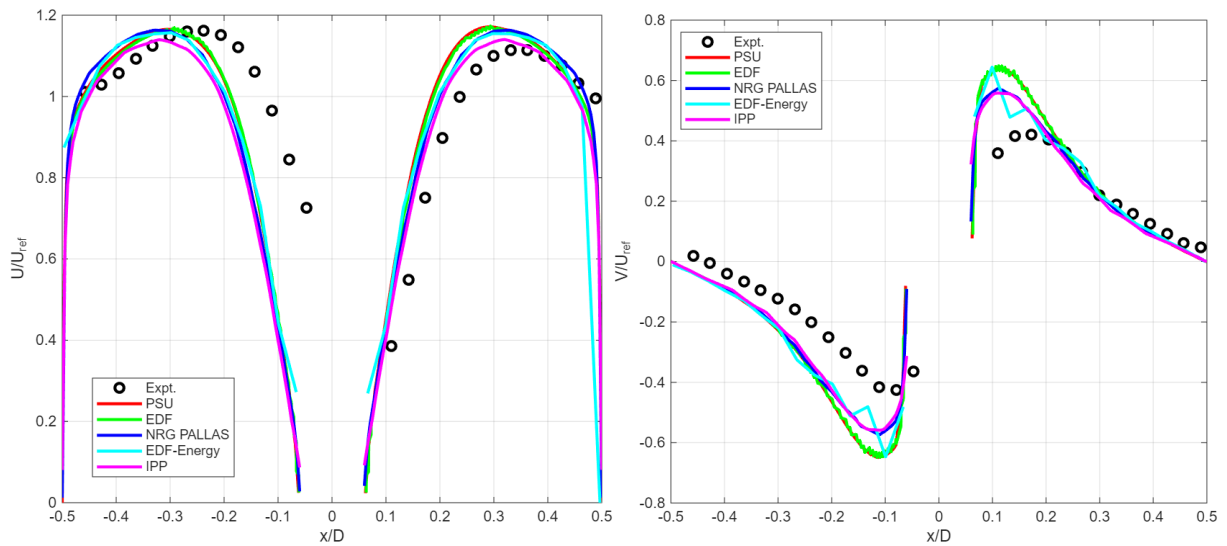


Figure 17. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619 mm Downstream of the Rod Tip

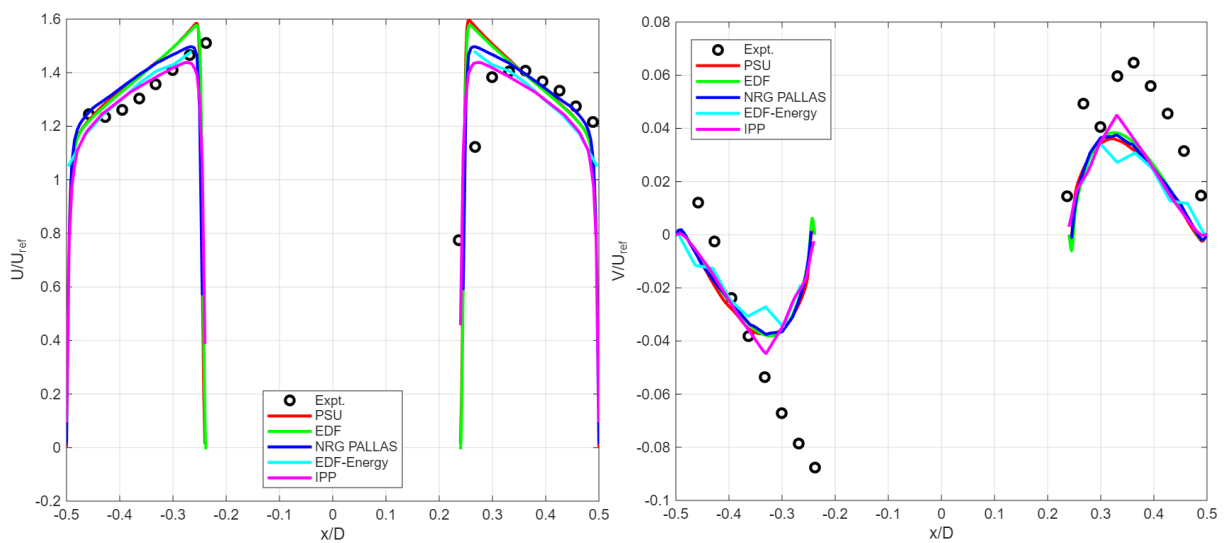


Figure 18. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip

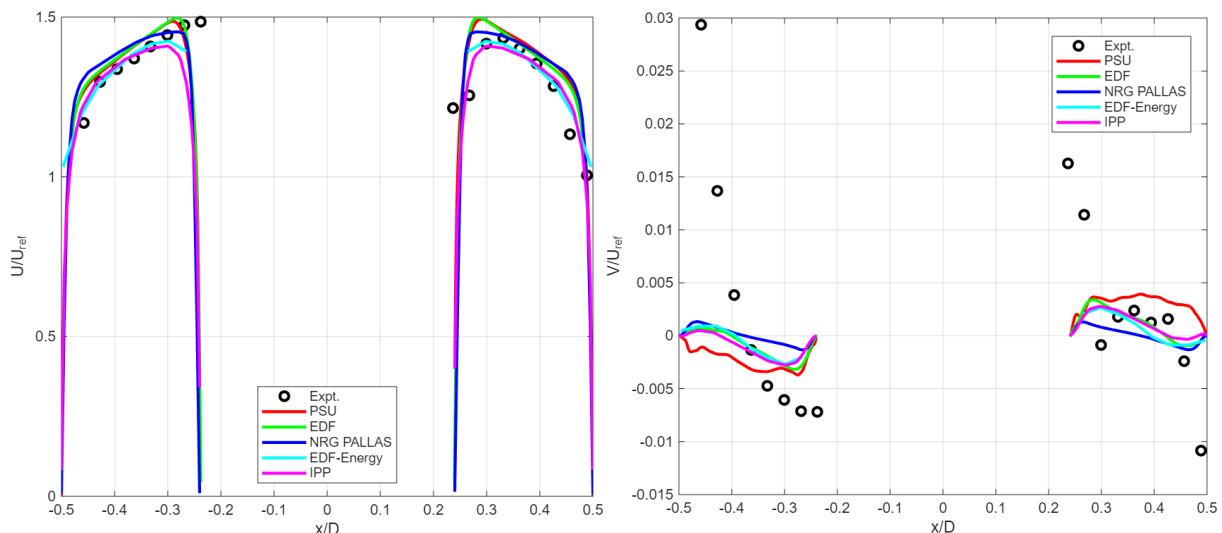


Figure 19. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip

Additional profiles concerning the time averaged second order statistics ($u'v'$ and turbulent kinetic energy, TKE or k) are presented in Figure 20 through Figure 25, for the participants using high fidelity approaches. Considering the experimental PIV results for the $u'v'$ data, they are observed to predict general trends, however owing to the aforementioned reasons, their magnitudes are expected to be severely underpredicted with respect to reality. The underestimation is likely most severe in the regions of highest energy and smallest scales least severe where turbulent scales are larger and more diffused.

This is also confirmed by observing the numerical results which provide similar profile trends but much higher magnitudes. The second order statistics clearly reveal the evolution of the boundary layer at the confining pipe wall as well as the rod surface through the peaks. Comparing the two high fidelity approach solutions, they show excellent agreement between them for the $u'v'$ except for the location 29 mm downstream (Figure 25) and mostly showing good agreement also for the TKE, except the peaks in the boundary layer where the gap between the predictions of DNS and LES appear to increase moving downstream.

The curves of DNS also appear to be less smooth compared to LES, except at the location 0.16 mm downstream (Figure 23) where the LES result near the rod shows wiggles while DNS does not. DNS appears rougher because it resolves all scales (the true chaotic reality). The roughness suggests that the DNS simulation, despite its high cost, was perhaps not run for a long enough time to achieve perfect statistical convergence for the $u'v'$ quantity in the highly unsteady region. As for the LES solution being jagged in Figure 23 could be an indicator of an issue related to the WR-LES filtering process or the resolution of the grid. The wiggle is thus numerical solution attempting to reconcile the physical turbulence with the limitations of the local grid.

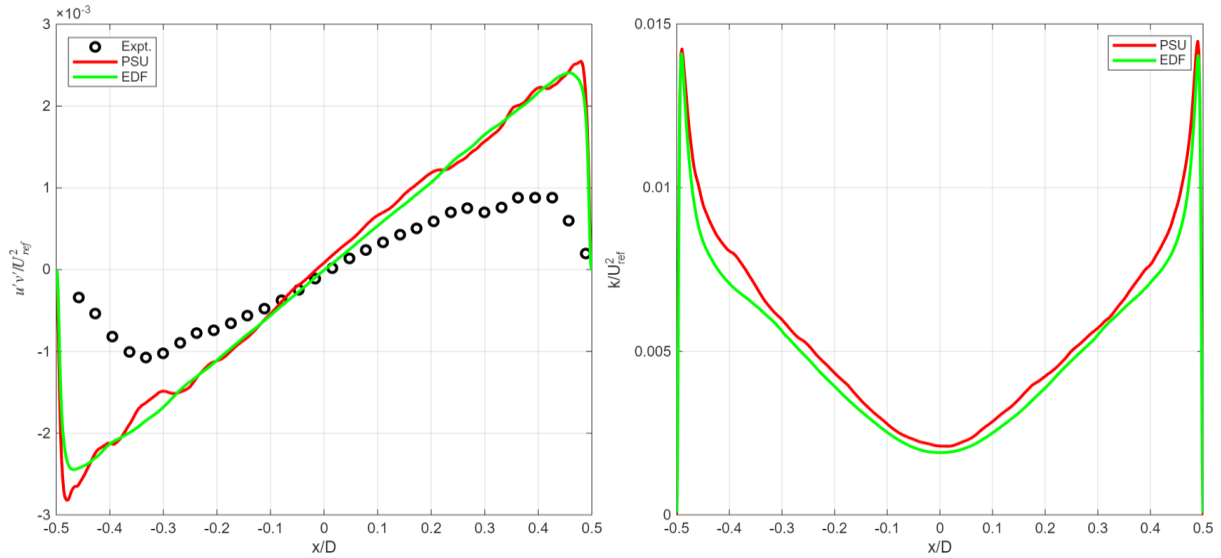


Figure 20. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 15.63786 mm Upstream of the Rod Tip

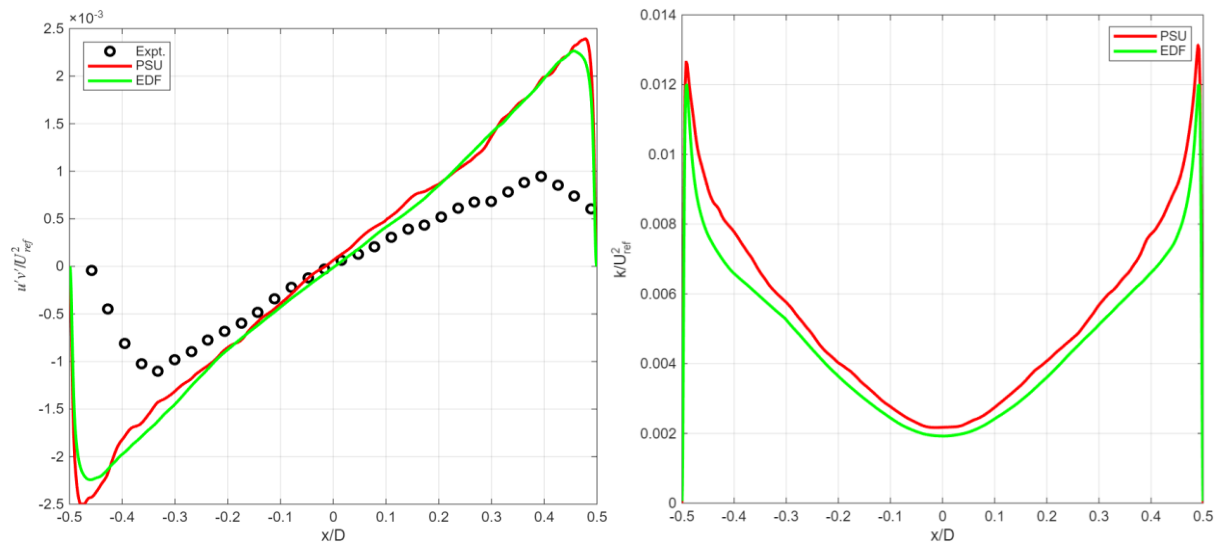


Figure 21. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 3.786006 mm Upstream of the Rod Tip

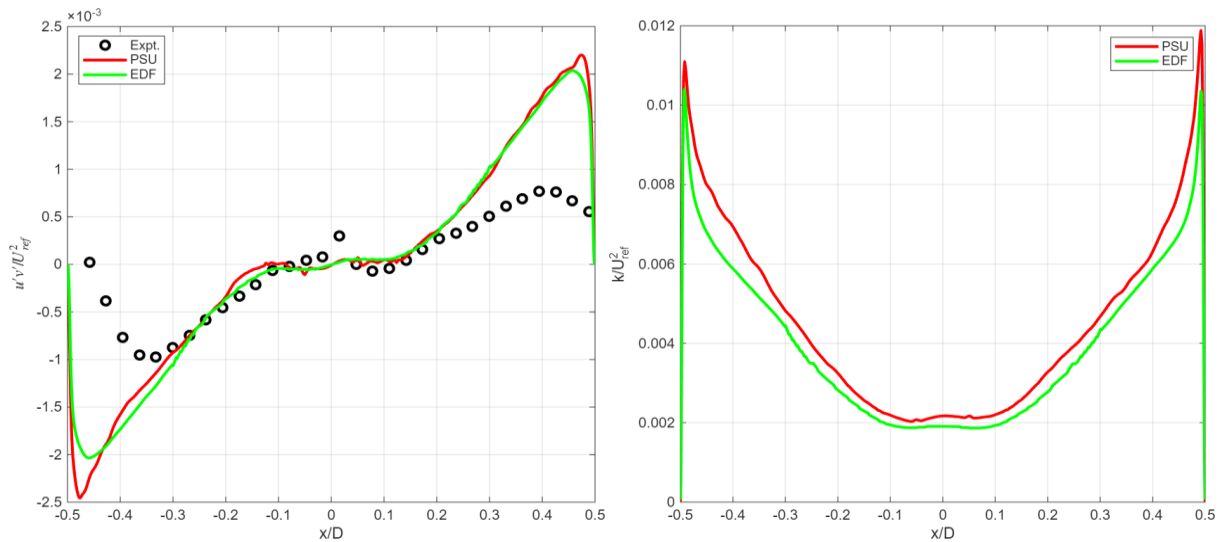


Figure 22. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 0.493836 mm Upstream of the Rod Tip

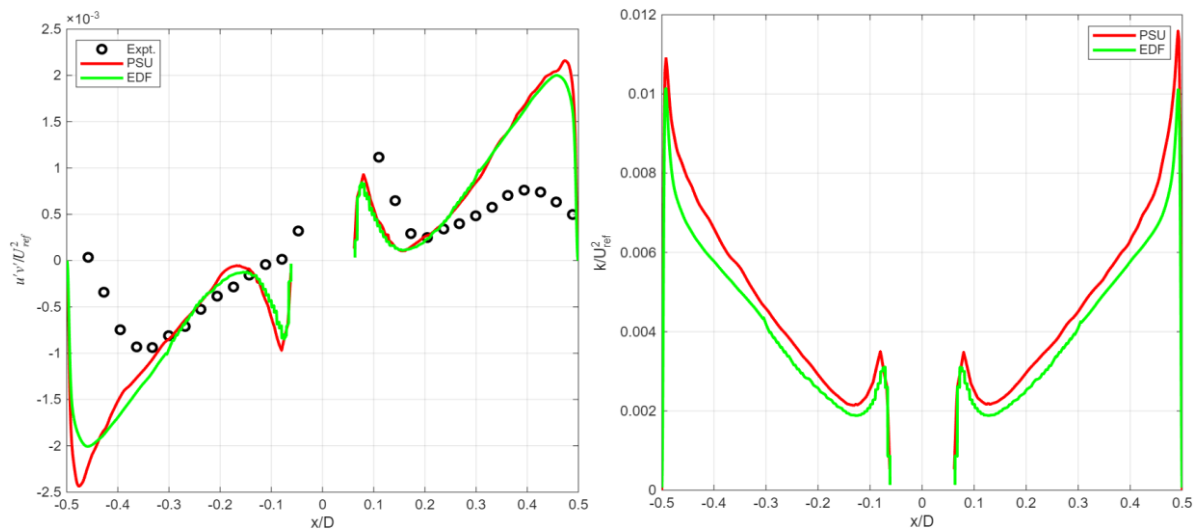


Figure 23. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 0.164619 mm Downstream of the Rod Tip

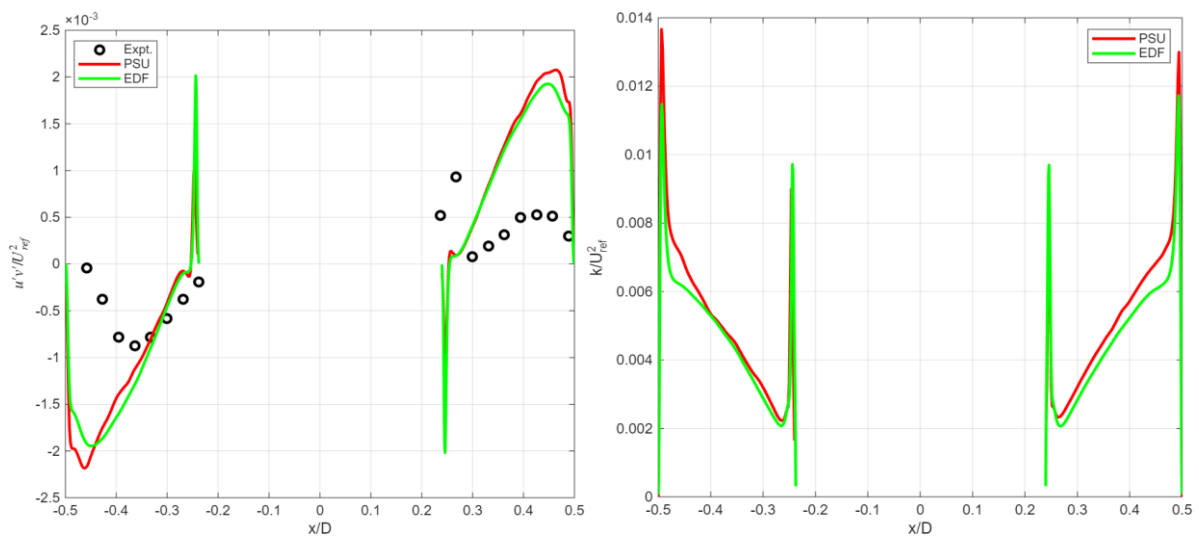


Figure 24. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 6.74898 mm Downstream of the Rod Tip

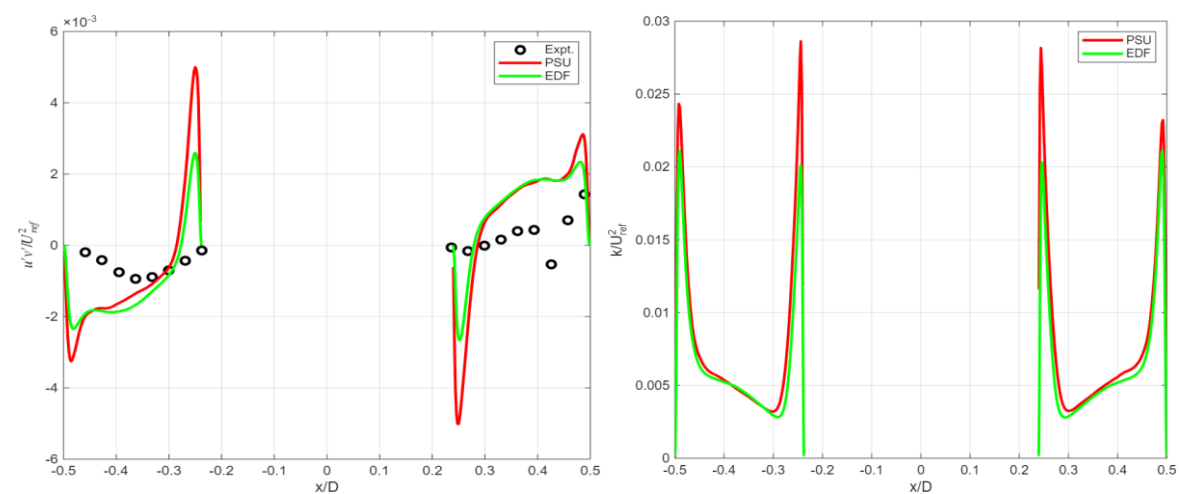


Figure 25. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles 29.135799 mm Downstream of the Rod Tip

Vertical time averaged profiles at the PIV measurement locations moving upwards from the rod tip height (non-dimensionalized by the rod diameter) are considered. These were recorded at different radial locations by the participants and are tabulated along with the locations where experimental data is available in Table 11. High fidelity solution data is available only for the positive locations while medium resolution solution data is also available at the negative locations. EDF provided data at positive radial locations slightly different from the rest of the participants as seen in Table 11.

Table 11. Vertical Profile Data Collection

Institution	Measurement Location, l (mm)	
	- : Left of Rod	+ : Right of Rod
PSU*		+5.583627, +6.242061, +7.558929
EDF		+5.60465, +6.27455, +7.56725
NRG PALLAS	-7.585095, -6.926661, -5.609793	+5.583627, +6.242061, +7.558929
EDF-Energy	-7.585095, -6.926661, -5.609793	+5.583627, +6.242061, +7.558929
IPP	-7.585095, -6.926661, -5.609793	+5.583627, +6.242061, +7.558929
Experiment	-7.585095, -6.926661, -5.609793	+5.583627, +6.242061, +7.558929

* Locations correspond to unscaled geometry. To get the location in the scaled geometry in 'm', use $l_{\text{scaled}} = l/21$

The solutions at the positive radial locations for the velocity components are presented in the top row in Figure 26 through Figure 28 while those for the negative radial locations are presented in the bottom row of the same figures. The experimental data for this comparison can only be used to determine the expected trend of the solution but not the expected magnitude. This is because some data suffers from metallic reflections and simply cannot be used, as in top plots of Figure 26 and Figure 27 for example, while the remaining data is not representative of the expected symmetric annular gap in the simulations owing to the rod tip offset, leading to unequal gaps on either side of the vibrating rod. This is confirmed by comparing the corresponding similar positive and negative locations in the experimental data, for example at +7.558929 mm and -7.585095 mm (Figure 28).

Focusing on the numerical results, it is observed that for the positive locations, the high fidelity solutions agree well with each other for both velocity components, taking into account the slightly different measurement location of EDF. For the medium resolution approaches, the agreement with the high fidelity data varies depending on the measurement location. Near the rod wall at a position of $\sim +5.6$ mm, the results of NRG PALLAS and EDF-Energy agree well with each other upto a height of $z/d=2$, beyond which differences are observed where the EDF-Energy solution predicts a rate of decrease of axial velocity more in line with the high fidelity solution. IPP predicts the right trend but consistently underpredicts the axial velocity. For the radial velocity component, all solutions are well in agreement with each other with differences noticed for the peak velocity. Here, IPP results match more closely to the high fidelity than those of NRG PALLAS and EDF-Energy which underpredict the peak radial velocity.

Moving further away from the rod wall, at $\sim +6.2$ mm, the axial velocity solutions in order of decreasing agreement (increasing underprediction) with the high fidelity results are NRG PALLAS, EDF-Energy and IPP. But for the radial velocity, the observation is the same as that of the $\sim +5.6$ mm location.

Moving even further away from the rod wall at a location $\sim +7.5$ mm which is approximately halfway between the rod wall and confining pipe wall, the axial velocity plot reveals an almost overlapping

profile of NRG PALLAS compared to the high fidelity solution followed by EDF-Energy and IPP in order of increasing underprediction. For the radial velocity, the observation is different than the previous locations. While the general trends all agree well with each other, it is observed that for the peak radial velocity, NRG PALLAS agrees in the prediction to the high fidelity solution, while EDF-Energy underpredicts and IPP overpredicts the same.

For the negative radial locations, the trends of the axial velocity are found to be similar to the positive location counterparts while the radial velocity shows a different profile but this is simply due to the sign convention where the flow now moves leftward in this region.

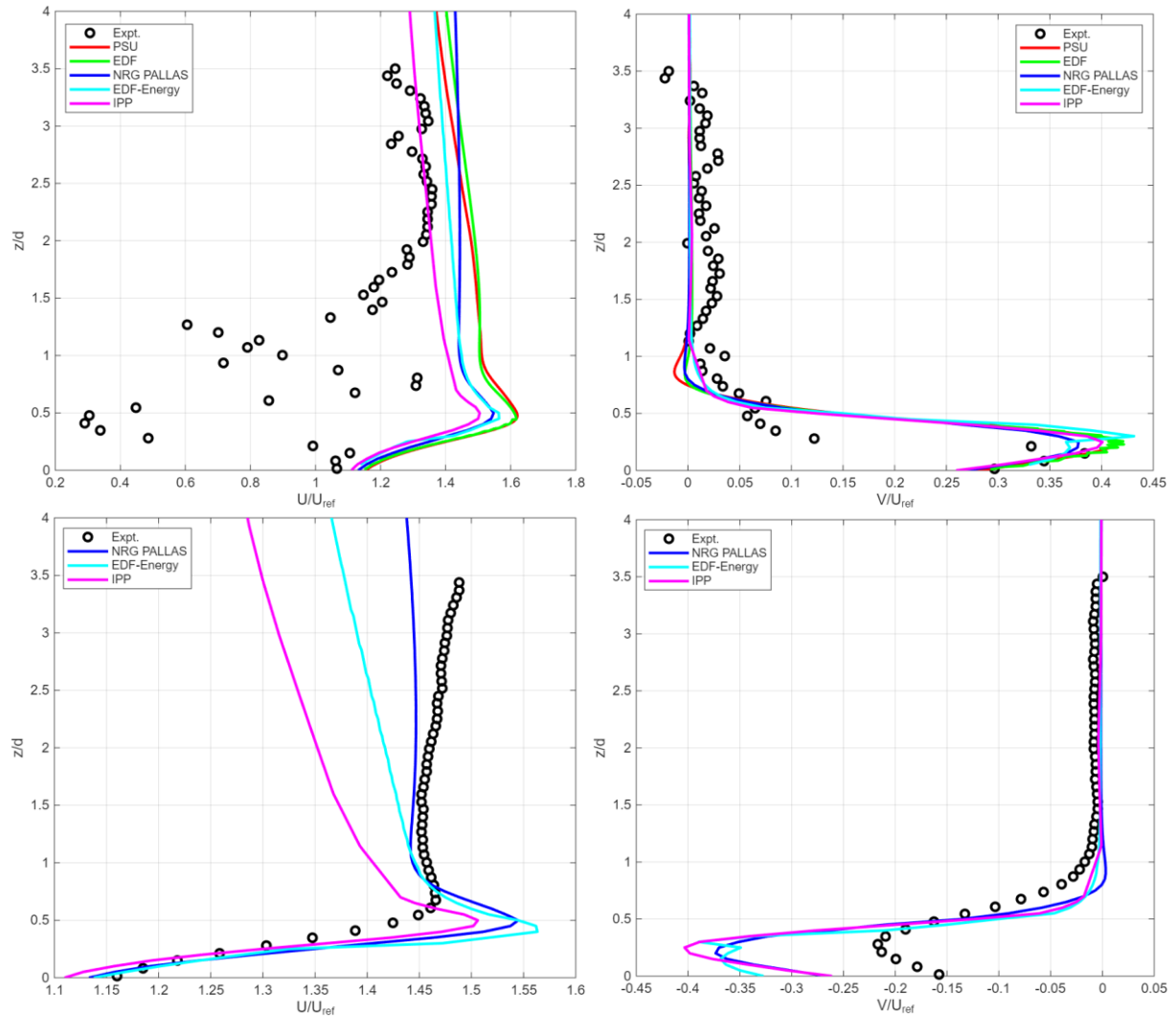


Figure 26. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles. Top: at Locations +5.583627 mm (Experiment, PSU, NRG PALLAS, EDF-Energy, IPP) and +5.60465 mm (EDF); Bottom: at Locations -5.609793 mm (Experiment, NRG PALLAS, EDF-Energy, IPP)

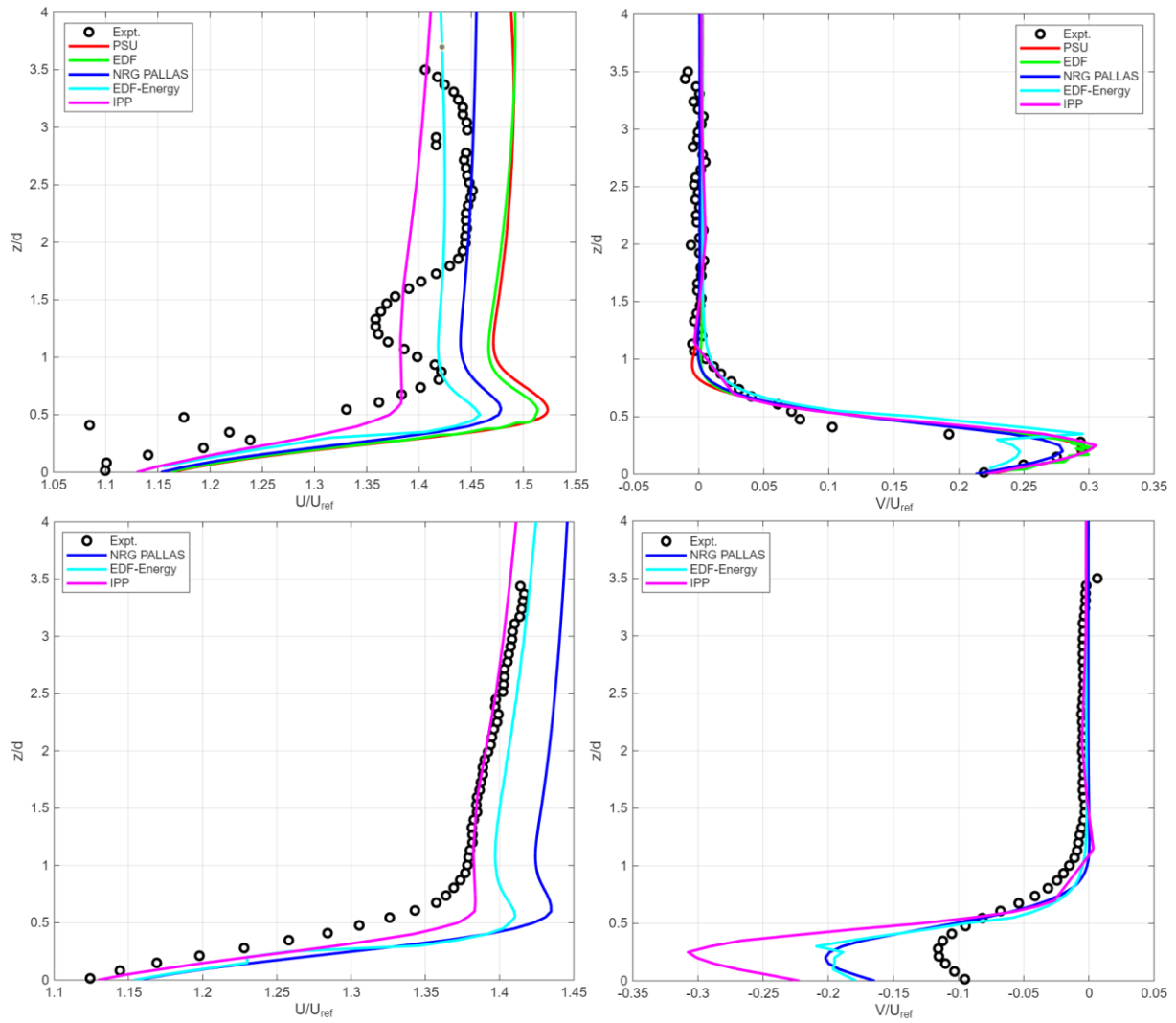


Figure 27. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles. Top: at Locations +6.242061 mm (Experiment, PSU, NRG PALLAS, EDF-Energy, IPP) and +6.27455 mm (EDF); Bottom: at Locations -6.926661 mm (Experiment, NRG PALLAS, EDF-Energy, IPP)

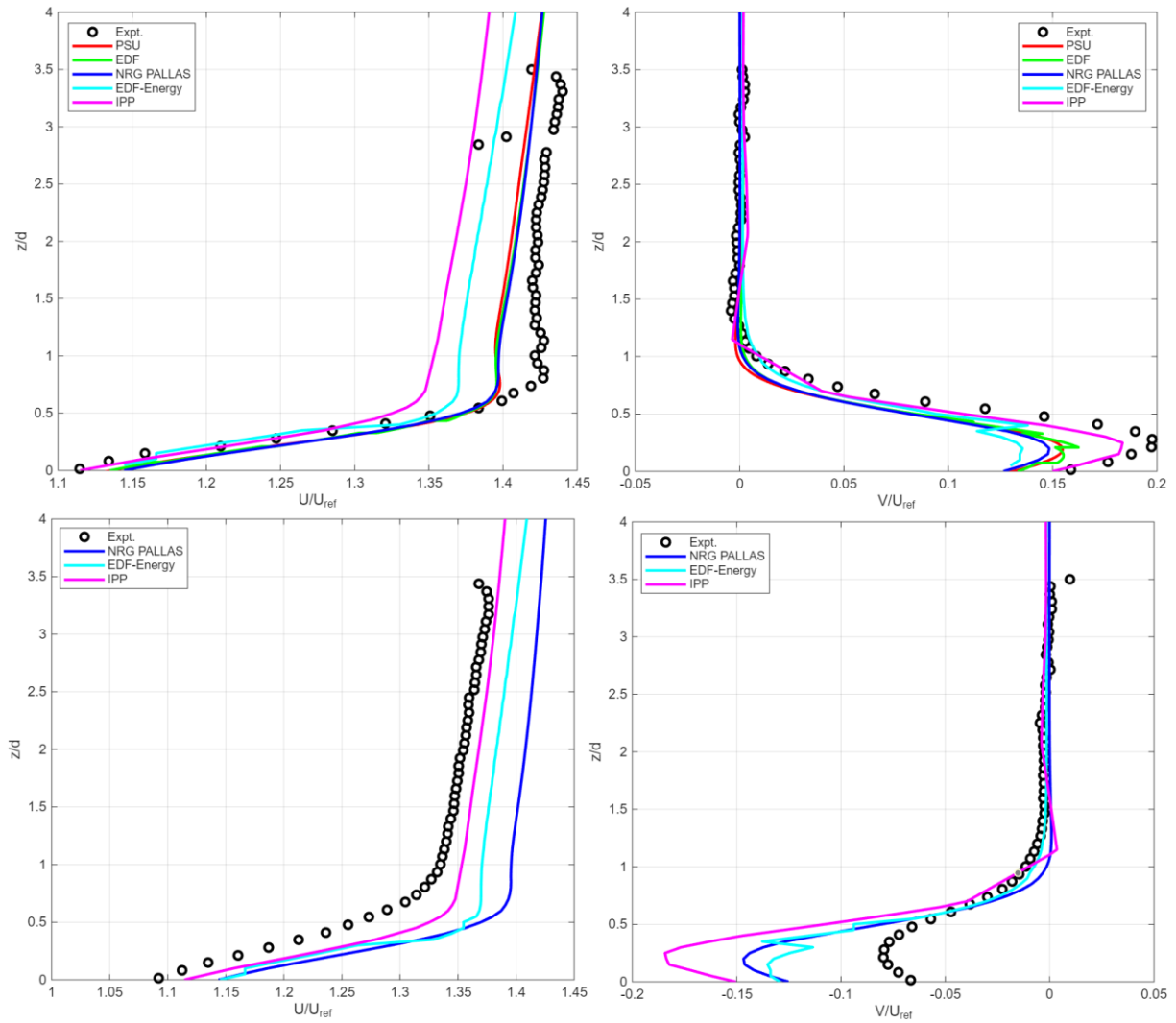


Figure 28. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles. Top: at Locations +7.558929 mm (Experiment, PSU, NRG PALLAS, EDF-Energy, IPP) and +7.56725 mm (EDF); Bottom: at Locations -7.585095 mm (Experiment, NRG PALLAS, EDF-Energy, IPP)

At the positive locations where high fidelity data is available, second order statistics ($u'v'$ and TKE) are also considered along with experimental data. These are plotted in Figure 29 through Figure 31. The experimental results cannot be used for validation of numerical results owing to the rod tip offset in the experiment as well as the aforementioned fact that PIV is not very reliable concerning second order statistics. The two high fidelity results agree well in terms of the observed trends and also for the magnitude of the evolution of $u'v'$ which gets better in agreement moving away from the rod wall. However, the TKE plots show solutions of similar magnitudes but with the EDF solution consistently underpredicting compared to the PSU solution. While part of the discrepancy can be explained by the slight mismatch in measurement location, it is not sufficient since gaps were also observed in the TKE comparison of the horizontal profiles at the same measurement locations.

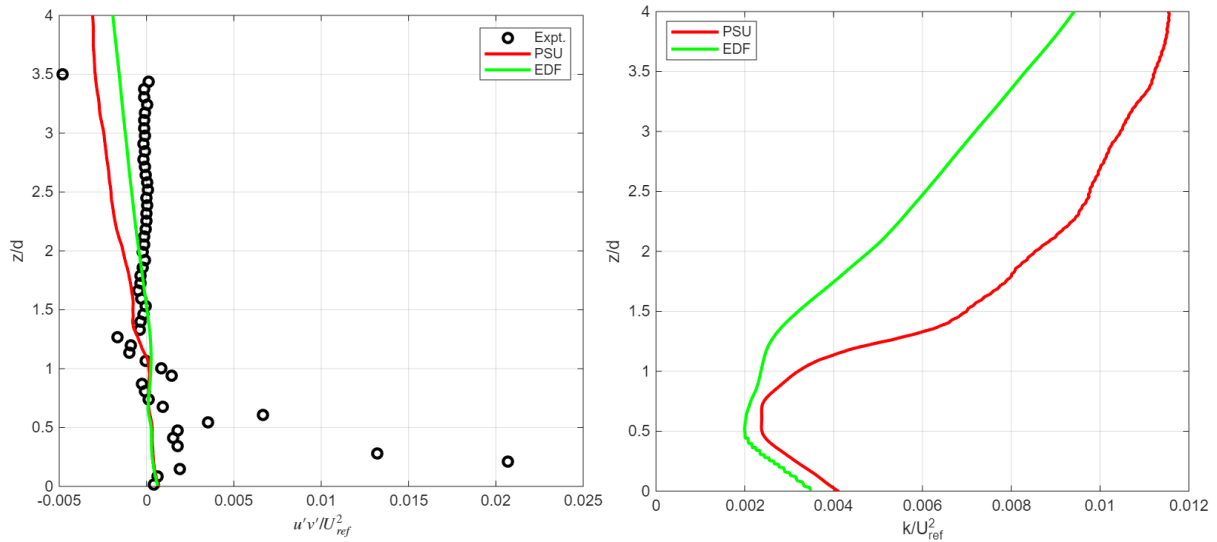


Figure 29. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles at Location +5.583627 mm (Experiment, PSU) and +5.60465 mm (EDF)

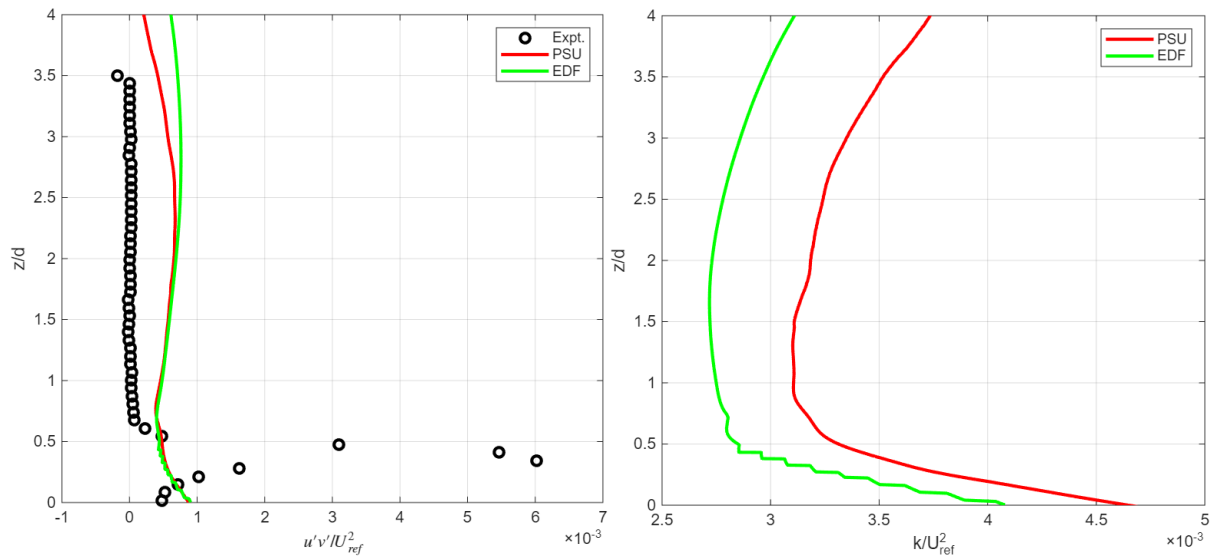


Figure 30. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles at Locations +6.242061 mm (Experiment, PSU) and +6.27455 mm (EDF)

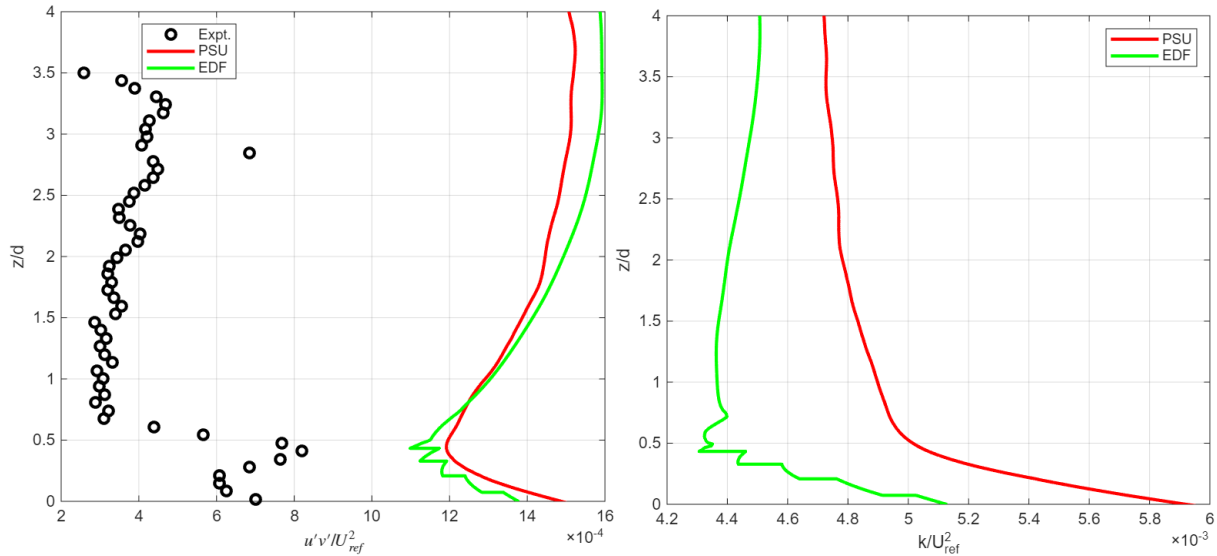


Figure 31. Time-Averaged Non-dimensional Reynolds Stress Component ($u'v'$) (left) and Turbulent Kinetic Energy (right) Profiles at Locations +7.558929 mm (Experiment, PSU) and +7.56725 mm (EDF)

4.3.2 Vibration Data Comparison

The motion of the rod tip has been monitored during the simulations for all participants except PSU who performed pure CFD simulations. Each participants performed simulations for unequal durations of physical time depending on the available resources. The time histories obtained from the simulations are compared against the experiment in Figure 32. 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. This is unique for each submission depending on the simulation strategy.

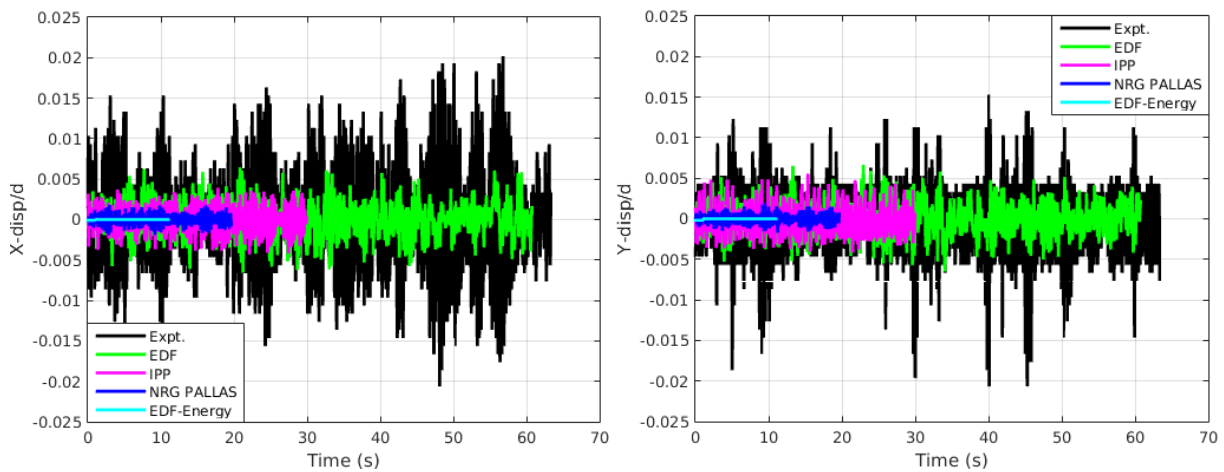


Figure 32. Time History of the Non-dimensional Rod Tip Front (left) and Side (right) Displacements

The obtained time histories are also presented as 3D plots in Figure 33 without removing the mean for the numerical results. Qualitatively, the numerical results show the same chaotic pattern as the experiment with the exception of EDF-Energy, whose displacements are very small compared to the already small numerical offset.

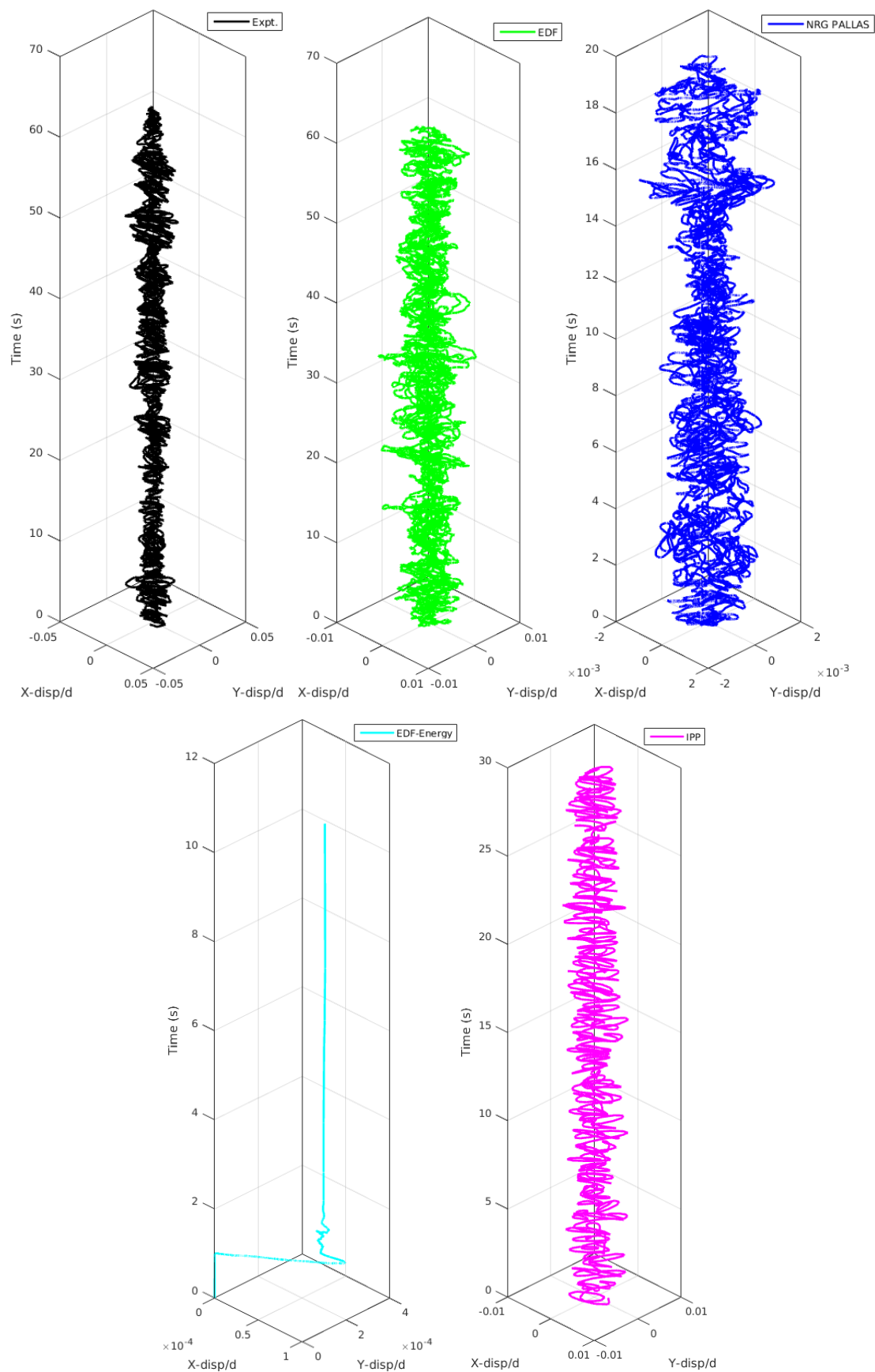


Figure 33. 3D Time History of Rod Tip Displacements



During the procedure, it was found that the rod tip was oscillating at an offset in the experiment and the numerical results. This is tabulated in Table 12. The numerical offsets are much smaller in comparison to the experiment and are considered to be a numerical artifact.

Table 12. Observed Rod Tip Offset

Institution	Front-Offset (mm)	Side-Offset (mm)	Net Offset (mm)
EDF	0.00088232	0.001089	0.0014
NRG PALLAS	0.0018765	-0.0015263	0.0024
EDF-Energy	0.0008038	0.0018157	0.0020
IPP	0.0021148	-0.00036823	0.0021
Experiment	1.47675	0.02654	1.4770

Key results obtained from the time history are presented in Table 13. The RMS polar displacement is numerically predicted to be smaller than the experiment. The predictions are closest by EDF which is expected given the use of the high resolution approach, with an underprediction of a factor of 2.87. The next is IPP with displacement in the same order of magnitude as EDF which is remarkable considering the use of medium resolution approach and a coarse time step size. The underprediction is by a factor of 3.34. NRG PALLAS who also use a medium resolution approach but using a composite rod model underpredict the displacement by an order of magnitude or a factor of 9.84. EDF-Energy underpredicts vibrations by 4 orders of magnitude or a factor of 9492.73.

The RMS front (x-) and side (y-) displacements are found to be very dissimilar for the experiment and the numerical predictions except for EDF. This could stem from the rod tip offset which was significant for the experiment. For the numerical results, the asymmetry in the simulated test section of NRG PALLAS and IPP may have had a role. However, this is not applicable for EDF-Energy who also predict different displacements in both directions, although these displacements can be considered to be in the range of numerical noise.

Table 13. Obtained Displacement Results:

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

The Power Spectral Density (PSD) of the rod tip front (x-) displacement is compared to the experimental data in Figure 34. The number of time steps simulated by the participants affects the frequency range which can be represented. Since IPP simulated only 1500 time steps, only frequency data upto 25 Hz is available. In general, the numerical trends themselves agree well with the experiment and with each other. However, the numerical results are mostly found to be lower in energy content compared to the experiment to varying degrees depending on the participant.

For the first peak, EDF and IPP show similar underpredictions followed by NRG PALLAS. For the second peak, EDF slightly overpredicts the energy level while NRG PALLAS slightly underpredicts it followed by IPP who have larger underpredictions. For the remaining frequency range, NRG PALLAS makes predictions in the same ball park as EDF but with the discrepancy increasing with increasing frequency.

EDF Energy is observed to consistently show severe underprediction of the energy content throughout the frequency range which is expected considering the RMS values.

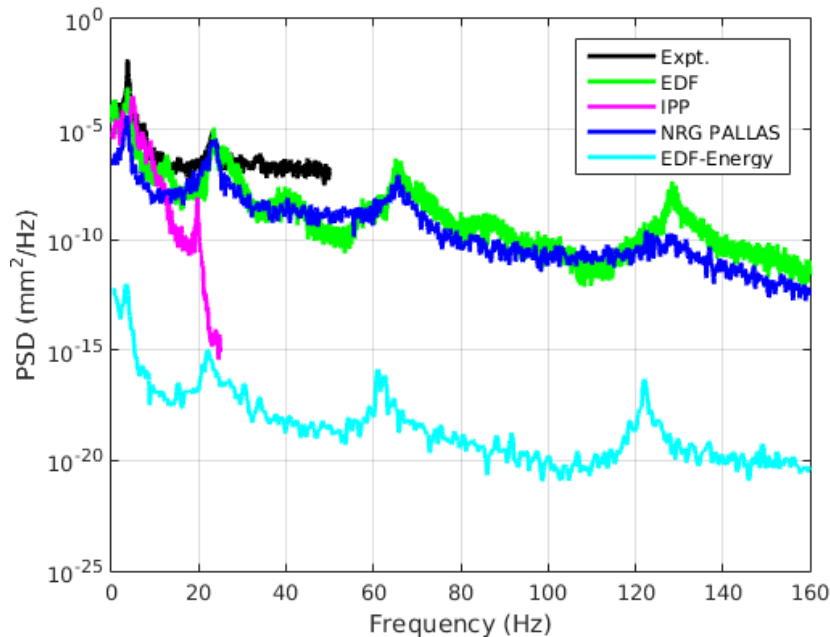


Figure 34. PSD of the Tip Displacement in the Front (x-) Direction

The frequencies of the peaks obtained numerically, experimentally and analytically via classical Euler-Bernoulli beam theory (refer section 6.2.1) are provided in Table 14. For the first two peaks available in the experimental data, the numerical predictions show agreements in the decreasing order of EDF, NRG PALLAS, EDF-Energy and IPP. For the higher frequency peaks, NRG PALLAS and EDF show good agreement with each other followed with the predictions being lower by EDF Energy. In all cases, the peak frequencies significantly differ from the analytical solution at higher modes of natural frequencies.

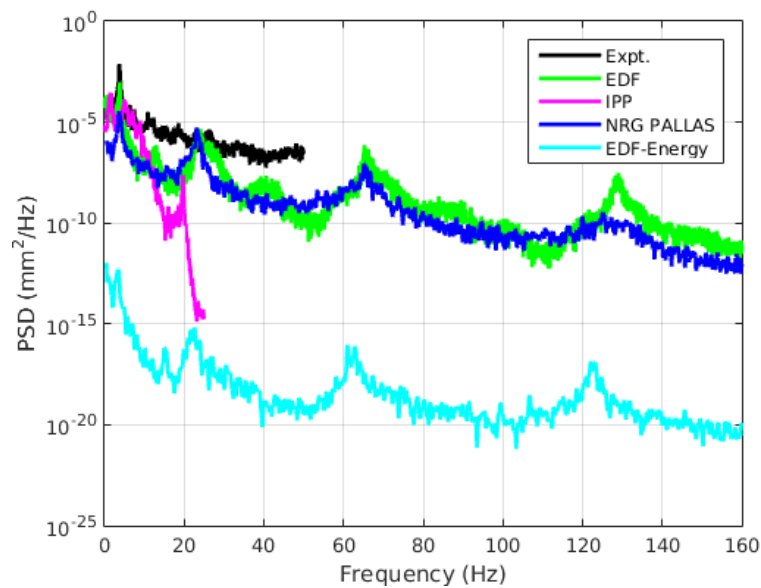
Table 14. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically

Approach	#1 (Hz)	#2 (Hz)	#3 (Hz)	#4 (Hz)
EDF	3.70	23.42	65.13	128.4
NRG PALLAS	3.64	23.81	65.68	128.2
EDF-Energy	3.50	22.12	61.00	122.0
IPP	5.00	19.67	N/A	N/A
Experiment	3.76	23.47	N/A	N/A
Analytical	3.71	25.6	71.8	141

A similar look is also taken for the rod tip side (y-) displacement as shown in Figure 35. 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 34). The first mode frequency for both directions is summarized in Table 15. The frequencies obtained from the front and side displacement are similar in the experiment and EDF-Energy results, show slight dissimilarity for EDF and NRG PALLAS and show significant dissimilarity for IPP results. For the average frequency, the numerical predictions show agreements in the decreasing order of EDF, NRG PALLAS, EDF-Energy and IPP to the experimental and analytical value.

Table 15. Frequencies of the Numerical and Experimental Side Displacement PSD Peak #1 Compared to the Beam Natural Frequencies Obtained Analytically

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

**Figure 35. PSD of the Tip Displacement in the Side (y-) Direction**

4.4 Benchmark summary and conclusions

In this chapter, the benchmark case, chosen to be the free-clamped flow at $Re=21.2k$ for the rod with a hemispherical tip, was discussed. Five participants provided their results for comparison: PSU, EDF, NRG PALLAS, EDF-Energy and IPP. Each participant used different modelling approaches: pure CFD vs two-way FSI, the simulated geometry respecting the experimental test section or modifying it, modifications to the vibrating rod, differences in material properties, meshing strategy and lastly but most importantly the turbulence modelling of the fluid. Such a wide range of differences make the benchmark quite rich, but it also brings forth the challenge of identifying a single aspect of the simulation strategy that can be said to make or break the accuracy of the calculations.

However, based on the compared results the following key comments can be made:

- Flow Field Accuracy:
 - To obtain high accuracy results with a focus only on flow field data, comparing solely the high fidelity approaches, it appears to be more fruitful to go for a pure CFD approach with DNS than a two-way FSI with LES.
 - From the medium resolution approaches of SSG-RSM, K-Epsilon (cubic), and K-Omega SST, there is no clear “best” model that provided solutions close to the high fidelity solution at all locations.

- What was noticed is that for horizontal measurement profiles, the classical eddy viscosity models (K-Epsilon (cubic) and K-Omega SST) performed similarly while differences were observed in the vertical measurement profiles. SSG-RSM performed either slightly better than the eddy viscosity models or somewhere in between in terms of accuracy.
- Vibration Accuracy:
 - As one expects, the better your flow prediction, in particular, pressure and stresses is over the rod, the better would be the accuracy of the vibration amplitudes. The LES results conformed to this expectation with being the closest to the experimental data amongst the participants. However, it is still to be remarked that this “best solution” still underpredicted the RMS vibration amplitude by a factor of 2.8 with subsequent participants’ medium resolution solutions further degrading in accuracy.
 - However, the above expectation is not met within the medium resolution results. SSG-RSM approach ended up with vibration amplitudes several orders of magnitude lower than the experimental results. This is suspected to be connected to the mesh being wall modelled rather than wall resolved. The eddy viscosity models approach were much better but still either underpredicted the amplitude by an order of magnitude for the K-Epsilon (cubic) approach or by a factor of 3.3 for the K-Omega model, which is quite good considering how LES performed.
 - Concerning the vibration frequency, all approaches provide a good estimate for the same compared to the experiment except the one using the K-Omega SST approach. It is quite peculiar that for this approach, the prediction of vibrating frequency in the two principle directions were so starkly different. Of course, it is not just the turbulence model which could have led to this result. The general accuracy ranking for the vibration frequency is LES, K-Epsilon (cubic), SSG-RSM and K-Omega SST.
- Other considerations:
 - The effect of replicating the test section exactly as well as modelling the vibrating rod with a full CSM structural model appears to yield low increase in accuracy, if any, compared to using a single inlet-outlet configuration and beam model respectively. The effect of the chosen turbulence model overshadows any possible impact of these model choices. Making this simplification eases mesh constraints, and can potentially speed up the calculations as remarked by the computational cost comparison.
 - While there is suspected to be an effect of the meshing strategy taken by the participants, the current comparison cannot shed light on the superiority of any approach. However, what can be observed from the medium resolution results is that the participants using a wall resolved approach in conjunction with eddy viscosity models were able to predict vibrations to a large extent as opposed to the participant who used a better SSG RSM model but in conjunction with a wall modelled mesh. This indicates that the wall treatment in combination with corresponding turbulence model plays an essential role for the prediction accuracy.
 - The impact of the different material properties chosen by the participants also appears to be overshadowed. The vibration tests of the rod in air/vacuum and water are a good approach to validate chosen material properties.

5. Simulations performed by PSU

5.1 Introduction

Pressurized Water Reactors (PWRs), constituting roughly three-quarters of the world's 400 nuclear reactors, play a vital role in global electricity production. Essential to their operation are densely arranged fuel rods, which are susceptible to Flow-Induced Vibrations (FIV), a complex Fluid-Structure Interaction (FSI) phenomenon. Turbulent flows, characterized by fluctuating pressures and velocities, can induce resonant oscillations in fuel rods, leading to structural damage such as Grid-to-Rod Fretting (GTRF). GTRF remains the leading cause of fuel defects in PWRs, resulting in economic losses and operational disruptions. Consequently, accurately predicting FIV is critical for the safe and economical operation of nuclear reactors.

Historically, nuclear fuel failure mechanisms have evolved significantly. Initial reactor designs used stainless steel cladding, soon replaced by zirconium alloys for improved neutron economy. During the 1970s and 1980s, manufacturing defects and material issues such as hydriding and Pellet–Cladding Interaction dominated fuel failures. Although advancements reduced many of these issues, GTRF has remained the leading cause of fuel defects in PWRs since the 1990s. Preventing GTRF depends on accurate prediction of FIV, a task made difficult by the reactor's complex geometry and turbulent flow.

This study contributes to addressing these challenges through Direct Numerical Simulation (DNS) of turbulent flow around a cantilevered rod within a confining pipe, replicating an experiment from the University of Manchester (UoM) as part of the GO-VIKING consortium within WP2. The project aims to enhance understanding and modeling of FIV in nuclear reactors. By generating high-fidelity fluid-only data, this DNS work provides an avenue for benchmarking and validating lower costing turbulence models used and developed by GO-VIKING partners. Although the current simulations do not include structural dynamics, the minimal experimentally observed rod displacement justifies a rigid-body assumption, allowing detailed fluid analysis to inform future coupled FSI studies. Ultimately, this work supports efforts toward safer, more reliable, and economically viable reactor fuel designs.

5.2 Methodology

5.2.1 Configurations

Two free-clamped configurations are considered for the DNS, fluid only, simulations:

- Rounded-end with an annulus Reynolds number of 11,105 and 21,200.
- Blunt-end with an annulus Reynolds number of 12,220.

The geometries of each case are set based on the experiments conducted by the University of Manchester (UoM) and the annular Reynolds numbers originally aimed to match the Reynolds numbers of experimental cases ($Re=21.2k$ for the curved end and $Re=23.33k$ for the blunt end) for direct comparison and simulation benchmarking. However, an error was made in the setup and the simulated cases were at a factor of 0.5238 of the targeted Re . The following



subsections will further detail the setup and the reason for the mismatch based on the experiments conducted at UoM as well as the DNS simulations.

5.2.1.1 University of Manchester Experiments

UoM designed and implemented a flow loop experiment featuring a cantilevered rod within a confining pipe to simulate FIV, as shown in Figure 36 (Cioncolini, et al., 2023). The rod is fixed at one end and free at the other, with the free end configurable in various shapes (e.g., semi-spherical, conical, or blunt). The primary objective of this experiment is to investigate FIV on the cantilevered rod, which serves as an approximation for a fuel rod, from the effects of axial water flow.

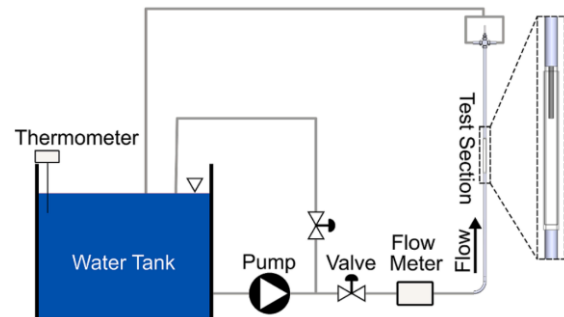


Figure 36. UoM Experimental Setup.

5.2.1.2 Simulation Setups

The simulation domain was deliberately reduced compared to the experimental setup to minimize computational costs without compromising the accuracy in the region of interest. In the experiment, the rod length is 1.045 mm measured from the fixed end, and the confining pipe length is 2.044 mm. It is noted that the rod length used in the simulation is approximately one-fifth of the experimental rod length. This shorter length was specifically selected because the flow at this position is predominantly developed (free from geometric contraction effects), and the magnitude of rod displacement due to FIV is greatest near the free end, diminishing significantly toward the fixed end. Additionally, the pipe domain upstream of the rod is less than 10% of the experimental length. This reduction is enabled by employing inlet flow recycling over a length equivalent to four pipe diameters, ensuring fully developed turbulent pipe flow conditions. By significantly reducing the domain size in this manner, the simulation achieves the resolution required for DNS while minimizing computational resources. These general dimensions are detailed in Table 16 where an additional line is included for the rounded-end case.

Table 16. Geometric details of simulations.

Parameter	Symbol	Value (mm)	Value ($1/D_h$)
Rod Diameter	D_i	10	0.47619
Pipe Diameter	D_o	21	1.0
Rounded Tip	R	5	0.238095
Rod Length (without tip)	L_r	220	10.47619
Confining Pipe Length	L_p	346	16.47619
Hydraulic Diameter	D_h	D_o	1.0

The simulations are dimensionless, with the pipe inlet bulk Reynolds number set by the viscosity. To nondimensionalize, the domain coordinates are divided by the hydraulic diameter, D_h . The hydraulic diameter is now unity. The density and bulk inlet velocity ($U_b = U_{ref}$) are set to one. From here, the viscosity is determined by the area ratio between the pipe and annulus and the target annulus Reynolds number. Originally, the following expression was used to set the viscosity:

$$\mu = \frac{1}{Re} \left(\frac{A_{pipe}}{A_{annulus}} \right)$$

However, it was only after simulations for the curved and blunt end were complete that the above expression was found to be incorrect. The correct expression below was thus adopted, and only the $Re=21.2k$ case for the curved end was rerun. Note that in this chapter of the report, the comparisons to the experiment are made between the results of the first approach and the original targeted experimental Re . For the benchmark comparison in chapter 4, the correct $Re=21.2$ case is used.

$$\mu = \frac{1}{Re} \left(1 + \frac{D_{rod}}{D_{pipe}} \right)^{-1}$$

The simulations are initialized with fully developed turbulent pipe flow mapped to the pipe inlet. A setup in the NekRS GitHub for turbulent pipe flow can be found [here](#). The Reynolds number of the inlet pipe flow is set using the viscosity with the annular Reynolds numbers mentioned above. To flush out any flow development effects, the simulation was run for five to ten flow throughs before averaging, collecting turbulence statistics, and collecting time history data. The boundary conditions are as follows: no-slip walls to accurately represent wall flow behavior, a Dirichlet inlet with recycling to achieve and maintain fully developed turbulent flow, and stabilized outflow. The inlet recycling length is equivalent to four pipe diameters.

5.2.2 Numerical Simulation Approach

This study leverages Direct Numerical Simulation (DNS) to accurately capture turbulent fluid flow by resolving all relevant time and length scales without turbulence modeling. DNS, representing the highest-fidelity method in Computational Fluid Dynamics (CFD), serves as a benchmark for validating and refining more cost-effective turbulence modeling approaches. Using NekRS, an open-source Spectral Element Method (SEM) code optimized for high-performance computing (HPC), this study directly solves the Navier-Stokes equations from fundamental input parameters, requiring significant computational resources due to the necessity of resolving the smallest turbulent scales. This section details the governing equations, key turbulence analyses, code used, and computational requirements.

5.2.2.1 Equations & Turbulence Analyses

The governing equations for the simulations are the incompressible Navier-Stokes equations.

$$\nabla \cdot \mathbf{U} = 0$$

The continuity equation where $\mathbf{U}$ is the fluid velocity vector.

$$\frac{DU}{Dt} = -\frac{1}{\rho}\nabla p + \nu\nabla^2 U$$

The momentum equations where ρ is the density, p is the pressure, and ν is the kinematic viscosity. The simulations assume constant fluid properties and no external forces.

The Q-criterion is a widely used vortex identification technique in CFD, effective in distinguishing vortical structures from regions dominated by shear. The criterion mathematically identifies areas where rotational motion exceeds strain using the velocity gradient tensor, expressed as:

$$Q = \frac{1}{2}(\|\Omega\|^2 - \|S\|^2)$$

Where Ω and S represent the rotation rate (antisymmetric) and strain rate (symmetric) components, respectively. Positive values indicate vortices. Due to computational constraints, this analysis focuses only on the rod's rounded-end configuration. Visualization using 124 terabytes of DNS simulation data, presented in a video (Bistany, 2024), clearly demonstrates the coherent vortical structures as they stretch and dissipate upon entering the contraction region near the rod tip, providing valuable insights into turbulence dynamics.

Turbulence can be categorized as either isotropic, where properties are uniform in all directions, or anisotropic, where turbulent fluctuations vary by direction. Further divided into three groups:

- x_{1C} : One-component turbulence (rod-like) where fluctuations occur predominantly in a single direction.
- x_{2C} : Two-component, axisymmetric turbulence (pancake-like), where fluctuations occur in two directions with equal magnitude.
- x_{3C} : Three-component, isotropic turbulence, where fluctuations exist in all three directions with equal magnitude.

These classifications are visually represented by the barycentric triangle seen in Figure 37.

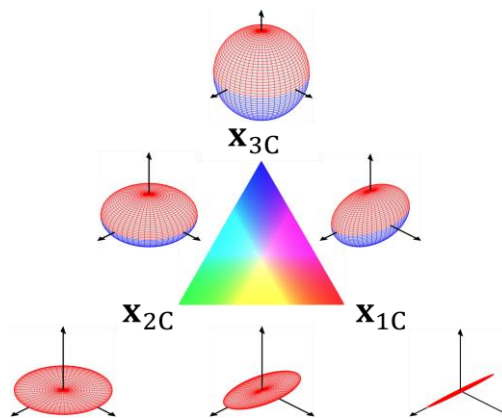


Figure 37. Barycentric triangle representation.

Quantitatively, turbulence anisotropy is described using the anisotropy tensor, derived from the Reynolds stress tensor, defined as:

$$a_{ij} = \langle u_i u_j \rangle - \frac{2}{3} k \delta_{ij}$$

Where a_{ij} captures the anisotropic components, $\langle u_i u_j \rangle$ is the Reynolds stress tensor, k is the Turbulent Kinetic Energy (TKE), and $\frac{2}{3} k \delta_{ij}$ represents the isotropic part.

TKE, defined as the mean kinetic energy per unit mass within the fluctuating velocity field, provides essential insights into turbulent flows. The TKE budget, representing a balance of production, dissipation, and turbulent transport terms, is fundamental for understanding energy transfer mechanisms and validating turbulence models. The condensed form of the TKE budget equation is expressed as:

$$\frac{\partial k}{\partial t} + \langle \mathbf{U} \rangle \cdot \nabla k = \wp - \varepsilon - (\nabla \cdot D_T + \nabla \cdot D_p + \nabla \cdot D_v)$$

Where the left side of the equation represents the local change in TKE plus the convection by the mean flow and on the right side of the equation we have the following budget terms from left to right: production, dissipation, turbulent diffusion, pressure diffusion, and viscous diffusion. The turbulence statistics contributing to each budget term are extensively detailed in the literature. TKE budgets are only collected for the rounded-end benchmark case.

Turbulence statistics are evaluated along the rod's axial length for radial positions spanning the pipe wall to the rod wall (Table 17), and along vertical lines from the rod tip to roughly $5D_h$ downstream (Table 18). The origin for these line plots is taken at the rod tip without the semi-spherical rounded-end for consistency between the rounded-end and blunt-end cases.

Table 17. Radial line plot locations as a function of axial distance along rod. Origin is rod tip without rounded-end.

ID	P1	P2	P3	P4	P5	P6	P7	P8	P9	P10
(mm)	0.0005	0.004	0.008	0.014	0.02	0.03	0.05	0.1	0.15	0.2

Table 18. Vertical line plot locations as a function of radial distance normal to rod wall, without the addition of the rod radius (e.g. $R9 = \frac{D_i}{2} + 5 \times 10^{-3}$). Origin is rod tip without rounded-end.

ID	R1	R2	R3	R4	R5	R6	R7	R8	R9
(mm)	1×10^{-5}	2×10^{-5}	5×10^{-5}	1×10^{-4}	2×10^{-4}	5×10^{-4}	1×10^{-3}	2×10^{-3}	5×10^{-3}

5.2.2.2 NekRS & Computational Resources

The code used in this work is NekRS, an open-source SEM CFD solver specifically designed for modeling and simulating complex turbulent fluid flows at high



fidelity. Built upon its CPU-oriented predecessor Nek5000, NekRS incorporates advanced optimizations for GPU architectures, dramatically accelerating performance and enabling simulations at previously unattainable scales and resolutions. Utilizing GPU-based HPC, NekRS enables DNS of complex geometries and flow conditions that were previously computationally infeasible or prohibitively costly. This capability allows for capturing the complete turbulence energy spectrum without empirical modeling, thus significantly enhancing the accuracy and fidelity of simulation results.

The spectral element method is central to NekRS, enabling to perform DNS of turbulent flow in complex problems at a low computational cost (Fischer, et al., 2022). SEM integrates the geometric flexibility of finite elements with the exponential convergence of spectral methods. Each element in the computational mesh supports is discretized with a tensor-product of high-order Lagrange polynomials, defined on Gauss-Lobatto-Legendre (GLL) collocation points. This approach allows for exceptionally accurate spatial discretization while maintaining computational efficiency (Fischer, et al., 2022). A notable advantage of SEM, detailed by (Fischer, et al., 2022), is permitting matrix-free operations and tensor contractions, which are highly efficient and scalable on GPU-based HPC systems (Fischer, et al., 2022). The solution fields (e.g., velocity and pressure) are expanded:

$$u(x) = \sum_{j=1}^n u_j \phi_j(\mathbf{x})$$

where ϕ_j are high-order basis functions and u_j are unknown basis coefficients (Fischer, et al., 2022). To enforce incompressibility, NekRS solves a pressure Poisson equation, which arises from the continuity constraint in the governing equations and ensures a divergence-free velocity field. In discrete form this becomes $A p = b$ with A the stiffness matrix and b the data vector derived from the velocity field (Fischer, et al., 2022). The pressure Poisson equation governs the fastest modes in the system and therefore represents the stiffest substep. (Fischer, et al., 2022) demonstrates that it often accounts for the majority of the computational time per timestep.

The high resolution and moderately high Reynolds number in this study require exceptionally small timesteps to maintain a CFL number near unity, which would be computationally impractical without the computational speed provided by NekRS. The simulations presented were conducted using the Frontier supercomputer at Oak Ridge National Laboratory, the world's fastest supercomputer at the time of these simulations. For the most computationally intensive (requiring the longest time averaging) flow statistics—specifically, the TKE budgets for the rounded-end configuration—data were collected over approximately 25 domain flow-through cycles, requiring about 25,000–30,000 node-hours on Frontier.

5.2.2.3 Numerical Mesh

In DNS, the computational grid must be fine enough to resolve the Kolmogorov scale η throughout the domain.

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

This requirement is particularly strict near wall regions, where the smallest turbulent structures form. NekRS requires the exclusive use of hexahedral elements. Figure 38 illustrates these elements for the rounded-end, detailing all domain boundaries and the fluid volume (sections of elements removed for clarity). The blunt-end holds a similar meshing strategy with the only difference being the end configuration.

Each element is subdivided using an N^{th} order polynomial, and because the spectral element method employs GLL points, each element contains $(N + 1)^3$ computational grid points. It is well established in the literature that DNS grid resolution requirements scale with Reynolds number raised to a fractional power. Figure 39 compares the spacing between GLL points and the local Kolmogorov length scale across a cross-section of the annulus where the Reynolds number is highest. From this comparison, it is demonstrated that the grid spacing is sufficiently fine to capture the smallest turbulent scales defined by the Kolmogorov length scale.

Although DNS resolves all turbulent scales down to the Kolmogorov length scale, a Grid Convergence Index (GCI) study is included to quantify numerical uncertainty in integral metrics and verify asymptotic convergence. The pipe friction Reynolds number is used as the reference quantity:

$$Re_\tau = \frac{u_\tau \delta}{\nu}$$

Where u_τ is the friction velocity and δ is the pipe half-width. The analytical Re_τ is 484.57. Using Richardson extrapolation, the finest grid ($N = 7$) deviates by just 0.072%, while the extrapolated value differs by less than 0.001%. With a safety factor of 1.25, the GCI confirms the solution lies in the asymptotic convergence range.

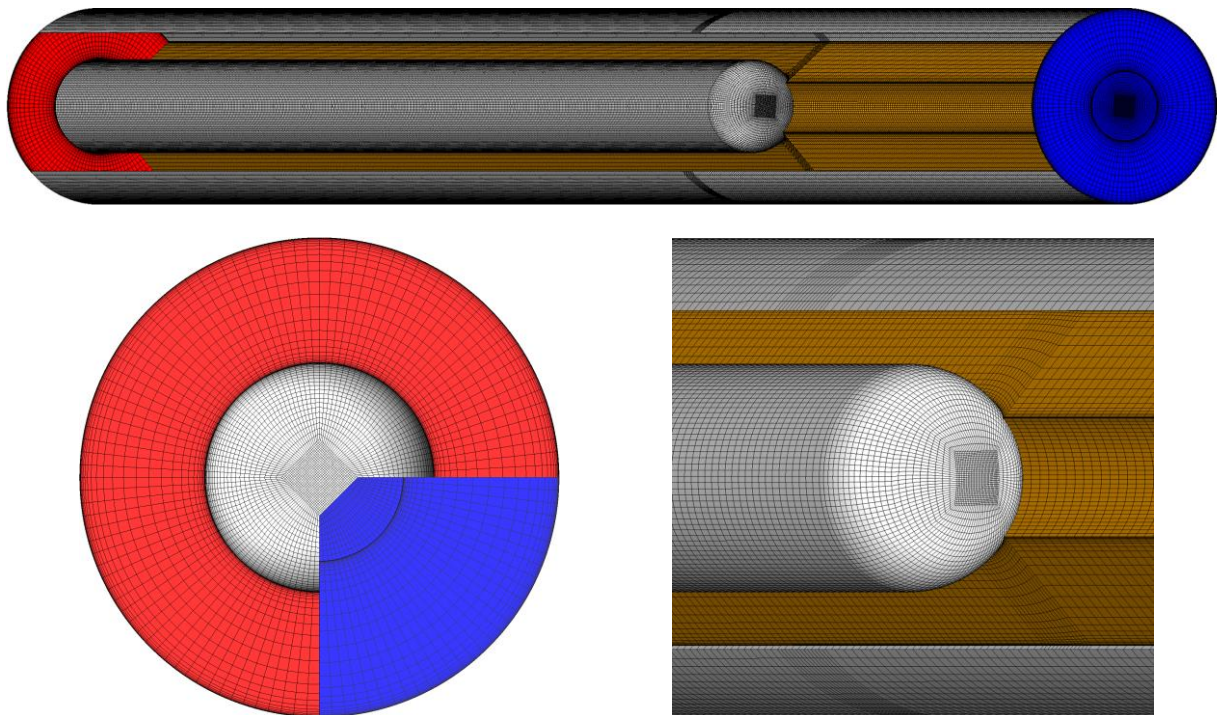


Figure 38. Rounded-end configuration mesh element structure: inlet (blue), walls (white), outlet (red), & fluid (orange).

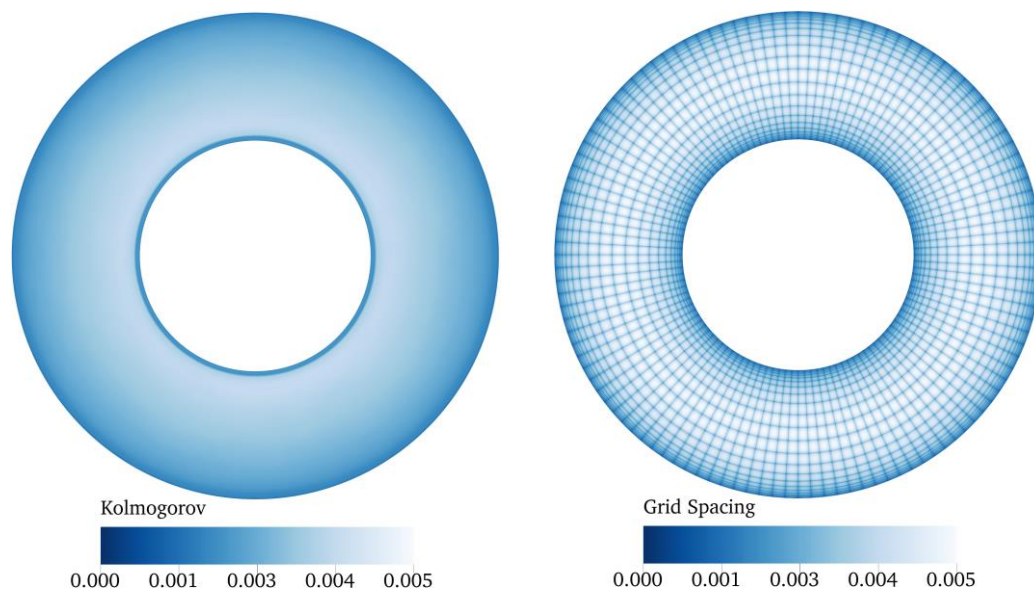


Figure 39. Comparison of Kolmogorov length scale to the distance between GLL points.

While DNS inherently captures all turbulent scales, the GCI analysis offers a complementary check on numerical convergence. It verifies that variations across grid resolutions are minimal and that the selected mesh provides sufficient accuracy—both physically and numerically—for evaluating key quantities like turbulence budgets and Reynolds stresses.

The grid details are shown in Table 19. All turbulence statistics for the rounded-end are computed with the $N = 7$ mesh. The $N = 5$ mesh for the rounded-end was used to capture a particular time history frequency (100 hz) over 10 dimensional seconds. Nonetheless, the statistics are also provided for the rounded $N = 5$ mesh excluding Q-criterion analysis and TKE budgets. The blunt-end case was computed with $N = 5$ due to computational resource consumption, the same statistics are provided for the blunt-end as the rounded $N = 5$ case.

Table 19. Grid details of fluid meshes.

Configuration	Polynomial Order (N)	Elements	Grid Points
Rounded	5	2.65×10^6	5.71×10^8
Rounded	7	2.65×10^6	1.35×10^9
Blunt	5	2.54×10^6	5.49×10^8

5.2.3 Verification & Validation

5.2.3.1 Pipe Flow

The verification with pipe flow consists of obtaining expected canonical behavior in first- and second-order statistics. The first step in verification focuses on resolving near-wall turbulence before the flow interacts with the annular region containing the cantilevered rod. Figure 40 presents the mean velocity profile expressed in wall units, where y^+ is the wall-normal distance normalized by the viscous length scale, and U^+ is the mean velocity normalized by the friction velocity. This analysis confirms accurate representation of the turbulent boundary

layer subregions, including the viscous sublayer and the logarithmic region. Verification is further supported by examining second-order statistics—specifically, the RMS of axial velocity fluctuations in Figure 40 and the TKE budgets in Figure 41. Both figures are validated with experimental and pipe flow data and canonical turbulent pipe flow is verified through expected statistics behavior and a numerical comparison.

This pipe flow analysis serves as the $10D_h$ upstream reference for the comparison with experimental data. Given the extensive investigation of pipe flow throughout fluid mechanics history, the turbulence statistics presented in Figure 40 & Figure 41 provide strong confidence in the simulation's accuracy, supporting further analysis near the rod tip and along the rod length.

Legend: Present study (—), UoM (Cioncolini et al., 2023) ($\blacktriangle$), Toonder & Nieuwstadt (1997) $Re = 17,800$ ($\square$), Khoury et al. (2013) $Re = 19,000$ ($\times$), $U^+ = y^+$ (---), & Log Law (---).

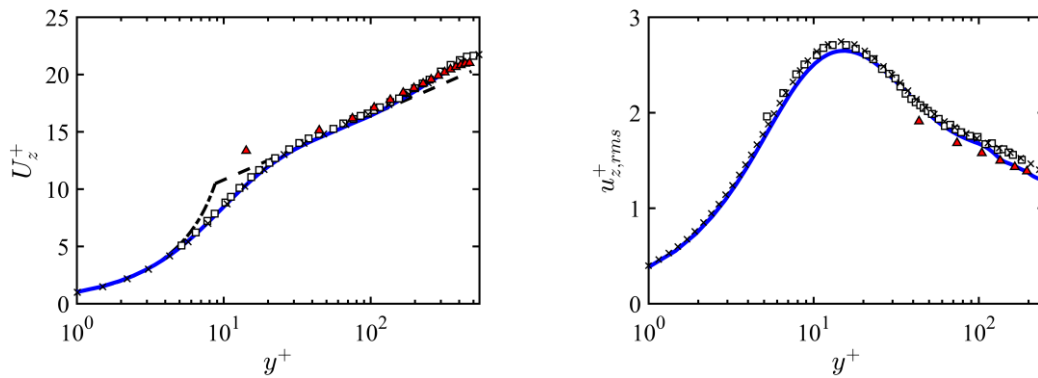


Figure 40. Fully developed turbulent pipe flow. Mean axial velocity profile (left) and wall-normal RMS of axial velocity fluctuations (right). Comparisons with: (Cioncolini, et al., 2023), (Toonder & Nieuwstadt, 1997) , & (Khoury, et al., 2013).

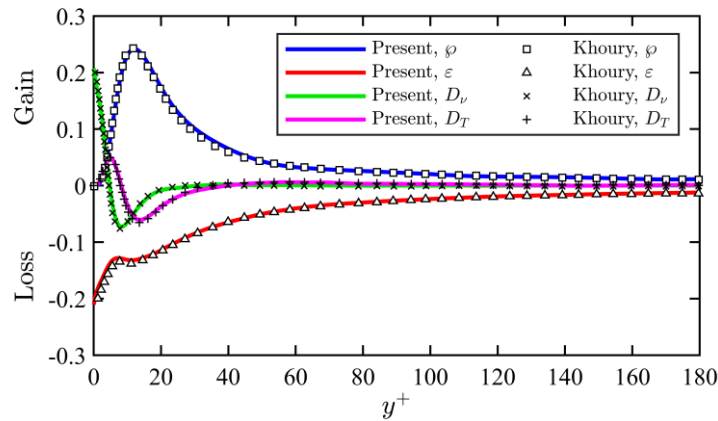


Figure 41. Comparison of pipe wall-normal TKE budget terms normalized by u_τ^4/ν compared with (Khoury, et al., 2013). Pressure diffusion, D_p , omitted for clarity.

5.2.3.2 Comparisons with Experiment

To begin this comparison, the contour plots of velocity with the rounded-end case are compared in Figure 42. Overall, good visual agreement is met for the flow field.

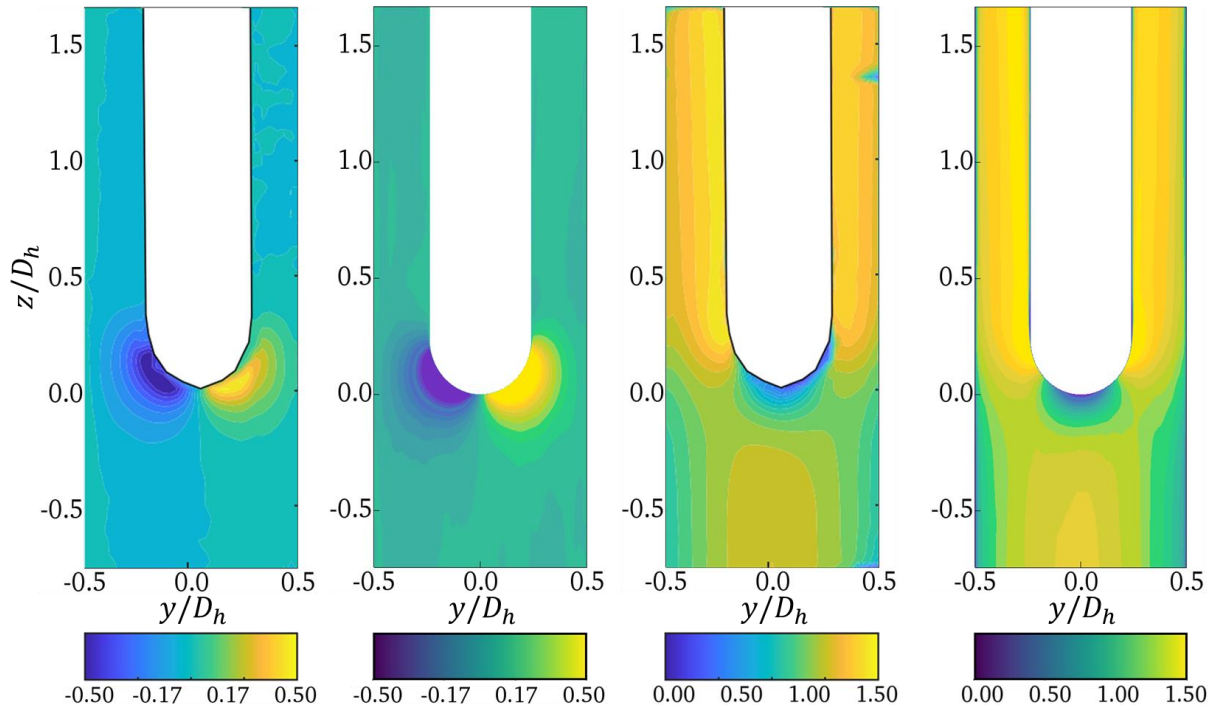


Figure 42. Time-averaged dimensionless velocity contours for Rounded-end. From left to right: UoM radial, Present radial, UoM axial, & Present axial.

A series of horizontal (radial) line plots are generated and directly compared with available experimental data for both configurations. Table 20 & Table 21 list the locations of these line plots, with the first column indicating the line ID and the second showing the requested comparison locations in dimensional form. For the rounded-end case, the experimental data are not normalized by U_{ref} (but it is provided and used to normalize for comparison), and domain coordinates are normalized by the pipe diameter. For the blunt-end, the data is normalized by the bulk velocity, however, domain coordinates appear to be normalized by the annulus hydraulic diameter ($D_{h,a} = 11$ mm). Therefore, for the blunt-end, the locations are normalized by $D_{h,a}$ then rescaled and normalized by D_h (pipe diameter) for consistency. Locations from the PIV data within the rod should be ignored but are not removed in the subsequent plotting. Figure 43 through Figure 52 plot the following time averaged data for both cases: axial velocities, radial velocities, axial normal stresses, radial normal stresses, and radial (wall-normal) shear stresses. For these figures, the blue line is the present work, and the white squares are the experimental data. The experimental data for the blunt-end included only the axial and radial velocity components; however, the Reynolds stresses from the present work are still provided for future comparisons.

Additionally, a series of vertical (axial) line plots are generated and directly compared with available experimental data for both configurations. Table 22 and Table 23 list the locations of these vertical lines. The same scaling process for selecting the available locations as in the two above tables is used here. Figure 53 through Figure 62 plot the following time averaged data for both cases: axial velocities, radial velocities, axial normal stresses, radial normal stresses, and radial (wall-normal) shear stresses.

Table 20. Horizontal rounded-end line plot locations. Origin is tip of rounded-end. Positive indicates downstream (up the rod) of origin.

Line ID	Location (mm)	Location ($1/D_h$)
HR1	-16.296294	-0.776014
HR2	-3.786006	-0.180286
HR3	-0.493836	-0.023516
HR4	0.164619	0.007839
HR5	6.74898	0.321380
HR6	29.1358	1.387419

Table 21. Horizontal blunt-end line plot locations. Origin is tip of blunt-end. Positive indicates downstream of origin.

Line ID	Location (mm)	Location ($1/D_{h,a}$)	Location ($1/D_h$)
HB1	-15.697941	-1.4270864525	-0.747521
HB2	-4.0041015	-0.3640093465	-0.1906715
HB3	-0.7750826706	-0.0369086986	-0.0369086986
HB4	0.164619	0.0039788829	0.007839
HB5	6.74898	0.6172925982	0.32138
HB6	29.1358	2.2119082581	1.387419

Table 22. Vertical rounded-end line plot locations. Origin is tip of rounded-end. Positive indicates right of confining pipe axis. Taken from $1D_h$ upstream of tip and $5D_h$ downstream.

Line ID	Location (mm)	Location ($1/D_h$)
VR1	5.583627	0.265887
VR2	6.242061	0.297241
VR3	6.900495	0.328595

Table 23. Vertical blunt-end line plot locations. Origin is tip of blunt-end. Positive indicates right of confining pipe axis. Taken from $1D_h$ upstream of tip and $5D_h$ downstream.

Line ID	Location (mm)	Location ($1/D_{h,a}$)	Location ($1/D_h$)
VB1	5.6220423	0.5110947627	0.2677163
VB2	6.0718056	0.5519823442	0.2891336
VB3	6.9713322	0.6337575055	0.3319682

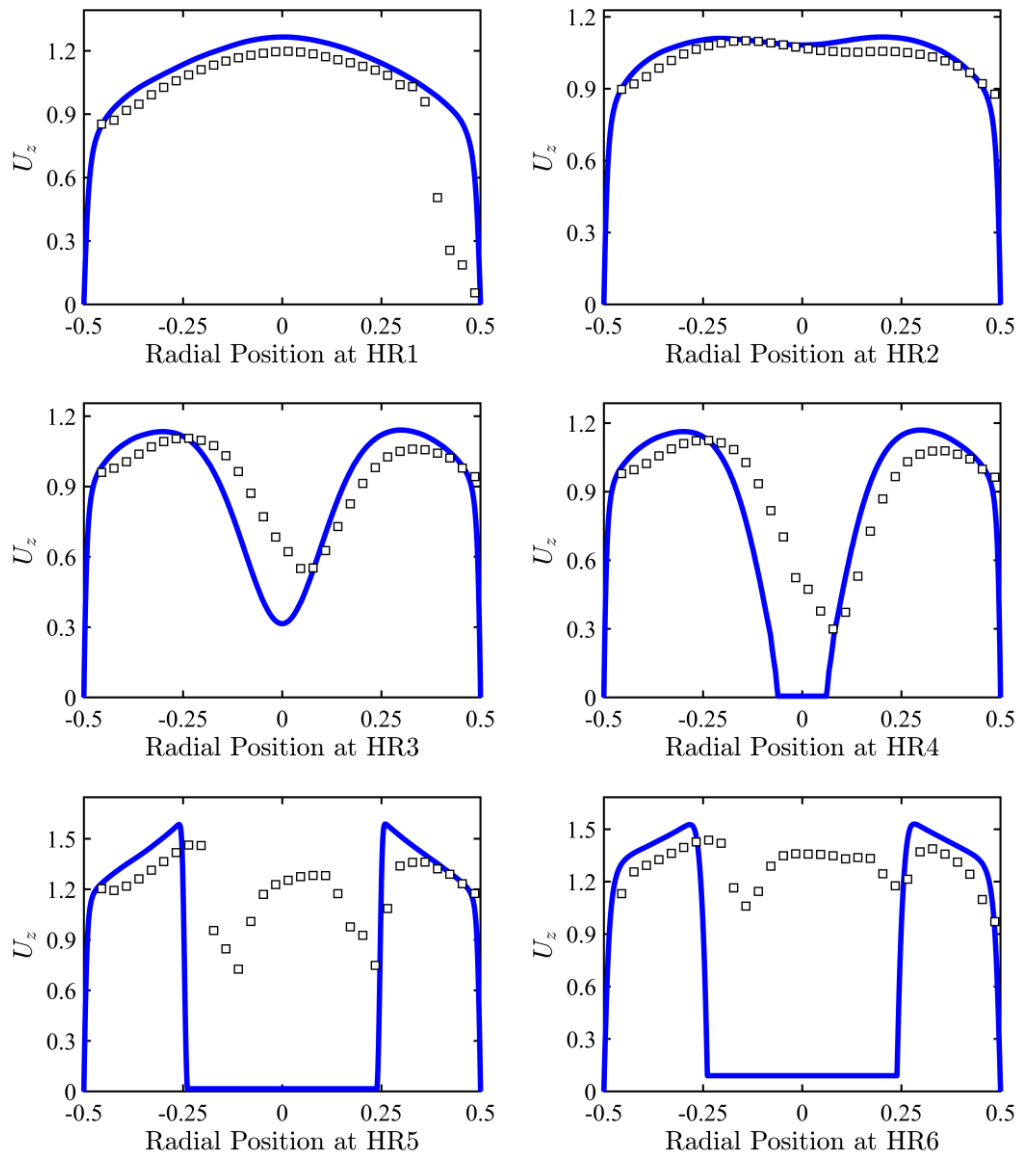


Figure 43. Rounded-end axial velocities corresponding to locations in Table 20.

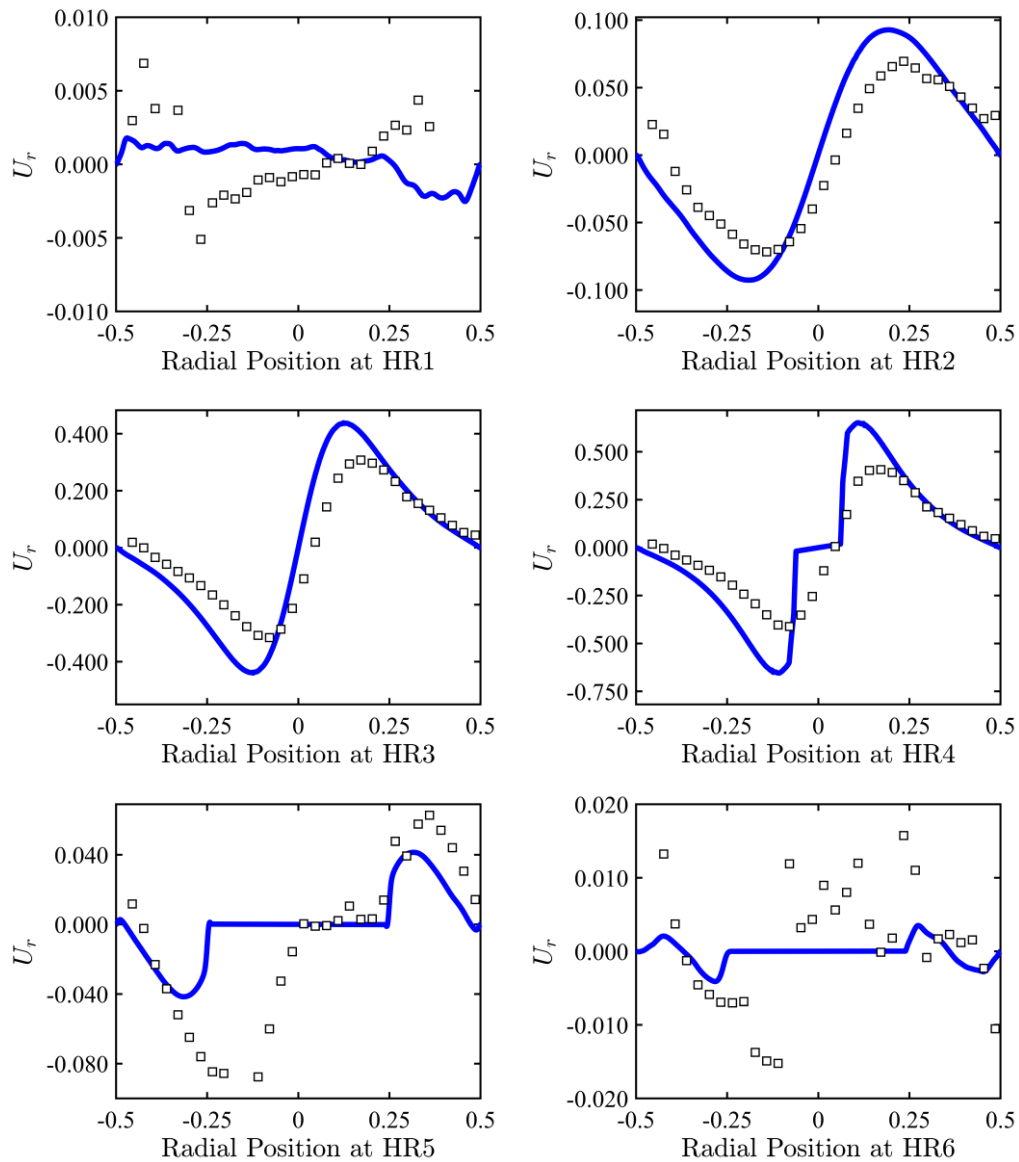


Figure 44. Rounded-end radial velocities corresponding to locations in Table 20.

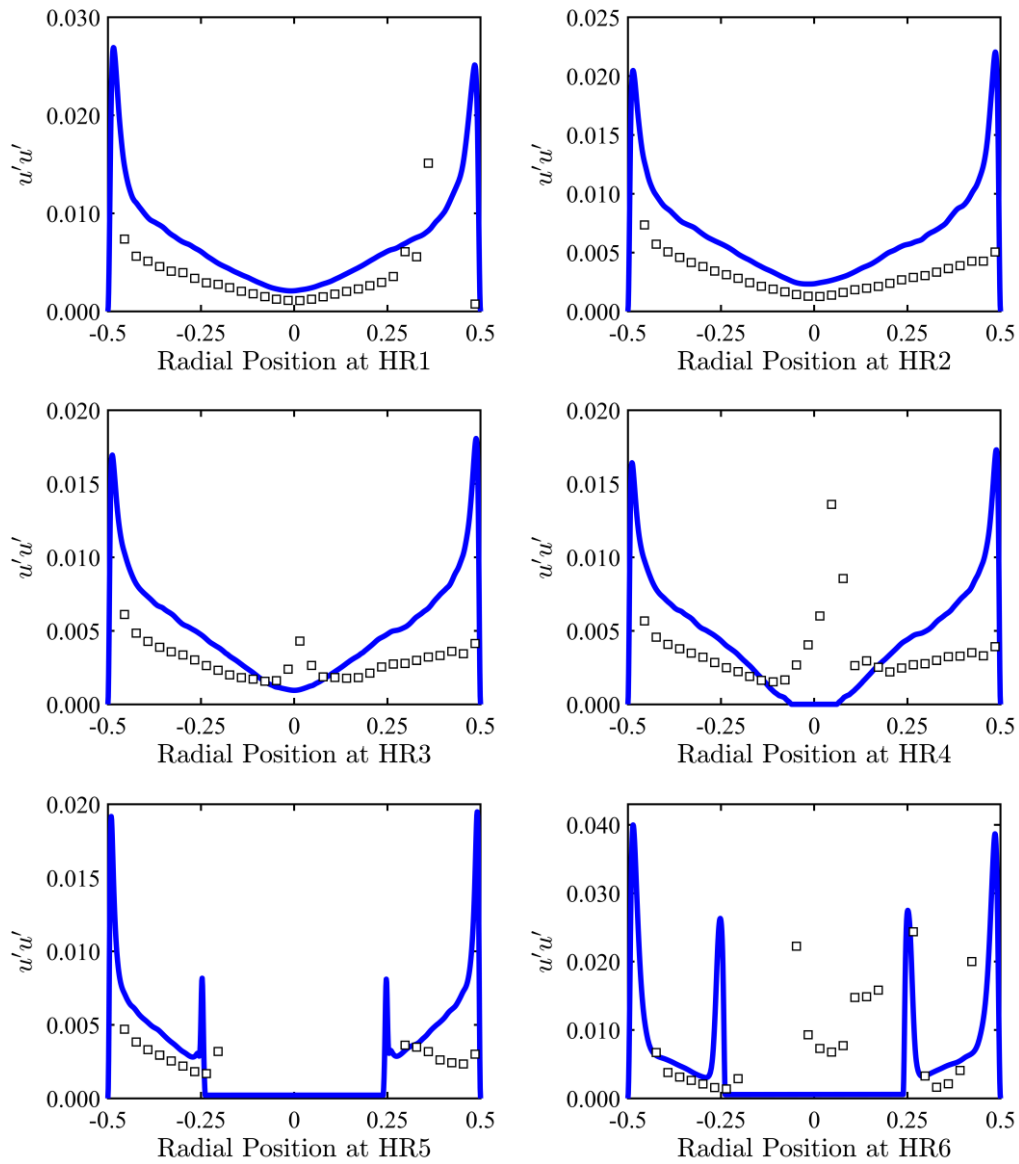


Figure 45. Rounded-end axial normal stresses corresponding to locations in Table 20.

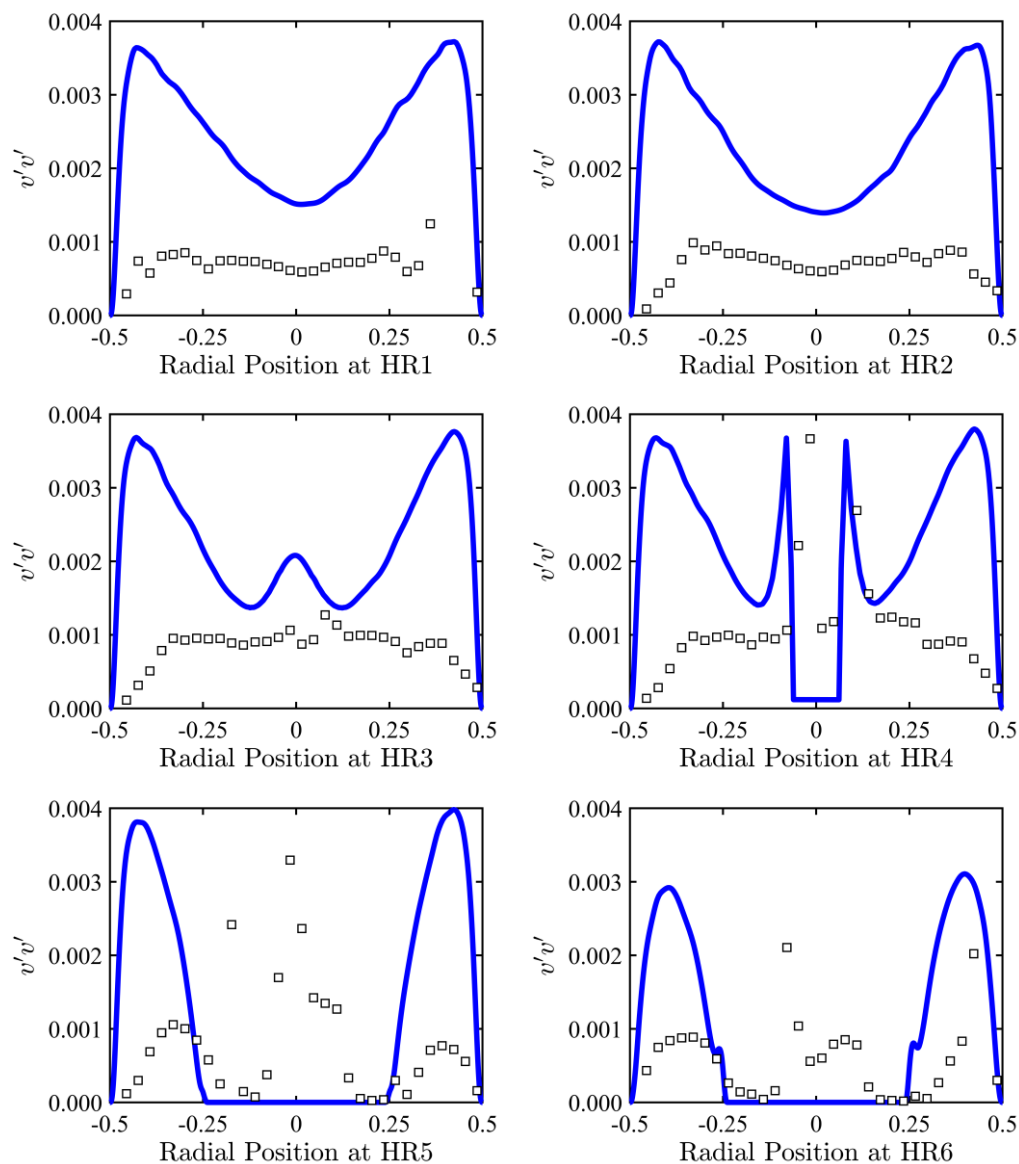


Figure 46. Rounded-end radial normal stresses corresponding to locations in Table 20.

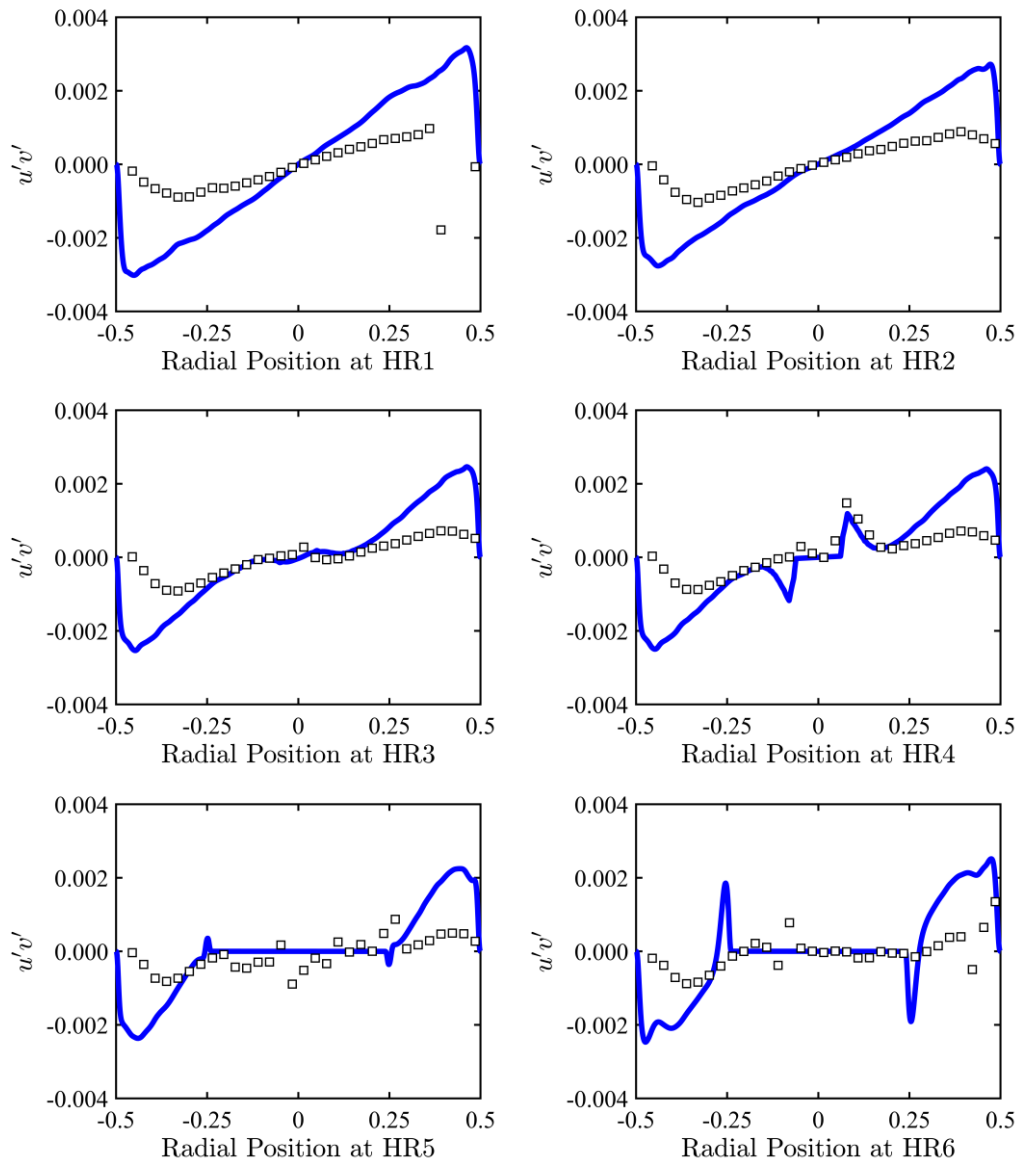


Figure 47. Rounded-end radial (wall-normal) shear stresses corresponding to locations in Table 20.

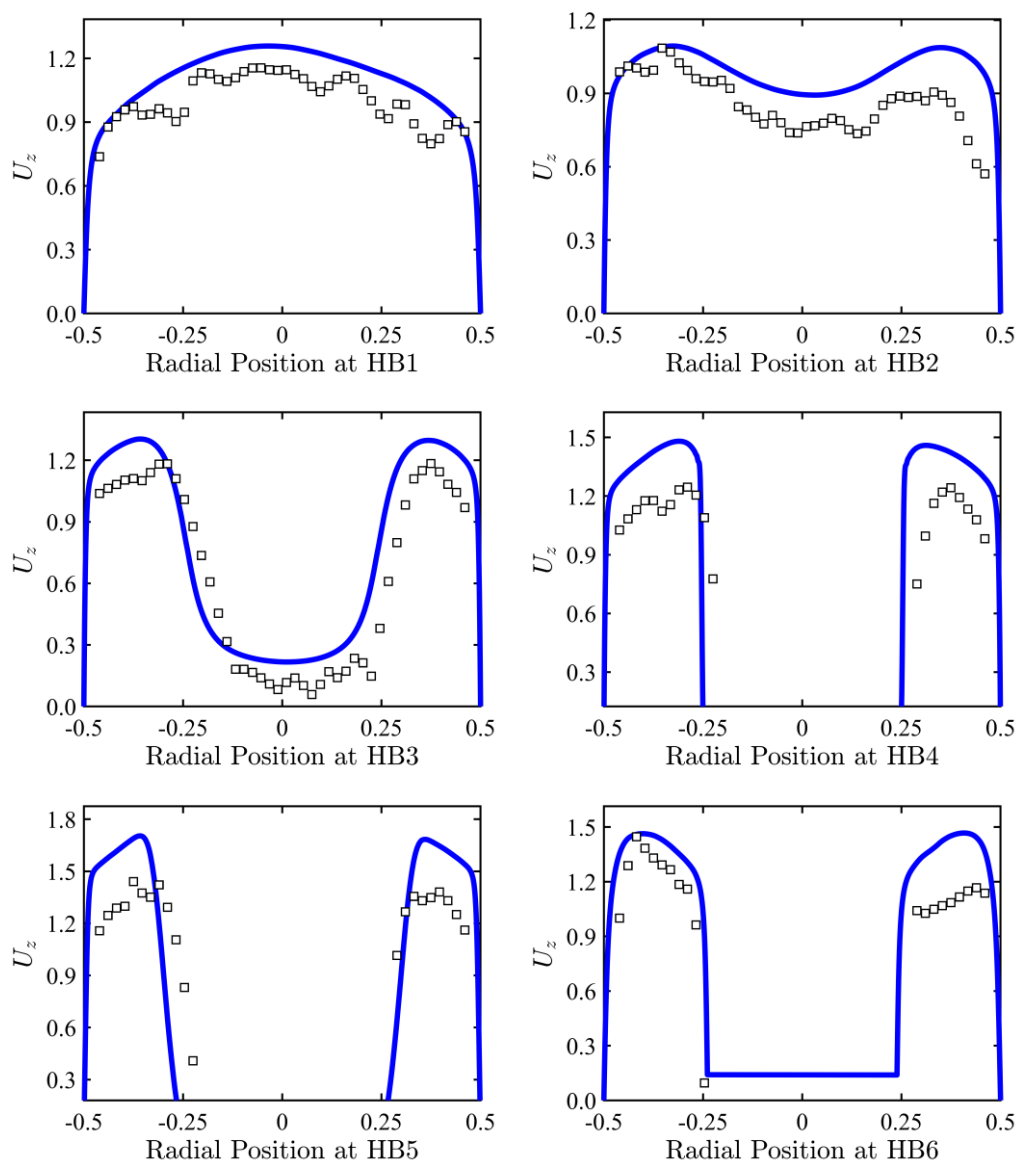


Figure 48. Blunt-end axial velocities corresponding to locations in Table 21.

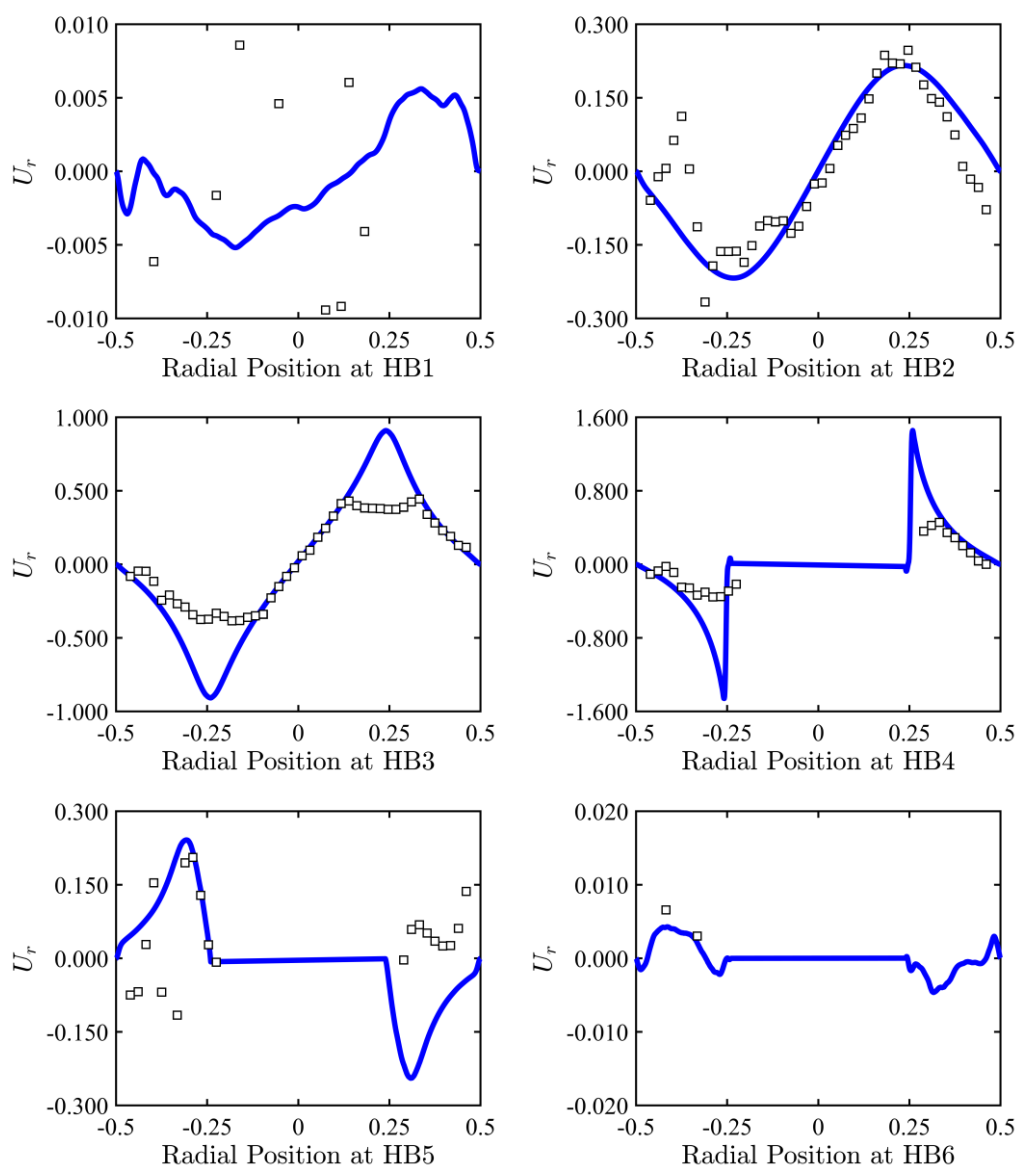


Figure 49. Blunt-end radial velocities corresponding to locations in Table 21.

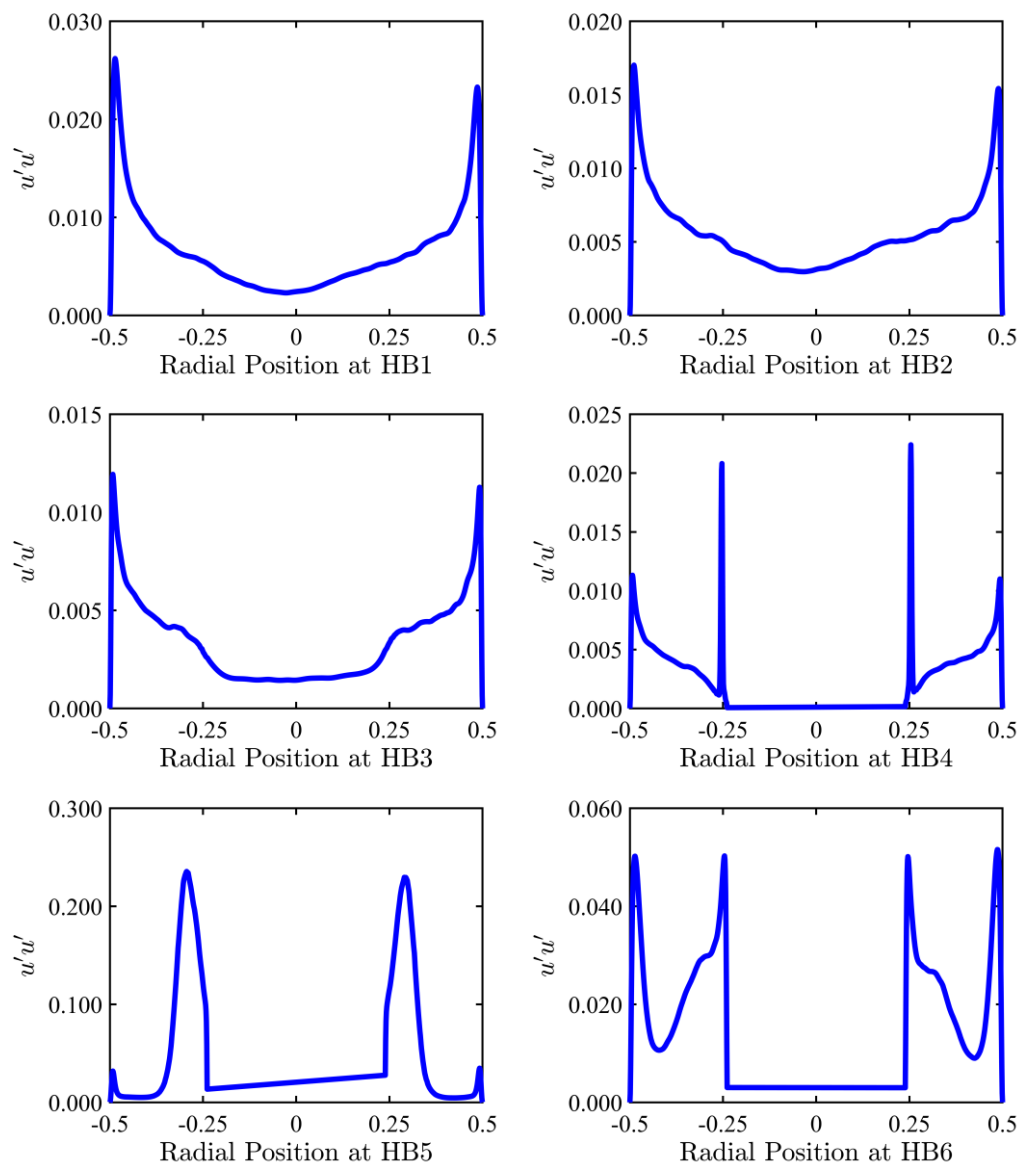


Figure 50. Blunt-end axial normal stresses corresponding to locations in Table 21.

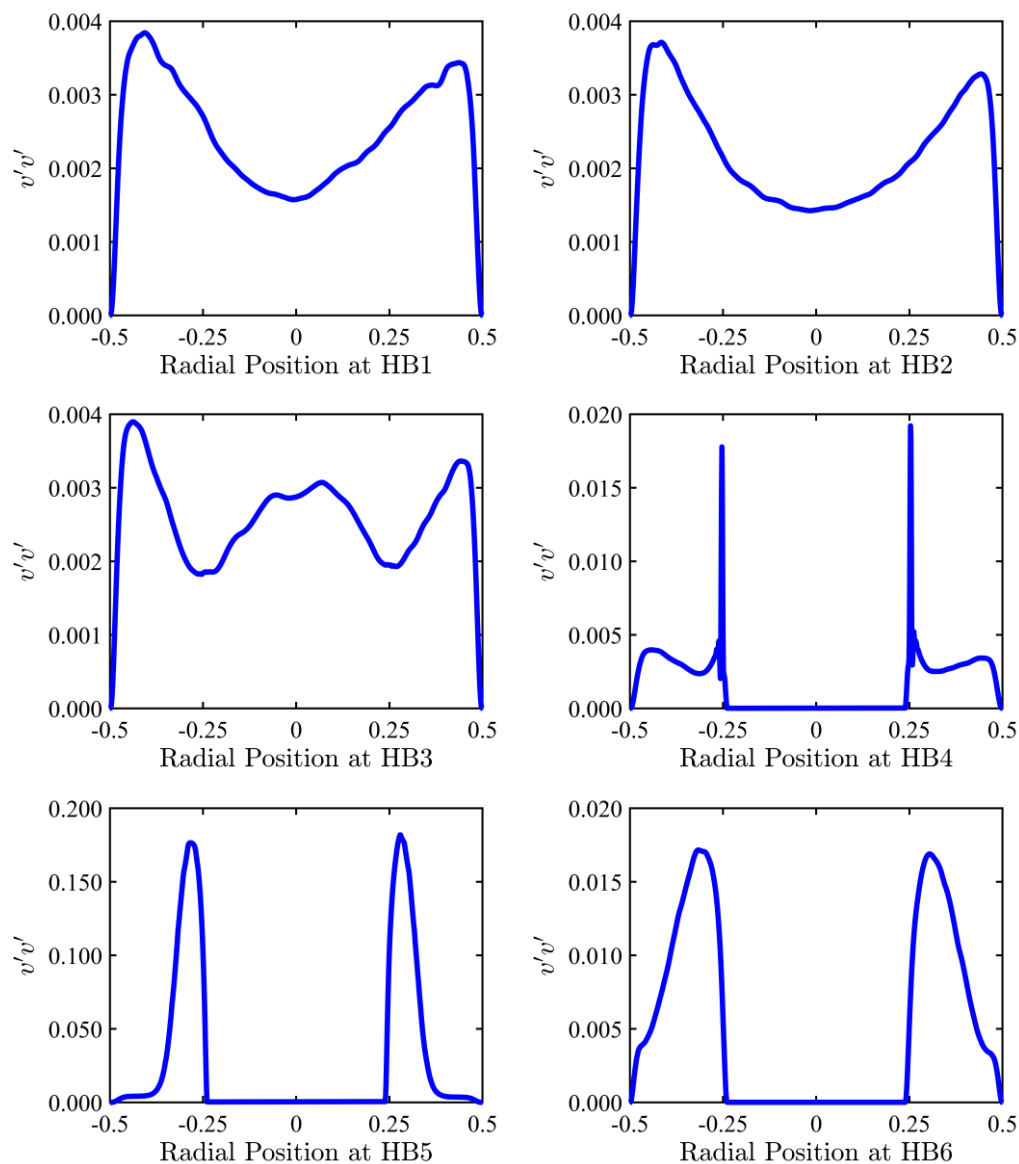


Figure 51. Blunt-end radial normal stresses corresponding to locations in Table 21.

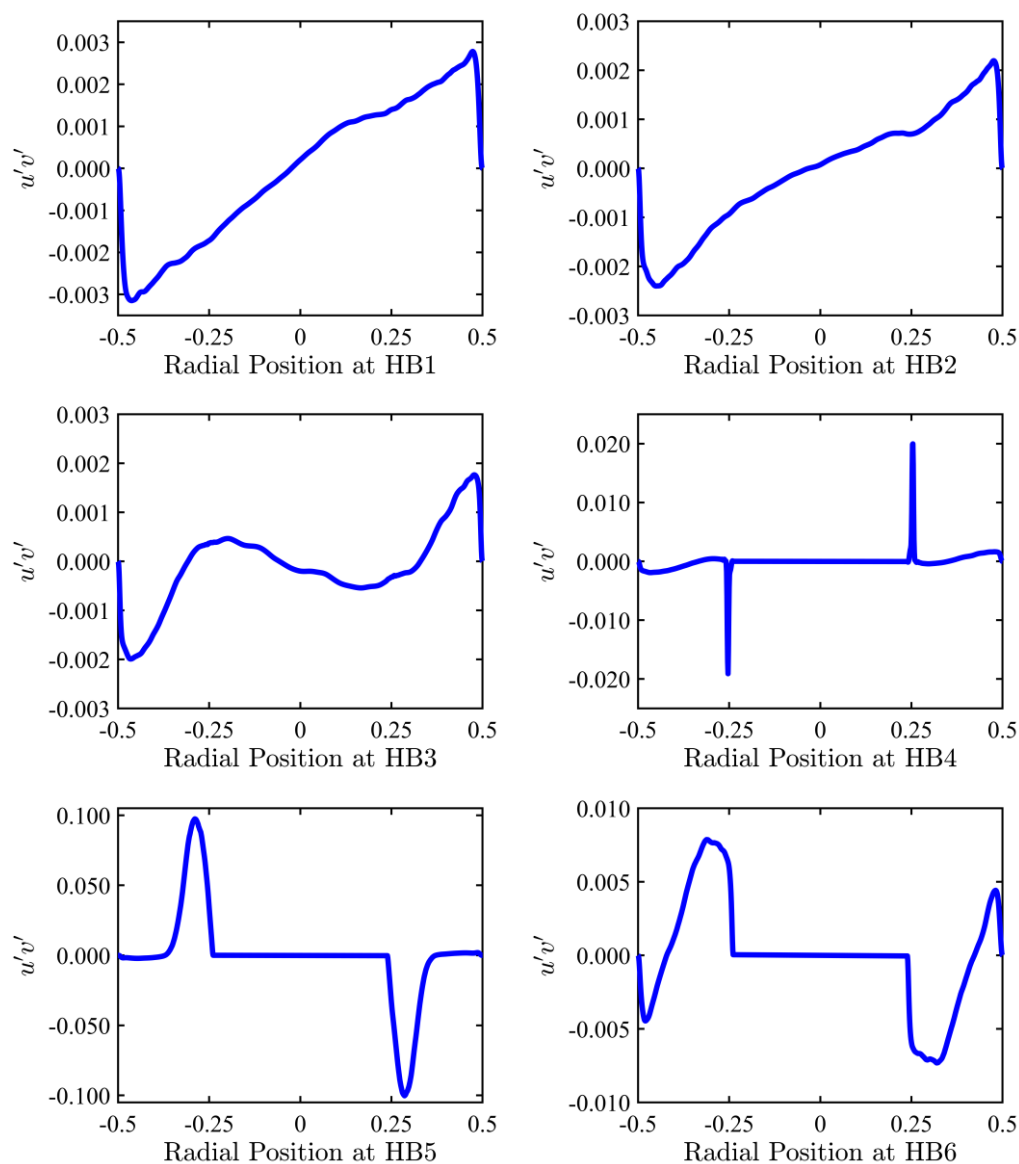


Figure 52. Blunt-end radial (wall-normal) shear stresses corresponding to locations in Table 21.

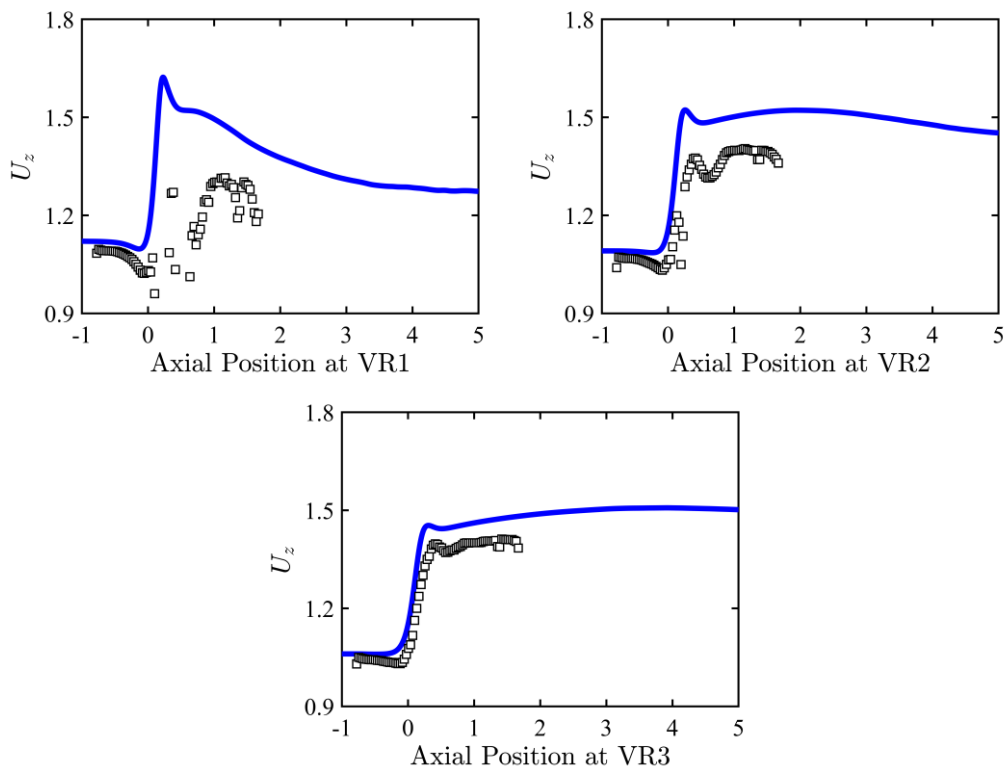


Figure 53. Rounded-end axial velocities corresponding to locations in Table 22.

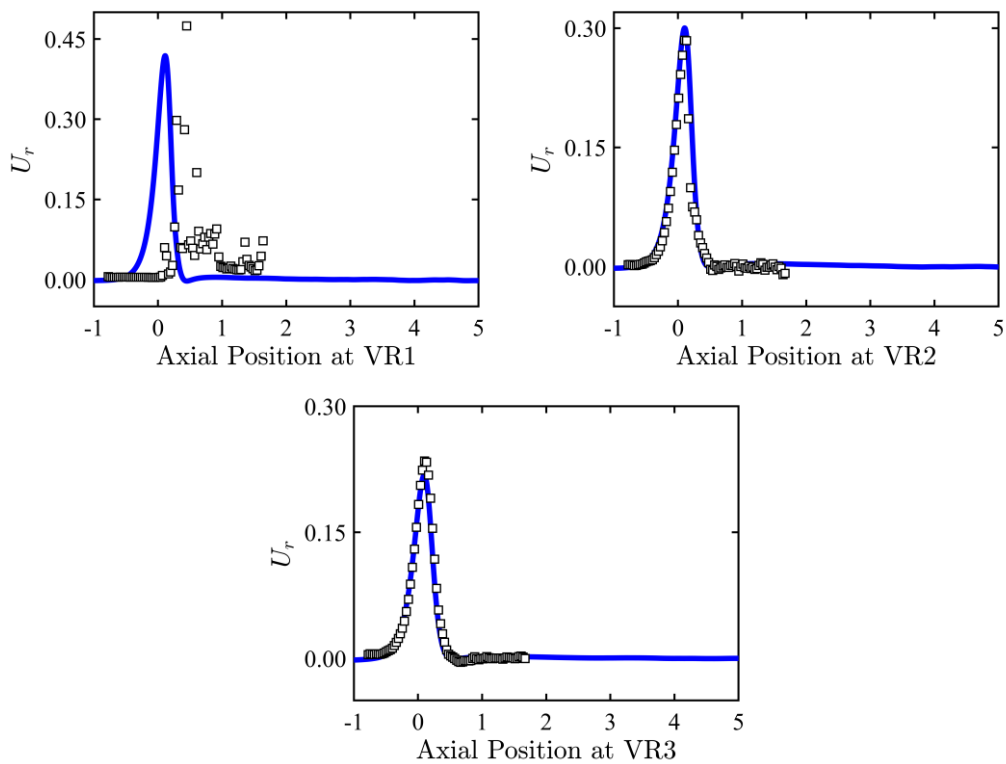


Figure 54. Rounded-end radial velocities corresponding to locations in Table 22.

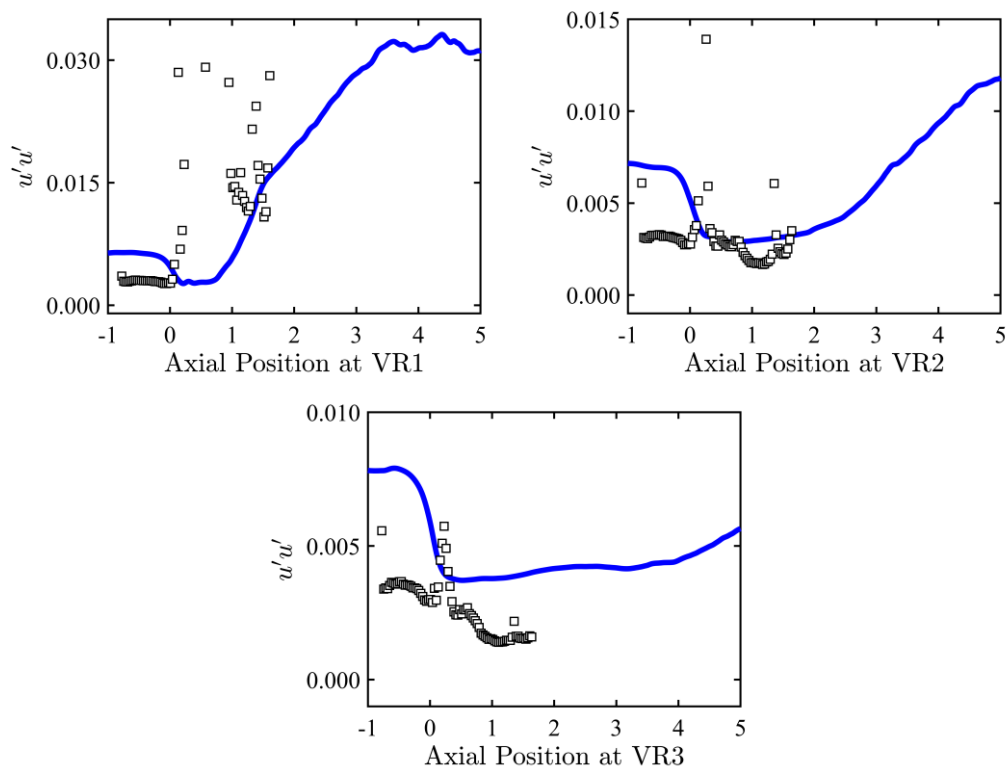


Figure 55. Rounded-end axial normal stresses corresponding to locations in Table 22.

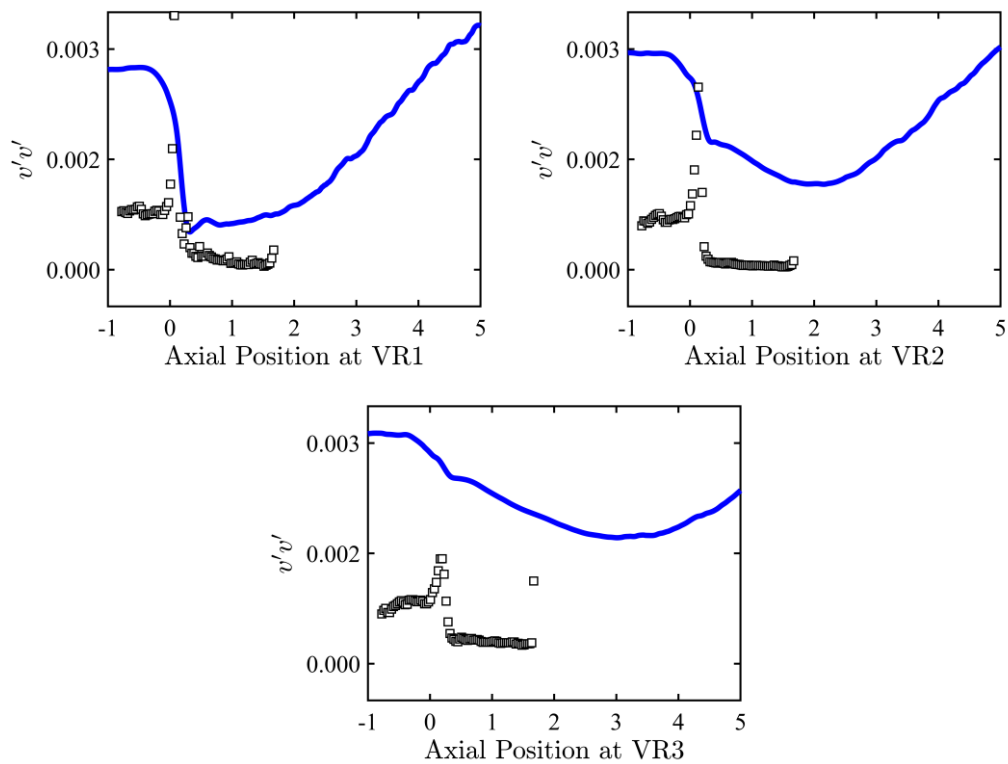


Figure 56. Rounded-end radial normal stresses corresponding to locations in Table 22.

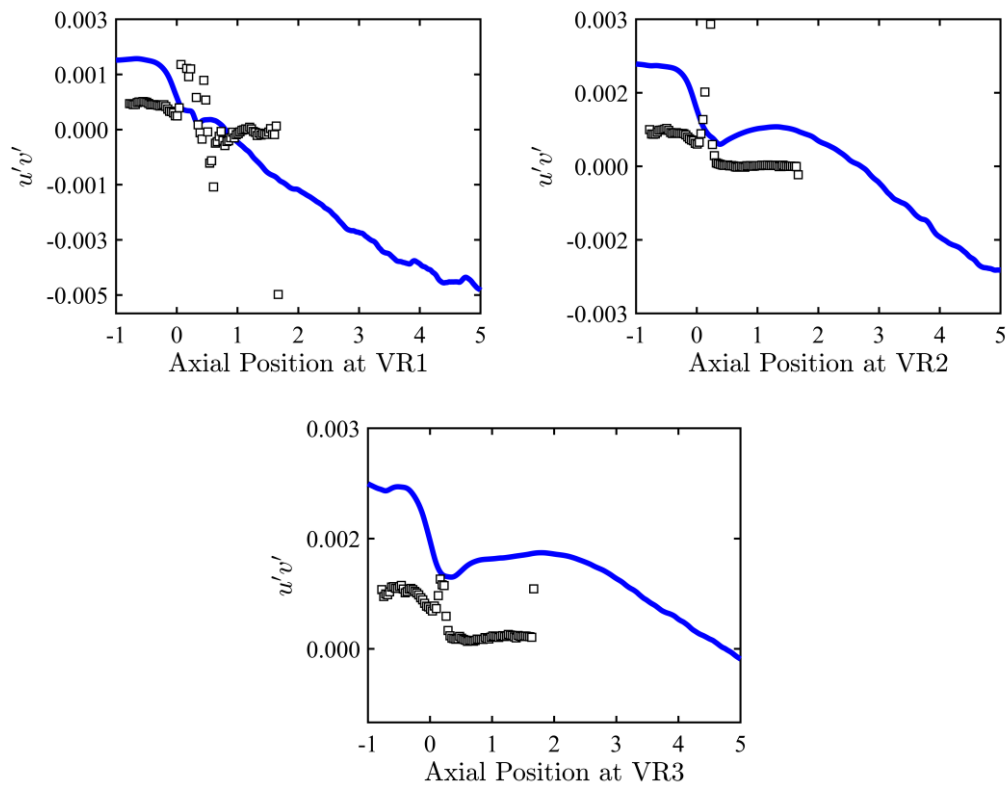


Figure 57. Rounded-end radial (wall-normal) shear stresses corresponding to locations in Table 22.

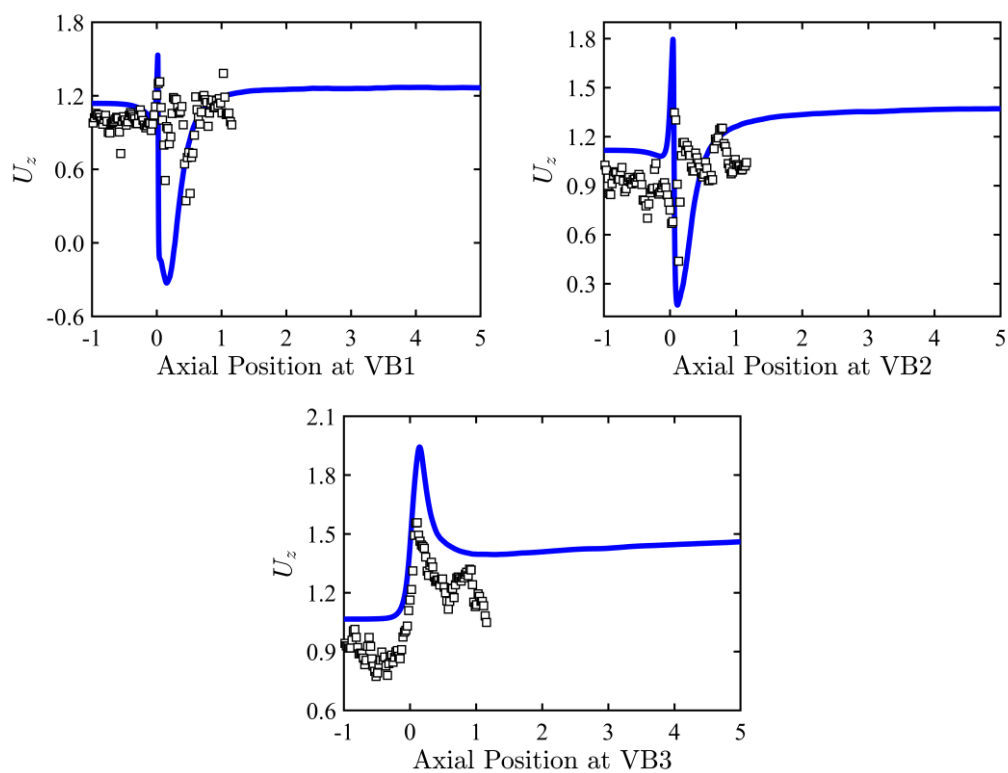


Figure 58. Blunt-end axial velocities corresponding to locations in Table 23.

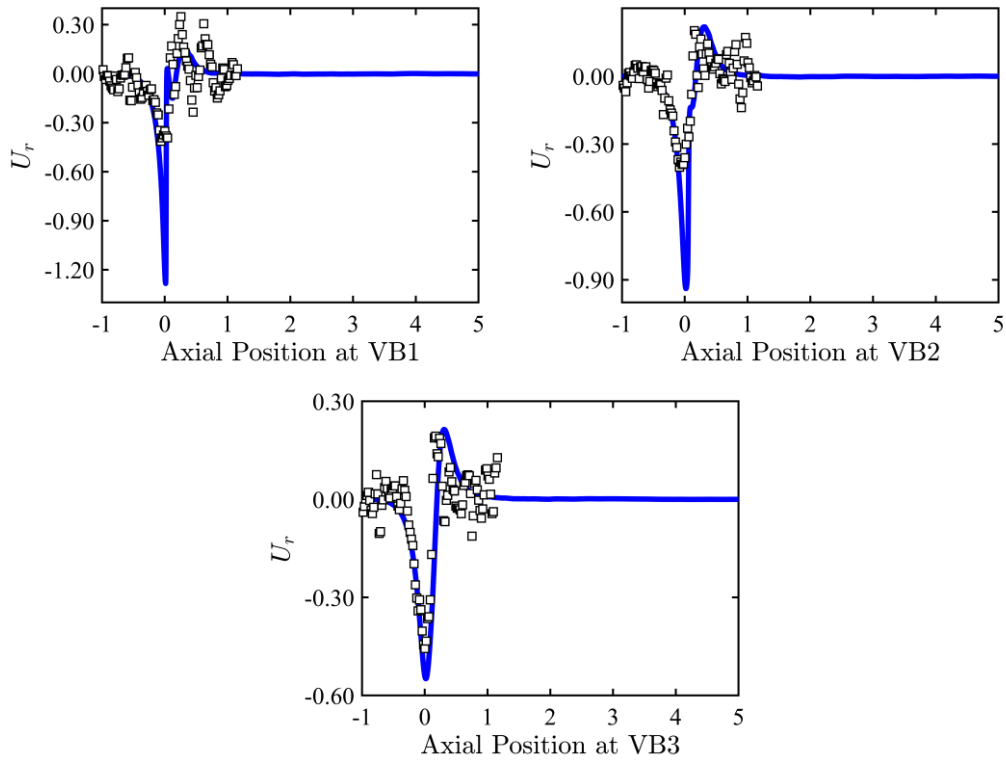


Figure 59. Blunt-end radial velocities corresponding to locations in Table 23.

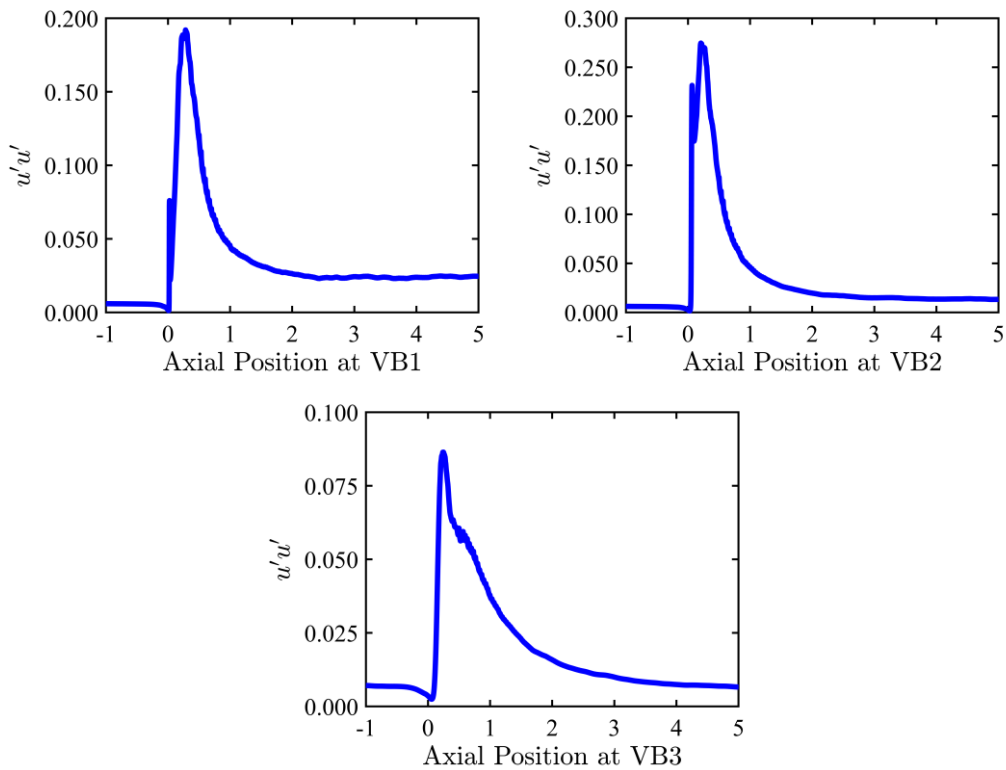


Figure 60. Blunt-end axial normal stresses corresponding to locations in Table 23.

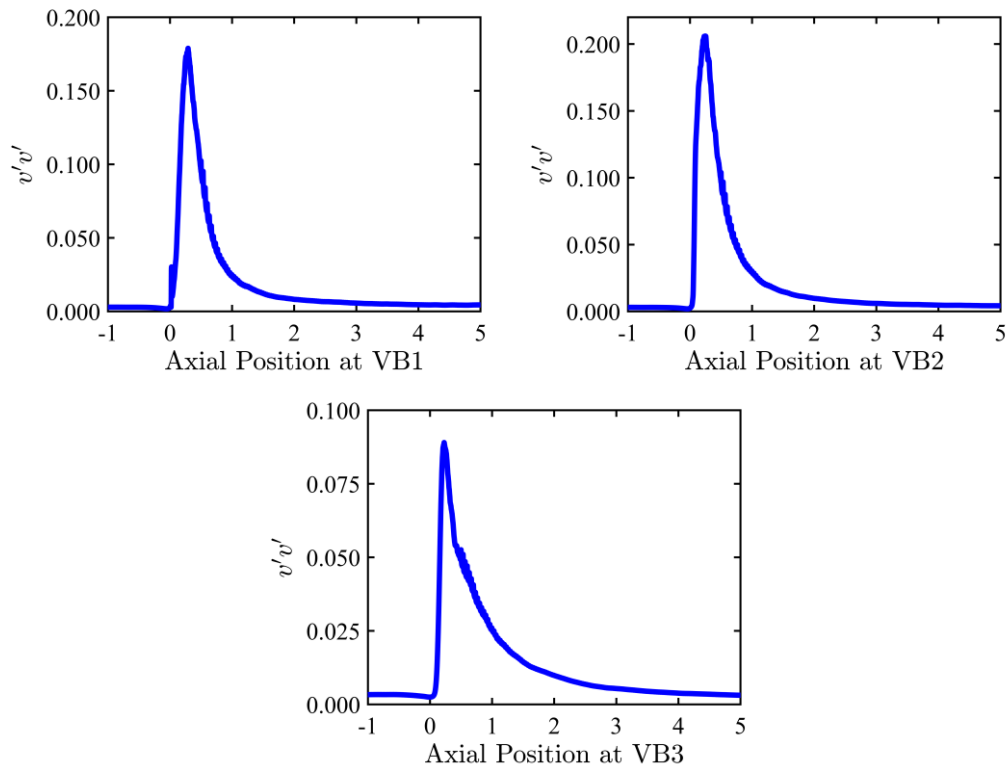


Figure 61. Blunt-end radial normal stresses corresponding to locations in Table 23.

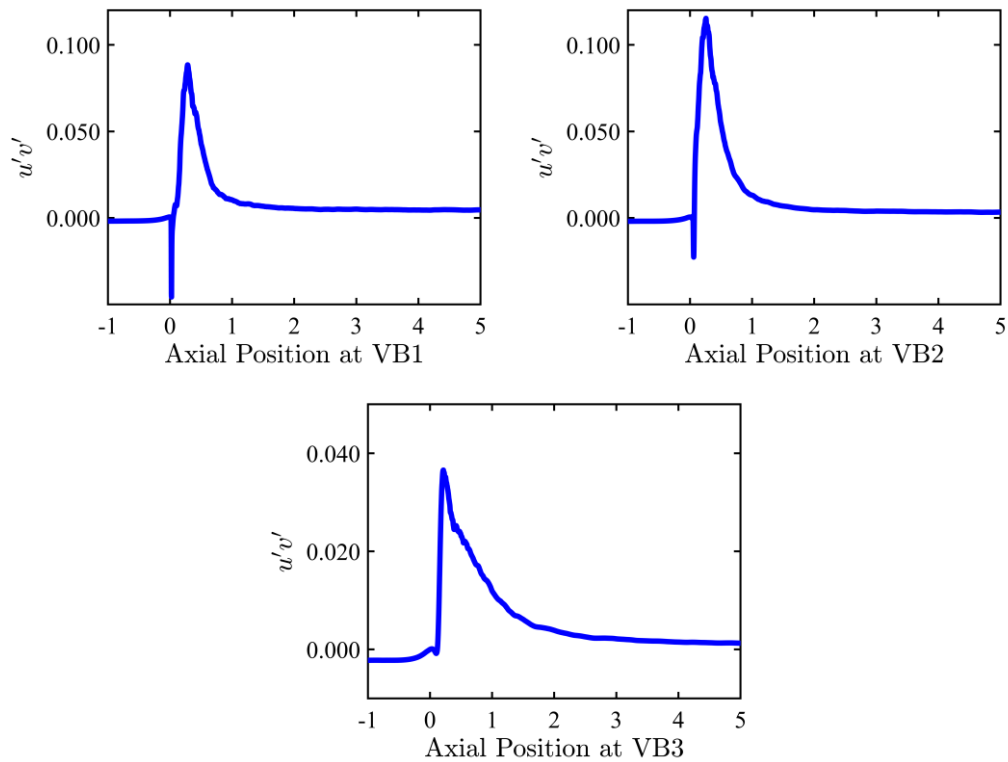


Figure 62. Blunt-end radial (wall-normal) shear stresses corresponding to locations in Table 23.

5.3 Simulation Results

All statistics and geometric parameters are reported in dimensionless form unless otherwise specified. Because $U_{ref} = 1$, normalization by U_{ref} is trivial and thus omitted. The data can be easily rescaled for comparison, provided the Reynolds numbers are consistent. Section 5.3.1 & 5.3.2 present the simulation results for both the rounded-end and blunt-end cases, respectively.

5.3.1 Rounded-End

5.3.1.1 Axial Cross-Sectional Fields

Figure 63 through Figure 69 show two-dimensional cross-sections centered on the rod's rounded tip, highlighting both first- and second-order flow statistics. Each axial slice spans roughly five pipe diameters—about $1.5D_h$ upstream of the tip and $3.5D_h$ downstream.

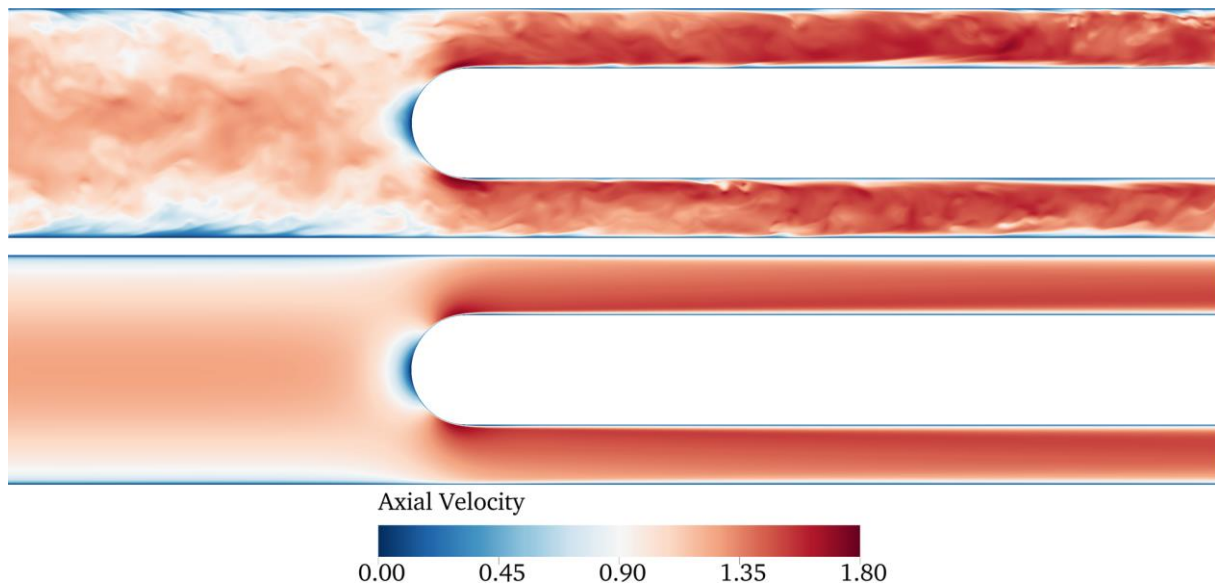


Figure 63. Rounded-end axial velocity: instantaneous (top) & time averaged (bottom).

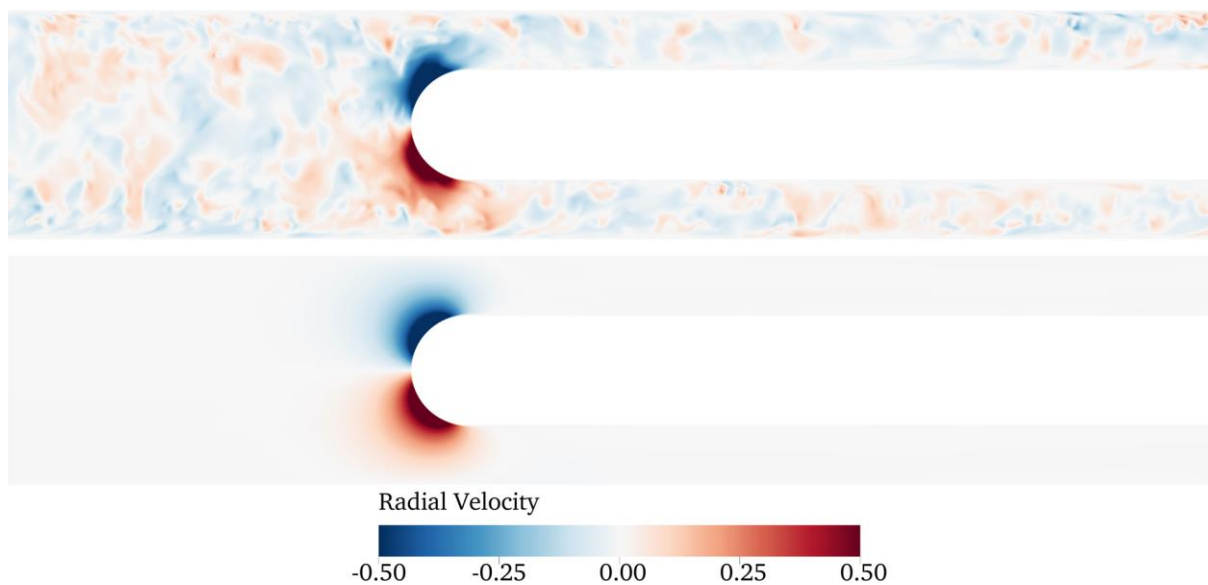


Figure 64. Rounded-end radial velocity: instantaneous (top) & time averaged (bottom).

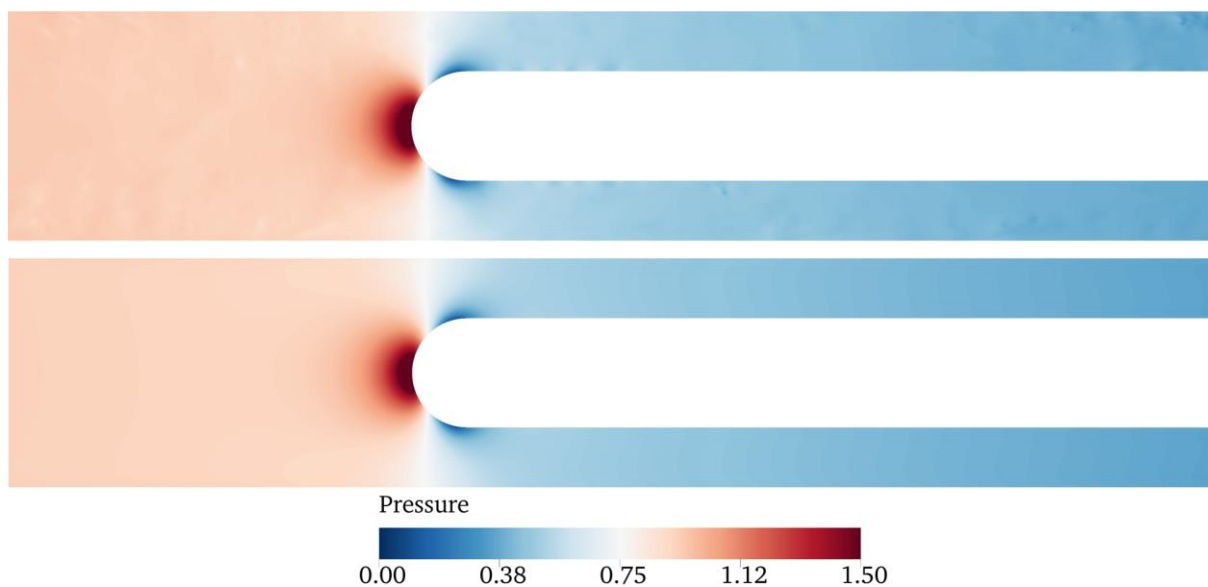


Figure 65. Rounded-end pressure: instantaneous (top) & time averaged (bottom).

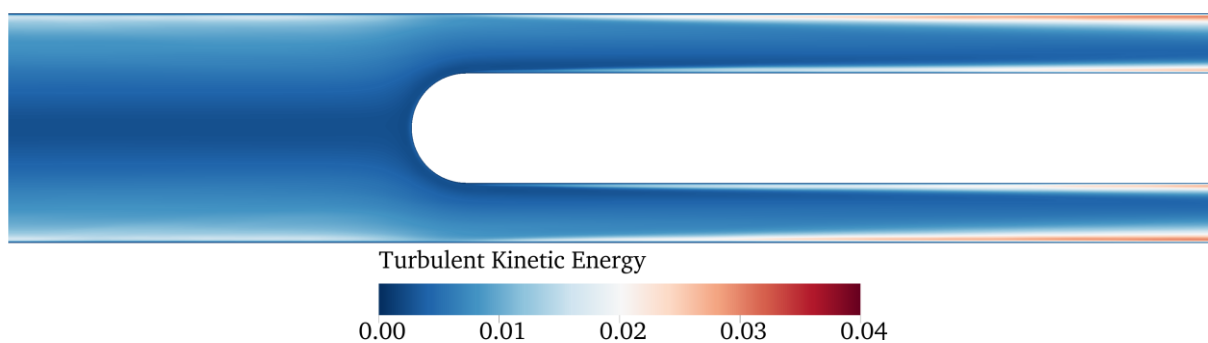


Figure 66. Rounded-end TKE.

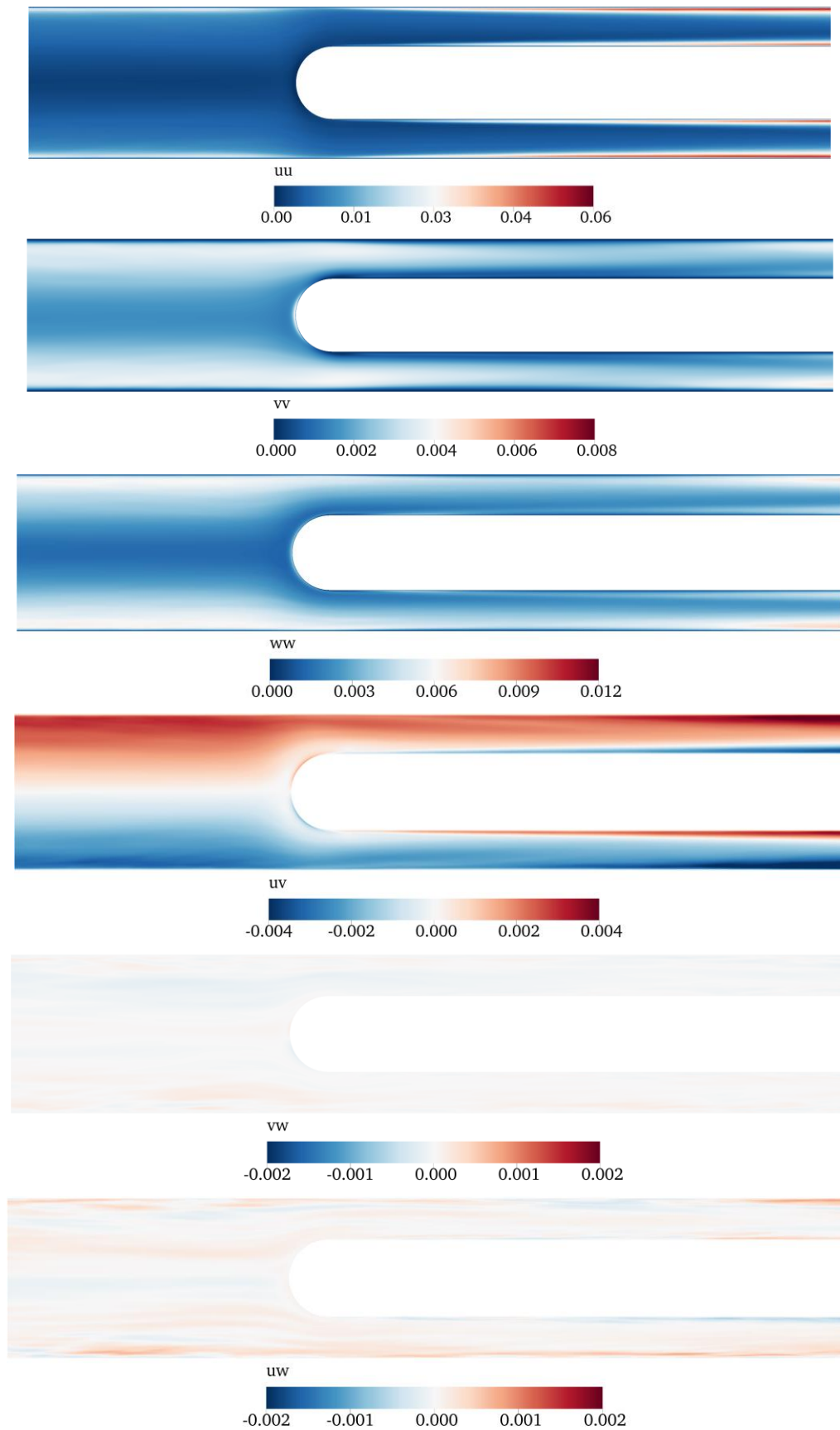


Figure 67. Rounded-end Reynolds stresses.

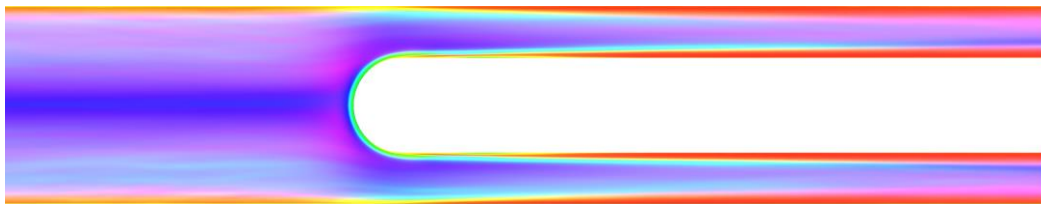


Figure 68. Rounded-end turbulence anisotropy.

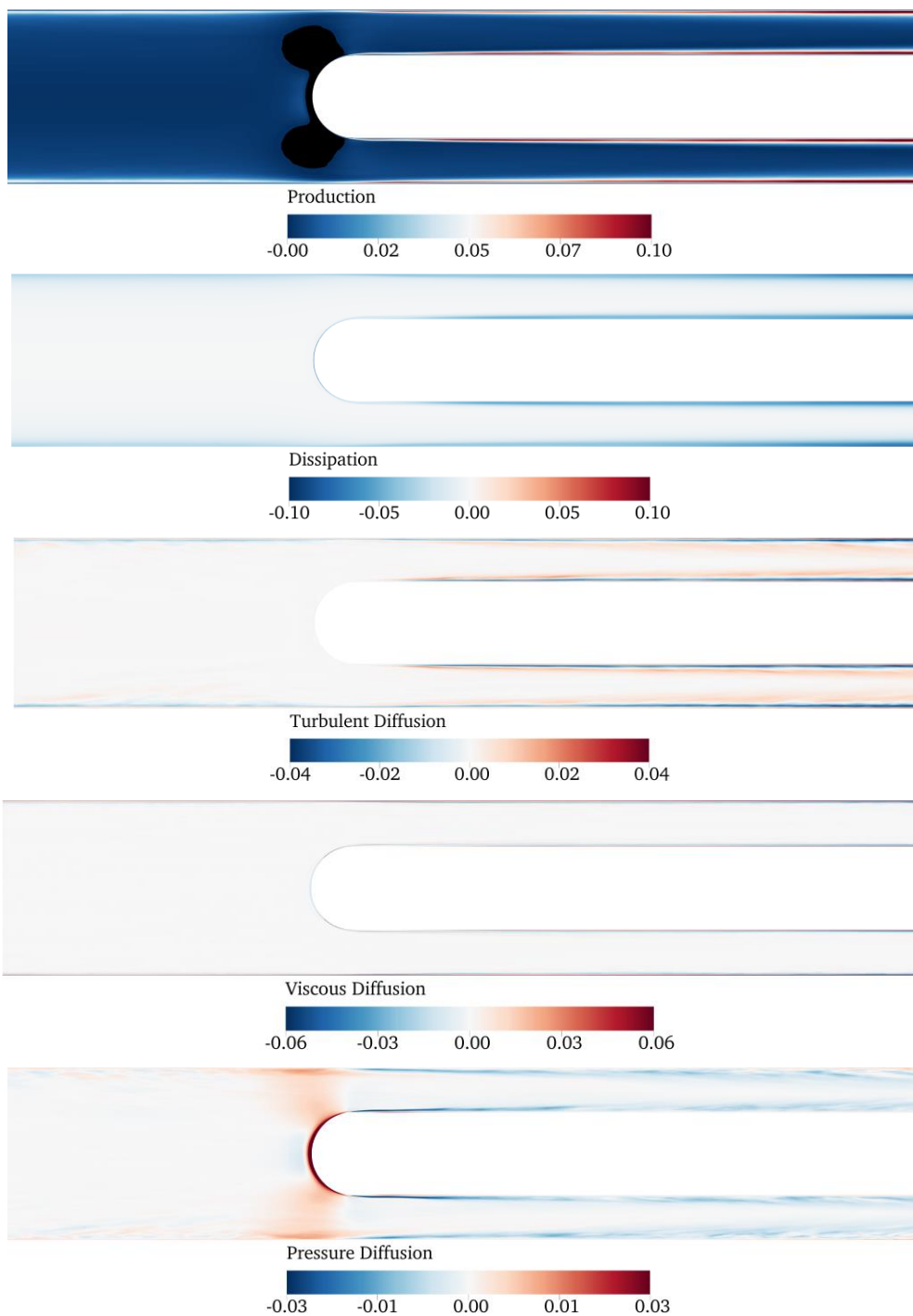


Figure 69. Rounded-end TKE budget terms. Negative production region indicated in black.

5.3.1.2 Q-Criterion

Figure 70 & Figure 71 show the computed Q-criterion for the rounded-end case. The structures are observed to stretch and dissipate in the contraction zone supporting the suppression of axial normal stresses and local partial laminarization. Additionally, a video is provided (Bistany, 2024).

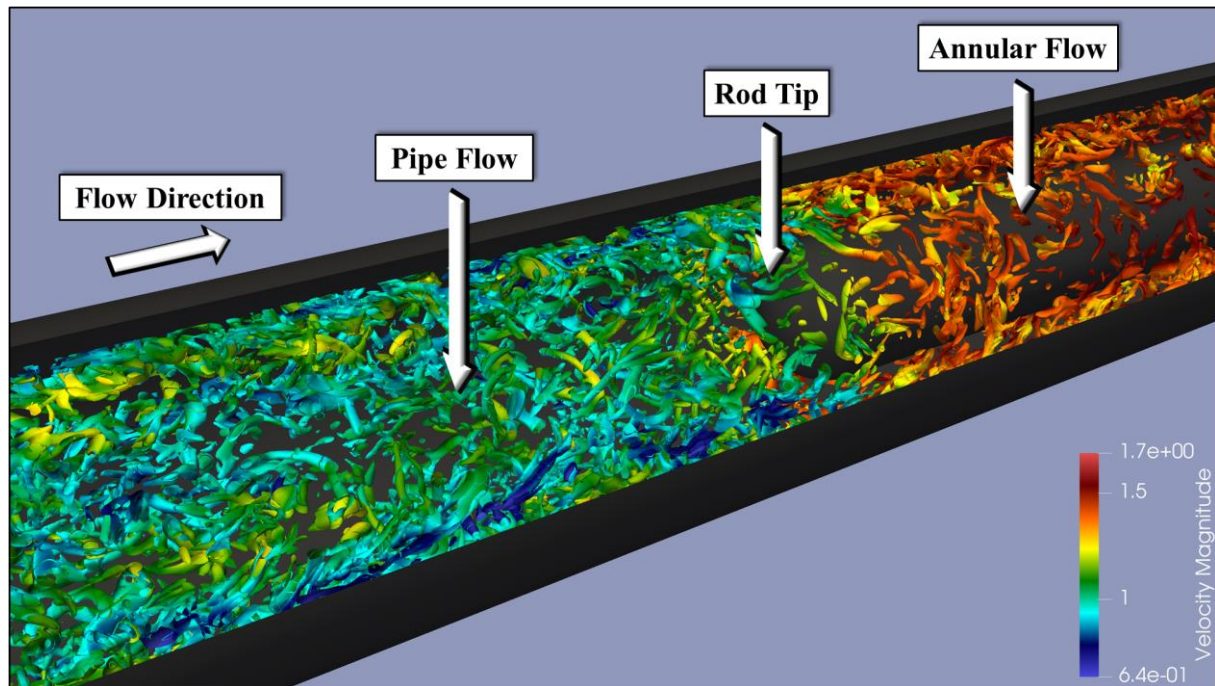


Figure 70. Q-criterion revealing coherent structures.

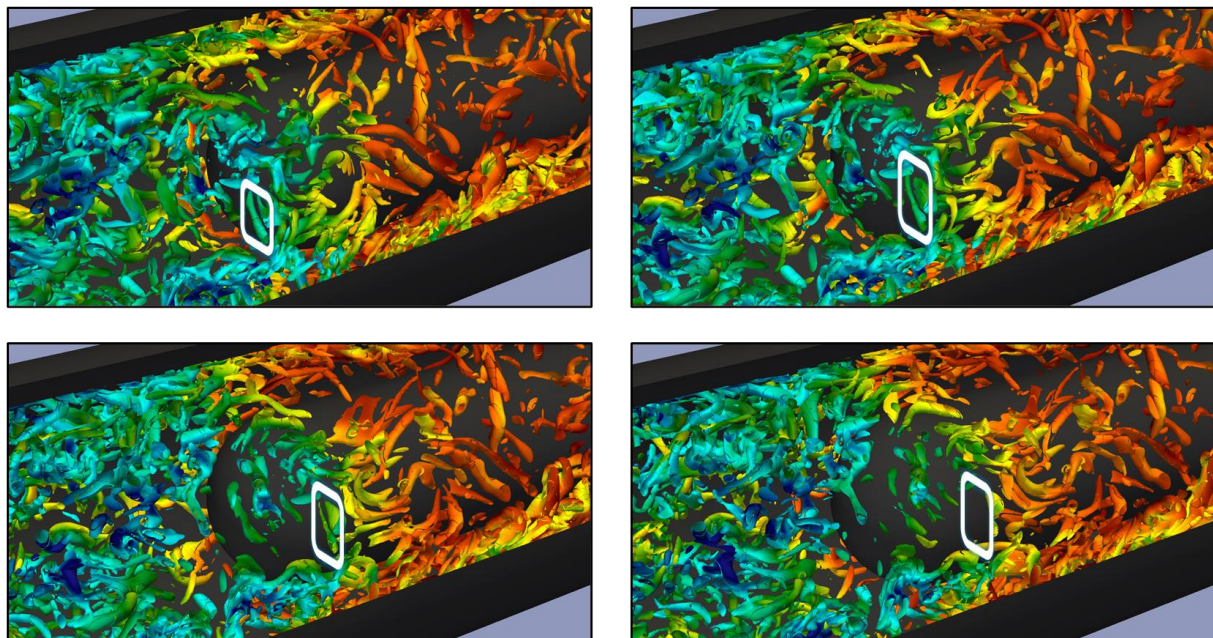


Figure 71. Comparison of Q-criterion isosurfaces at four different times, T , tracking the stretching and decay of a turbulent structure in the contraction region. T_1 (top left), T_2 (top right), T_3 (bottom left), & T_4 (bottom right).

5.3.1.3 Turbulence Anisotropy

Figure 72 shows the entire domain using the Barycentric colormap to highlight the axial evolution of turbulence anisotropy along the rounded-end rod. Just downstream of the contraction (P1–P4, Figure 73), anisotropy shifts leftward on the Barycentric map toward two-component turbulence along the rod wall, where partial laminarization and vortex stretching suppress TKE and axial fluctuations before gradually returning to canonical behaviour farther downstream (P5–P8, Figure 73).

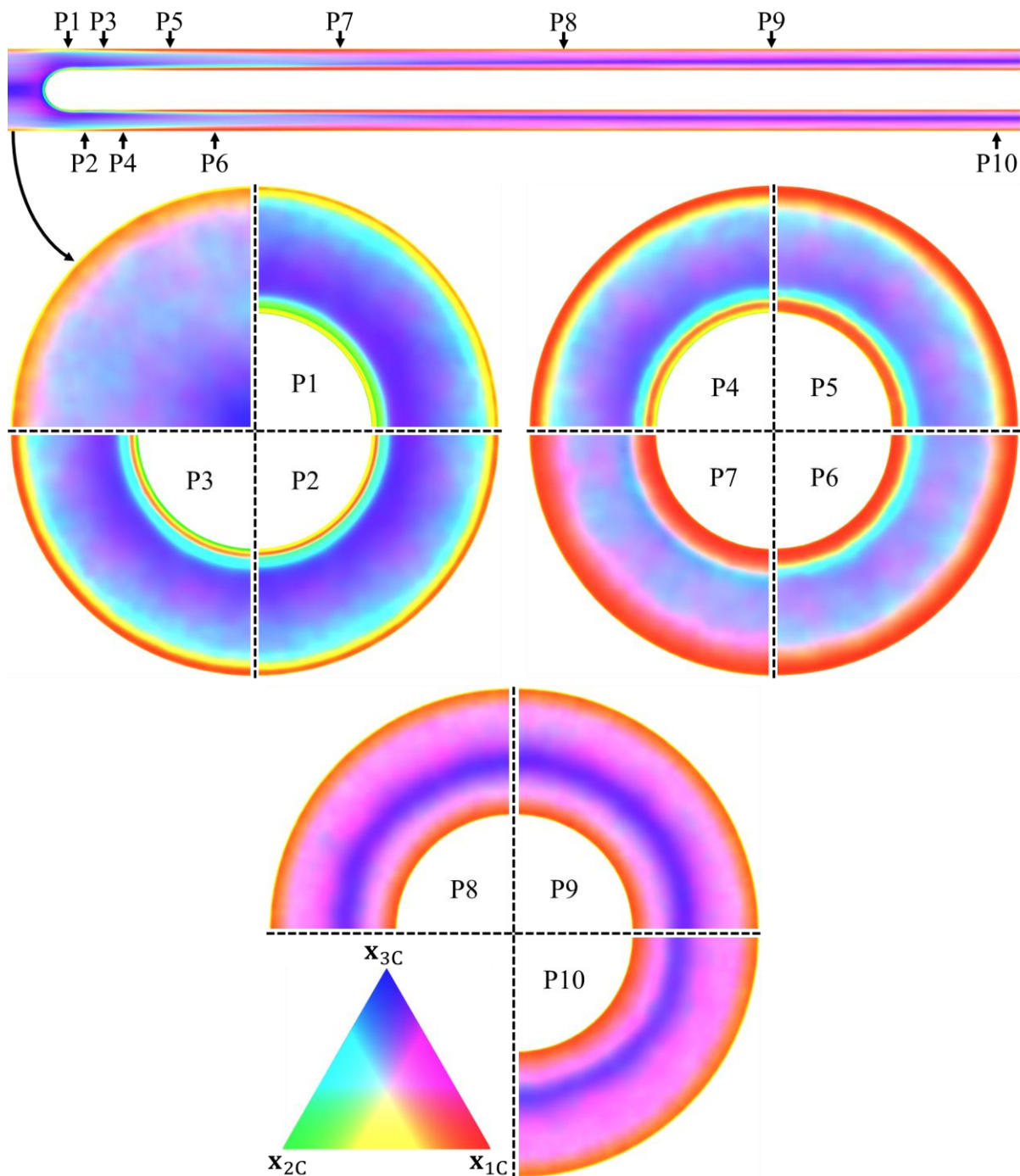


Figure 72. Axial evolution of rounded-end turbulence anisotropy using Barycentric color mapping.

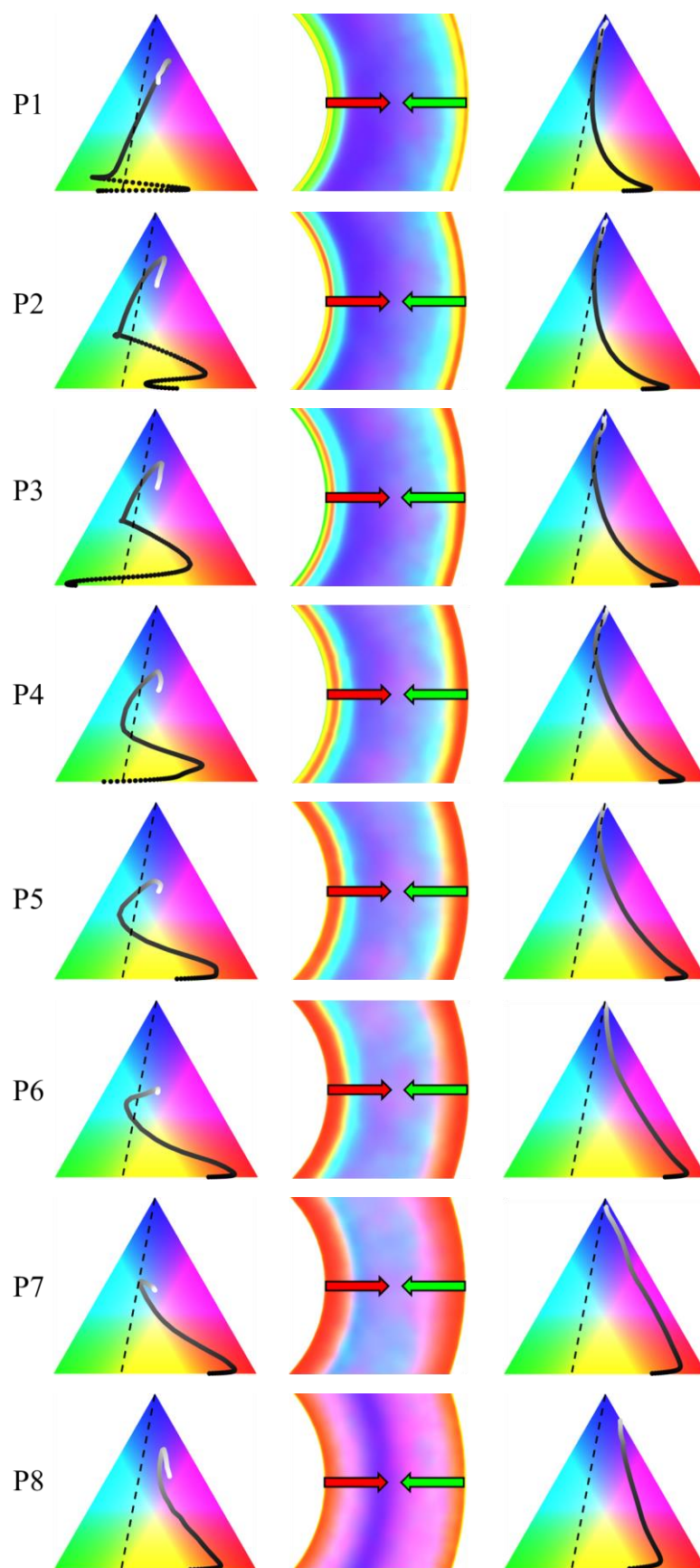


Figure 73. Rounded-end Barycentric triangle tracing at axial locations (black is wall). Red arrow is rod wall (left column) & green arrow is pipe wall (right column).

5.3.1.4 Turbulence Statistic Plots

Finally, for the rounded-end, the turbulence statistics, visually represented in section 5.3.1.1, are plotted in Figure 74 through Figure 78 showing their evolution in both axial and radial directions. For all plots, the origin is the rod tip without the rounded-end, see Table 17 & Table 18 for line ID details. For all plots showing the evolution of the radial profile with different axial locations, $\frac{y}{D_h} = 0$ is confining pipe wall extending to the rod wall. For all plots showing the evolution of the vertical profile with different radial locations, $\frac{z}{D_h} = 0$ is rod tip extending $z = 100$ mm downstream of rod tip.

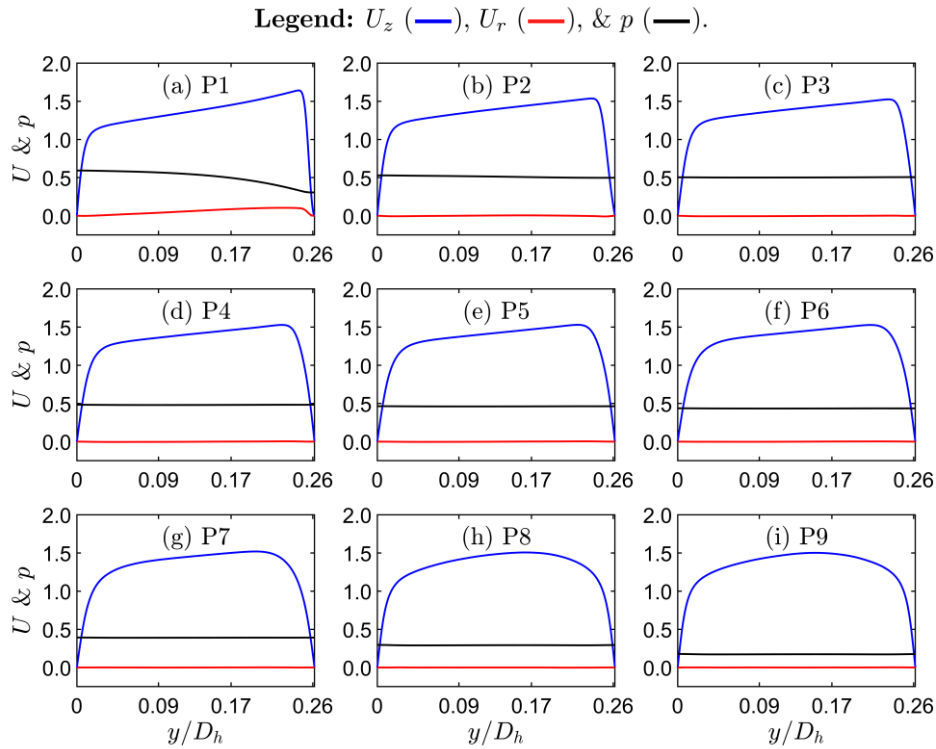


Figure 74. Axial evolution of Rounded-end velocity & pressure profiles.

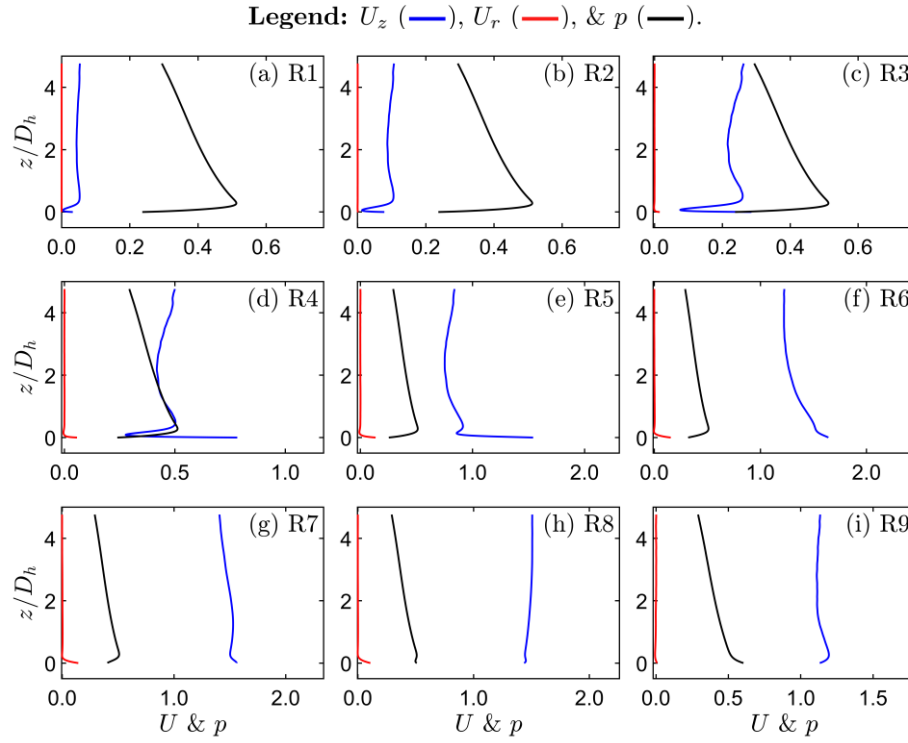


Figure 75. Radial evolution of rounded-end velocity & pressure profiles.

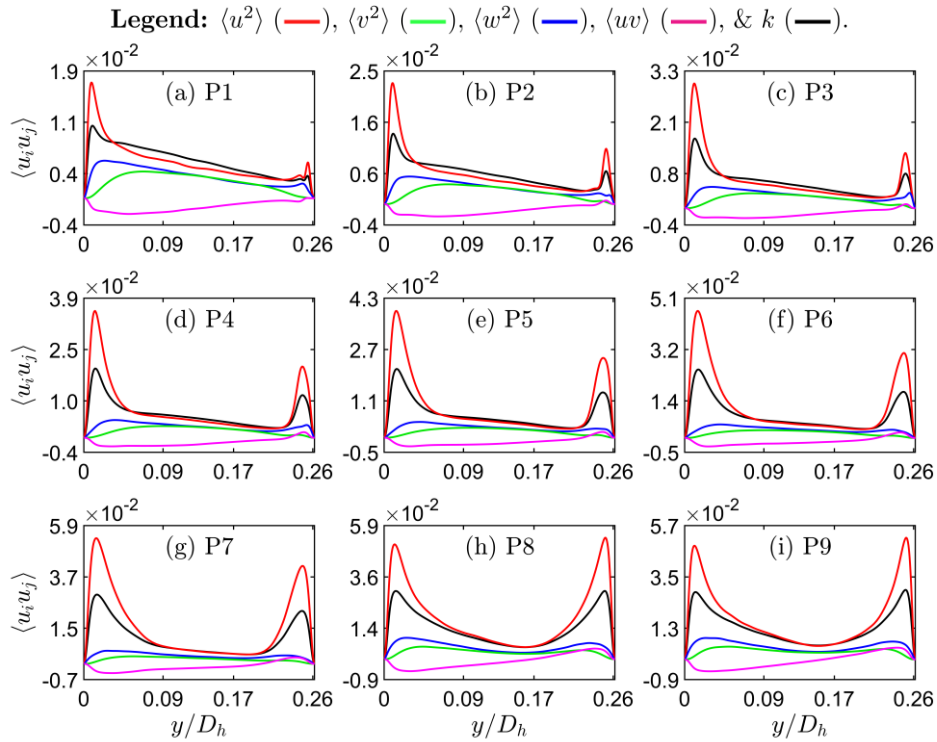


Figure 76. Axial evolution of rounded-end Reynolds stresses & TKE.

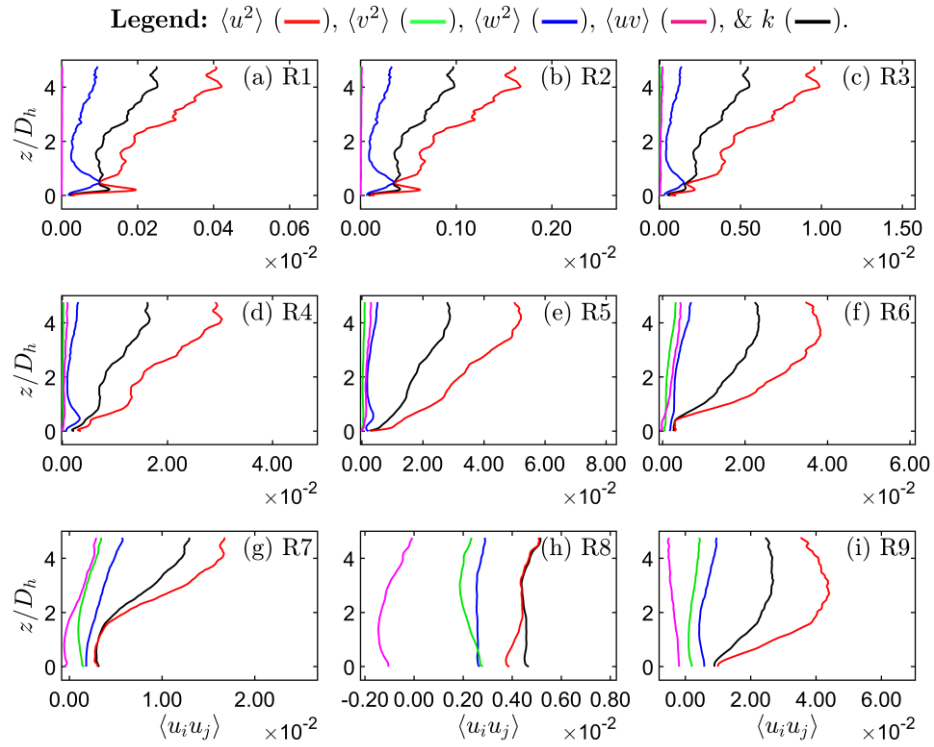


Figure 77. Radial evolution of rounded-end Reynolds stresses & TKE.

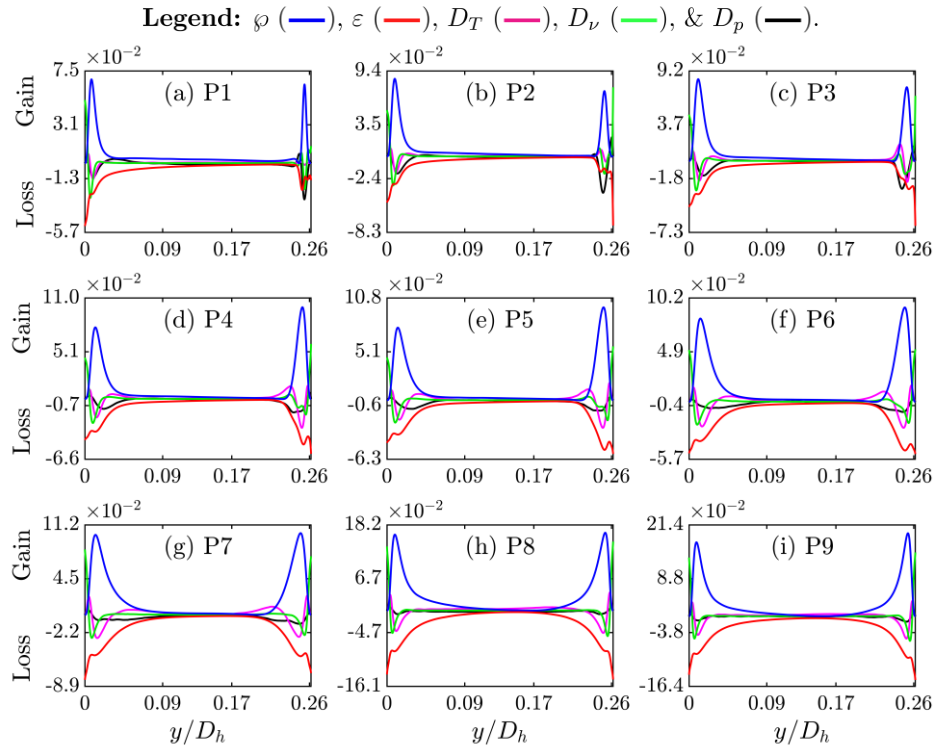


Figure 78. Axial evolution of rounded-end TKE budget terms.

5.3.2 Blunt-end

5.3.2.1 Axial Cross-Sectional Fields

Figure 79 through Figure 84 show two-dimensional cross-sections centered on the rod's rounded tip, highlighting both first- and second-order flow statistics. Each axial slice spans roughly five pipe diameters—about $1.5D_h$ upstream of the tip and $3.5D_h$ downstream. The blunt-end case mirrors the rounded-end analysis, except that the Q-criterion and TKE-budgets are not computed.

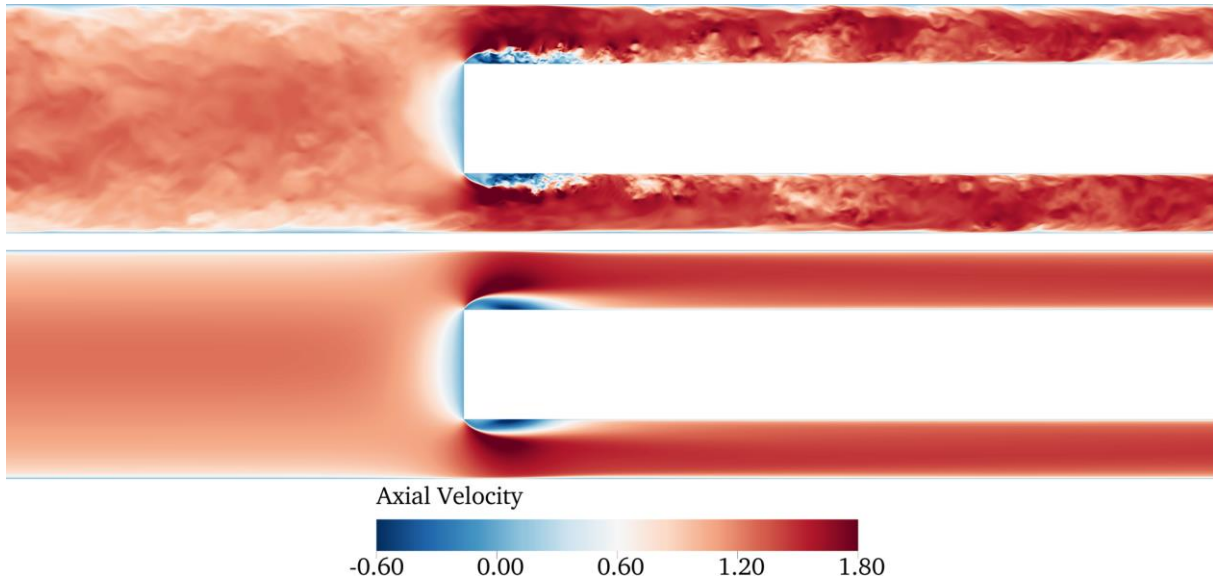


Figure 79. Blunt-end axial velocity: instantaneous (top) & time averaged (bottom).

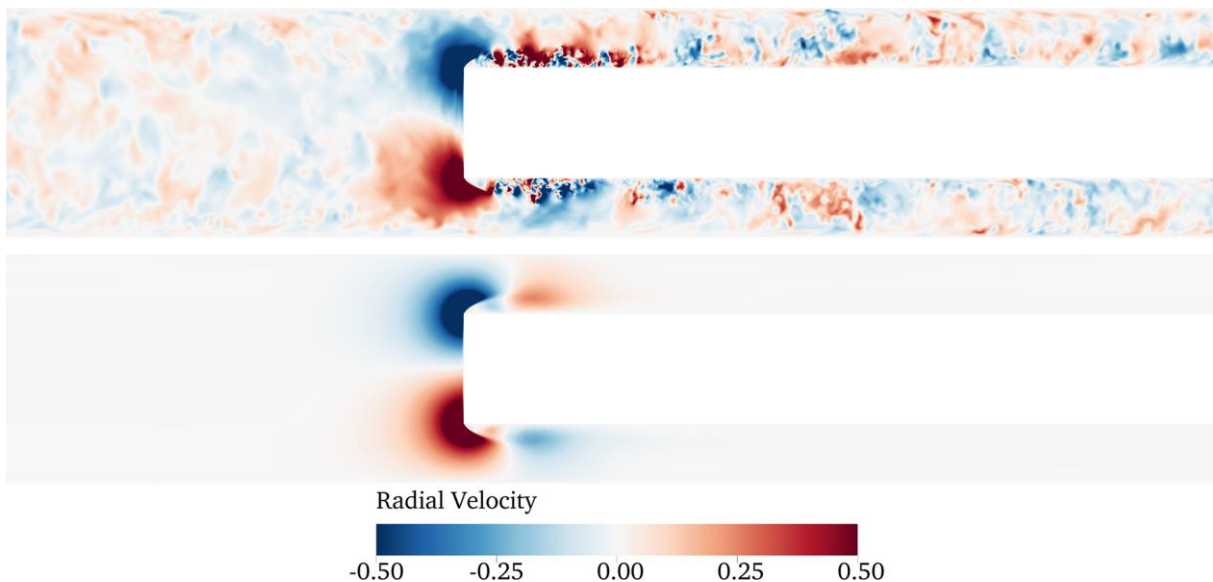


Figure 80. Blunt-end radial velocity: instantaneous (top) & time averaged (bottom).

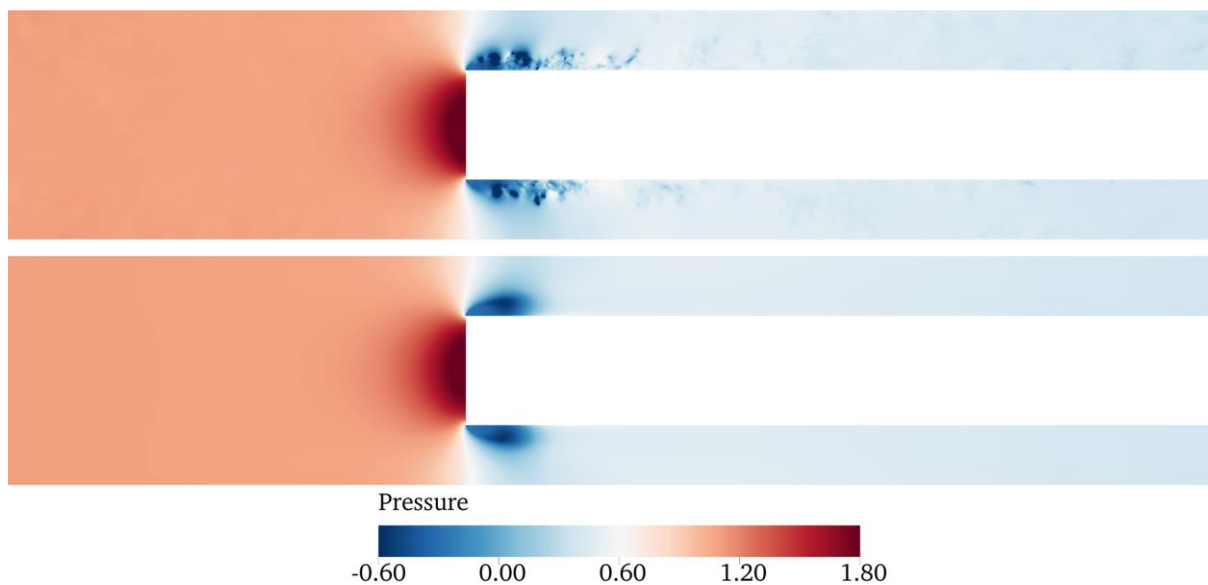


Figure 81. Blunt-end pressure: instantaneous (top) & time averaged (bottom).

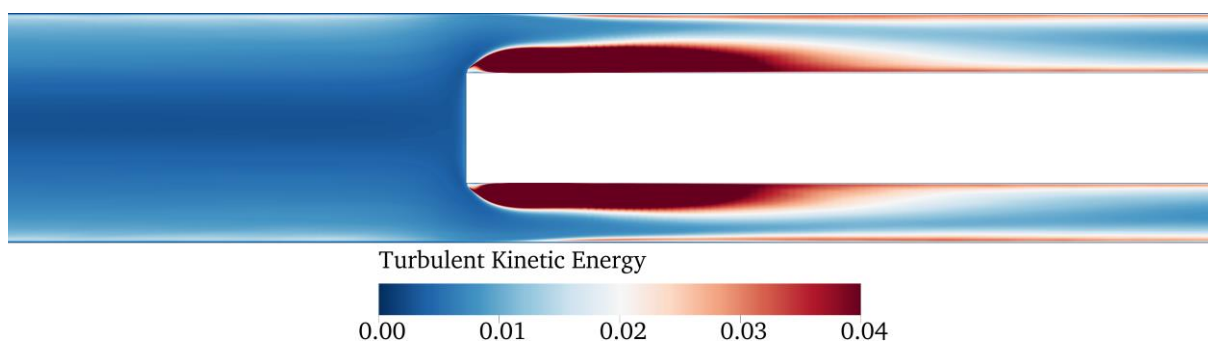
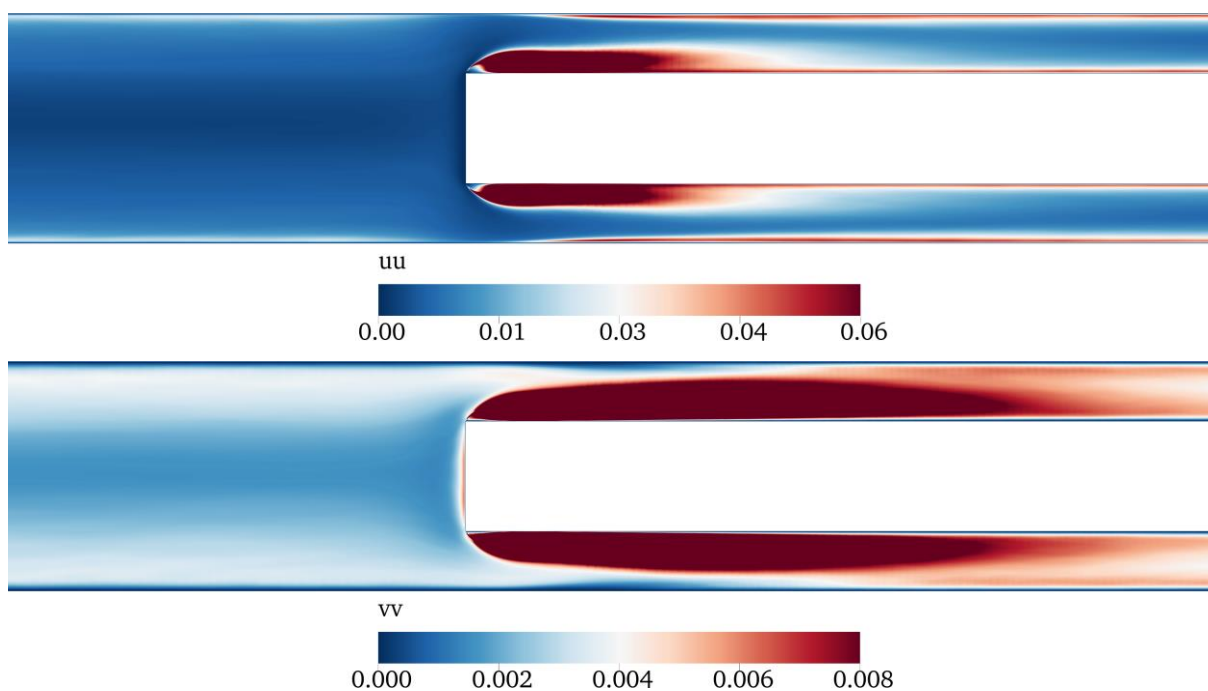


Figure 82. Blunt-end TKE.



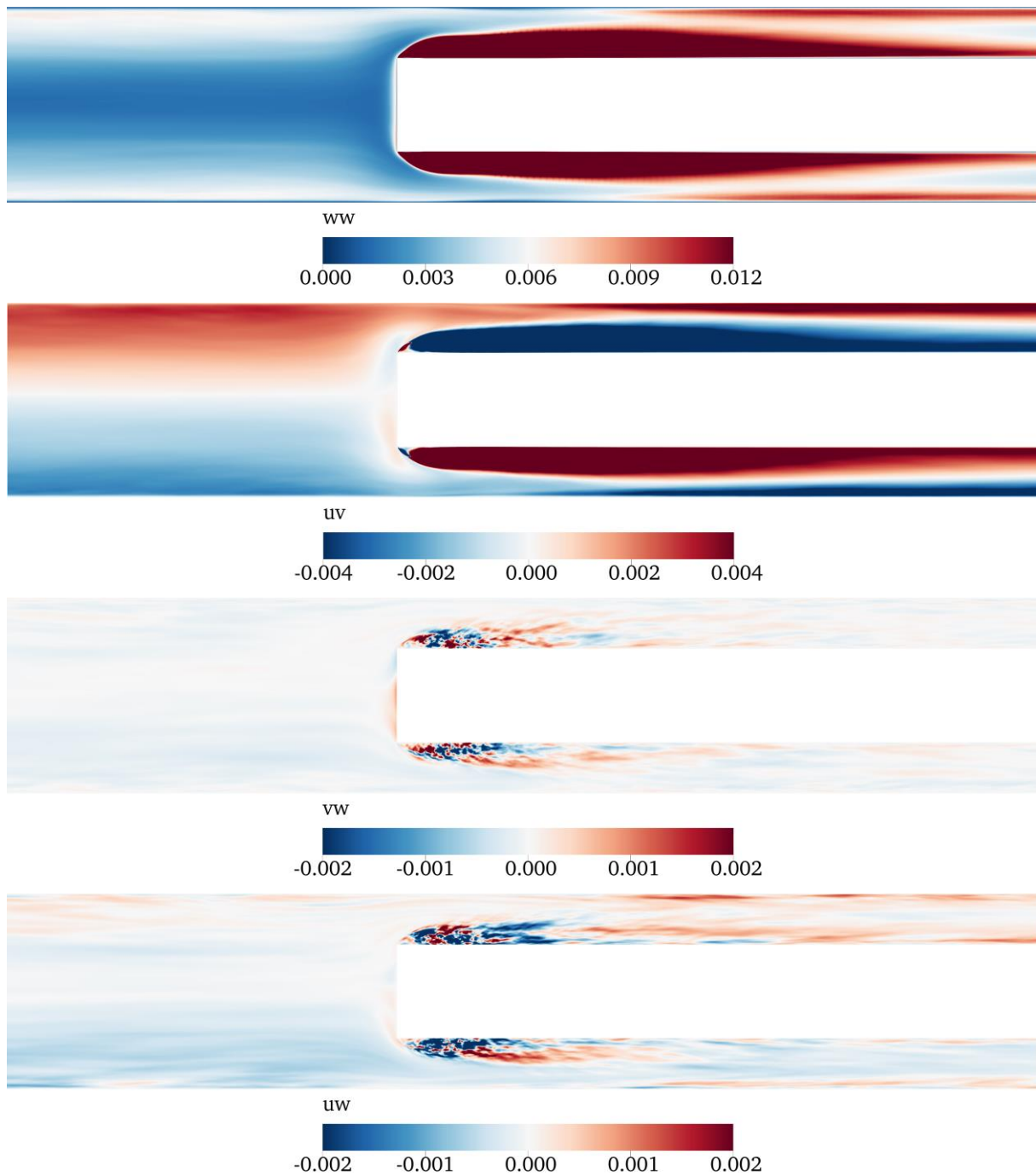


Figure 83. Blunt-end Reynolds stresses.

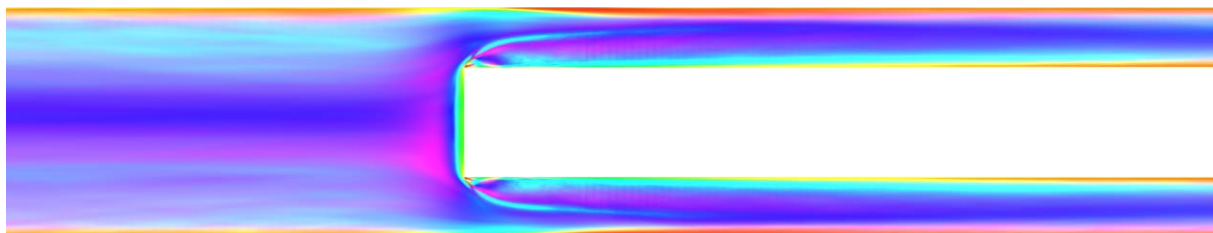


Figure 84. Blunt-end turbulence anisotropy.

5.3.2.2 Turbulence Anisotropy

Shown in Figure 85, turbulence anisotropy near the blunt-end tip differs significantly from that at the rounded-end, owing to the sharp geometric contraction of the blunt-end versus the gradual contraction of the rounded-end. Figure 86 emphasizes this difference by tracing the Barycentric triangles at the same locations as in Figure 73. These changes stem from the large recirculation region and flow separation at the tip—features absent in the rounded-end.

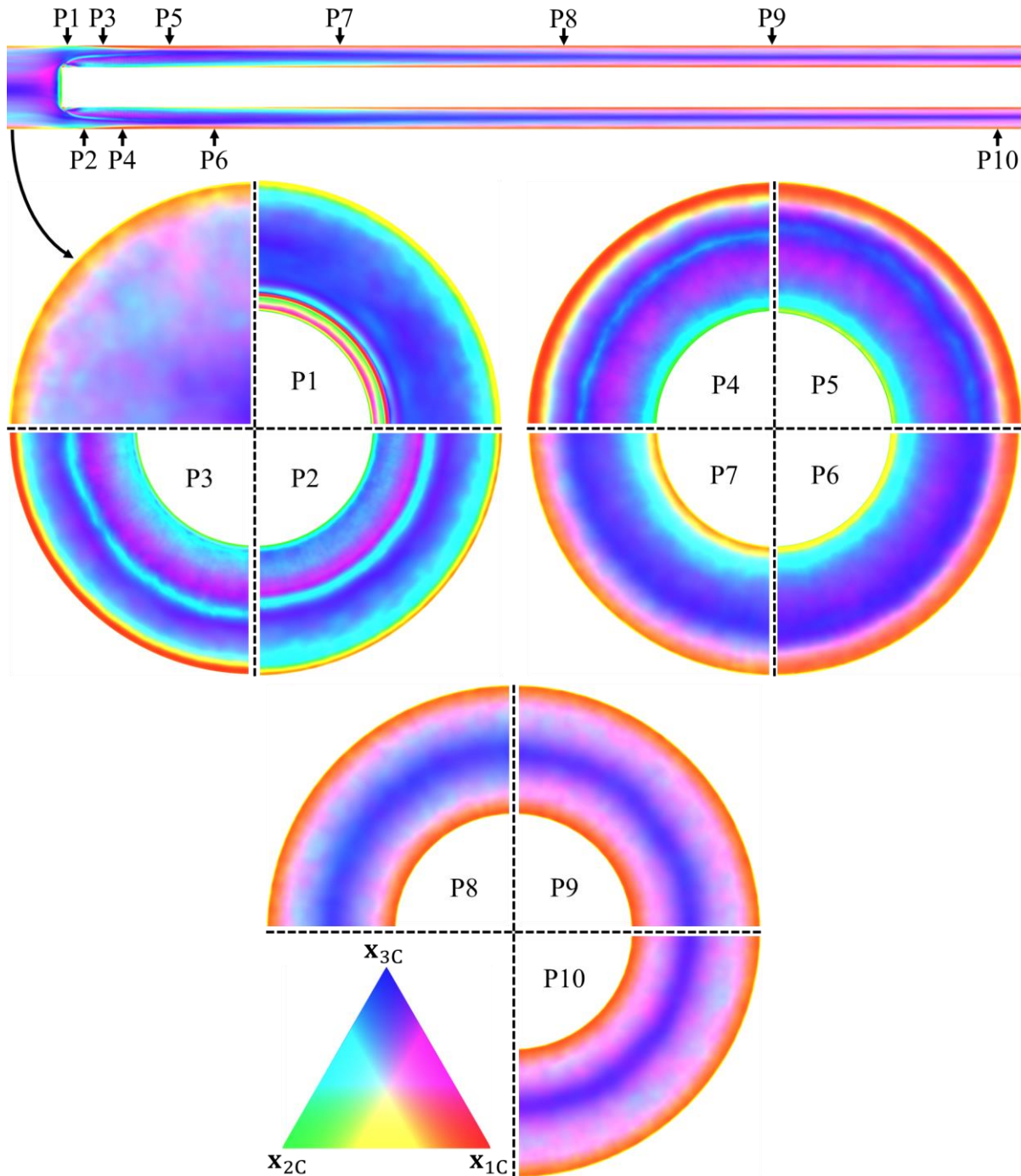


Figure 85. Axial evolution of blunt-end turbulence anisotropy using Barycentric color mapping.

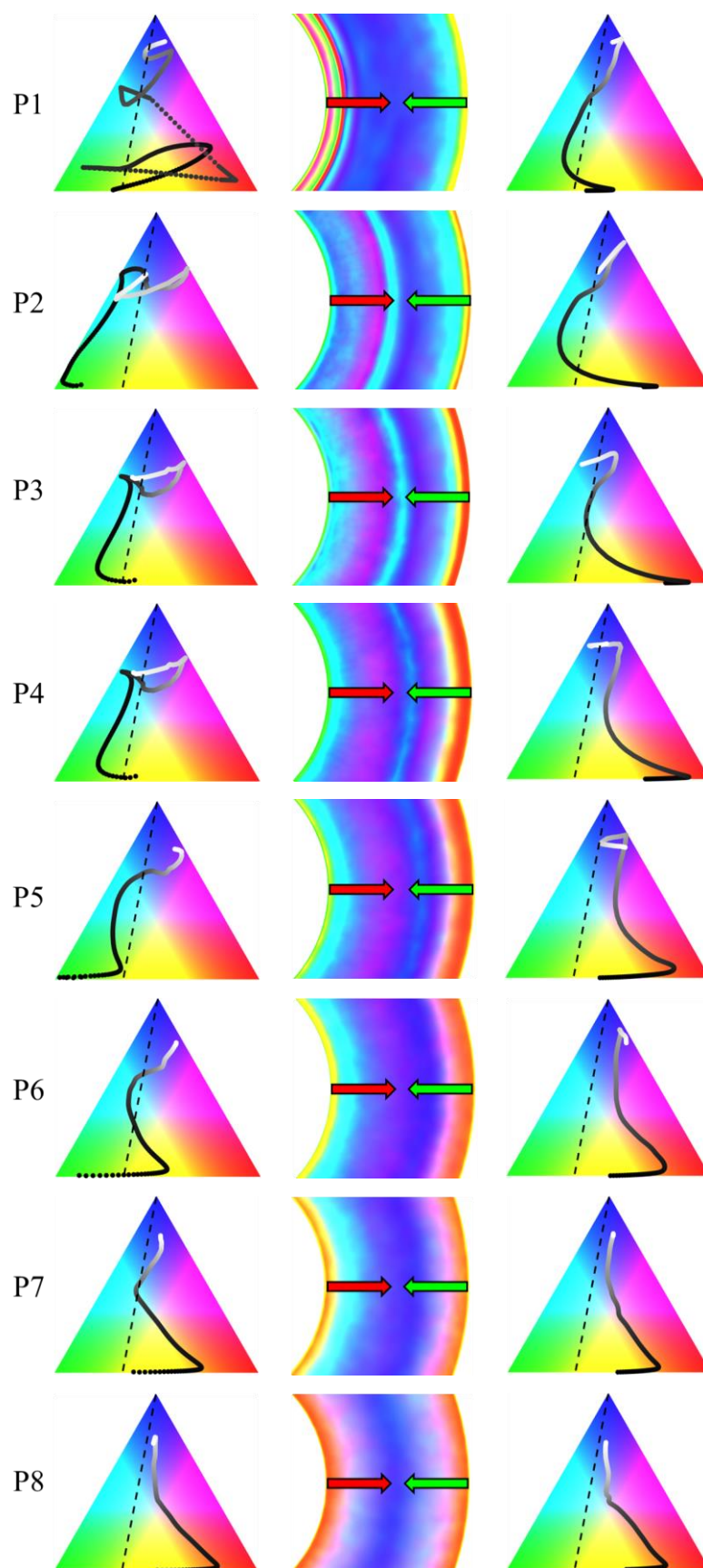


Figure 86. Blunt-end Barycentric triangle tracing at axial locations (black is wall). Red arrow is rod wall (left column) & green arrow is pipe wall (right column).

5.3.2.3 Turbulence Statistic Plots

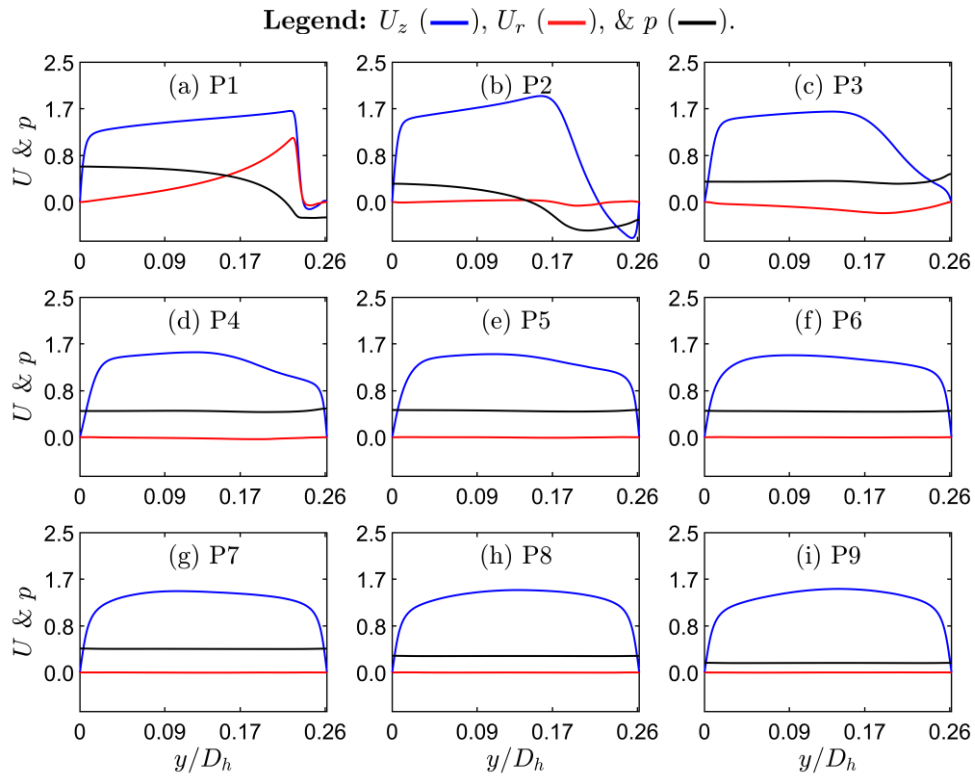


Figure 87. Axial evolution of Blunt-end velocity & pressure profiles.

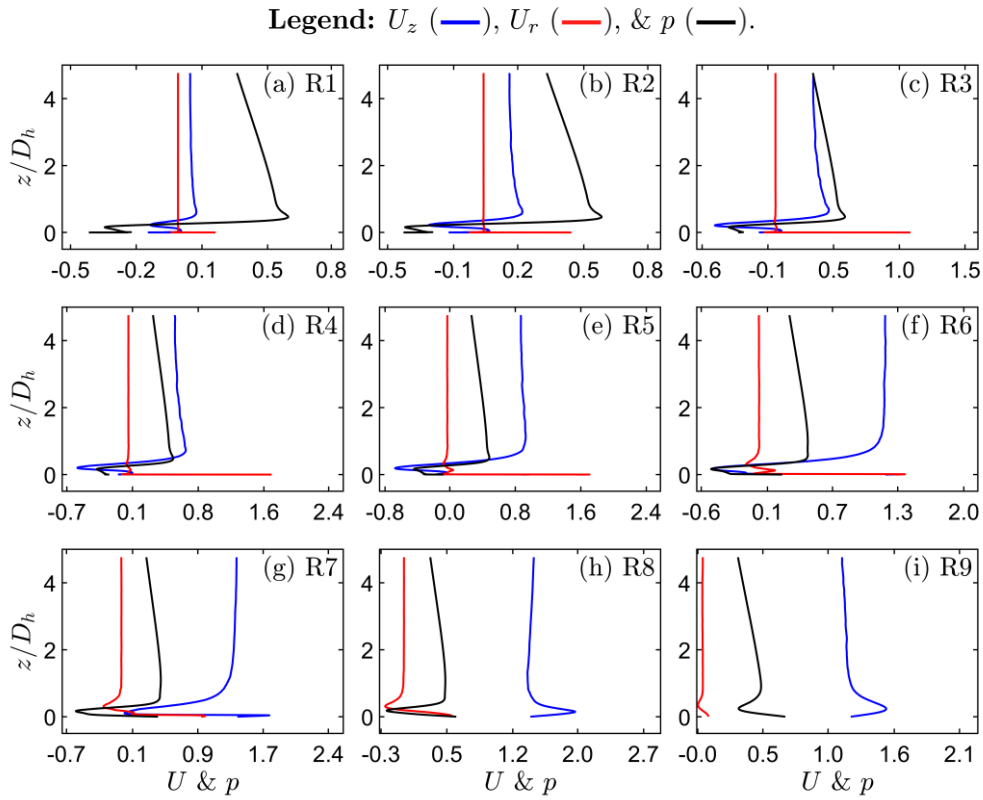


Figure 88. Radial evolution of blunt-end velocity & pressure profiles.

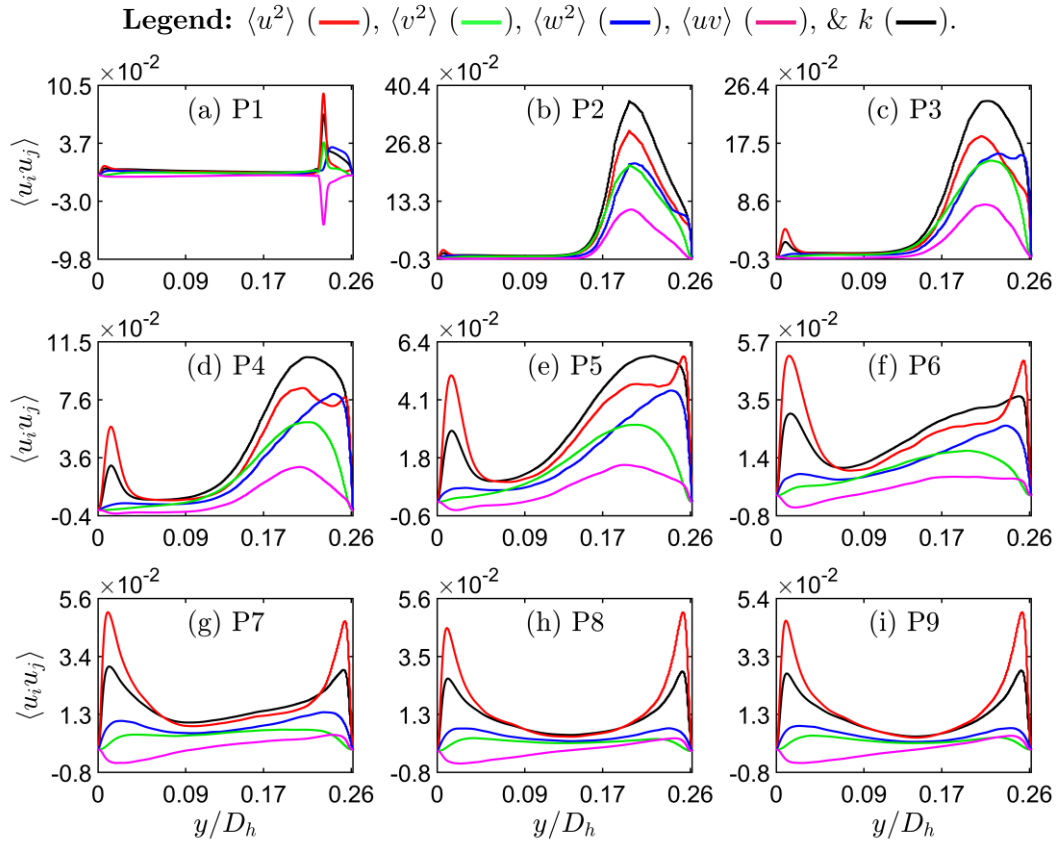


Figure 89. Axial evolution of blunt-end Reynolds stresses & TKE.

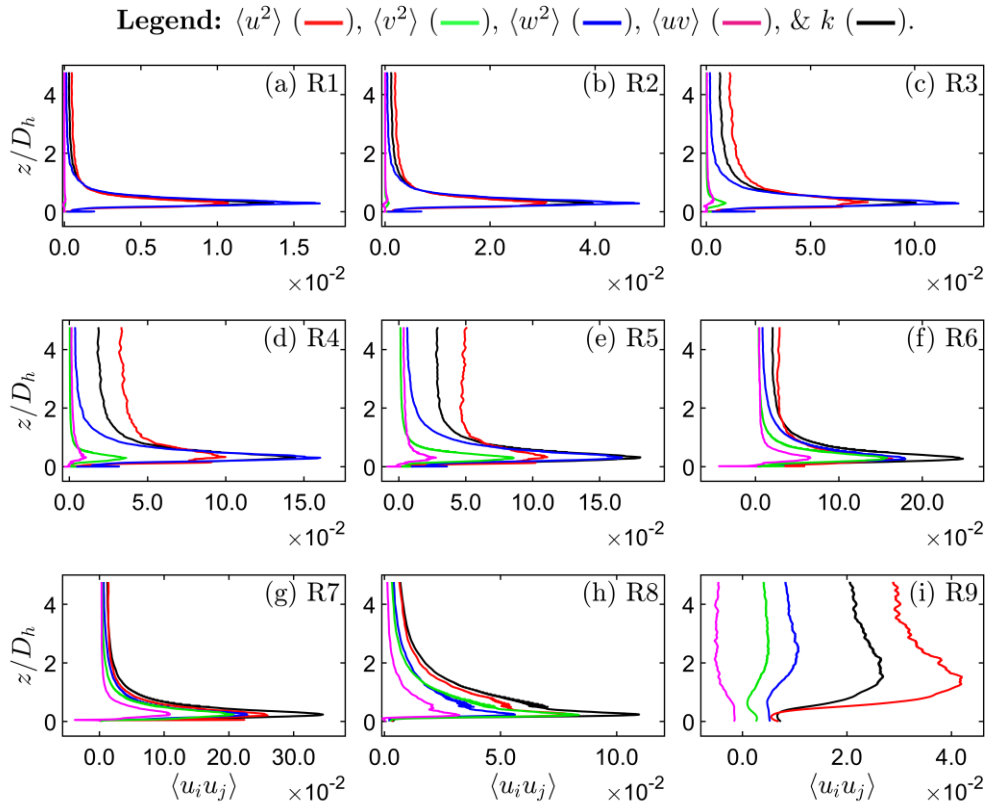


Figure 90. Radial evolution of blunt-end Reynolds stresses & TKE.

5.3.3 Wall Shear Stress Comparison

The wall shear stress along the rod wall is presented in Figure 91 directly comparing the rounded-end and blunt-end cases. The wall shear stabilizes five diameters downstream of the tip for both cases. However, due to the different end configuration geometries having substantial effects on the flow, the wall shear behaviour is dramatically different near the rod tip. The negative TKE production around the rounded tip, leading to partial local laminarization of the flow, results in the near zero wall shear stress at the tip. The large negative wall shear near the blunt-end tip indicates flow reversal, characteristic of the flow separation resulting in the recirculation zone.

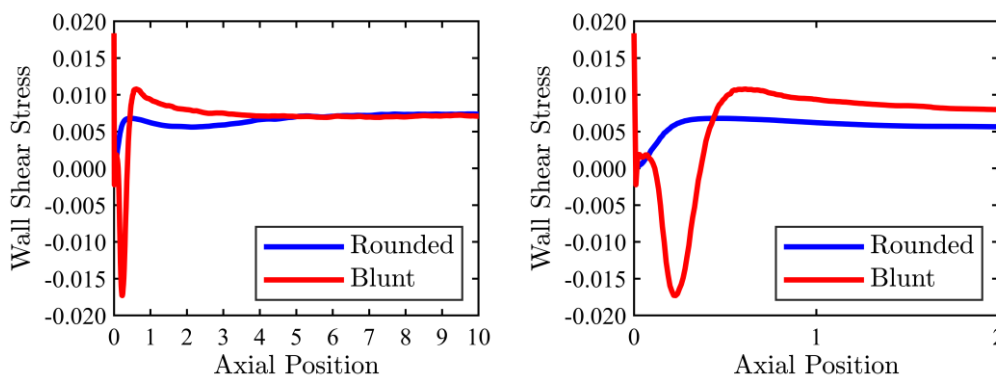


Figure 91. Dimensionless wall shear stress along entire rod wall (left) and zoomed in near free end (right). Axial position zero is rod tip without rounded-end.

5.4 Conclusions and Future Work

This study presents DNS results for turbulent flow around a free-clamped cantilevered rod in two fluid-only configurations: a rounded-end case at an annulus Reynolds number of 11,104 and a blunt-end case at 12,220. The simulations were first validated and verified with canonical pipe flow and showed excellent agreement. Then, many experimental comparisons were conducted to $Re = 21.2k$ and $23.33k$ for the curved and blunt end respectively, showing good agreement for first-order statistics at some locations and decent agreement with Reynolds stress profile shapes. The simulation was rerun for the rounded-end case at $Re=21.2k$ and was used for the benchmark comparison as presented in chapter 4.

The Q-criterion exposes coherent vortical structures and their distortion as the flow contracts from the pipe to the annulus—behavior consistent with literature on contracting flows—and zones of negative TKE production in the rounded-end case result in a region of partial local laminarization. The TKE-budget analysis further shows pronounced shifts in production and dissipation terms along the rod, with elevated pressure diffusion tied to strong pressure gradients that peak near the contraction and rod tip; the diffusing turbulent pressure field there generates the random pressure fluctuations that buffet the rod and drive FIV.

For both cases, two-dimensional cross-section fields are presented with a focus near the rod tip. They visually display the different end configuration effects on turbulence statistics and are further quantified by the turbulence statistic plots for vertical and radial locations along the rod wall. The evolution of statistics is remarkably different near the rod tip and for a

considerable distance along the rod wall, different for each case, before returning to expected annular flow statistic behaviors. These statistics provide high fidelity DNS data for benchmarking and validating future, lower costing, turbulence modeling tools.

The turbulent anisotropy differs sharply between the two contraction geometries at the rod tip, highlighting the effects of each on the flow as it contracts from pipe flow to annular flow. Future work will incorporate a cone-ended configuration—potentially evaluated at several cone heights—to extend the anisotropy analysis across all three geometries. The current lack of second-order tensor visualization tools in CFD underscores the importance of such analyses.

6. Simulations performed by EDF

6.1 Introduction

In the context of nuclear engineering, turbulence induced vibration of fuel rods can lead to mechanical wear, which can be responsible for safety issues and significant maintenance costs in Nuclear Power Plants (NPPs). The GO-VIKING European project aims at developing Fluid Structure Interaction (FSI) methods on simpler geometries. Both experimental and reference numerical data are used to develop engineering tools and assess industrial numerical models. In the context of WP2, a benchmark has been set up on a cantilever beam configuration, in a confined tube and hydraulic diameter representative of a fuel rod. The main objective is to compare the results of high fidelity and medium resolution approaches with the available experimental data. A moderate Reynolds number piece of experiment is chosen, such that the high fidelity simulations are affordable. The outputs of the high-resolution approaches will be used to feed simpler models.

Hereby, two-way coupling simulations, without inner iterations, of a cantilever rod under turbulent axial flow have been performed at Reynolds number 21 200. In the context of the vibration impact in nuclear power generation (VIKING) consortium (Zwijssen & al., 2022), previous simulations with a fixed rod and a fluid solver alone were performed at higher Reynolds number ($Re_H = 58\,200$). The interested reader may refer to (Norddine & Benhamadouche, 2023).

6.1.1 Executive Summary of the Current Study

CFD with finite volume method and wall-resolved large eddy simulation (WR-LES) turbulence model is used for the flow, along with an in-house Euler-Bernoulli beam equation solver for the rod. Free vibration tests have been performed for the calibration of the mechanical damping of the beam and validation purposes. The beam motion is accounted for using an Arbitrary Lagrangian Eulerian (ALE) method. The meshes are made of 144 million hexahedra for the flow and 3800 grid points for the beam, respectively. The beam motion and the flow statistics are both investigated; comparisons are drawn with CFD simulations and a fixed rod in central position as well as experimental results from the literature. The lowest theoretical and experimental natural frequencies are recovered by the simulation, as well as the shape of the power spectral density (PSD) of the tip displacement. However, the amplitude of the displacement and its root mean square (RMS) is smaller in the simulation than in the experiment. This might be due to a misalignment of the rod in the experiment. Then, the flow statistics are compared to the PIV results. Apart from the asymmetry in the PIV profiles, there is a good agreement for the mean velocity field and the turbulent kinetic energy. Additional simulations without FSI and a fixed in central position were performed for comparison. The flow statistics obtained with FSI and a fixed rod in central position are very similar. Therefore, the rod vibration has a very small effect on the flow.

6.2 Physics and Numerical Modelling

6.2.1 Euler-Bernoulli beam equation theory



The solid rod is modelled as an Euler-Bernoulli beam (valid for slender objects and small displacements). The Euler-Bernoulli beam theory is briefly recalled in this section, with several analytical results. The natural frequencies of a cantilever beam, with cantilevered Boundary Conditions (BCs) at one side and free BC at the other, are:

$$f_n = \beta_n^2 \sqrt{\frac{EI}{\rho_l + \rho_{add}}},$$

with ρ_l the solid lineic mass and ρ_{add} the lineic added mass of the beam induced by the displacement of the surrounding fluid. The added mass for a beam in a cylinder is estimated as follows (Blevins, 2001):

$$\rho_{add} = \frac{\pi}{4} \rho D_o^2 \frac{1 + \left(\frac{D_o}{D_c}\right)^2}{1 - \left(\frac{D_o}{D_c}\right)^2},$$

where ρ denotes the fluid density, $D_o = 10.01mm$ is the rod outer diameter and $D_c = 20.86mm$ is the confining pipe inner diameter. The β_n coefficients are the solutions of the following equation:

$$\cosh(\beta_n L) \cos(\beta_n L) + 1 = 0.$$

The first four solutions are: $\beta_1 = 0.596864 \frac{\pi}{L}$, $\beta_2 = 1.49418 \frac{\pi}{L}$, $\beta_3 = 2.50025 \frac{\pi}{L}$, $\beta_4 = 3.49999 \frac{\pi}{L}$. In the present configuration, $E = 1.93 \times 10^{11} Pa$, $I = 1.94 \times 10^{-10} m^4$, $\rho_l = 0.588 Kg \cdot m^{-1}$. Hence, the four first natural vibration frequencies are: $f_1 = 3.714 Hz$, $f_2 = 25.63 Hz$, $f_3 = 71.75 Hz$, $f_4 = 140.6 Hz$. The mode shapes of the displacement of the cantilever beam are:

$$w_i = A \left[\cosh(\beta_i x) - \cos(\beta_i x) + \frac{\cosh(\beta_i L) + \cos(\beta_i L)}{\sin(\beta_i L) + \sinh(\beta_i L)} (\sin(\beta_i x) - \sinh(\beta_i x)) \right]$$

6.2.2 Numerical model

The calculations are performed with the version 8.0 of code_saturne (Archambeau, et al., 2004), an open source collocated finite volume CFD code developed at EDF R&D. The fluid solver has been coupled with an in-house Euler-Bernoulli beam equation solver. The beam motion is accounted for explicitly in the fluid domain, at every time step, through an ALE method which allows to displace the CFD mesh during the simulation. The coupling is explicit and without inner iterations.

6.2.2.1 Fluid model

Incompressible WR-LES is performed with the Smagorinsky subgrid scale turbulence model (and Van-Driest wall-damping function). The fluid is liquid water at $T = 18.2^\circ C$. Constant density and dynamic viscosities values are considered in the present simulations, respectively equal to $\rho = 997.567 kg \cdot m^{-3}$ and $\mu = 1.047133 \times 10^{-3} Pa \cdot s$. The values were obtained

with the Cathare EOS library (CEA, 2004). The corresponding kinematic viscosity is $\nu = 1.05 \times 10^{-3} \text{m}^2 \cdot \text{s}^{-1}$. The meshes are conformal and made of structured hexahedra. They are generated with the open-source meshing software GMSH 4.11.1 (Geuzaine & Remacle, 2009). The surface mesh of the curved end-piece is provided in Figure 92 and cut views of the fluid mesh are given in Figure 93.

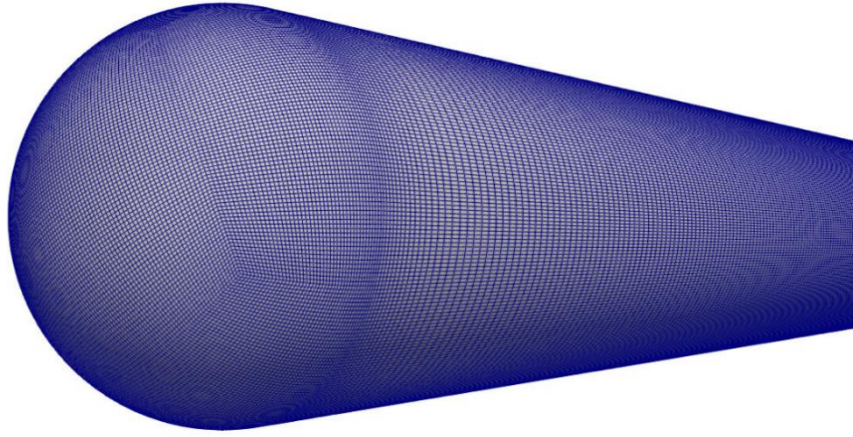


Figure 92. Surface mesh of the curved end-piece of the rod.

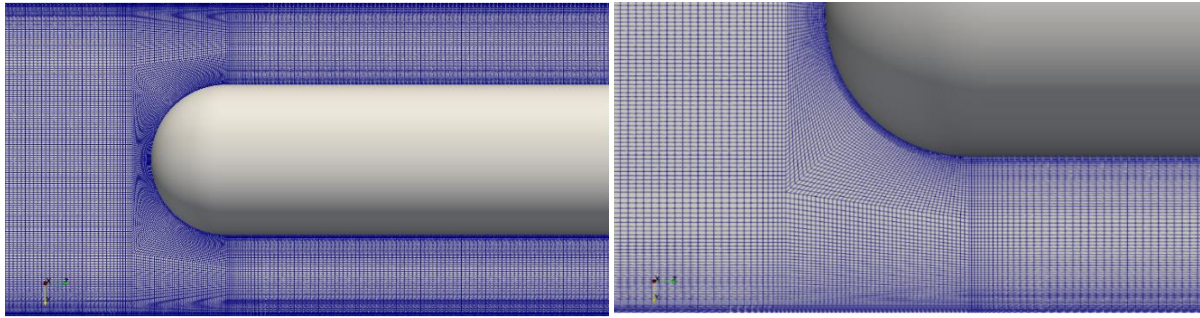


Figure 93. Cut view of the fluid mesh at the tip of the rod (left) and zoom on the end-piece (right).

A cut view of the full simulation domain is provided in Figure 94. The full-length beam is represented. The union cross at the outlet of the domain has been discarded in the present simulations. We assumed that the local effects closed to the cantilevered side of the beam does not have much effect on its motion. The fluid upstream of the end-piece has been represented on a length $L_{upstream} = 6D_c$. In addition, a $6D_c$ precursor domain is used to produce fully developed turbulent inlet conditions. Periodic inlet outlet boundary conditions are applied to the precursor domain and a volume source term is added to the momentum equation to impose a constant mass flow rate. The fluid quantities at the outlet of the precursor are mapped to the inlet of the computational domain, dynamically. A sketch of the precursor strategy is provided in Figure 95.



Figure 94 : Cut view of the fluid computational domain. Instantaneous velocity magnitude.

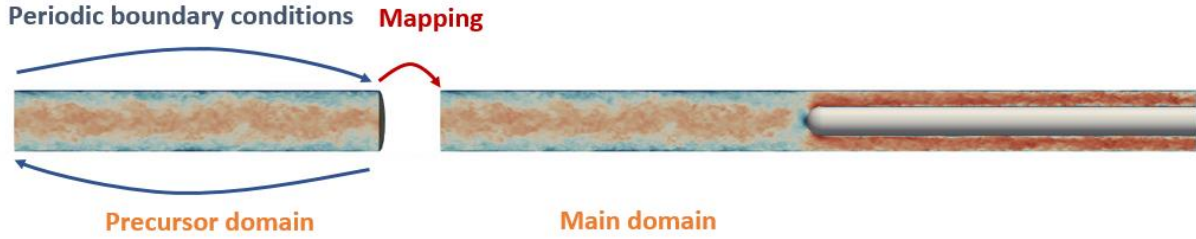


Figure 95. Sketch of the precursor calculation strategy. Instantaneous velocity magnitude.

The inlet bulk velocity, upstream of the cantilever is $U_b = 1.57 \text{ m} \cdot \text{s}^{-1}$. The bulk velocity in the annular domain is $U_{b,H} = 2.03 \text{ m} \cdot \text{s}^{-1}$. The Reynolds number, based on the annular bulk velocity $U_{b,H}$ and annulus hydraulic diameter $D_H = 11 \text{ mm}$, is $Re_H = 21\,200$. The targeted wall-normal non-dimensional distance of the first cell centre is $y^+ = 0.5$, where $(\cdot)^+ = \frac{(\cdot)u^*}{\nu}$ denotes the non-dimensional distance associated to the distance $(\cdot)$. The Colebrook relation has been used to estimate the distance to the walls Δy . We expect y^+ to be lower than 1 almost everywhere. A geometric progression in the wall-normal direction with ratio 1.1 is used in the vicinity of the walls and lower than 1.05 further. The prescribed stream-wise and azimuthal non-dimensional cell-sizes are respectively $\Delta z^+ = 20$ and $R\Delta\theta^+ = 10$, respectively. The level of refinement at the walls have been analysed a posteriori from the coupled FSI simulations, a histogram of the y^+ values is provided in Figure 96. As expected, there is a peak close to 0.5 and most of the values are below 1. A cut view of the y^+ surface field is provided in Figure 97, the largest values are localized on the curved end-piece and remains below 1.4.

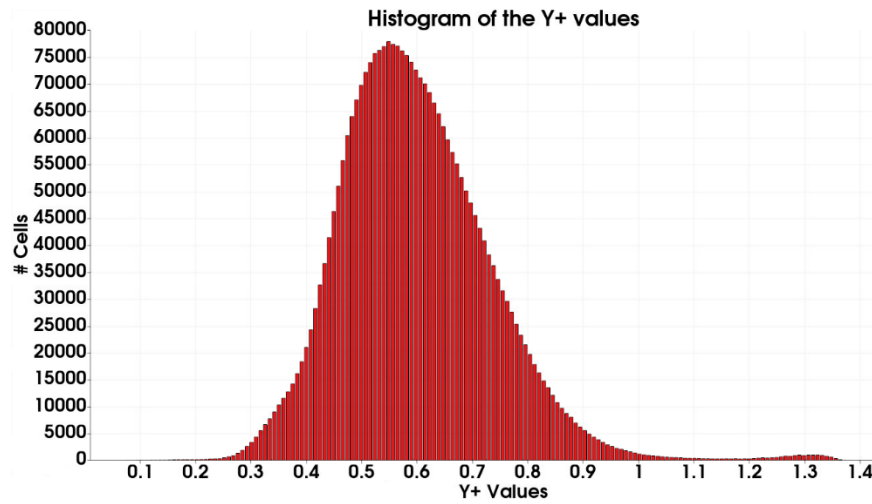


Figure 96. Histogram of the instantaneous y^+ values in the fluid domain.

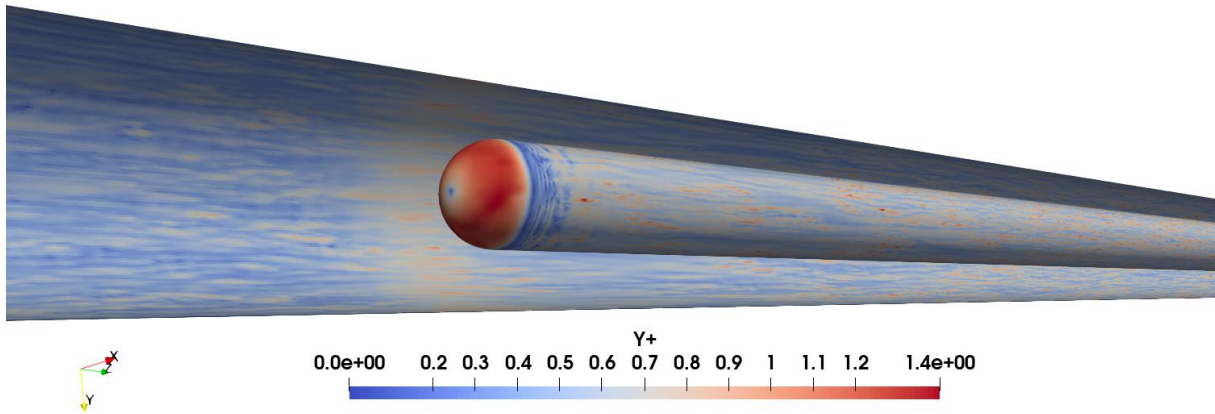


Figure 97. Cut view of the instantaneous y^+ values in the fluid domain.

A constant reference time step $\Delta t = 1.5 \times 10^{-5}$ s is chosen such that the mean CFL remains below 1.0 in most of the cells. A histogram of the cell CFL is provided in Figure 98. Most of the cells have a CFL value below 0.35 and a few cells have a CFL number between 0.35 and 0.8.

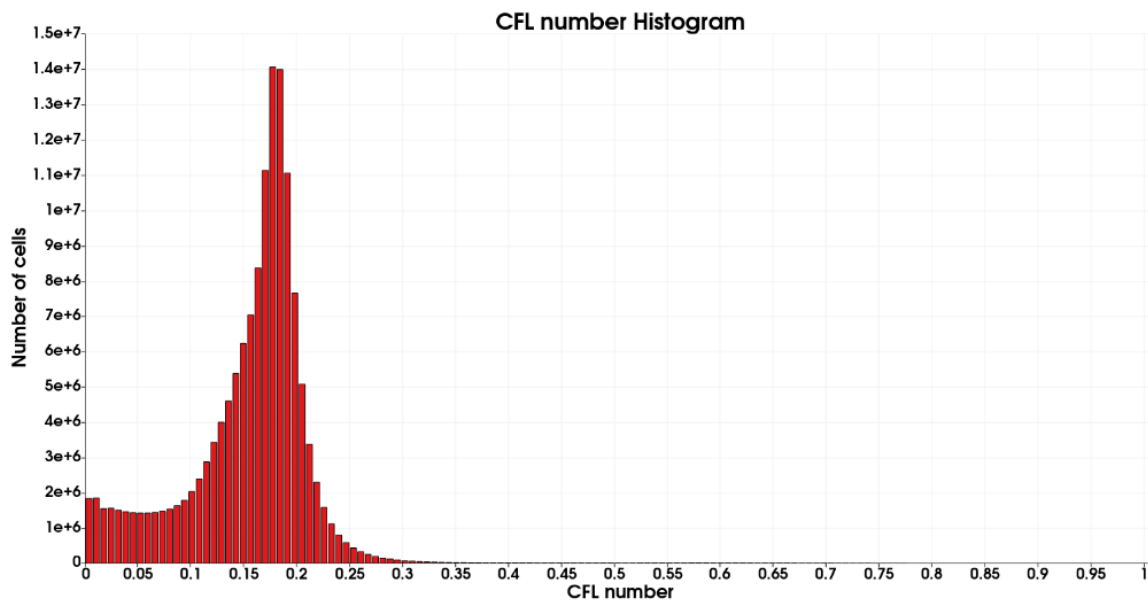


Figure 98. Histogram of the CFL number.

A cut view of the Courant number field, at the vicinity of the curved end-piece, is provided in Figure 99. The largest cell values of the Courant number are obtained close to the tip, because of the higher velocity values and the mesh topology. A cut view of the Courant number field with the mesh grid and a zoom on the tip is provided in Figure 100.

The numerical options used for all the calculations are the ones used by default with the version 8.0 of code_saturne with LES turbulence models. The SIMPLEC predictor corrector algorithm without outer sub-iterations is used. Centred fluxes with implicit Gauss gradient reconstruction along with a second order centred time-stepping is used. The solver tolerances are set to 10^{-5} for both the velocity and the pressure resolutions. A maximum number of 5 and 10 iterations are used for the right-hand side reconstruction of the pressure and velocity variables, respectively.

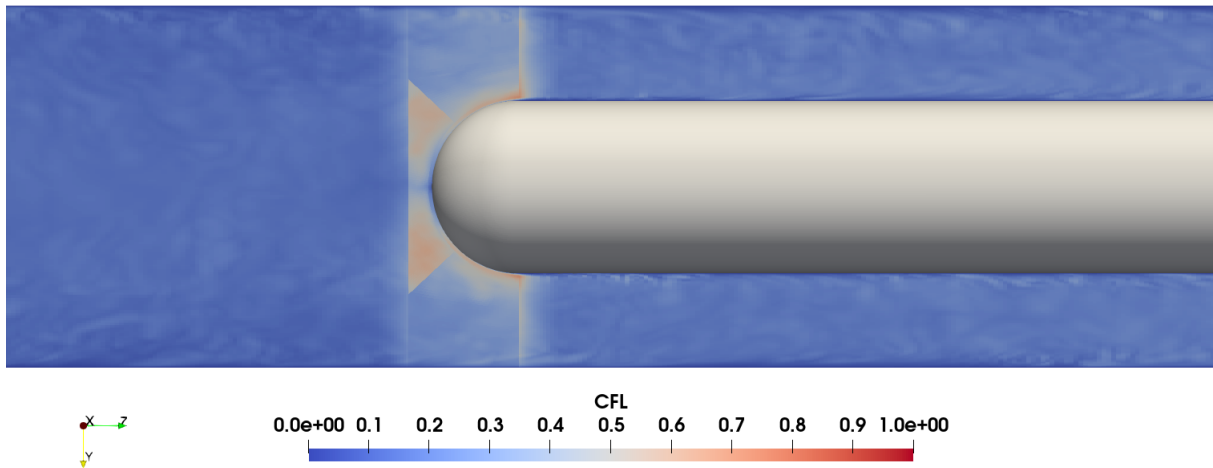


Figure 99. CFL number at the vicinity of the curved end-piece.

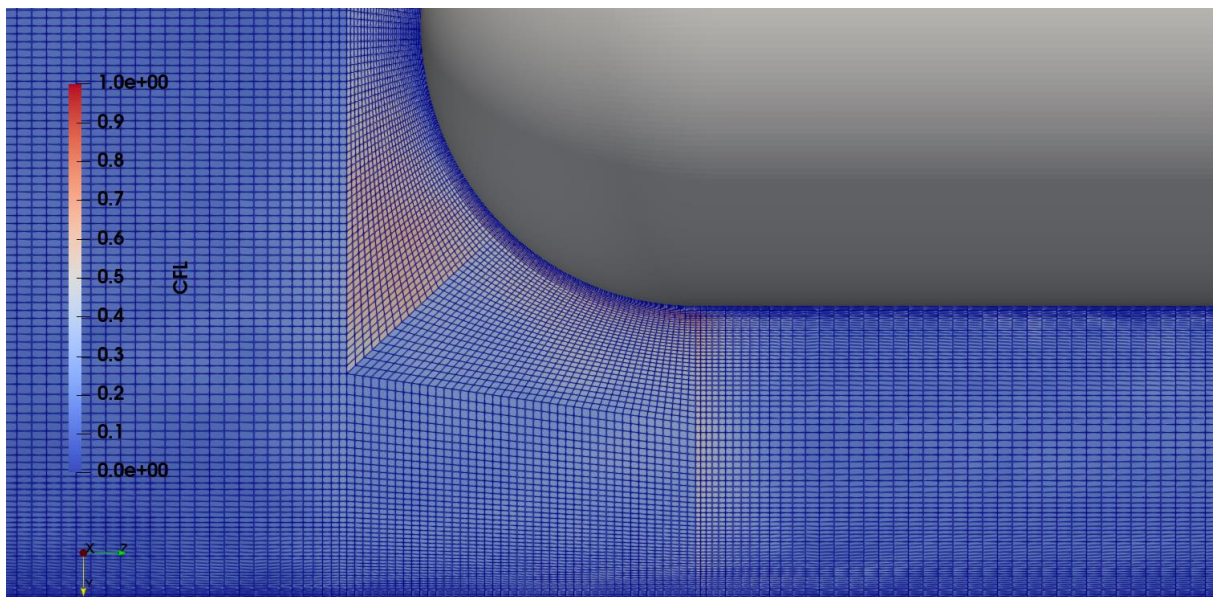


Figure 100. Zoom in the zones with highest CFL numbers. Cut view of the CFL number field above the mesh grid.

6.2.2.2 Solid model

An in-house beam equation solver with damping has been developed in a C language for this application. The Euler-Bernoulli equation with mechanical damping is solved, which is suitable for a slender beam. The solver works both as a standalone solver or in a coupled fashion with code_saturne V8.0. Both steady and unsteady version are implemented along with the standard beam boundary conditions. The solver works with both regular and unregular meshes, which allows us to be conformal with the CFD mesh. The following equation is solved by the mechanical solver:

$$\rho_l \frac{\partial^2 w}{\partial t^2} + EI \frac{\partial^4 w}{\partial x^4} + \mu_m \frac{\partial w}{\partial t} = f$$

With ρ_s the volumetric mass, S the beam cross section, E the Young's modulus, I the second moment of inertia, μ_m the mechanical damping factor and f the lineic load. The values selected for ρ_l , E , and I are the experimental ones, recalled in section 6.2.1.

Finite differences method with a numerical scheme at fractional time step $n - \frac{1}{2}$ is used, the scheme is second order in space and first order in time. The order of convergence has been verified through the simulation of steady and unsteady test cases with analytical solutions. An explicit coupling scheme with the fluid is used, without inner iterations between the fluid and the solid. The solid mesh is defined by the fluid one: the lineic forces f are computed over crowns of cells in the CFD solver. The centre of gravity of each of the crowns corresponds to a solid grid point, which avoid the use of interpolation. The resulting mesh is made of 3800 grid points. The solid solver runs on a single MPI process in the coupled calculation.

6.2.2.3 Coupling methodology

The lineic forces are computed by the CFD solver at each time step and given to the mechanical one. The displacement is computed by solving the Euler-Bernoulli beam equation and the mesh CFD mesh is explicitly deformed through an ALE methodology. The displacement at the nodes of the surface mesh of the rod are imposed. A mesh viscosity equal to 1.0×10^8 is used in the vicinity of the rod, to prevent its deformation while a mesh viscosity of 1.0 is applied in the rest of the domain to prevent the occurrence of negative volume cells. The mesh viscosity field is represented in Figure 101. There is no sub-iteration between the fluid and solid solvers inside each time step to save CPU time, since the displacements and the time steps are both very small. The solid mesh is defined by the fluid one, hence there is no need for interpolation between the solvers.

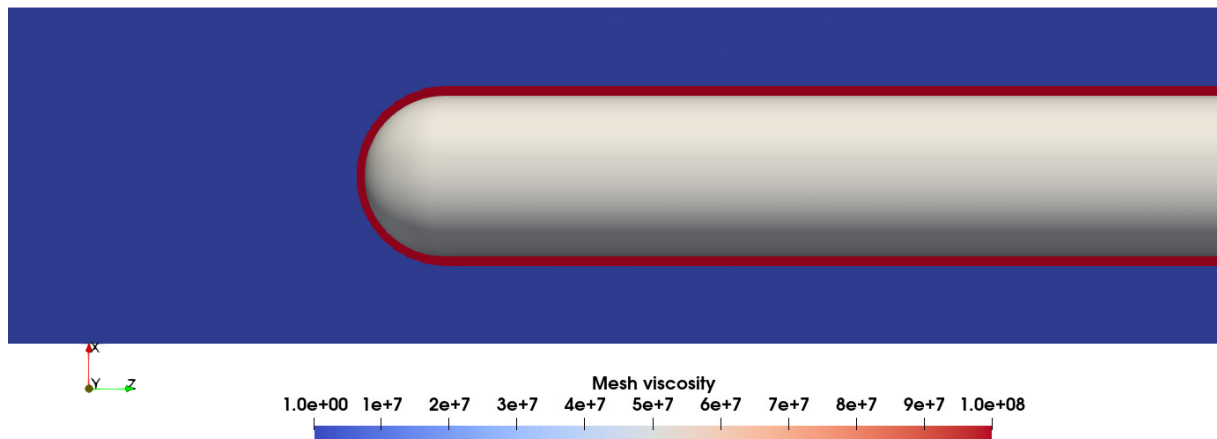


Figure 101 : Cut view of the mesh viscosity field in the fluid domain for the ALE methodology.

6.2.2.4 High Performance Computing

3150 MPI ranks (90 nodes) are used on the EDF R&D supercomputer Gaïa to perform the coupled calculations. The averaged time per iteration is 1.35s, corresponding to 1 day for simulating 1 second in physical time. Hence, the cost for a 60s coupled simulation is estimated to 4.7 million CPU hours and takes around 60 days. As a comparison, the elapsed time per

iteration for the fixed rod simulations is 0.785s per iteration, the coupling increases the CPU cost by 70%.

6.2.2.5 Post-treatments

In the context of the GO-VIKING European project specific post-treatments has been set up in the present fine calculation for benchmarking and validation of low-resolution methodologies: (i) 3D fields and profiles of the velocity, Reynolds stresses, pressure and pressure variance (ii) Time record of the displacement of the beam (iii) Surface and lineic stresses exerted along the beam. All the data were provided as 50GB archive to the partners.

In addition, power spectral densities (PSD) are computed, the Welch method with a Hanning window has been used. The length of the Hanning window is chosen as 1/16 of the dataset lengths. The `scipy.signal.welch` function, embedded in the Python3 library `scipy.signal` has been utilized.

6.2.3 Calibration of the mechanical damping with the simulations of vibration tests in still water

Simulations of a vibration test in still water have been performed to (i) verify the correct behaviour of our numerical model (ii) calibrate the mechanical damping coefficient μ_m to recover the analytical or experimental damping ratio ζ . The damping ratio of a vibrating rod confined in a cylinder and surrounded by still water has been estimated analytically with the following relation (Rogers, et al., 1984) (Blevins, 2001) :

$$\zeta = \frac{\pi}{2\sqrt{2}} \frac{\rho D_o^2}{\rho_t + \rho_{add}} \frac{1}{Re_{vib}^{0.5}} \frac{1 + \left(\frac{D_o}{D_c}\right)^3}{\left[1 - \left(\frac{D_o}{D_c}\right)^2\right]^2}$$

Where $Re_{vib} = \frac{\pi \rho f_1 D_o^2}{2\mu}$ denotes the vibrational Reynolds number.

The value obtained for the cantilever rod is 0.012. The damping ratio value has also been estimated experimentally (Cioncolini, et al., 2023) with the free vibration motion of the cantilever rod, with flat end-piece and an initial displacement. Using the logarithmic decrement of the envelope of the displacement, (Cioncolini, et al., 2023) estimated $\zeta = 0.015 \pm 0.001$. The experimental value is considered as the reference one in the following.

Coupled FSI simulations of the vibration experiment in still water were performed with several values for the mechanical damping coefficient μ_m ranging from 0 to 1. Several initial displacements ranging from 0.0001 m to 0.001 m in the x and y directions have been considered as well as several levels of mesh refinement, provided in Table 24

Table 24. Let us remark that mesh #4, corresponds to the same level of mesh refinement as the one used for the FSI calculation at $Re = 21200$ (section 6.3).

Table 24. Fluid and solid meshes used for the damping ratio sensitivity study

Mesh	# 1	# 2	# 3	# 4
# Fluid cells	350k	43.2M	51.9M	118M
# Beam grid points	1046	7800	3084	3884

The logarithmic decrement for two adjacent peaks with displacements w_{max}^0 and w_{max}^1 is:

$$\delta = \ln \left(\frac{w_{max}^0}{w_{max}^1} \right).$$

It can be generalized for a logarithmic decrement between a 0^{th} peak and a N^{th} peak:

$$\delta_N = \left(\frac{1}{N} \right) \ln \left(\frac{w_{max}^0}{w_{max}^N} \right).$$

The damping ratio is obtained from the logarithmic decrement:

$$\zeta_N = \frac{\delta_N}{\sqrt{\delta_N^2 + 4\pi^2}}.$$

The damping ratio has been estimated with several values of the mechanical damping coefficient μ_m in the Euler-Bernoulli equation with the meshes #1 to #4 (Table 24). The time record of the tip displacement (left) and damping ratio (right) are provided in Figure 102 to Figure 105 for the mesh #1 to #4, respectively. As expected, the larger is the mechanical damping coefficient μ_m , the smaller are the amplitudes of the movements. The damping ratio between the multiple peaks are constant with meshes 1, 2 and 3 while it is fluctuating with mesh 4. One needs to consider a few peaks to obtain a correct estimation of the damping ratio.

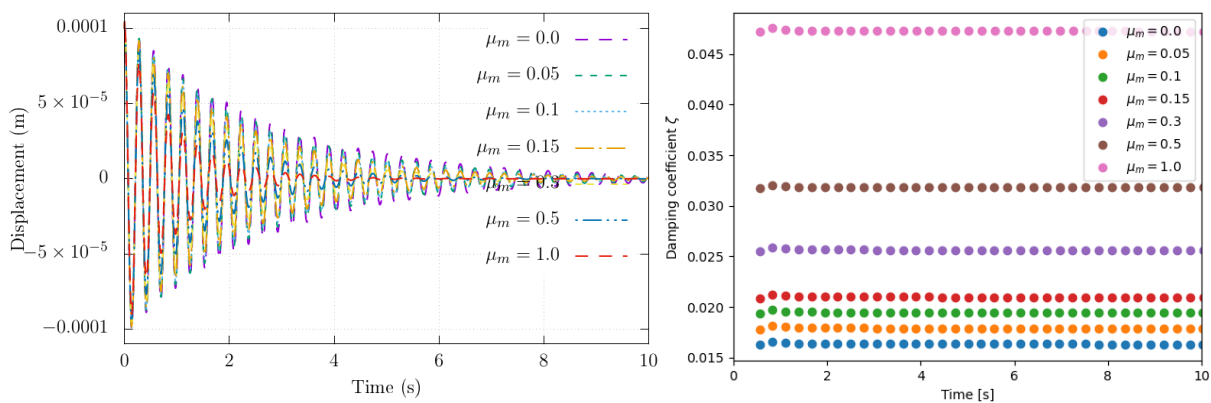


Figure 102. Time record of the displacement of the tip (left) and damping ratio estimated with the logarithmic decrement method. Free vibration simulations with mesh #1 and initial displacement $x_0 = 0.0001\text{ m}$. Comparisons with several μ_m coefficients.

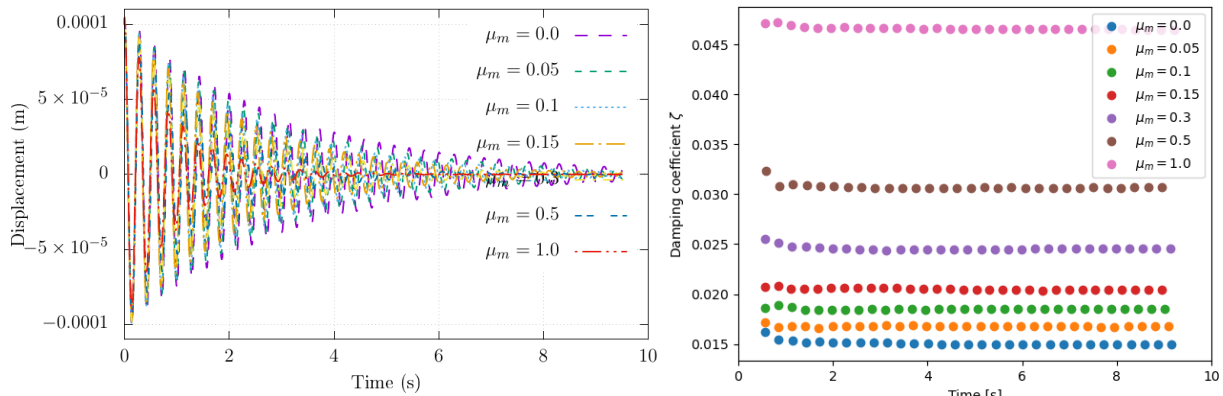


Figure 103. Time record of the displacement of the tip (left) and damping ratio estimated with the logarithmic decrement method. Free vibration simulations with mesh #2 and initial displacement $x_0 = 0.0001\text{ m}$. Comparisons with several μ_m coefficients.

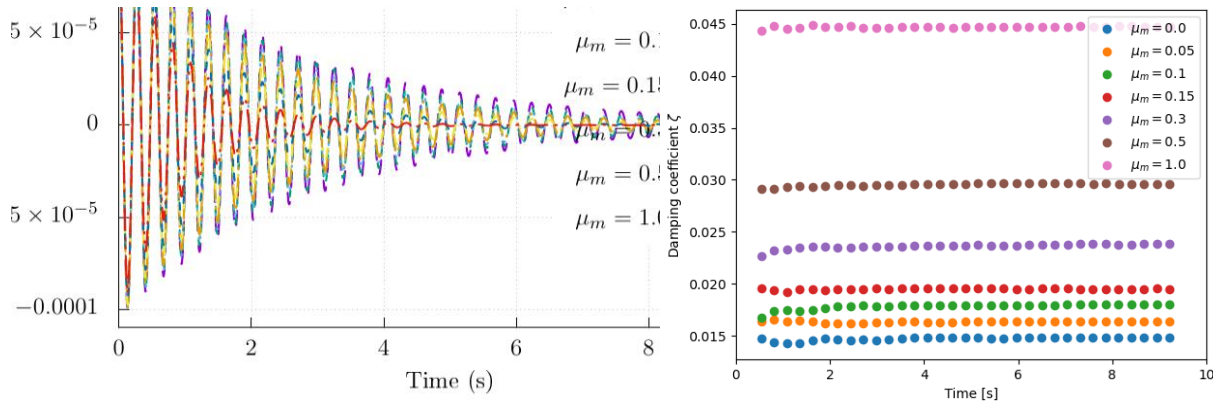


Figure 104. Time record of the displacement of the tip (left) and damping ratio estimated with the logarithmic decrement method. Free vibration simulations with mesh #3 and initial displacement $x_0 = 0.0001\text{ m}$. Comparisons with several μ_m coefficients.

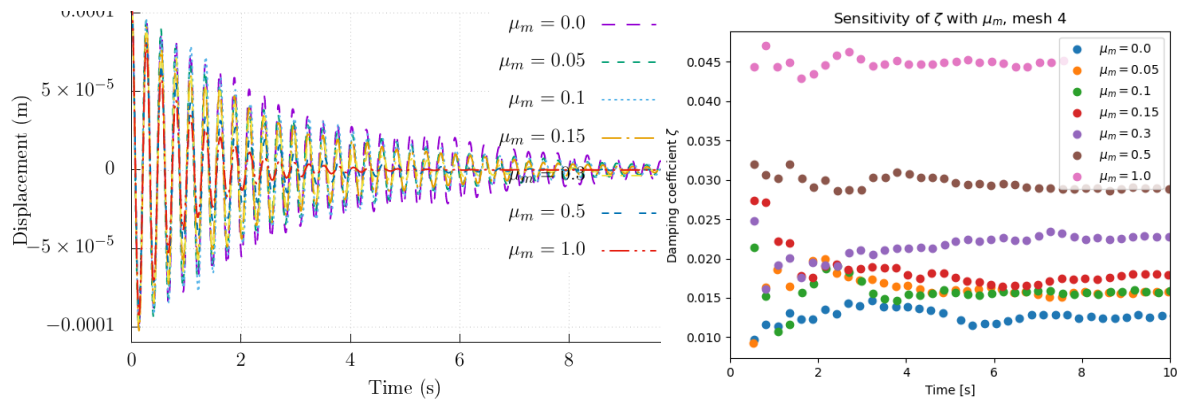


Figure 105. Time record of the displacement of the tip (left) and damping ratio estimated with the logarithmic decrement method. Free vibration simulations with mesh #4 and initial displacement $x_0 = 0.0001\text{ m}$. Comparisons with several μ_m coefficients.

The sensitivity of the damping coefficient ζ with respect to the mesh refinement is investigated. The results obtained with $\mu = 0 \text{ kg} \cdot \text{s}^{-1}$ and $\mu = 0.1 \text{ kg} \cdot \text{s}^{-1}$ are provided in Figure 106. The numerical values for μ_m ranging from 0 to $1 \text{ kg} \cdot \text{s}^{-1}$ are provided in Table 25. We notice that the finer the fluid mesh, the smaller the damping ratio, especially the refinement in the wall-normal and azimuthal directions. The small differences between the meshes 2 and 3 results shows that there is a low dependency to the solid refinement and the fluid refinement in the axial direction.

Table 25. Estimation of the damping ratio (mesh sensitivity) with $x_0 = 0.0001 \text{ m}$

$\mu_m \text{ (kg} \cdot \text{s}^{-1}\text{)}$	0.0	0.050	0.10	0.15	0.30	0.50	1.0
Mesh 1	0.0164	0.0179	0.0194	0.0210	0.0256	0.0318	0.0414
Mesh 2	0.0149	0.0168	0.0185	0.0204	0.0245	0.0307	0.0465
Mesh 3	0.0148	0.0164	0.0180	0.0195	0.0237	0.0296	0.0447
Mesh 4	0.0127	0.0146	0.0157	0.0176	0.0216	0.0253	0.0449

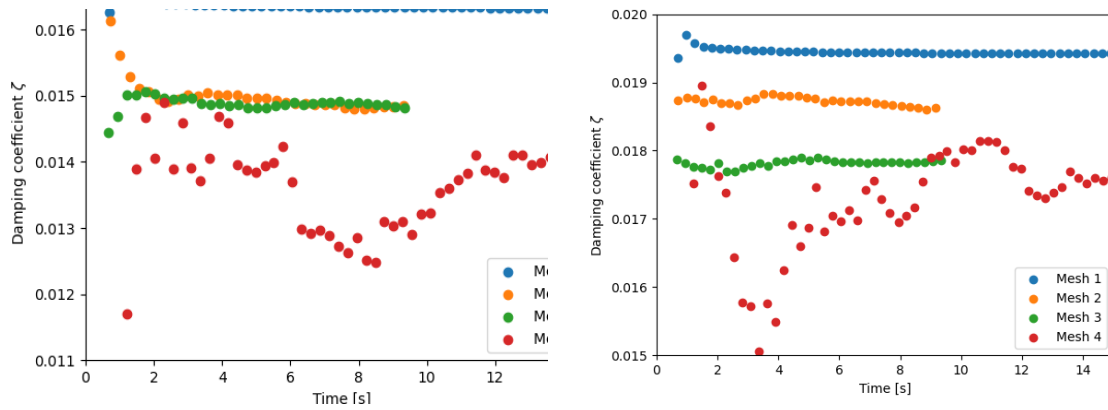


Figure 106. Mesh sensitivity study of the damping ratio coefficient with $\mu_m = 0 \text{ Kg} \cdot \text{s}^{-1}$ (left) and $\mu_m = 0.1 \text{ kg} \cdot \text{s}^{-1}$ (right).

The sensitivity to the initial displacement is investigated in Figure 107. The damping ratio is independent to the mesh displacement in the range $x_0 \in [0.0001, 0.001] \text{ m}$.

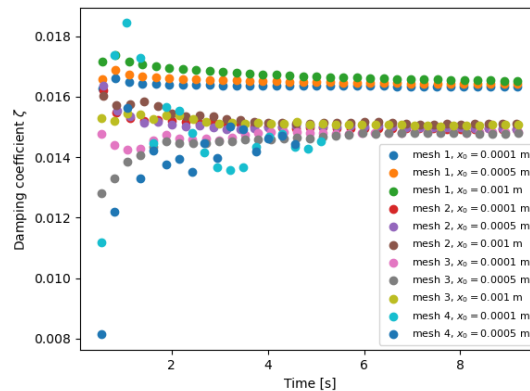


Figure 107. Damping ratio sensitivity to the initial displacement x_0 with $\mu_m = 0 \text{ kg} \cdot \text{s}^{-1}$.

Finally, we verified that the natural frequencies of the beam are recovered with the numerical model. Additionally, we verify that they are not affected by the choice of the μ_m coefficient. The PSD of the tip displacements for several μ_m coefficients ranging from 0 to $1 \text{ kg} \cdot \text{s}^{-1}$ are provided in Figure 108. The Welch method with a Hanning window has been used to filter the data.

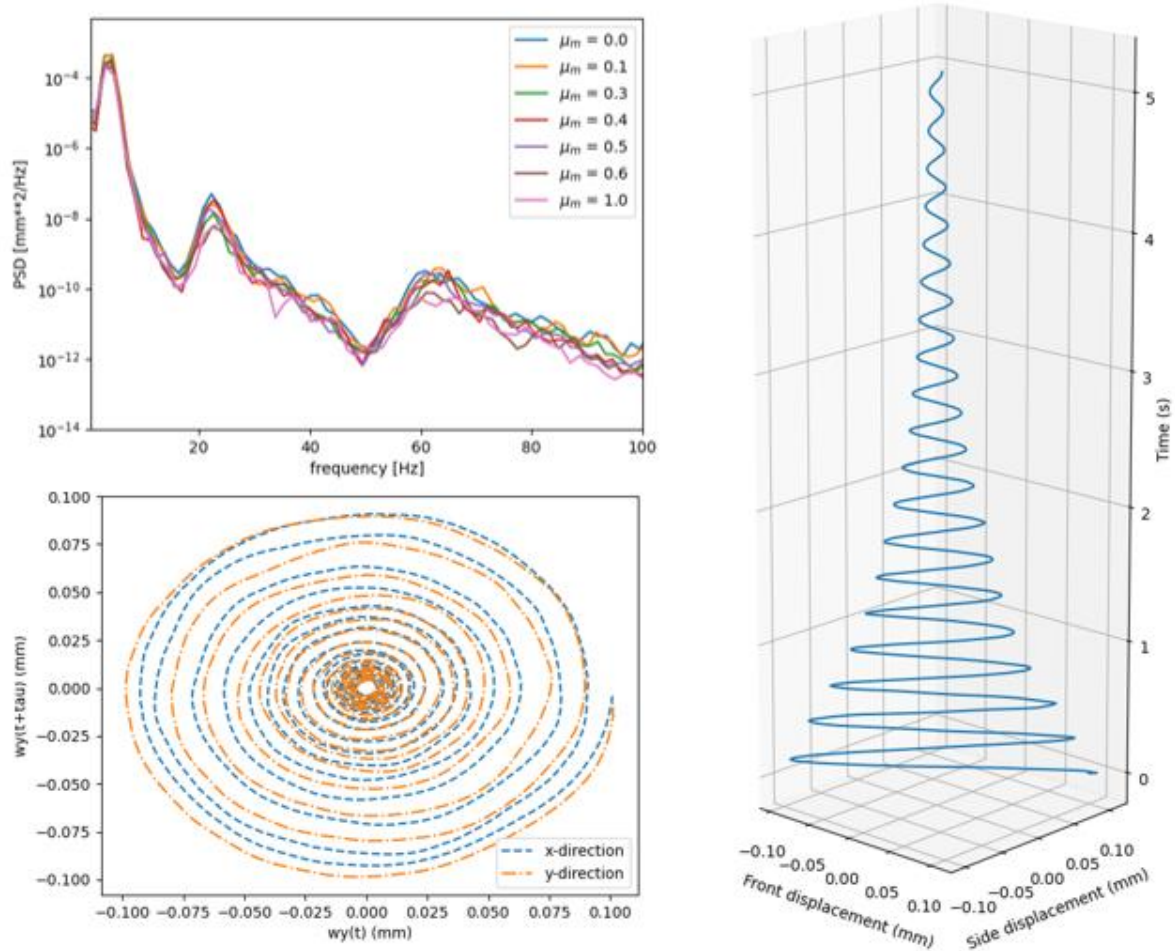


Figure 108. Tip displacement analysis of the free vibration tests with mesh #4. PSD with several μ_m coefficients (top left). Attractor of the displacement in the two span-wise directions, $\mu_m = 0 \text{ kg} \cdot \text{s}^{-1}$ (bottom left). 3D representation of the displacement in the span-wise directions with respect to time, $\mu_m = 0 \text{ kg} \cdot \text{s}^{-1}$ (right).

From these results presented in Table 25, the μ_m coefficient is set to $\mu_m = 0.05 \text{ kg} \cdot \text{s}^{-1}$ in the following since the LES mesh used with FSI calculations as a level of refinement equivalent to the one of mesh 4.

6.2.4 Assessment of the inlet turbulent conditions

There is no experimental data available at $Re = 21200$ regarding the inlet turbulence. Hence, we performed additional simulations at $Re = 15.7k$ with the WR-LES precursor alone to compare the numerical and experimental inlet turbulence. The first optical window is located $10 D_H$ upstream of the end-piece (see Figure 2), which corresponds to the location of the precursor domain in the full-length simulations. A periodic cylindrical pipe with diameter



$D_c = 0.021 \text{ m}$ and length $L = 6D_c = 0.126 \text{ m}$ has been considered with a bulk velocity $U_b = 1.13 \text{ m} \cdot \text{s}^{-1}$ imposed with a momentum source term.

The bulk velocity profiles obtained with the experiment and the fully developed LES are provided in Figure 109. The axial velocity profiles are in a good agreement. The mean radial velocity is almost zero in the simulation, which indicates that the averages are well converged.

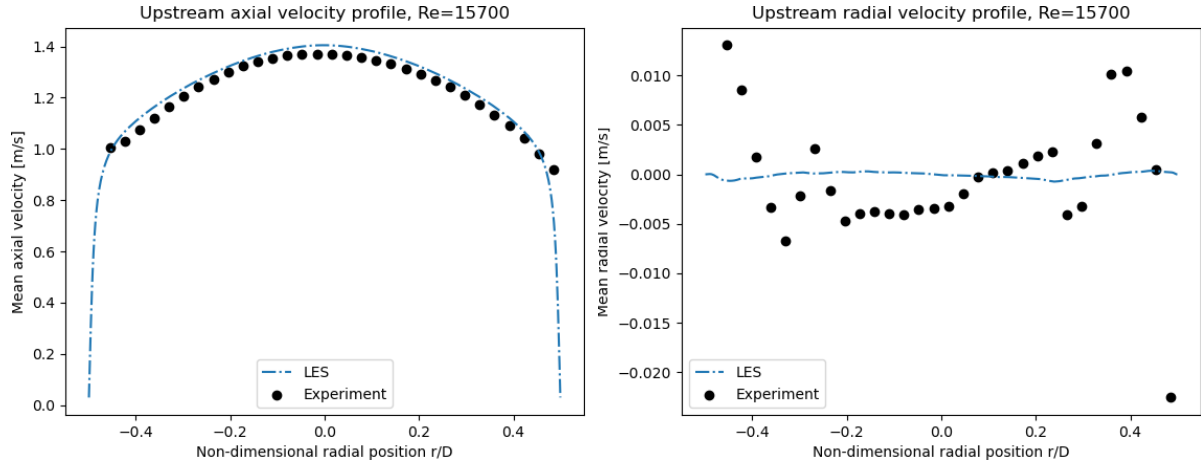


Figure 109. Comparison of the experimental upstream velocity profile and the one obtained in the LES precursor domain.

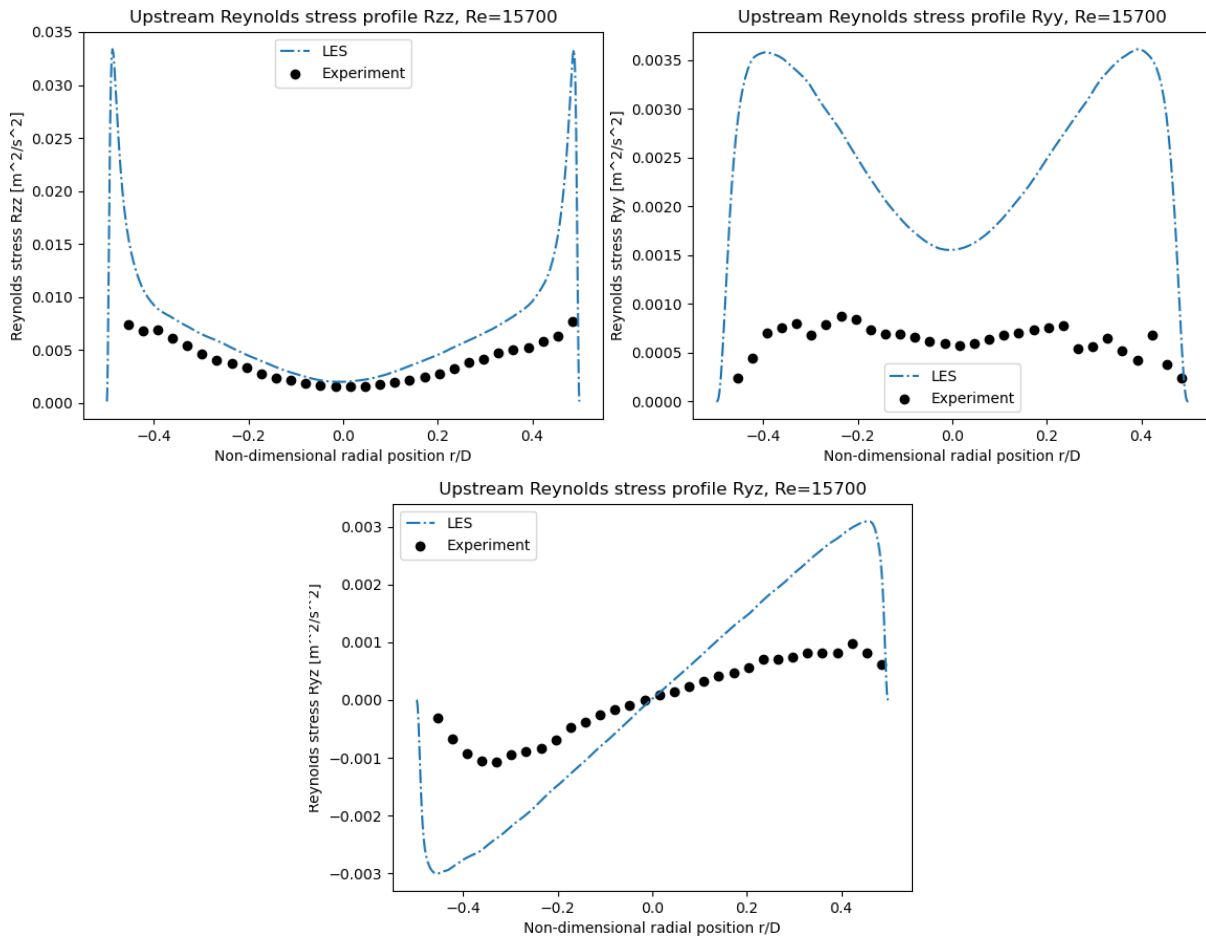


Figure 110. Comparison of the experimental upstream Reynolds stress profile and the one obtained in the LES precursor domain.

The Reynolds stress profiles are provided in Figure 110. There is quite a good agreement at the center of the pipe for the streamwise component R_{zz} , and consequently there is a good agreement on the turbulence kinetic energy. However, the turbulent intensity at the vicinity of the walls is less significant in the experiment than in the LES simulations. Similarly, the amount of turbulence is less significant in the experiment for the R_{yy} and R_{yz} components. This probably means that the inlet turbulence is not fully developed in the experiment. Nevertheless, the precursor strategy remains the more accurate and convenient way to produce unsteady inlet turbulent conditions for the WR-LES simulations.

6.3 Numerical FSI simulations of the cantilevered beam experiment at $Re=21200$

The numerical simulations of the cantilevered beam have been performed in three steps:

1. Initialization of the precursor domain alone for generating the turbulent inlet conditions.
2. Simulation of the full domain with a fixed rod to initialize the flow in the entire domain and compute statistics with a rod in central position. This simulation restarts from the simulation of step 1. with the precursor alone.
3. Coupled FSI simulation of the cantilevered beam, departing from the fixed rod numerical simulation of step 2.

The simulation with the precursor domain alone (step 1.) has run 10s in physical time to obtain a fully developed turbulent field. Then, the simulation with a fixed rod (step 2.) has run 12 more seconds departing from the precursor alone to initialize the coupled calculation. Finally, the rod is free to move at $t=22s$. The coupled FSI simulation has run for 60s departing from the fixed rod simulation (step 3.). The overall simulation time (including initialization) is 82s.

6.3.1 Precursor calculation alone

First, the precursor calculation ran alone for 10 physical seconds with the mass flowrate corresponding to $Re_{D_H} = 21\,200$ in the annular domain. This allows to initialize the turbulent inlet conditions generator without having to simulate the full domain (at higher computational cost). The chosen time step is higher than the one of the simulations with the cantilever rod since the maximum velocity is lower and the cell lengths are greater, especially in the vicinity of the end-piece. A time step of $\Delta t = 1.0 \times 10^{-4}s$ was used. The precursor mesh alone has 11.6M cells. The simulated physical time represents around 125 flow passes through the computational domain, which is sufficient to obtain a fully developed turbulence field. The flow establishment was verified with monitoring probes. It took around 7.5 hours to perform the simulation on the Cronos supercomputer, with 30×48 cores. The mean elapsed time

per iteration corresponds to 0.27 s. The velocity field in the precursor domain, at the end of the calculation is provided in Figure 111.

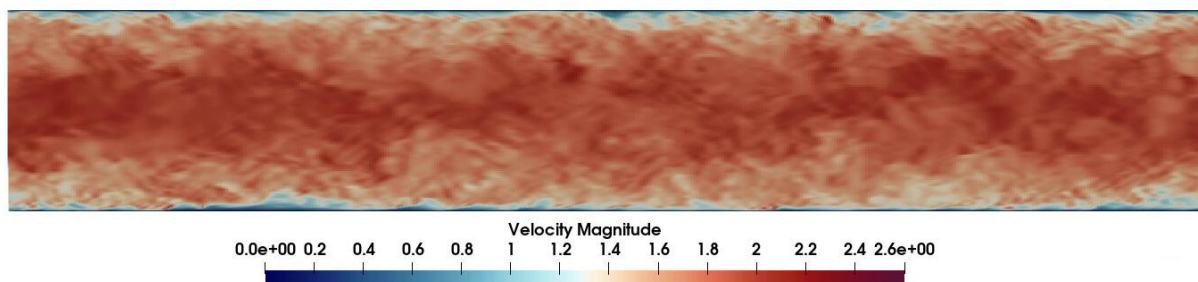


Figure 111. Cut view of the velocity field in the precursor domain.

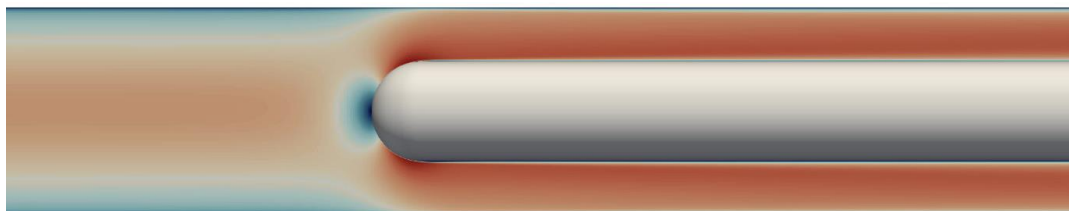
6.3.2 Calculation with a fixed rod

6.3.2.1 Simulation

Then, the calculation began on the full domain with a “restart on different mesh” option from the precursor calculation alone. The fully developed turbulent field of the precursor is mapped to the inlet of the full domain while it keeps running. During the beginning of the calculation, the cantilever is in a fixed position and does not move. A constant time step $\Delta t = 4 \times 10^{-5} \text{ s}$ is used at the beginning of the simulations to initialize the flow, then it has been switched to $\Delta t = 1.5 \times 10^{-5} \text{ s}$ to get a maximum CFL number lower than 1 in the entire computational domain, before averaging. The simulation with a fixed rod has run for 12 seconds in physical time before averaging. It takes around 0.6s for the flow to pass through the entire computational domain. Hence, 20 flow passes have been simulated with the rod in central position, which is sufficient for both reaching a statistically steady state. In addition, the fields have been time averaged for 23s in a separate computation workflow. The statistics of the velocity and pressure fields were post-processed with a fixed rod in central position to compare with the FSI results and estimate the rod motion effect on the flow.

6.3.2.2 Numerical results

Cut views of the time-averaged and an instantaneous velocity fields are provided in Figure 112.



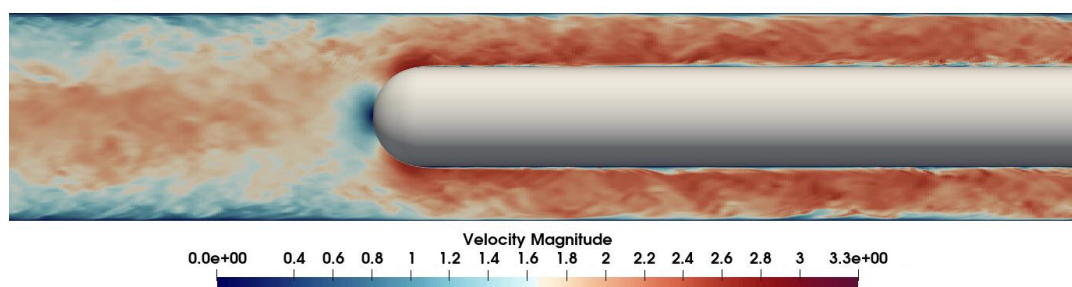


Figure 112. Cut view of the time averaged (top) and instantaneous (bottom) velocity fields close to the tip of the rod. CFD simulations with a fixed beam in central position.

The turbulent kinetic energy and Reynolds stress components fields are provided in Figure 113 and Figure 114, respectively. We do not observe any flow separation in the vicinity of the tip with the CFD simulations. The turbulent kinetic energy is maximum in the vicinity of the walls, in the annulus domain where the geometry is more constrained.

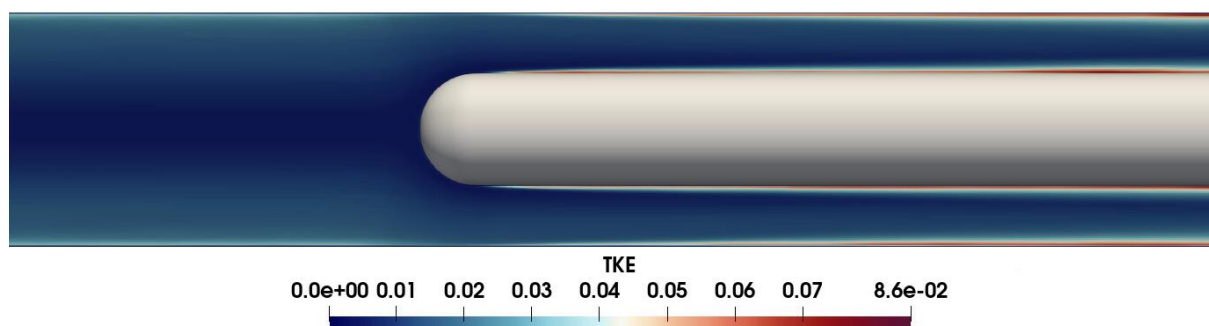
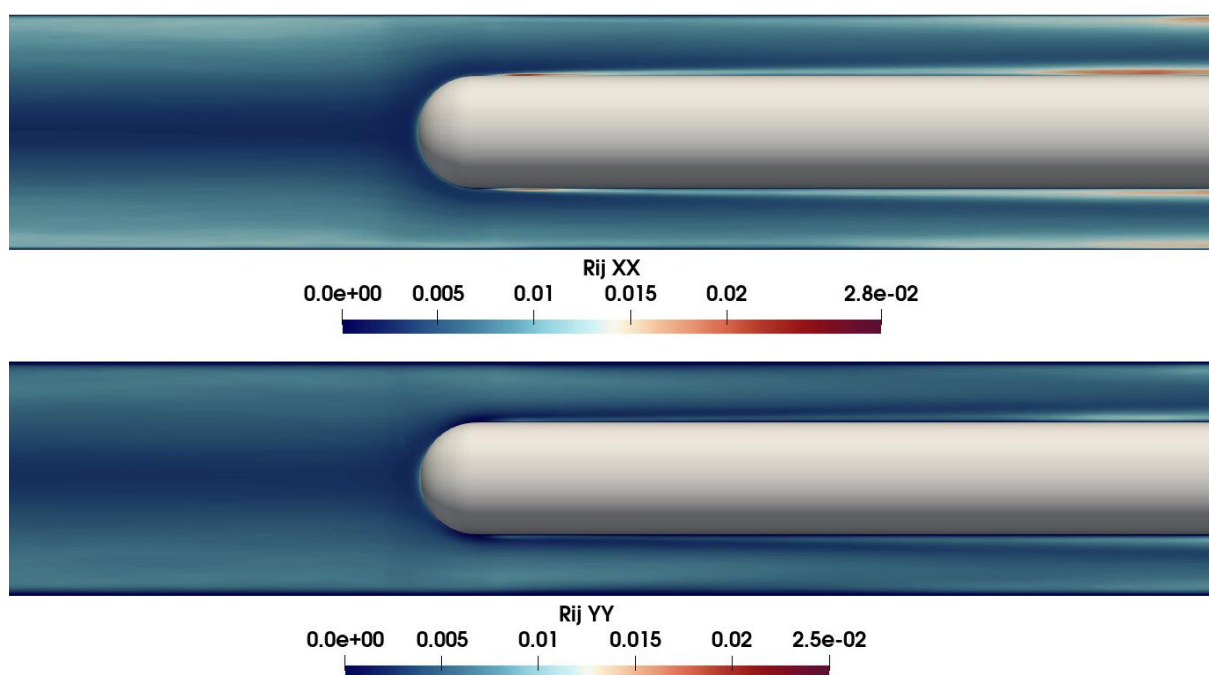


Figure 113. Cut view of the turbulent kinetic energy field close to the tip of the rod. CFD simulations with a fixed beam in central position.



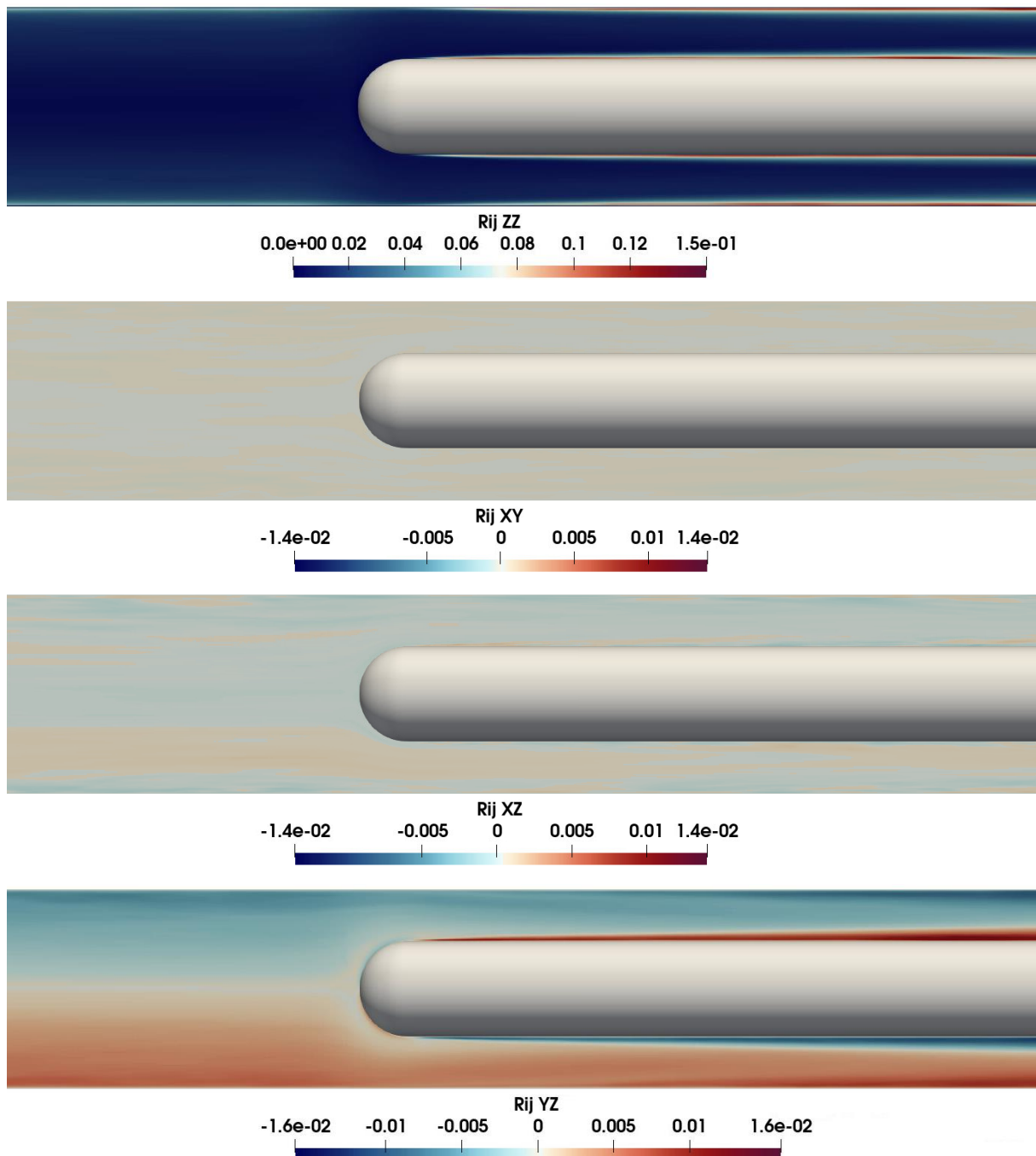


Figure 114. Cut views of the 6 components of the resolved part of the Reynolds stresses close to the tip of the rod. CFD simulations with a fixed beam in central position.

6.3.3 Calculation with FSI

Finally, departing from the results of the simulations with the rod in central position described in Section 6.3.2, we allowed the beam to move in the two spanwise directions with the weak two-way coupling methodology described in Section 6.2.2. The simulation has run for 60s in physical time and the averaging process began after 10s, for the beam dynamics to reach a nominal regime. Hence, the time acquisition for the statistics with FSI is 50s long.

6.3.3.1 Analysis of the flow



Funded by
the European Union

EUDuC = 0E001

Both the fixed rod and FSI CFD results are compared with the PIV experimental data. Several radial profiles at the PIV measurement locations are provided in Figure 115 to Figure 120. The PIV data inside of the beam (due to the reflection on the metallic surface) are discarded. The time-averaged axial and orthoradial velocity profiled as well as Reynolds stress components are drawn.

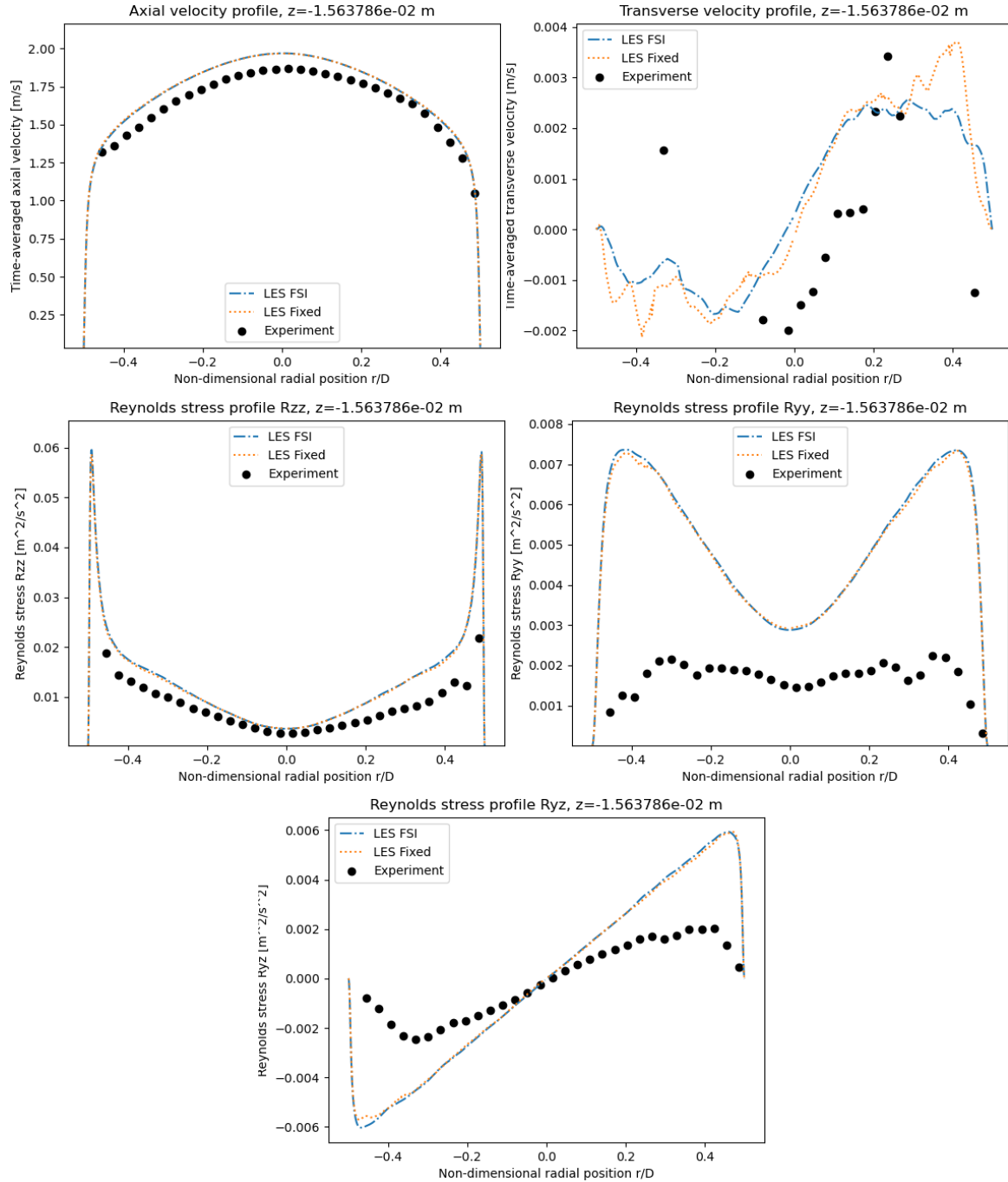


Figure 115 : Turbulent profiles $1.56 \times 10^{-2} \text{ m}$ upstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (R_{zz} : middle left, R_{yy} : middle right, R_{yz} : bottom).

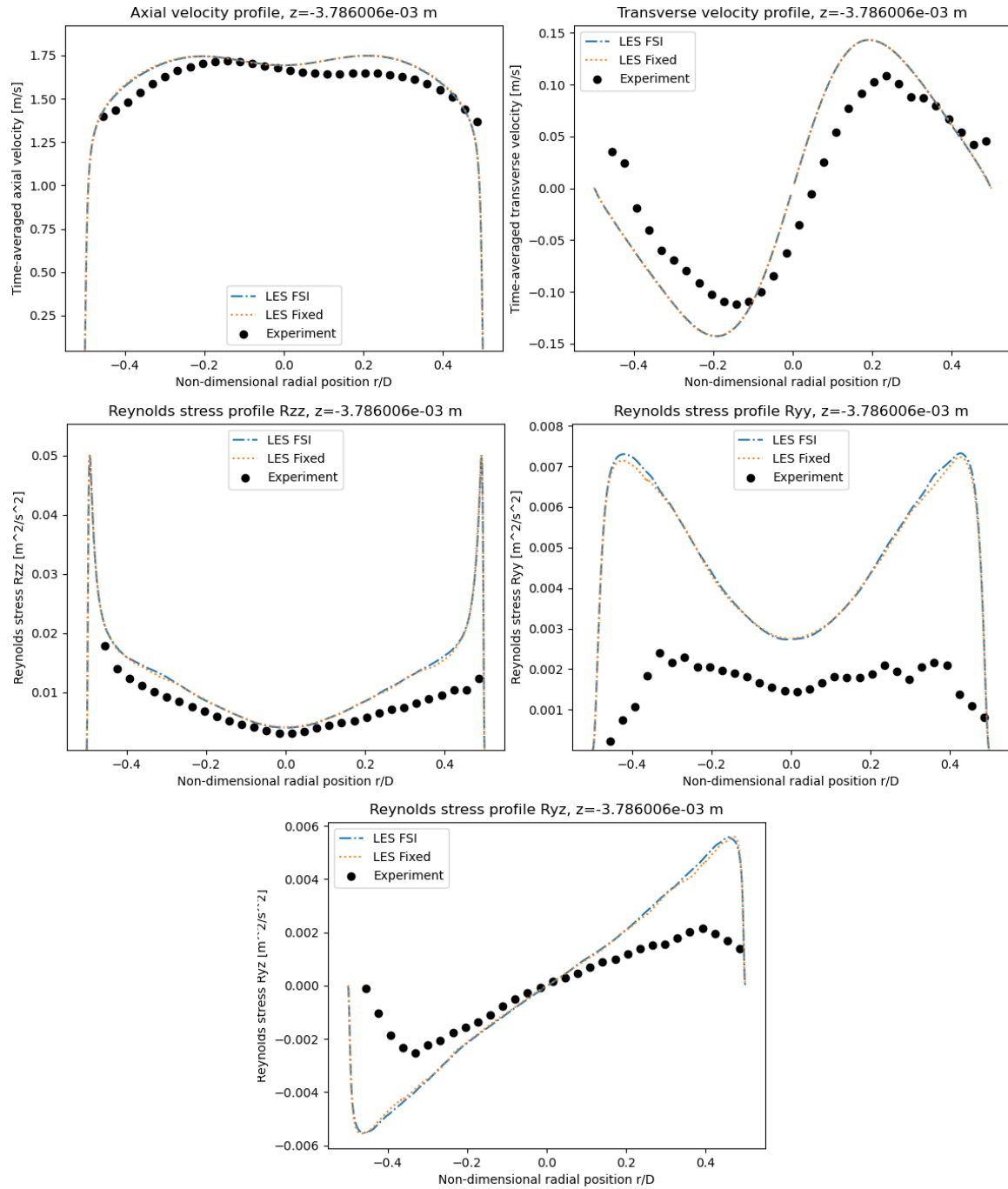


Figure 116 : Turbulent profiles $3.79 \times 10^{-3} \text{ m}$ upstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (R_{zz} : middle left, R_{yy} : middle right, R_{yz} : bottom).

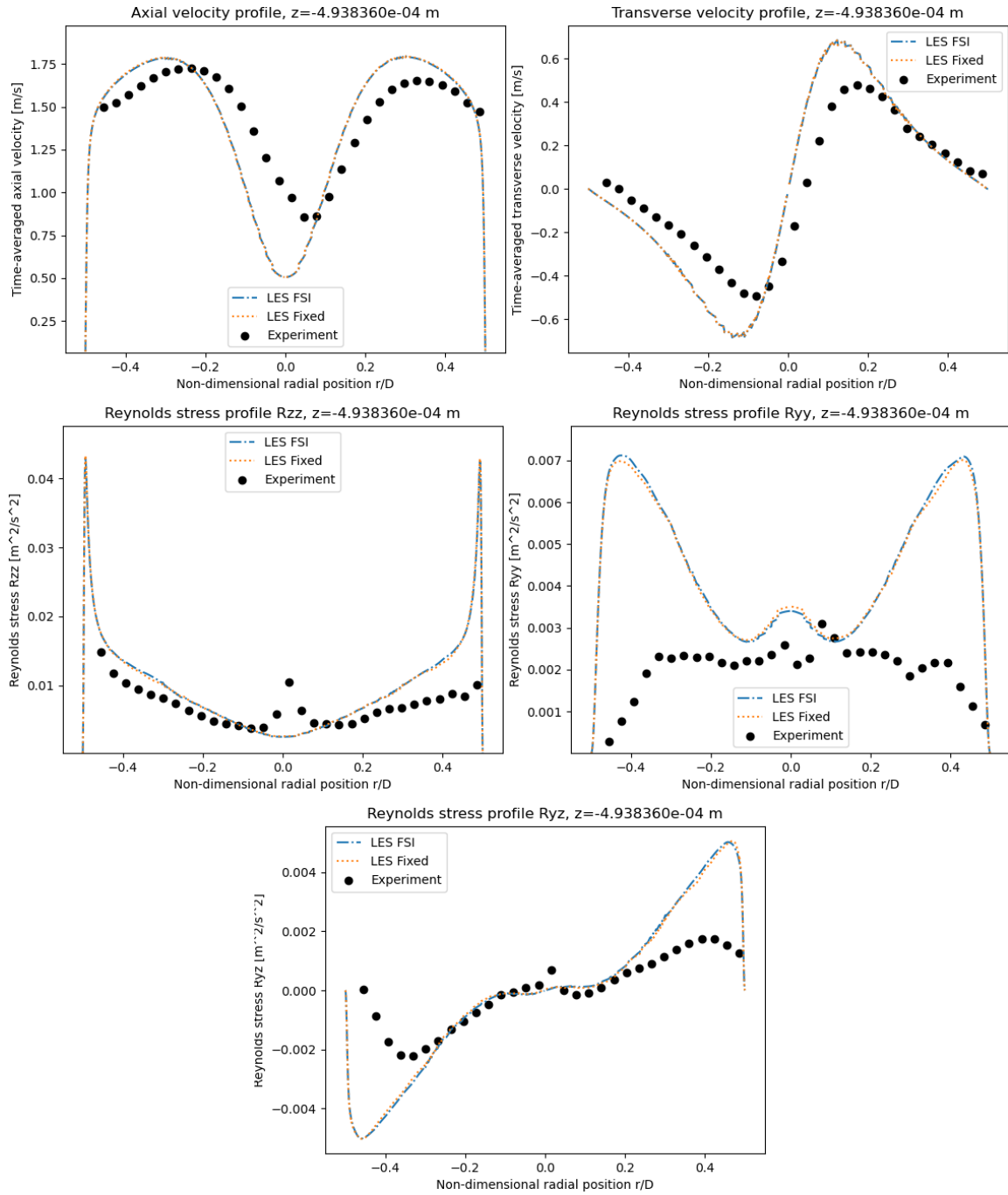


Figure 117 : Turbulent profiles $4.94 \times 10^{-4} \text{ m}$ upstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (R_{zz} : middle left, R_{yy} : middle right, R_{yz} : bottom).

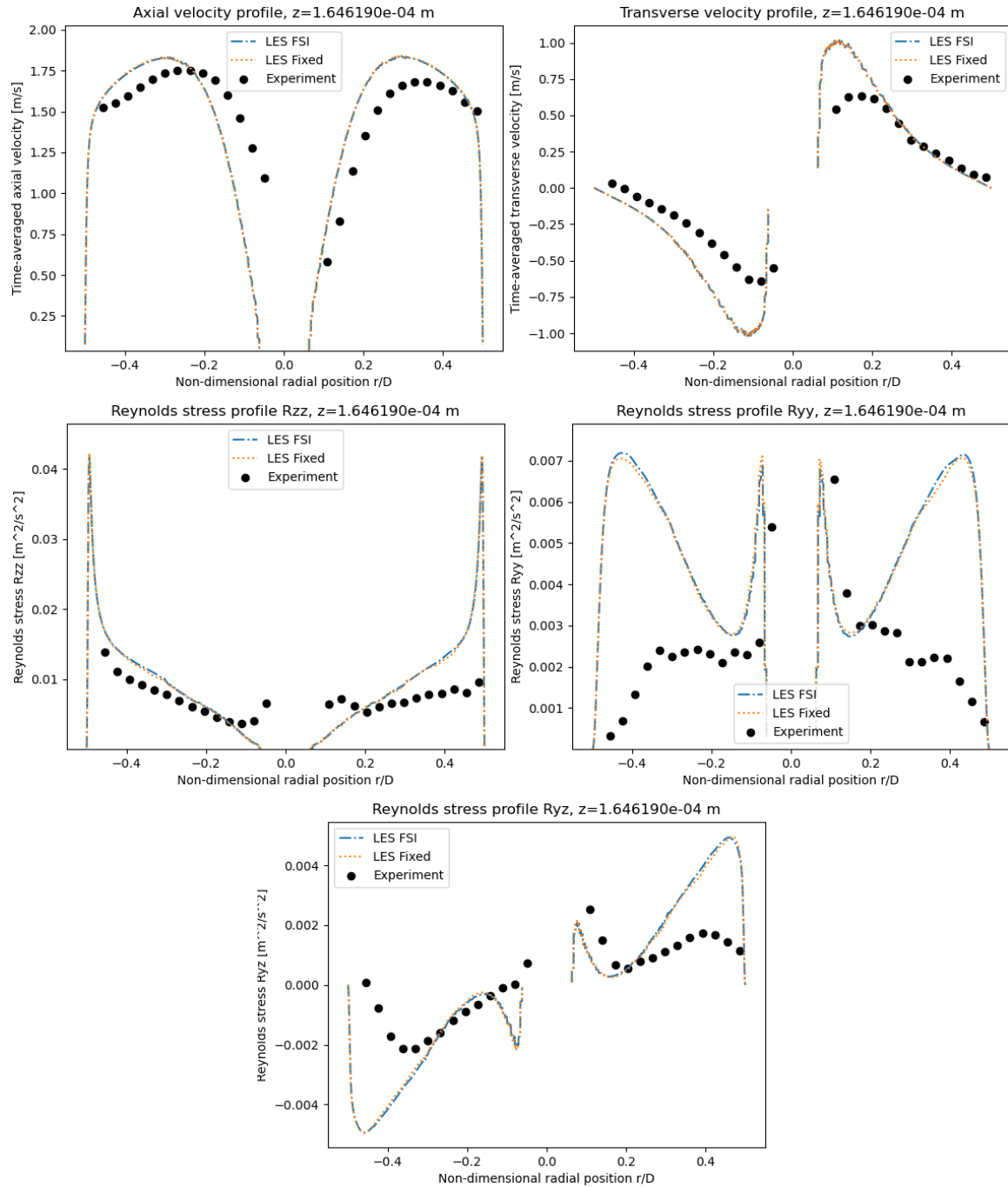


Figure 118 : Turbulent profiles $1.64 \times 10^{-4} \text{ m}$ downstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (R_{zz} : middle left, R_{yy} : middle right, R_{yz} : bottom).

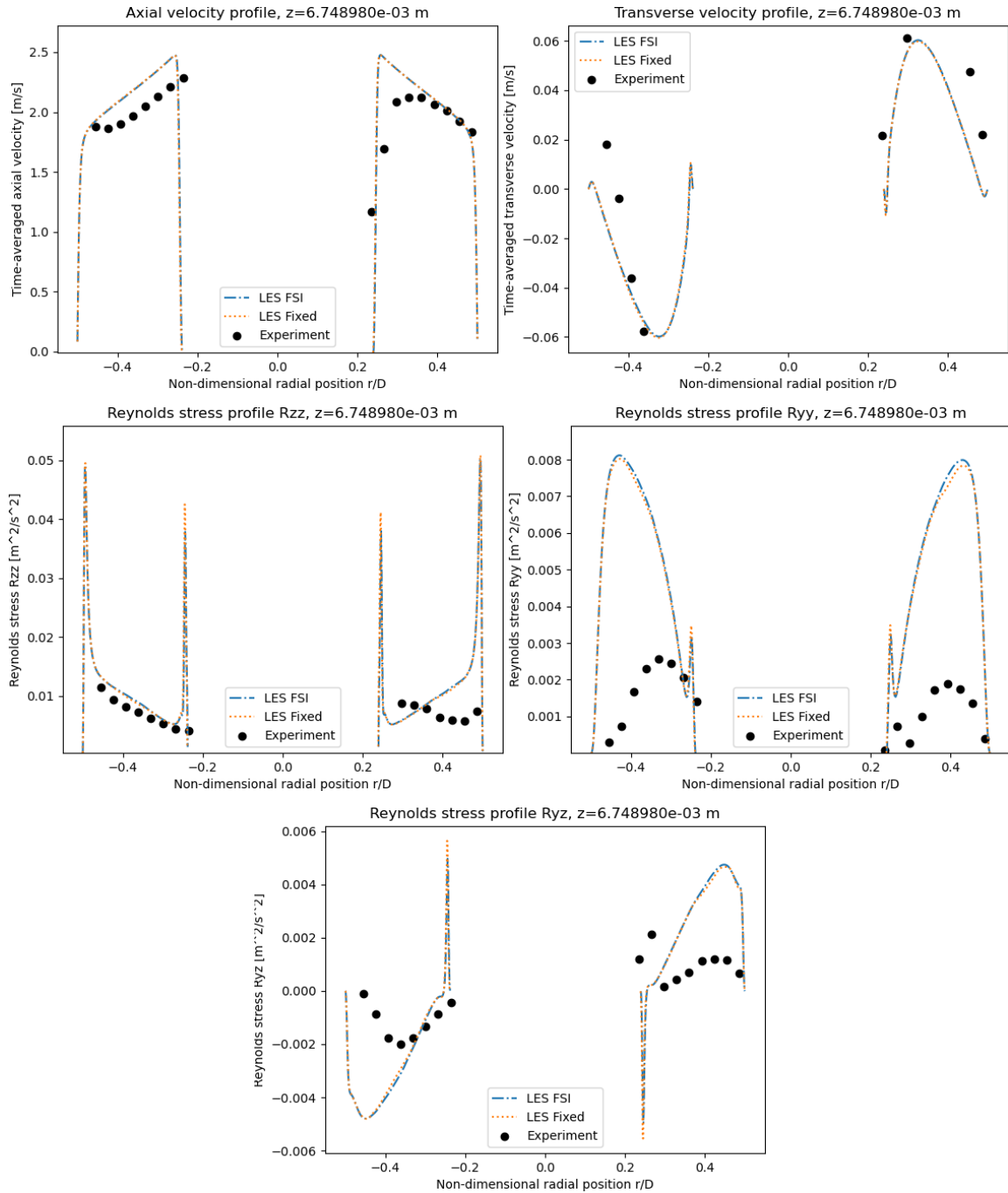


Figure 119 : Turbulent profiles $6.75 \times 10^{-3} \text{ m}$ downstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (R_{zz} : middle left, R_{yy} : middle right, R_{yz} : bottom).

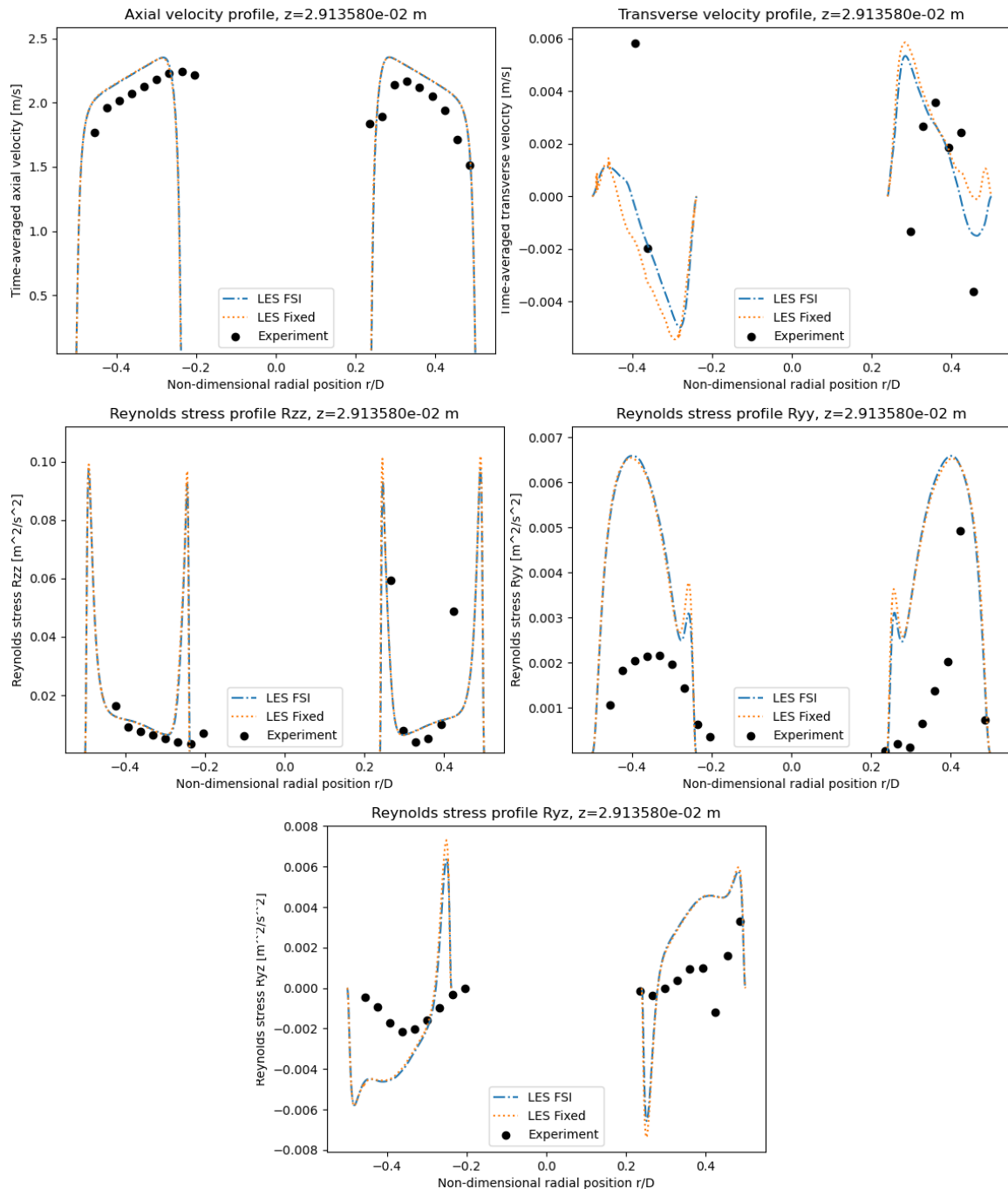


Figure 120 : Turbulent profiles $2.91 \times 10^{-2}m$ downstream of the tip. FSI simulations compared with experimental results. Mean axial (top left) and transverse (top right) velocity profiles and velocity covariance (R_{zz} : middle left, R_{yy} : middle right, R_{yz} : bottom).

There is a good agreement for the velocity. There is also a good agreement regarding the diagonal components of the Reynolds stress tensor and the turbulent kinetic energy. The LES simulation predicts a greater turbulent kinetic energy magnitude at the vicinity of the walls. The discrepancy close to the walls is probably due to the PIV limitations. When it comes to the cross-components of the Reynolds stress profiles, the LES simulation predicts a greater magnitude for every component, especially upstream of the tip (Figure 115 and Figure 116). The shape of the profiles is well recovered. The Reynolds stress components are more in

accordance with the PIV measurements in the vicinity and downstream of the tip (Figure 117 to Figure 120). The results obtained with the fixed rod simulation are very close to the ones obtained with FSI. Hence, the cut views of the velocity and turbulent quantities of the Figure 112 to Figure 114, obtained with the fixed simulation are the same as with the coupled FSI simulation. This is due to the very small amplitude of the beam displacement.

The evolution of the profiles with respect to the axial position has been investigated. Several radial profiles, from the confining tube to the rod, with axial positions given in Table 26 are drawn. The time-averaged velocity field, turbulent kinetic energy and Reynolds stress component R_{yz} are given in Figure 121.

Table 26. Axial positions of the $P_{r,i}$ radial profiles (The origin is taken at the tip of the curved end-piece)

Profiles	$P_{r,1}$	$P_{r,2}$	$P_{r,3}$	$P_{r,4}$	$P_{r,5}$
Axial position	$5.0 \times 10^{-4} \text{ m}$	$4.0 \times 10^{-3} \text{ m}$	$8.0 \times 10^{-3} \text{ m}$	$1.4 \times 10^{-2} \text{ m}$	$2.0 \times 10^{-2} \text{ m}$
Profiles	$P_{r,6}$	$P_{r,7}$	$P_{r,8}$	$P_{r,9}$	$P_{r,10}$
Axial position	$3.0 \times 10^{-2} \text{ m}$	$5.0 \times 10^{-2} \text{ m}$	$1.0 \times 10^{-1} \text{ m}$	$1.5 \times 10^{-1} \text{ m}$	$2.0 \times 10^{-1} \text{ m}$

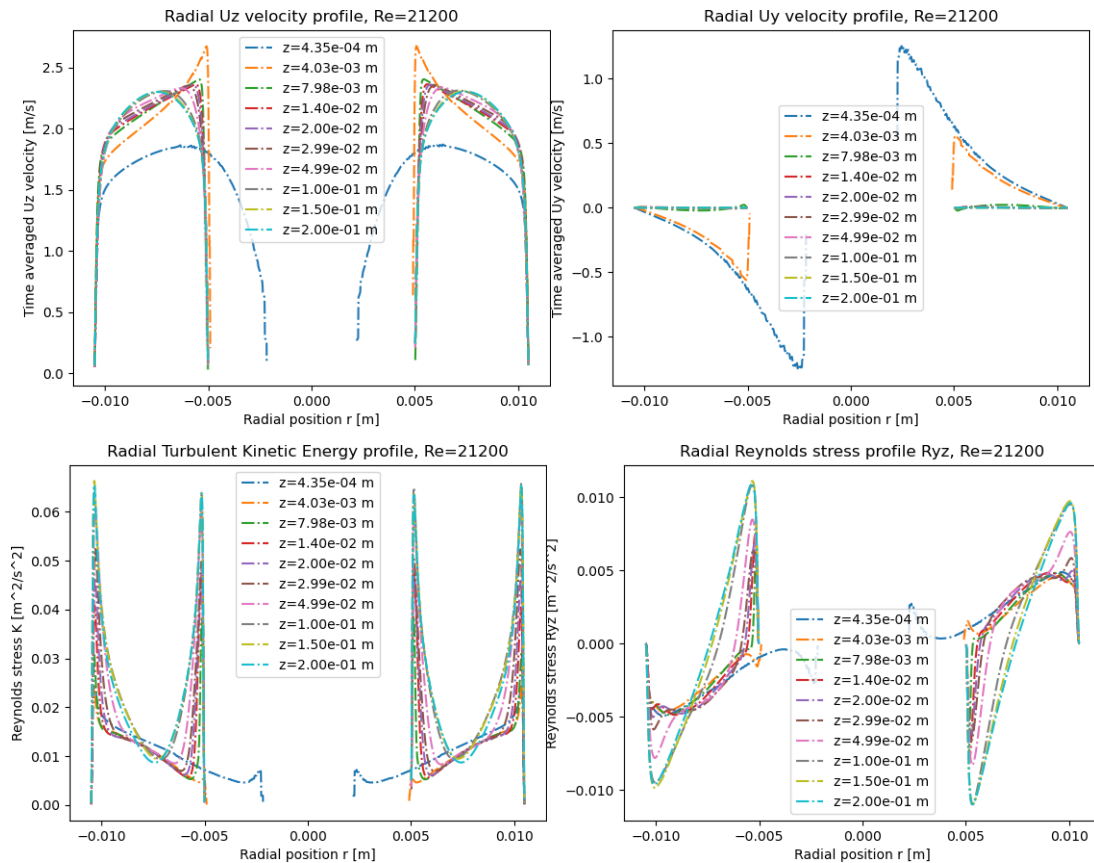


Figure 121. Radial profiles $P_{r,i}$. Time-averaged axial velocity (top left), time-averaged spanwise velocity (top right), Turbulence Kinetic Energy (bottom left) and Reynolds stress profile R_{yz} (bottom right).

The axial velocity profiles progressively reach those of an infinite annular domain in a few centimetres downstream of the tip. The recirculation in the span-wise direction is localized in the vicinity of the tip and becomes negligible from $z=8$ mm downstream of the tip. The shapes of the turbulent kinetic energy profiles are strongly affected by the tip and its magnitude becomes larger in the annular domain.

Similarly, axial profiles $P_{z,i}$, from $z = 0.0$ m to $z = 0.1$ m were defined, the radial positions of these profiles are given in Table 27.

Table 27. Radial positions of the $P_{z,i}$ axial profiles (The origin is taken at the tip of the curved end-piece)

Profile	$P_{z,1}$	$P_{z,2}$	$P_{z,3}$	$P_{z,4}$	$P_{z,5}$
Radial position	1.0×10^{-5} m	2.0×10^{-5} m	5.0×10^{-5} m	1.0×10^{-4} m	2.0×10^{-4} m
Profile	$P_{z,6}$	$P_{z,7}$	$P_{z,8}$	$P_{z,9}$	
Radial position	5.0×10^{-4} m	1.0×10^{-3} m	2.0×10^{-3} m	5.0×10^{-3} m	

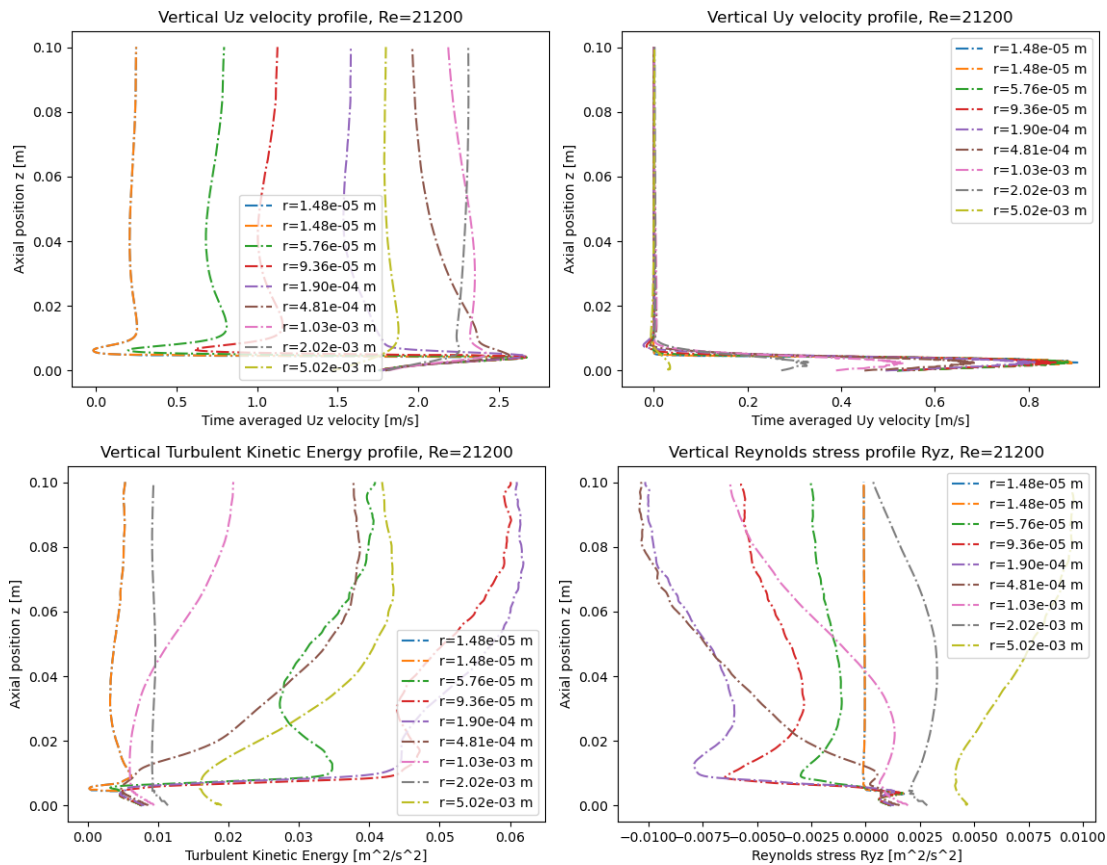


Figure 122. Axial profiles $P_{z,i}$. Time-averaged axial velocity (top left), time-averaged spanwise velocity (top right), Turbulence Kinetic Energy (bottom left) and Reynolds stress profile R_{yz} (bottom right).

6.3.3.2 Surface forces and stresses exerted over the rod

Cut views of the normal stress and the magnitude of the shear stress along the rod are given in Figure 12. The maximum values are obtained on the rounded end-piece for both components.

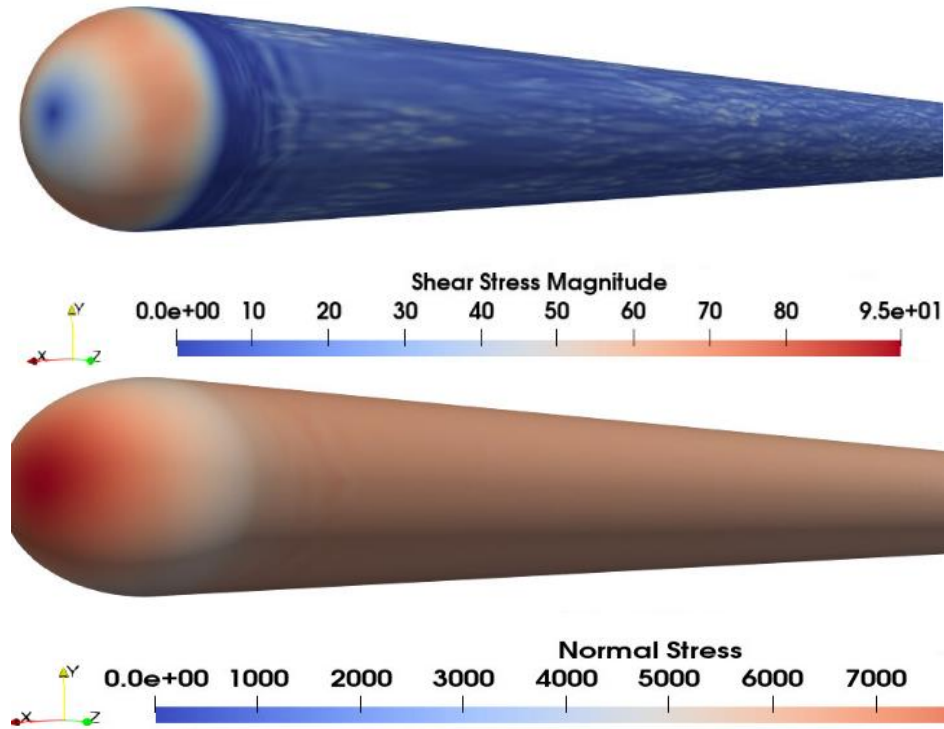


Figure 123 : Cut views of the instantaneous shear stress magnitude (top) and normal stress (bottom).

The total stresses are integrated over crowns of cells for the mechanical solver to compute the displacement. The time-averaged and standard deviation of the mean circumferential stress are given in Figure 124. The mean values are zero and the maximum standard deviation is obtained at the centre.

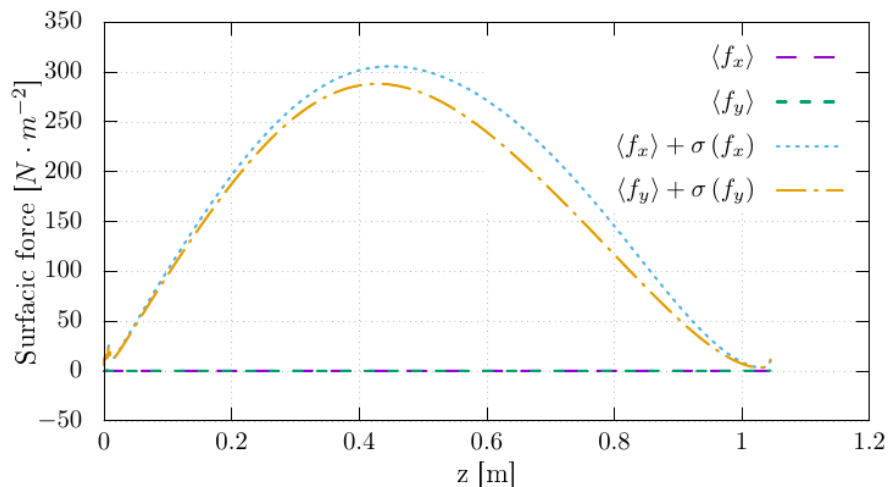


Figure 124 : Time average and standard deviation of the stress exerted over the rod with respect to position.

The axial component of the axis-symmetric stress exerted over the rod (along $\underline{e}_z$) is provided in Figure 125 and Figure 126 for the rounded end-piece and the rod, respectively. Both the time-averaged and standard deviation values are provided. The stress values observed on the rounded end-piece are much more important than those on the rod. The stress on the end-piece is due to the pressure effect, whereas it is due to the viscous effects along the rod. The time average and standard deviation values are very close, indicating a small level of fluctuations for the e_z component of the stress.

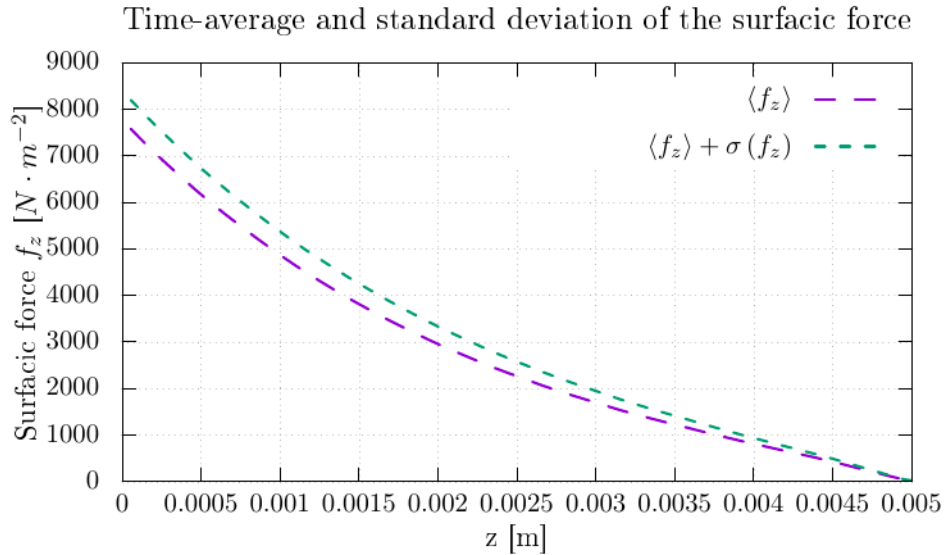


Figure 125. Axial component (along $\underline{e}_z$) of the axisymmetric stress on the rounded end-piece. Time-averaged values and standard deviation.

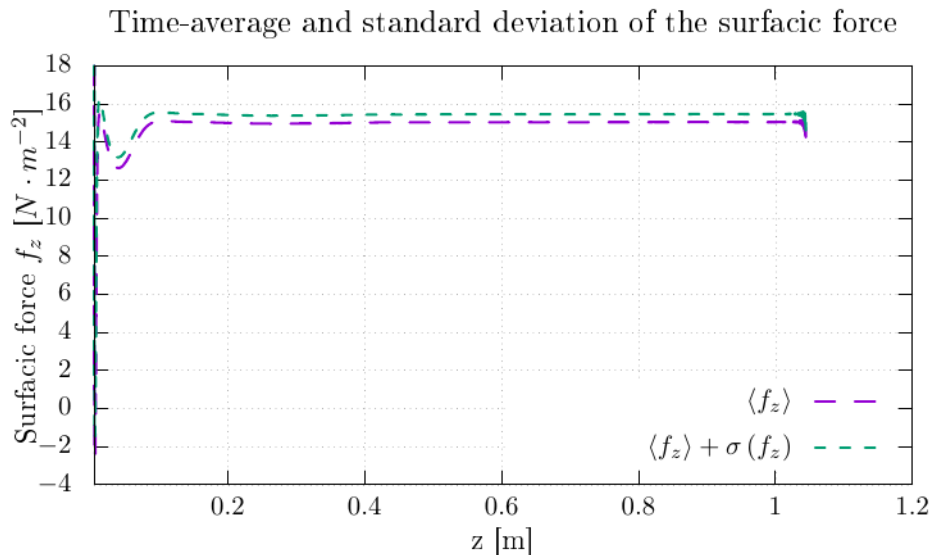


Figure 126. Axial component (along $\underline{e}_z$) of the axisymmetric stress on the rod. Time-averaged values and standard deviation.

6.3.3.3 FIV analysis and beam motion

The motion of the rod has been monitored during the simulation. The maximum amplitude of the displacement of the tip is 0.06 mm in the CFD simulation. The RMS. of the radial displacement estimated over 50s of simulation is 0.030 mm, lower than the 0.082 mm measured experimentally (Cioncolini, et al., 2023). The displacement of the tip of the beam in the front and side directions are provided in Figure 127. One may observe a dissymmetry in the experimental displacement: $RMS(U_{x,exp}) = 0.067 \text{ mm}$ while $RMS(U_{y,exp}) = 0.048 \text{ mm}$. The values obtained with the CFD are the same in both spanwise cartesian direction $RMS(U_{x,cf}) = RMS(U_{y,cf}) = 0.021 \text{ mm}$. This may be due to the tip misalignment observed in the PIV measurement of the Figure 118 to Figure 120. Nevertheless, the amplitude of the vibration remains lower in both directions with the CFD.

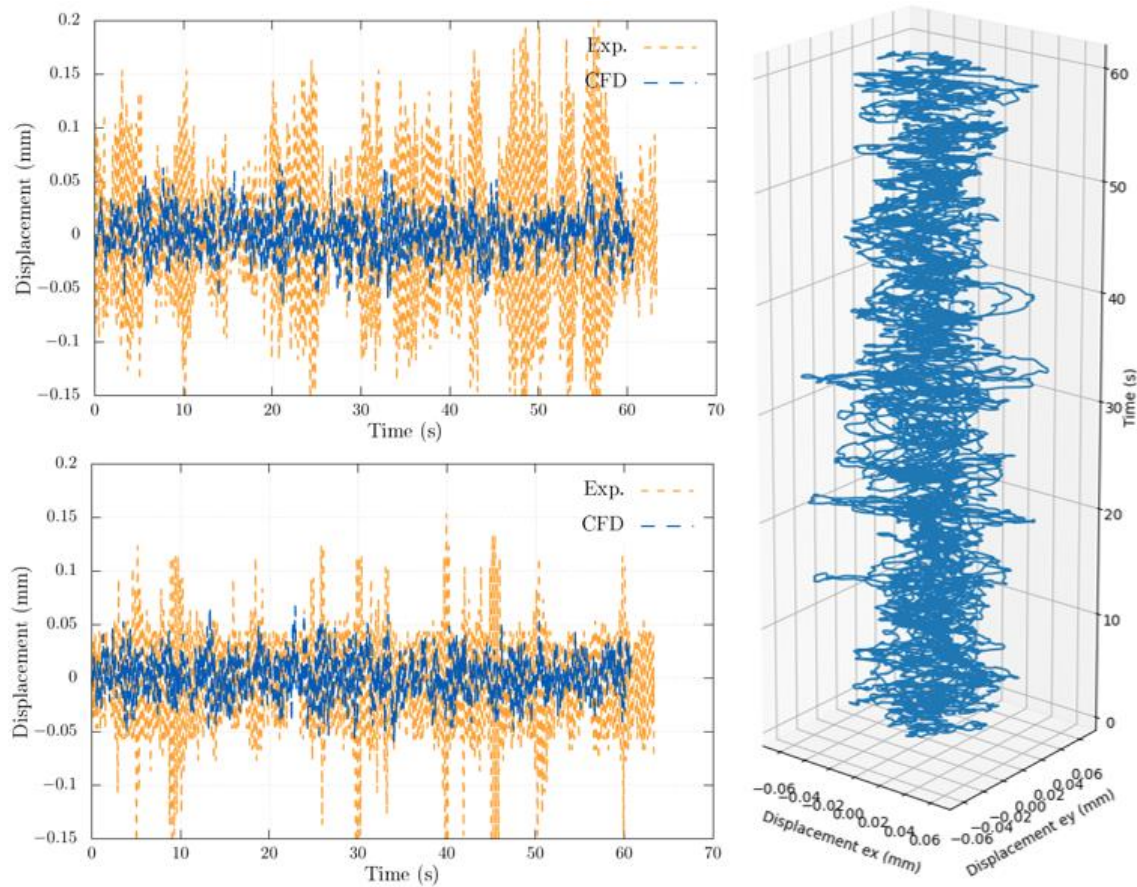


Figure 127 : Comparisons of the experimental and CFD tip displacements in the transverse e_x (top left) and e_y (bottom left) directions. 2D displacement of the tip obtained with CFD (right).

The Power Spectral Density (PSD) of the tip displacement in the x-direction is compared to the experimental data, the results are provided in Figure 128. The shapes of the PSD are in good agreement for $f \in [0\text{Hz}, 30\text{Hz}]$. The frequencies of the peaks obtained with the WR-LES, the experiment and the theoretical natural frequencies are provided in Table 28. The CFD and the experiment are in very good agreement for the two peaks available with the experimental data. The peaks frequencies slightly differ from the analytical natural frequencies.

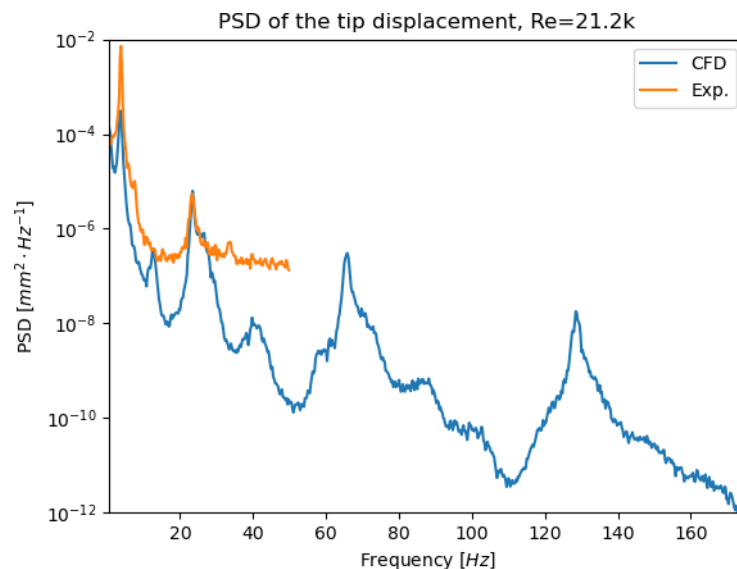


Figure 128. PSD of the tip displacement in the x-direction

Table 28. Frequencies of the numerical and experimental PSD peaks compared to the beam natural frequencies obtained analytically

Modes	#1	#2	#3	#4
CFD	3.70 Hz	23.3 Hz	66.0 Hz	129 Hz
Analytical	3.71 Hz	25.6 Hz	71.8 Hz	141 Hz
Experimental	3.72 Hz	23.3 Hz	N/A	N/A

6.4 Conclusions

A FSI coupling methodology between fine WR-LES simulations and a Euler Bernoulli beam equation model has been developed. The methodology does not impair dramatically the HPC performances of the LES simulations and allows one to take into account the solid motion in the calculation. The methodology has first been assessed with free vibration simulations in still water. The correct natural frequencies are recovered, and the mechanical model has been calibrated to recover the experimental damping ratio. Then, a turbulent cantilevered beam simulation at $Re_{p_H} = 21\,200$ has been performed. FSI and fixed rod simulations yield very close velocity and Reynolds stress profiles, which means that the rod vibration does not have a strong impact on the flow. There is a good agreement between the PIV results and the CFD ones although there is a dissymmetry in the experimental results, probably due to a misalignment of the beam in the experiment or a lack of convergence of the profiles. This dissymmetry is also noticeable in the time records of the tip displacement. The theoretical and experimental natural frequencies of the beam are recovered by the simulation and there is quite a good agreement with the experiment regarding the PSD shape. However, the amplitude of the displacements is smaller in the simulation than in the experiment by a 2.7 factor. This may be due to both the uncertainties at this Reynolds number value or a rod misalignment that disturbs the flow. The data post-treated in the present simulations are provided to the GO-VIKING partners for benchmarking and validation purposes in the context of the WP2 benchmark on the cantilever beam experiment.

7. Simulations performed by EDF-Energy

7.1 Introduction

The GO-VIKING project is a 4-year Horizon-funded project, a collaboration between many stakeholders across Europe, the US and Canada, tasked with investigating flow-induced vibration (FIV) in components found in nuclear power plants (steam-generators). Flow-induced vibration has received attention because of the long-term damage to structural components due to fretting and cracking incurred by this phenomenon. The project aims to develop methods and best practice guidelines to accurately predict FIV and its effects in components of nuclear power plants and assist the academic and industry experts of the GO-VIKING stakeholders in decision making. GO-VIKING is an extensive continuation of the VIKING project, which investigated similar themes.

EDF UK R&D contributes to two main work packages for the project: Work Package 2 (WP2) and Work Package 4 (WP4). WP2 is focused on the study of numerical Computational Fluid Dynamics simulations of flow-induced vibration of fuel assemblies subjected to single-phase axial flow, while WP4 investigates FIV in various configurations of multiphase flow.

The focus of this report is the numerical investigation of axial flow over a single cantilever rod, which falls under WP2, where experimental data is available (Cioncolini, et al., 2023) for the validation and verification.

The University of Manchester have conducted flow-induced vibration experiments of this test case and provide data for rod displacement to validate computational fluid dynamics (CFD) models (Cioncolini, et al., 2018) (Cioncolini, et al., 2023). This experiment is relevant for applications of flow-induced vibrations in water-cooled nuclear reactors, as the setup of the cantilever beam test rig mimics a fuel rod found in the fuel bundle section of a typical nuclear reactor core.

The experimental configuration consists of a vertical circular rod, as shown in the schematic of Figure 129. A vertical circular rod is inserted in a confining tube. The rod is clamped at the top end and free at the bottom end and allowed to vibrate, while the free-end of the vibrating rod may induce flow separation. In the experiment, there were three configurations of how the free-end was shaped: flat (blunt), hemispherical and conical. The vibrating rod is filled with lead shots to replicate the fuel pellets found in typical nuclear fuel bundles. The type of fluid used in the experiment was water. There were two configurations of the flow: flow from the fixed end to the free end (clamped–free) or flow from the free end to the fixed end (free–clamped) (Figure 130). This work focused on performing a three-dimensional numerical investigation of flow from the free end to the fixed end. The outer diameter of the vibrating rod is 10 mm, while the confining tube has an internal diameter of 21 mm resulting in an annulus hydraulic diameter of the gap between tube and rod being 11 mm. A flow straightener is inserted at the entrance of the confining tube upstream of the free-end of the vibrating rod. The flow straightener is used to reduce secondary flows and produce a uniform flow field upstream of the vibrating rod. The exit of the flow straightener is located 2.5 m away from the vibrating rod fixed-end, with the total length of the vibrating rod with the blunt end being 1.06 m.

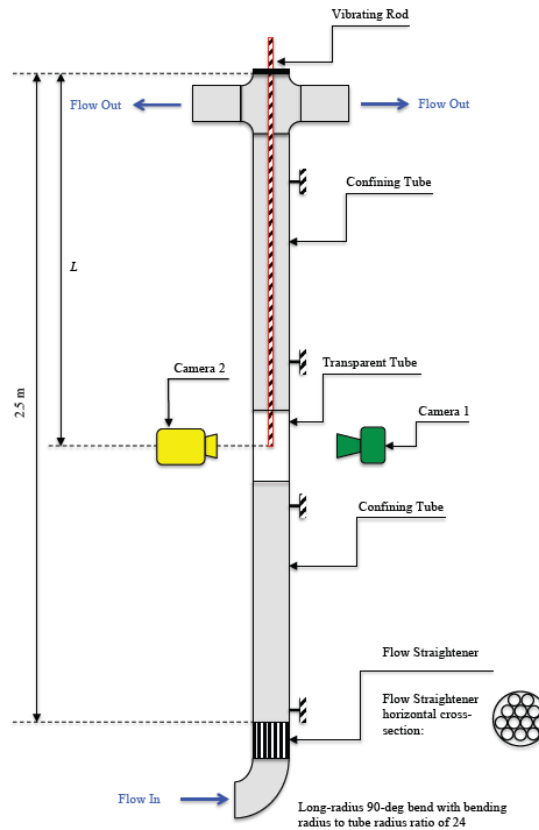


Figure 129: Schematic of the cantilever rod experiment (Cioncolini, et al., 2018)

This report demonstrates the validation of the results generated from the medium-resolution CFD models compared against the data produced by the experiments. This is important as the typical Reynolds numbers found in pressurised water reactors (PWRs) are very high, which precludes the use of high-fidelity numerical methods for the whole domain as it would be computationally expensive. Three simulations were conducted and validated against the experimental data available for three different Reynolds numbers. Two were done for the cantilever rod with the blunt end, while the third one was done for the rod with the hemispherical end.

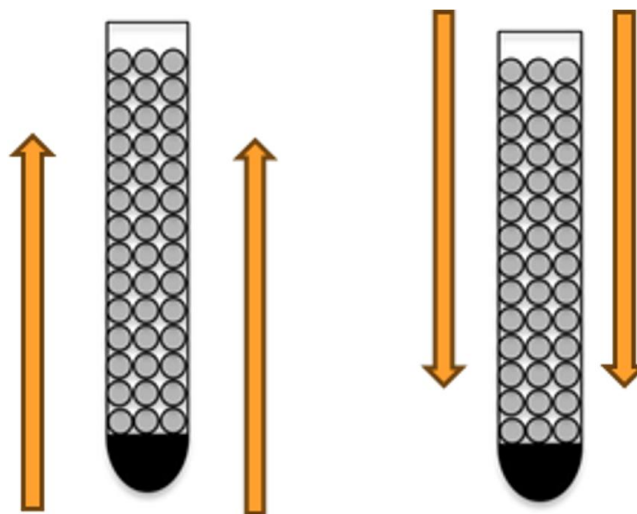


Figure 130: Flow configurations of Free-clamped (left) and clamped-free(right) .

Medium resolution validation of the cantilever rod experiment was done using the CFD package STAR-CCM+. STAR-CCM+ is used for the CFD computation of the fluid domain and finite element analysis (FEA) for the solid domain; thereby, the need to couple a separate fluid solver to a solid one for this numerical analysis is negated. Unsteady Reynolds-Averaged Navier Stokes (URANS) was used for the closure of the momentum transport equations for the fluid domain with the SSG Reynolds stress model (RSM).

This work demonstrates the suitability of medium resolution methods being able to capture the features of the axial flow through the cantilever rod. It also investigates the displacement dynamics of the cantilever rod for the coupled fluids/solids solver and verifies if the experimental data can be accurately predicted.

7.1.1 Scope of Current Work

This report presents the results of three numerical cases conducted. Three Reynolds numbers have been assessed for the test cases, as shown in Table 29. The Reynolds number is defined based on a constant-density fluid and the hydraulic diameter of the pipe. Two of the cases focused on the blunt end, while the third case used the hemispherical end.

Table 29: Characteristics of the three test cases

	Case 1	Case 2	Case 3
End type	Blunt	Blunt	Hemispherical
Reynolds number	16,000	42,000	21,000
Bulk velocity (m/s)	1.30	3.31	1.59
Fluid Domain (Water)			
Density (Kg/m³)	997.00	997.561	997.57
Dynamic Viscosity (Water) (Pa•s)	0.00089	0.001047	0.001047
Solid Domain (Stainless Steel)			
Density (Kg/m³)	7489	7490	7490
Young Modulus (GPa)	75.4	76.46	76.46
Poisson's Ratio	0.25	0.265	0.25



7.2 Methodology

STAR-CCM+ uses finite volume discretisation for the fluid domain, while finite element discretisation is used for the solid region. STAR-CCM+ version 17.04 was used and adopts a partitioned approach for fluid-structure interaction calculations where the fluid and solid domains are solved separately but coupled strongly. At the interface between the fluid and solid domains, energy and momentum are conserved.

7.2.1 Configurations and boundary conditions

7.2.1.1 Solid Domain

The Solid Stress model was used to perform the structural analysis of a cantilever beam in STAR-CCM+. For the Solid Stress Solver, the Max Norm was selected as the type of norm used in the stress residual calculation. The option for strain energy was deactivated. For all three cases, a stainless-steel material was selected for the rod. Note that the structure of the rod in the experiment consists of a shell filled with dense lead shots; however, we simplify the model by modelling the rod as a solid with homogeneous properties. The equivalent homogeneous properties were taken from (Salachna, et al., 2023): a density of 7490 kgm^{-3} and a Young's modulus of 76.4 GPa. Though these are not the real physical properties of the materials, they produce the same characteristic frequencies. A more detailed validation of the solid model was detailed in a prior progress report (Harris, 2023), which demonstrated suitable frequencies and damping coefficients of the structure in both vacuum and still water conditions. The material law models selected were: "Isotropic Linear Elasticity", "Linear Elasticity" and "Linear Strain (Small Strain)". The boundary condition for the cantilever rod was specified as symmetry, while an FSI Interface was set for the boundary condition of the remainder of the rod surface. A Fixed Solid Stress Constraint was specified for the cantilever rod base. The motion specification for the cantilever rod was set to Solid Displacement.

7.2.1.2 Fluid Domain

The unsteady Reynolds-average Navier-Stokes (URANS) method was followed to solve the fluid domain for all three cases. A segregated approach was adopted for the calculations. For the time scheme, the implicit unsteady solver was used with the temporal discretisation being second – order. This work made use of the SSG Reynolds Stress turbulence model (RSM) (Speziale, et al., 1991). As low – medium resolution computational grids were used for the fluid domains, the "High y^+ treatment" was selected. A MUSCL 3rd order central – differencing scheme was selected to discretise convective terms in the momentum transport equations. On the other hand, an upwind scheme was selected for convective terms for the Reynolds stresses transport equations. The gradient limiter selected was the Modified Venkatakrishnan, as this option was deemed less dissipative than the default option in unsteady turbulent simulations. The SIMPLE scheme was selected for the pressure – velocity coupling method.

A precursor simulation was conducted to generate the main inlet boundary conditions for the velocity, turbulent dissipation rate and Reynolds stress terms. The inlet condition is selected as "Velocity Inlet" with prescribed values of velocity and turbulence parameters at the inlet surface. Depending on the case, "Outlet" or "Pressure Outlet" are specified as the outlet boundary condition. The outlet is treated as a single boundary surface with the outlet

boundary being the annulus around the rod at the exit. This configuration differs from the experimental setup, which had two outlets in a union-cross formation, but it was shortened for simplicity under the assumption that flow downstream of the rod in the union cross would have little effect on turbulence at the rod tip one meter upstream. The fluid domain surface that interfaced with the cantilever rod was specified as an FSI interface.

7.2.2 Fluid – Structure Interaction Scheme

In STAR-CCM+, fluid-solid coupling is achieved through a partitioned approach for both the fluid and solid domain calculations. A strong coupling is used for the interface between the solid and fluid domains. This strong coupling is done by specifying the option "Fluid Load - Pressure & Wall Shear Stress" in the solid region parameters. This option ensures that fluid forces and solid forces are equal at the interface. Furthermore, mapping at the interface between the solid and fluid domains in STAR-CCM+ ensures that velocity and displacement are continuous across the interface, and overlapping of the domains is prevented.

The cases tested used both conformal and non-conformal interfaces between the solid and fluid domains. There was no noticeable difference in computational performance between the two approaches. The advantage of a non-conformal mapping is that it allows for a higher solid mesh quality for robustness, benefiting the FEA computation for the FSI part and a smaller mesh size for the fluid domain. Hence, a non – conformal interface was employed for two of the three cases, particularly as the fluid domain needed to be refined around the cantilever rod tip where flow acceleration and separation occurred.

7.2.3 Mesh Deformation

The mesh needs to deform for the fluid-structure interaction calculation, achieved by activating the option for deformation of the fluid and solid domains at every timestep. In StarCCM+, the deformation methods for the fluid and solid domains are activated in the "Tools Folder". The deformation parameter for fluid domain motion is called "Morphing" in STAR-CCM+, while the parameter for solid domain motion is called "Solid Displacement". After these deformation parameters have been activated, they must be specified in fluid and solid regions in STAR-CCM+ by labelling "Fixed" surfaces and morphing surfaces.

To improve the stability of the calculation, the additional effect of a displaced mass of fluid acting on the solid domain is computed. This displaced fluid mass is added to the solid domain. Prior progress reports detail the calculation of this "boundary added mass" from theoretical principles. STAR-CCM+ activates this additional effect through the specified value of the ratio of the displaced fluid volume to wetted surface area. It should be noted that STAR-CCM+ has the option of automatically calculating this value.

7.2.4 Hemispherical/round tip case

One hemispherical tip or rounded end case was investigated. The Reynolds number based on the annulus hydraulic diameter was 21,000. The fluid domain is made of a hexahedral grid as seen in Figure 131 and Figure 132. The mesh is heavily refined around the rod tip, as this is where gradients were strongest for the prior case. The axial refinement is constrained by the time step and the CFL limit of 1. Likewise, the radial refinement was dictated by the SSG wall

function requirement of $y^+ > 30$. Azimuthal refinement was chosen to be similar to the other directions to keep aspect ratios close to 1, under the assumption that azimuthal gradients will be smaller than other directions, given the highly axisymmetric nature of the geometry and bulk flow. Initial simulations with high wall refinement failed to capture the vibration behaviour.

Thus, based on the promising results of earlier work of (Salachna, et al., 2023) in modelling the blunt tip cases using wall functions, the team explored a similar approach, which limited wall y^+ to within the wall function range. Systematic mesh refinement studies were not performed. The number of cells in the fluid domain was 200,000 with no refinement near the walls; consequently, this was a medium resolution grid to investigate the suitability of a cost – effective URANS modelling methodology for accurately predicting flow-induced vibration of the cantilever beam subject to axial turbulent flow. The average y^+ was 31.6 with the maximum value being 39.4. Hence, standard wall function was used with the SSG RSM turbulence model. The timestep size used for the calculations was 0.00015 s to satisfy a CFL less than unity.

The solid domain had 62,500 cells. The cylindrical part of the rod used a structured hexahedral mesh, while the hemispherical tip used an unstructured tetrahedral mesh, as demonstrated in Figure 3 below. The choice of an unstructured tip mesh was justified, given that stresses and strains in the rod tip are near-zero and, as such, the main requirement for the tip hemisphere is to match the complex geometry rather than to accurately model any stresses that may affect the vibration characteristics of the rod. The parameters are summarised in Table 30.

Table 30: Summary table of fluid mesh properties for Round $Re = 21,000$ case

	Value
Fluid Mesh size	200000
Solid Mesh size	62500
Maximum Wall y^+	39.4
Surface Average Wall y^+	31.6
Near-wall y^+ model / Wall Treatment Model	Standard Wall function
Maximum Courant Number	0.5
Volume Averaged Courant Number	0.1
Number of inner iterations per time step	25
Time Step Size (s)	0.00015
Total Simulation Time (s)	11

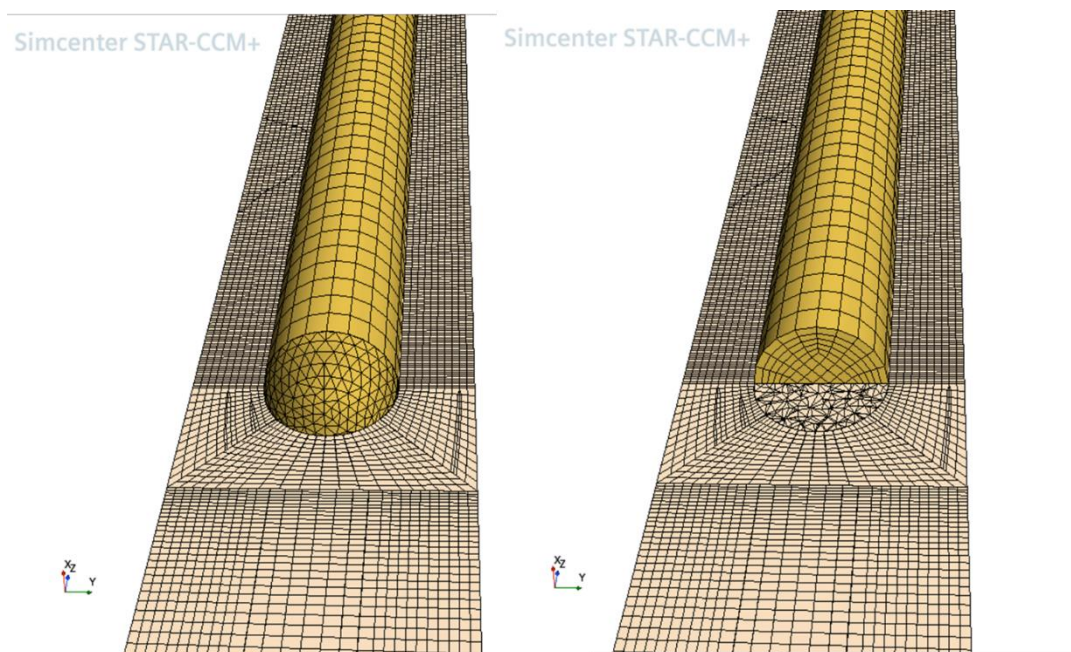


Figure 131: Mesh for the Hemisphere end for $Re=21000$ case: solid domain and section view of fluid domain in rod tip region. The tip region is included in both solid (left) and section (right) views.

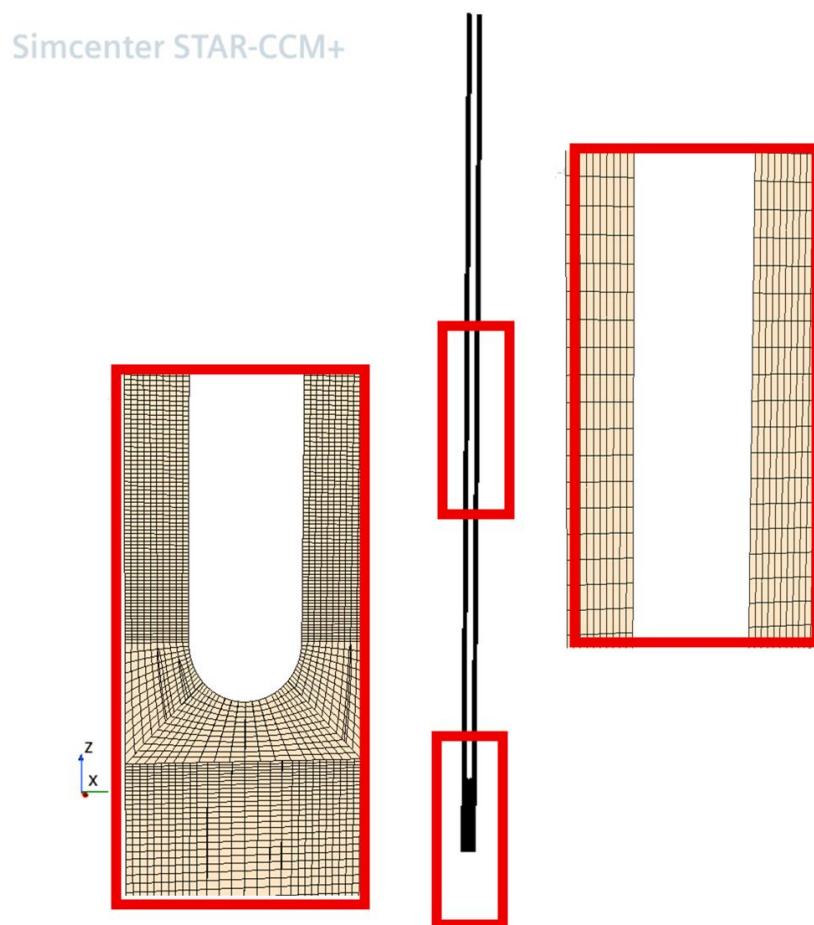


Figure 132: Close - up view of the structured mesh used for the fluid domain for the Hemisphere end $Re=21000$ case.

7.2.5 Blunt tip cases

Two blunt end cases were investigated. The two Reynolds numbers studied were 16,000 and 46,000. Details of the two cases are presented below:

7.2.5.1 $Re = 16,000$ Blunt tip case

For the $Re = 16,000$ blunt tip case, the fluid domain uses a structured hexahedral grid as seen in Figure 133. The number of cells in the fluid domain is 170,000, with no refinement near the walls; consequently, this mesh utilised a medium resolution grid. Similarly to the hemispherical meshing strategy, mesh refinement was concentrated at the rod tip where gradients are strong, while downstream of the rod tip, mesh refinement reduces toward the base of the rod. The average y^+ was 32, with the maximum value being 100. Hence, the standard wall function was used with the SSG RSM turbulence model. The timestep size used for the calculations was 0.00015s; the average and maximum Courant numbers for the computations were 0.0953 and 1.01, respectively. The fluid properties of this case are slightly different to the other two since this case was performed before standard values were agreed upon by all participants in WP2. The density and viscosity for this case are 997 kg m³ and 0.00089 Pas, respectively. The fluid properties of the other cases were set as 997.57 kg m³ and 0.001047 Pa s, for the density and dynamic viscosity respectively. Fluid mesh parameters are summarised in Table 31.

The solid domain remained very similar to that of the rounded tip case. It had 33,920 cells; this low grid resolution is justified due to the relatively simple nature of the solid domain aspect of the case when compared to the complexity of the fluid domain aspect. The solid domain made use of a structured hexahedral mesh. Unlike the rounded tip with its hemispherical geometry, this blunt tip has a simpler geometry such that the entire solid domain could be meshed as a single, unpartitioned, structured mesh.

Table 31: Summary table of Mesh Parameters for Blunt $Re = 16,000$ Case

	Value
Fluid Mesh size	166999
Solid Mesh size	33920
Maximum Wall y^+	101
Surface Average Wall y^+	32
Near-wall y^+ model / Wall Treatment Model	Standard Wall function
Maximum Courant Number	1.01
Volume Averaged Courant Number	0.0953
Number of inner iterations per time step	40
Time Step Size (s)	0.00015
Total Simulation Time (s)	10.5

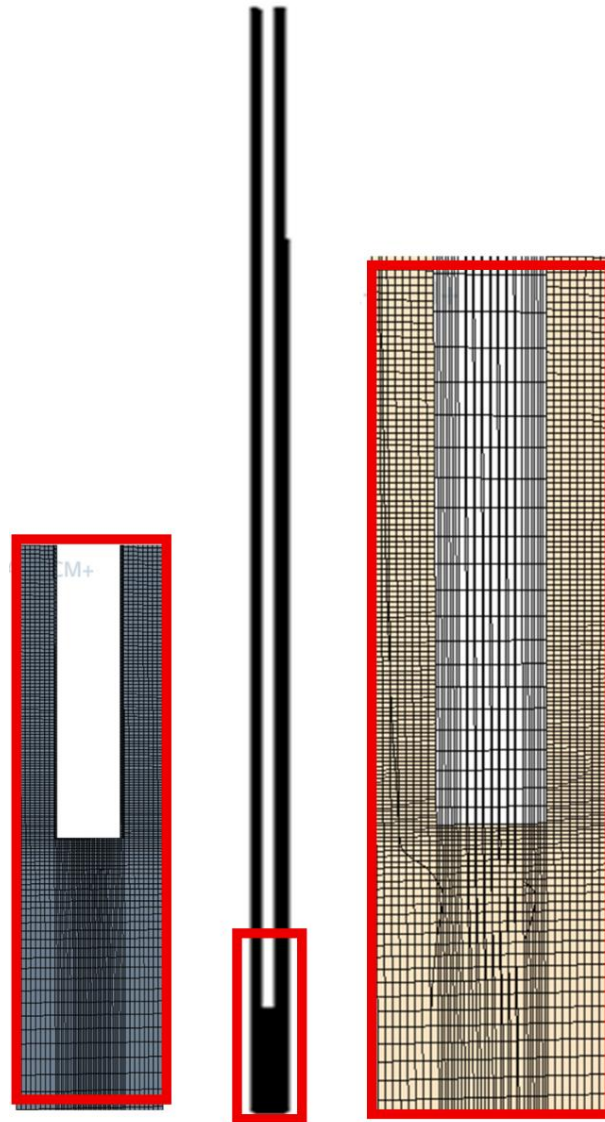


Figure 133: Mesh Structure for the Blunt 16k case: fluid domain in rod tip region (left), full fluid domain with tip region highlighted (centre), both fluid and solid domains in rod tip region

7.2.5.2 $Re = 46,000$ Blunt tip case

For the $Re = 46,000$ blunt tip case, the fluid domain uses a structured hexahedral grid. The number of cells in the fluid domain is 700,000, with no refinement near the walls (average y^+ of around 34); consequently, this mesh utilised a medium resolution grid. The mesh immediately upstream of the blunt tip is refined in the axial and radial directions to better capture the flow impingement on the free – end and the flow separation around the cantilever rod. Hence, the standard wall function was used with the SSG RSM turbulence model. The timestep size used for the calculations was 0.00005 s; the average and maximum Courant numbers for the computations were 0.9 and 1.7, respectively. The parameters are summarised in Table 32 below.

Table 32: Summary table of Mesh Parameters for Blunt $Re = 46,000$ Case

	Value
Fluid Mesh size	700000
Solid Mesh size	187000
Maximum Wall y^+	174
Surface Average Wall y^+	34.4
Near-wall y^+ model / Wall Treatment Model	Standard Wall function
Maximum Courant Number	1.7
Volume Averaged Courant Number	0.9
Number of inner iterations per time step	20
Time Step Size (s)	0.00005
Total Simulation Time (s)	20

The solid domain had 187,00 cells. The solid domain made use of a structured hexahedral mesh. The solid model for this case adopted a slightly different approach from the other cases. The solid model was designed to map conformally to the fluid domain at the FSI interface. Due to this conformality, the mesh has more cells than for the other cases to match the refinement of the fluid mesh at the interface.

7.3 Results and discussion

This section presents the results of the three cases of the cantilever beam subjected to axial turbulent flow. Figure 134-Figure 136 show the time series of the displacement of the free – end of the cantilever beam for the blunt tip $Re = 16,000$, Rounded tip $Re = 21,000$ and blunt tip $Re = 46,000$ cases, respectively. The Rounded tip $Re = 21,000$ case was observed to have negligible displacement of the free – end. Furthermore, the negligible displacement for the rounded tip case was still observed when the medium – resolution fluid domain mesh was refined to 450,000 and 642,000 cells. For the blunt end cases, vibration of the cantilever beam was observed. The simulation of the higher Reynolds number case of 46,000 was seen to become numerically unstable. On the other hand, the simulation of the Blunt tip $Re=16,000$ case maintained its stability throughout the entire duration of computation.

Table 33 summarises the reduced root mean square (RMS) amplitude (RMS amplitude/rod diameter)of the simulations against the experiment. As already noted, the reduced RMS amplitude of 0.000123 predicted for the round tip case using the medium-resolution grid is much smaller than the experiment value of 0.007. Conversely, the reduced RMS amplitude predicted by the blunt end cases is much closer to the experimental values. For the blunt tip $Re = 16,000$ case, the reduced RMS amplitude of 0.00269 is a close match to the experimental value range of 0.004 – 0.006. For the higher Reynolds number case of 46,000, the reduced RMS amplitude of 0.0928 is about three times the value from the experiment (0.02865). This overprediction is most likely due to the excess displacement of the cantilever rod during periods when the simulation became unstable. This poor comparison for the round tip case demonstrates the need for further investigation about the reasons why the medium resolution methodology using the SSG turbulence model is failing for this case, in contrast to the blunt end cases.

Table 33: Summary table of reduced RMS displacement amplitudes for each case compared to experimental values.

Reduced RMS displacement amplitude (A_{RMS}/d)	Blunt $Re = 16,000$ Case	Round $Re = 21,000$ Case	Blunt $Re = 46,000$ Case
X	0.00282	0.000075	0.0695
Y	0.00255	0.00017	0.1161
Mean	0.00269	0.000123	0.0928
Experimental Value	0.004 – 0.006 (Cioncolini, et al., 2018)	0.007 (Cioncolini, et al., 2023)	0.02865 Invalid source specified.

Table 34: Table of peak frequencies extracted for the PSD plots of each case.

Frequency	Blunt $Re = 16,000$ Case	Round $Re = 21,000$ Case	Blunt $Re = 46,000$ Case
X	3.25	3.5	3.4
Y	3.75	3.5	3.5
Mean	3.5	3.5	3.45
Experimental Value	3.6 – 3.75 (Cioncolini, et al., 2018)	3.5 – 3.9 (Cioncolini, et al., 2023)	3.45 Invalid source specified.

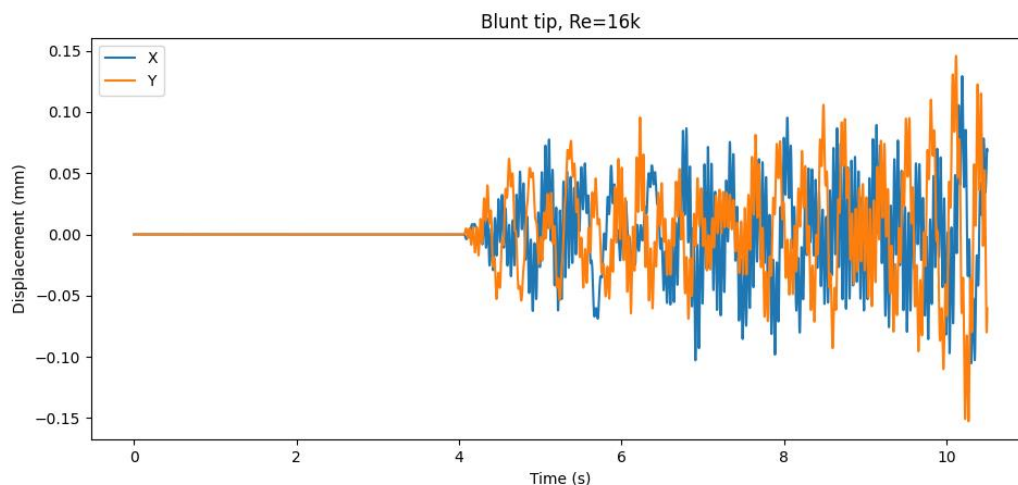


Figure 134: Blunt Tip $Re = 16,000$ case. Displacement of the free-end vibrating beam in the x and y directions.

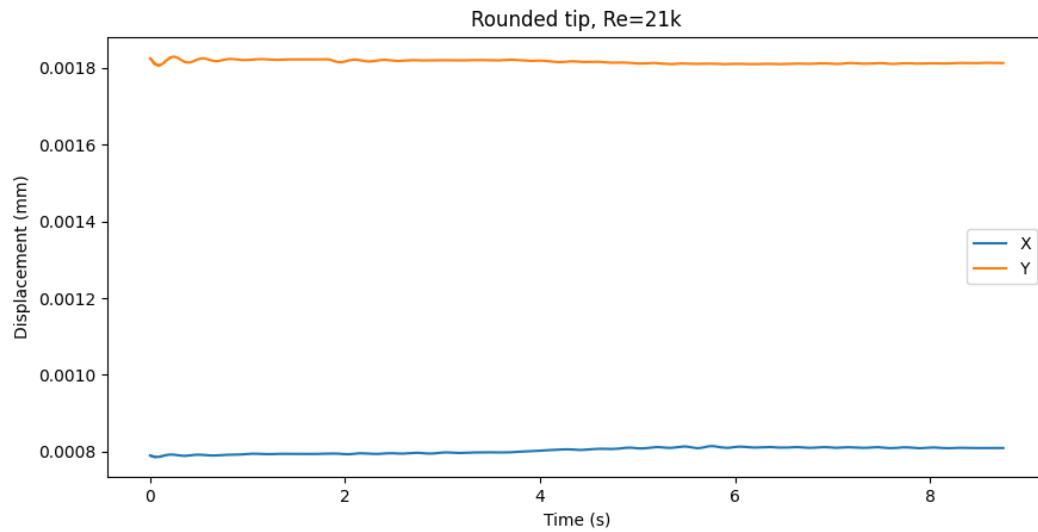


Figure 135: Rounded Tip $Re = 21,000$ case. Displacement of the free-end vibrating beam in the x and y directions.

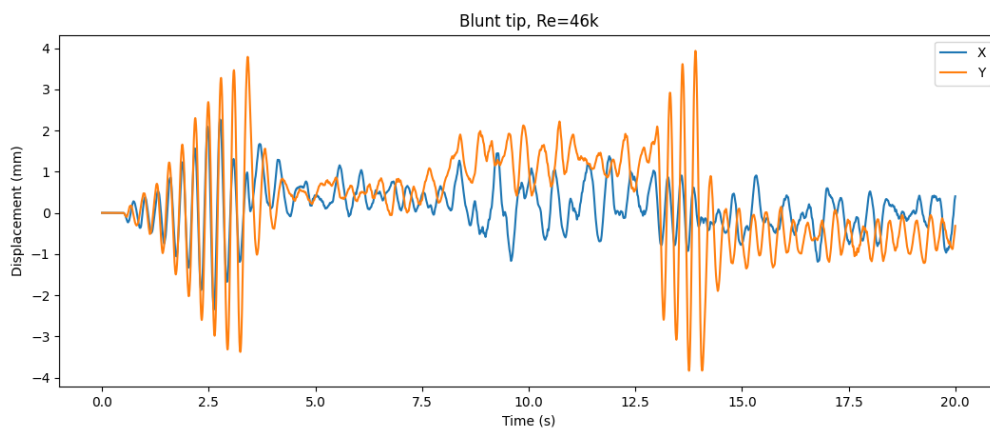


Figure 136: Blunt Tip $Re = 46,000$ case. Displacement of the free-end vibrating beam in the x and y directions.

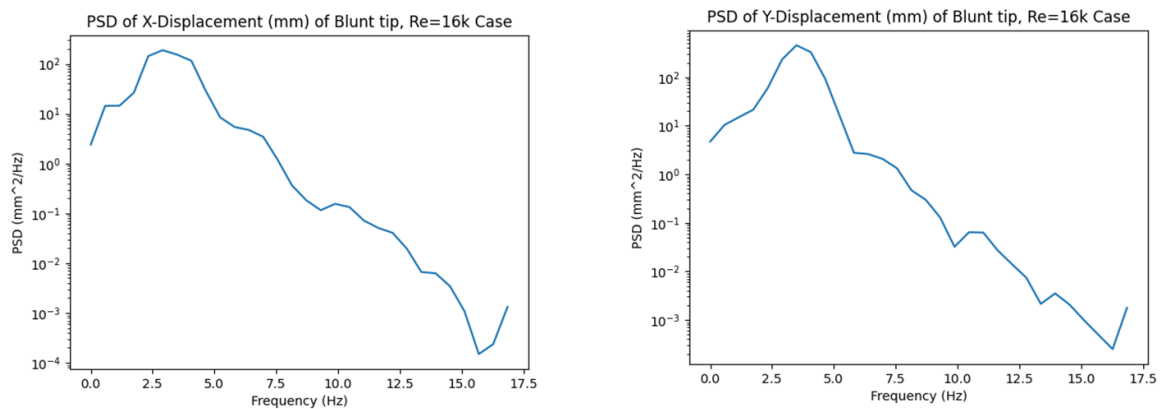


Figure 137: Plot of Power Spectral Density function of Displacement in X and Y axes for Blunt $Re = 16,000$ Case.

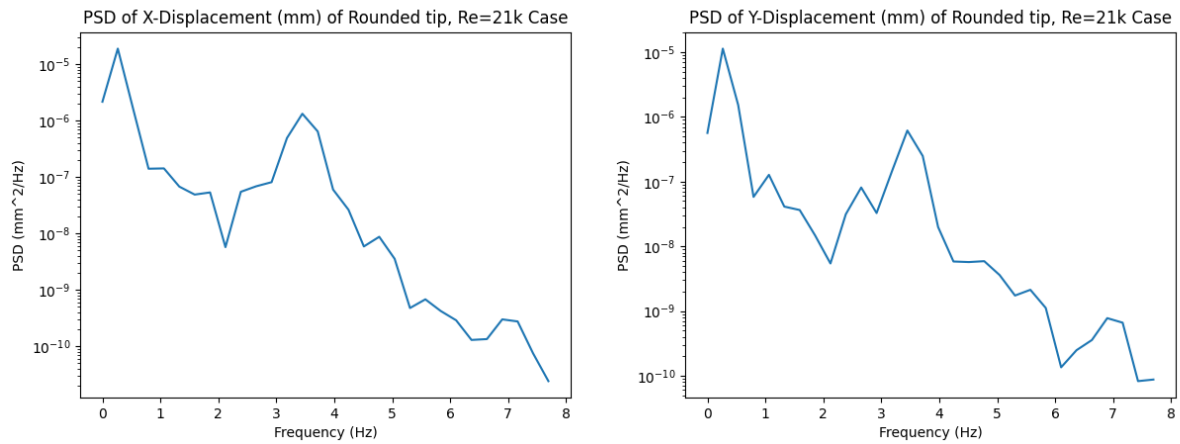


Figure 138: Plot of Power Spectral Density function of Displacement in X and Y axes for Round $Re = 21,000$ Case.

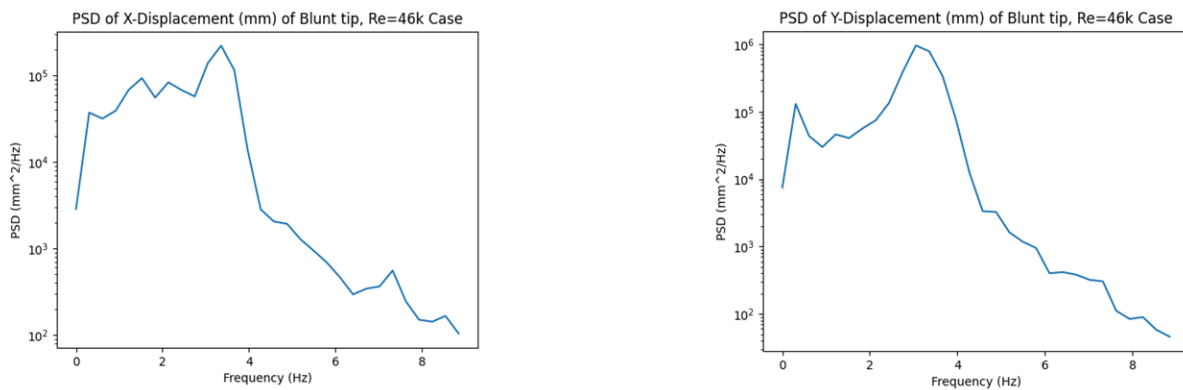


Figure 139: Plot of Power Spectral Density function of Displacement in X and Y axes for Blunt $Re = 46,000$ Case

Table 34 shows the summary of the natural frequencies predicted from all the test cases and a comparison against the experimental values. The plots of power spectral density of the displacement in the X- and Y- directions for the three cases are shown in Figure 137-Figure 139. The natural frequencies of the displacement of the cantilever rod predicted for the blunt end cases are in close agreement with the experimental data. The level of agreement is satisfactory considering the computationally cost-saving grid from the URANS approach adopted for the predictions of the blunt end geometry. It is worth noting that the natural frequency for the rounded tip case (~ 3.5 Hz), whose vibration of the rod is negligible, is in close comparison with the experimental value of 3.5 – 3.9 Hz. This shows that despite the medium resolution methods with URANS not being able to capture the amplitude of displacement of the round tip case, it can still accurately predict the vibration frequency of the rod.

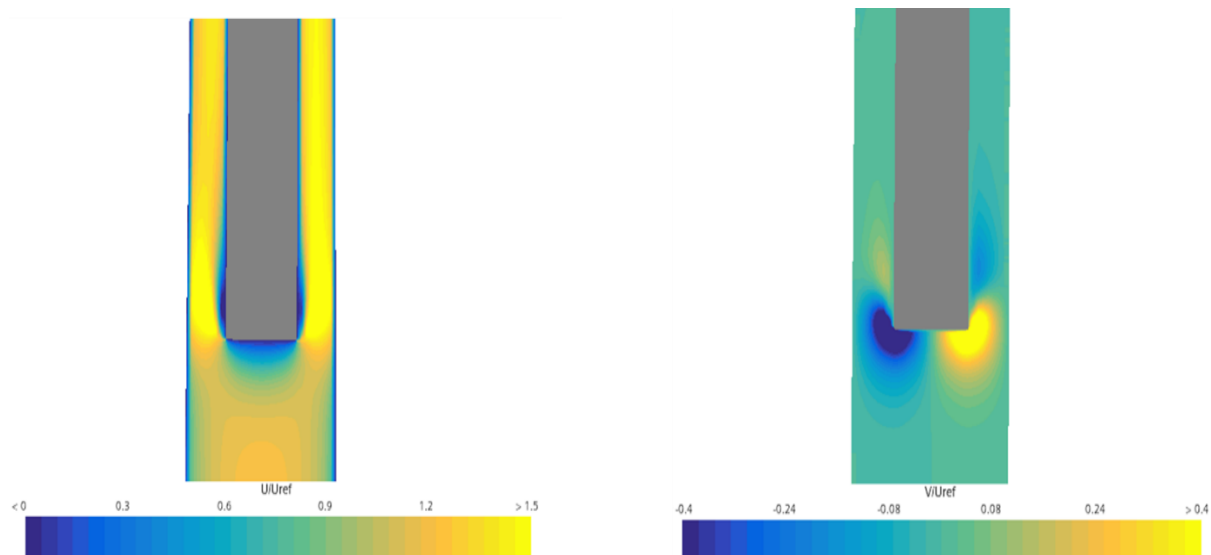


Figure 140: Contour plots of normalised axial (left) and radial (right) velocities for Blunt $Re = 16,000$ case.

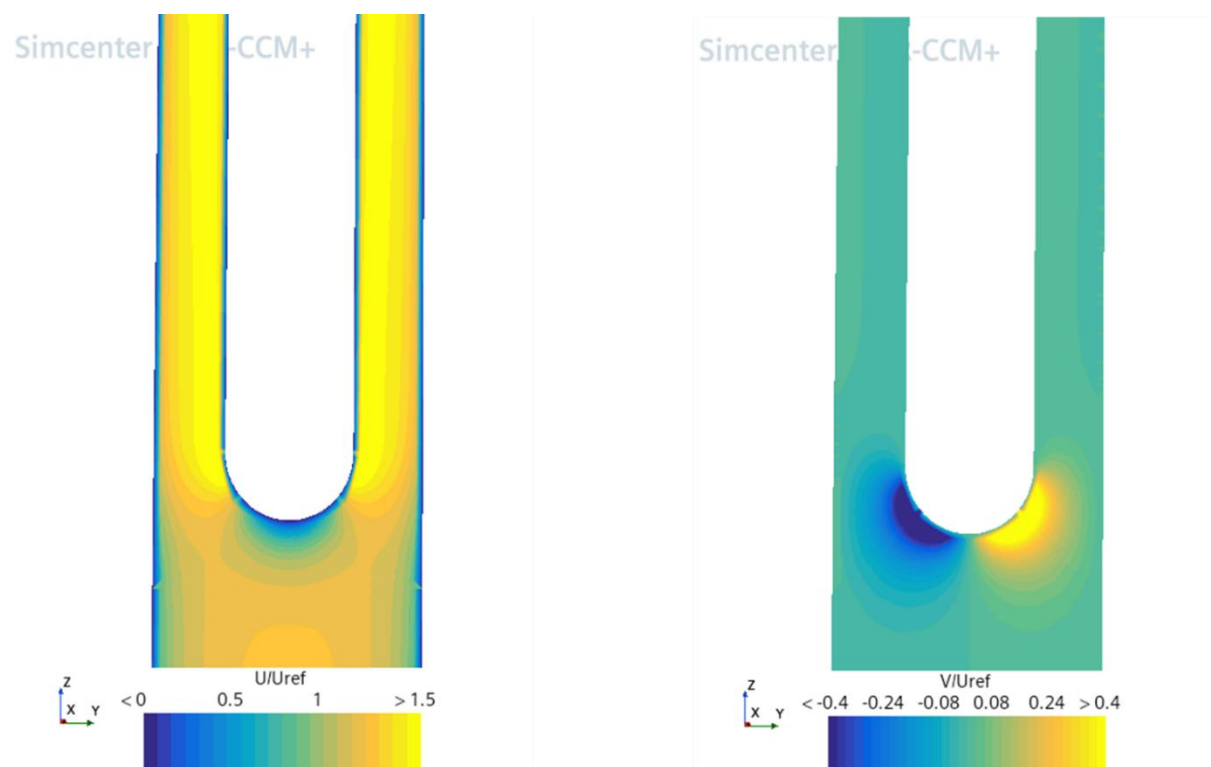


Figure 141: Contour plots of normalised axial (left) and radial (right) velocities for Round $Re = 21,000$ case.

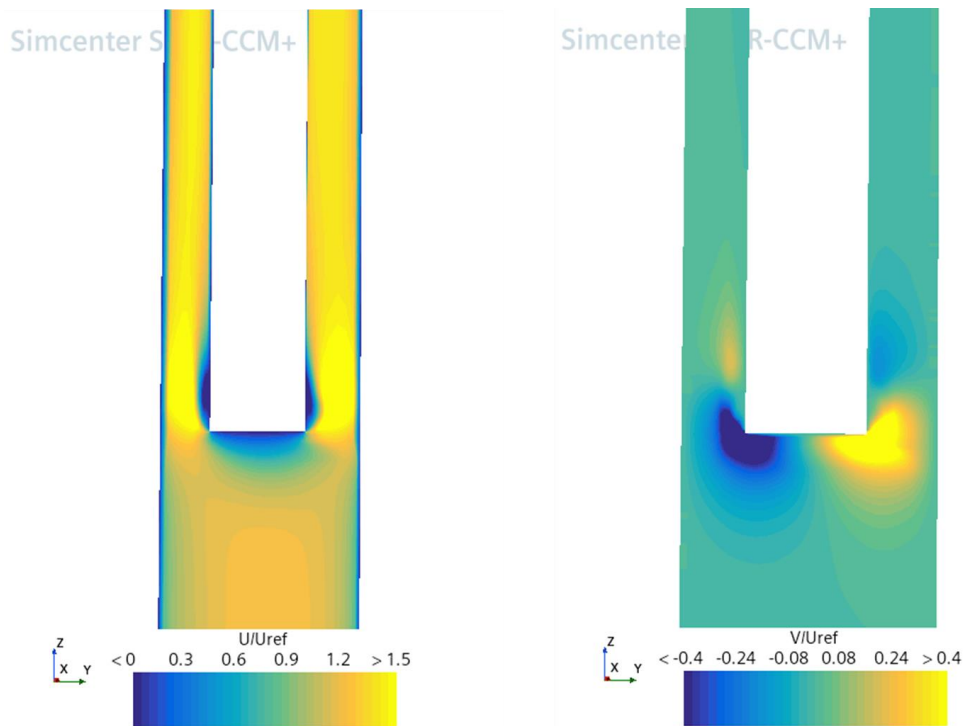


Figure 142: Contour plots of normalised axial (left) and radial(right) velocities for Blunt $Re = 46,000$ case.

Figure 140-Figure 142 show the contours of the normalised streamwise and cross-stream velocity for all three cases. For the blunt – tip cases, the streamwise velocity contour plots show a wider surface area of the impinging point, which is around the flat boundary of the free-end. The impingement surface area for the rounded tip is smaller compared to the blunt tip cases. A reason for this is that the changes in the flow direction and pressure in the rounded end are not predicted well with the current model, as in the blunt end case. This is reflected in Figure 143, where the amount of turbulent kinetic energy is underpredicted close to these regions compared to the other two test cases.

Furthermore, there is noticeable flow separation and reattachment immediately downstream of the cases of the blunt tip free – ends from the velocity contour plots. This flow separation and reattachment is not seen for the round tip case. Flow separation acts to increase turbulence in the flow, which leads to more excitation of the cantilever rod and greater displacement. As this feature is not observable for the round tip case, there is a possibility that the turbulent kinetic energy generated for this geometry is lower. The medium – resolution grid coupled with using a RANS turbulence model might struggle to predict this reduced turbulent kinetic energy around the round-tip free – end, which is leading to a much reduced vibration of the cantilever beam in the simulations. This observation is confirmed in Figure 143, where contours of the normalised instantaneous turbulent kinetic energy predicted by the SSG turbulence model for the three cases show a very suppressed turbulent kinetic energy profile for the round tip case. This suppression suggests that the Reynolds stress model is struggling to predict to correct levels of turbulence for the round tip case. There is a need to investigate if using much higher resolution grids, which can sufficiently resolve turbulence around the round tip with RANS methods, will improve the prediction of flow-induced vibration for this case. Although this strategy will significantly increase the computational costs of the URANS methodology.

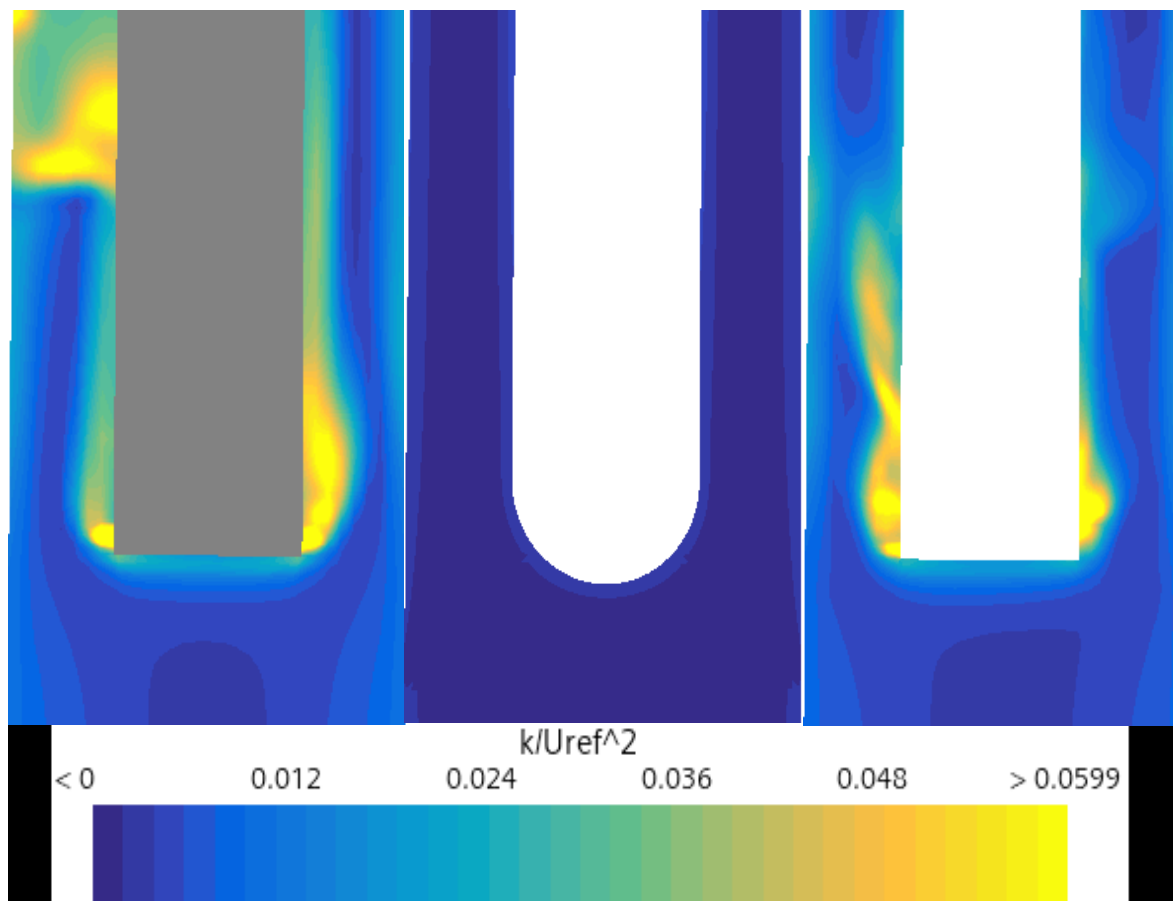


Figure 143: Instantaneous turbulent kinetic energy contour plots for annular Reynolds numbers of $Re = 16,000$ case (left), $Re = 21,000$ case (centre) and $Re = 46,000$ case (right)

Figure 144 includes plots of velocity profiles in the key region immediately downstream of the rod tip, included for completeness and reference. This figure further supports the discussions of the previous subsection.

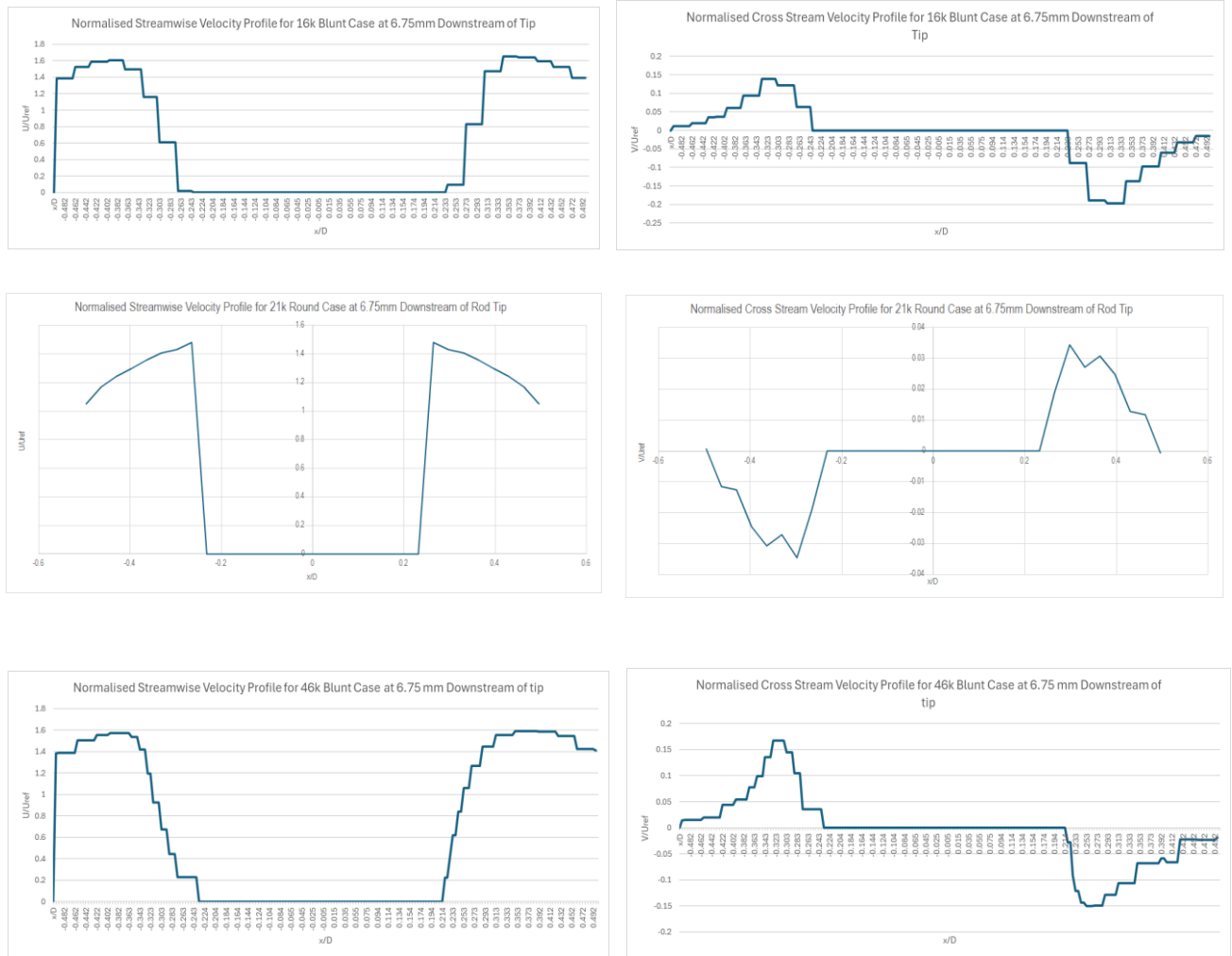


Figure 144: Plots of Horizontal Velocity profiles at 6.75mm downstream of the rod tip for streamwise (left column) and cross stream (right column). The plots are arranged in rows; Blunt $Re = 16,000$ (top row), Round $Re = 21,000$ (middle row), Blunt $Re = 46,000$ (bottom row).

7.4 Conclusions

The GO-VIKING project is tasked with investigating flow-induced vibration in components in nuclear reactors. This project is a collaboration between academic and industrial institutions found across Europe, the US and Canada. This project aims to develop best practice guidelines to better predict and understand the effects of flow-induced vibration in nuclear power plants.

The focus of this section is Work Package 2, where the flow-induced vibration of fuel assemblies subjected to single-phase axial flow is studied. EDF R&D UK is responsible for performing low/medium resolution computations and performing comparisons against the experimental data available for three different test cases.

In this section, we have presented the results for the fluid-structural interaction analysis of a cantilever rod subjected to axial flow with two different configurations: a cantilever rod with a blunt/flat free end, and another rod with a hemispherical or round free end. Three cases

were simulated in total; two were done for the blunt tip at Reynolds numbers based on the annulus hydraulic diameter, $Re = 16,000$ and $Re = 46,000$. One case was simulated for the round tip at $Re = 21,000$. We investigated the prediction of flow-induced vibration of the rod when using unsteady Reynolds-averaged Navier-Stokes equation methods in a low-resolution fluid domain. All simulations were conducted in STAR-CCM+ computational fluid dynamics software. STAR-CCM+ conducts CFD computations for the fluid domain using finite volume discretisation and finite element analysis of the solid domain using finite element discretisation. A strong two – way coupling between the fluid and solid domains was activated for all simulations. This is to ensure that forces and velocities at the interface between the two domains are balanced. The Reynolds stress turbulence model SSG was used in the fluid domain.

The medium resolution methodology was able to predict turbulent flow-induced vibration for the two blunt tip cases, with the natural frequency of the rod vibration being accurately predicted. For the blunt tip $Re = 16,000$ case, the root mean square (RMS) amplitude 0.00269 was in close agreement with the experimental data 0.004 – 0.006. For the higher Reynolds number blunt tip $Re = 46,000$ case, the RMS amplitude 0.0928 is slightly overpredicted compared to the experiment 0.02865. This overprediction was due to instabilities in the simulation for this case, which led to a marked increase in the displacement of the rod with the blunt – tip at certain phases of the calculation, thereby affecting accuracy.

Conversely, the round tip case failed to predict vibration of the cantilever rod when subject to axial turbulent flow; this failure led to the prediction of the RMS amplitude 0.000123 being much smaller than the experimental value of 0.007. Although the natural frequency of the cantilever rod vibration was well predicted. The failure of predicting the amplitude of displacement of the round tip free end was even observed when the medium resolution grid for the fluid domain was doubled and tripled. The failure of the round tip case in predicting flow-induced vibration using medium resolution grids compared to blunt tip cases must be noted, and can be attributed to the specific geometry configuration. The blunt tip case has a much larger surface area where the axial flow impinges onto it, compared to the round tip case. This larger impingement area for the blunt tip case leads to a higher generation of turbulent kinetic energy for this geometry compared to the round tip case. Furthermore, the medium resolution methodology predicted flow separation and reattachment around the blunt tip and none for the round tip. Consequently, the generation of turbulence around the cantilever free end, which is one of the instigating factors for vibration, is potentially lower for the round tip case than the blunt tip one. Subsequently, the medium resolution methodology struggled to accurately predict the turbulence levels around the rod free end for the round tip case; this led to flow-induced vibration not being predicted. As a result, for hemispherical tip configurations, there is a need to significantly refine the computational grid around the rod free-end to investigate if flow-induced vibration can be achieved, though this approach will markedly increase computational costs.

The low/medium resolution methodology that was assessed in STAR-CCM+ using the SSG RSM turbulence model was successful in predicting flow – induced vibration for the blunt tip cases. There is a need to explore other methods for improving the prediction capabilities for the round end cases without drastically increasing the computational grid requirements of the fluid region. Exploring data – driven methods could be an approach to improve accuracy while keeping computational costs low.

8. Simulations performed by NRG PALLAS

8.1 Introduction

The type of flow determines the physics of the FIV. For reactor cores, the flow is predominantly axial with respect to the fuel rods and thus the vibrations are primarily turbulence-induced. To approach this scenario, experimental and numerical efforts have been made in the research community on simpler single rod cases or bundles with smaller number of rods. Research on the single cantilevered rod configurations within GO-VIKING WP2 shows the effect of the choice of rod free end shapes (such as blunt, hemispherical, and tapered at different angles), the flow direction (towards or away from the rod free end) and the inflow speed itself on the vibration response (Cioncolini, et al., 2018) (Cioncolini, et al., 2023) (Li, et al., 2024) while the effect of multi-rod configurations have also been investigated in the research community (Liu, et al., 2013) (De Santis & Shams, 2017).

With such rich experimental data available, numerical Fluid Structure Interaction (FSI) simulations have also been attempted for the same. To numerically capture turbulence-induced vibrations of these configurations accurately, the turbulent fluctuations that cause flow unsteadiness pose a challenge. Numerical models generally require some treatment of turbulence to reproduce flow unsteadiness sufficiently to trigger and sustain the vibrations. To this end, high resolution Large Eddy Simulations (LES) turbulence modelling has been successfully used in a number of studies in literature to reproduce the flow unsteadiness (Benhamadouche, et al., 2009) (Norddine & Benhamadouche, 2023), which has then been coupled with structural solvers to predict the subsequent structural vibration (Elmahdi, et al., 2011) (Bakosi, et al., 2013). However, the fine mesh and small time step size requirements as well as the inherent computational cost of the approach, make this technique currently infeasible to be applied to complete fuel rod bundles.

The efforts of the current study thus lie towards employing a medium resolution Unsteady RANS (URANS) approach on the single cantilevered rod configuration having a hemispherical tip and blunt tip subjected to flow directed towards the rod free end for which experimental results are available as described in Chapter 3. The current study aims in establishing the current state-of-the-art and the gap that could be filled with higher resolution and computationally expensive approaches such as LES or hybrid methods.

The efforts made for the numerical simulations for the rod with hemispherical tip are provided in Sections 8.2 and 8.3, while those made for the rod with blunt tip are provided in Section 8.4. All calculations are performed using the commercial code Simcenter STAR-CCM+ (V2306) which has the capability to run strongly coupled FSI simulations.

8.2 Hemispherical Tip – Centered Rod

Simulations are carried out for the free-clamped flow at $Re=21.2k$ and $40k$ for the rod with a hemispherical tip. Note that the Re is defined using the velocity and distance in the annular gap between the rod and the confining pipe. Furthermore, while the numerical simulations are carried out at $Re=40k$, the experimental results used in the comparison are at $Re\approx 42.5k$ for the PIV and $Re=42.06k$ for the vibration results. Ideally, the numerical simulations should



have also been carried out at $Re=42k$, however, when starting the simulation the rounded value of $Re=40k$ as provided in D2.1 (Cioncolini, et al., 2022) was used. It was only after the simulations were complete that the clarification was made by experimentalists in a progress meeting. Given the timeline and computational cost of the calculations, a re-run could not be achieved using the precise Re . While this was also the case with $Re=21.2k$ being reported as $Re=20k$ in D2.1 (Cioncolini, et al., 2022), the precise Re value was clarified in time before the start of the numerical calculations and thus the precise $Re=21.2k$ was adopted in the numerical simulations.

Simulations were run with the rod being positioned exactly in the center of the confining pipe at $Re=21.2k$ and $40k$. These are discussed in this section. For the $Re=21.2k$ case, a simulation was also run where the rod was positioned off-centre. This is separately motivated and discussed in Section 8.3.

The geometry is created in STAR-CCM+. Only the experimental test section is considered (see Figure 2) including the union cross, part of the outlet pipes and excluding the curved inlet with flow straighteners of the experiment.

8.2.1 Solid Domain

Only the hollow cylinder and the aluminum plug are considered. The effect of the lead shots on the linear mass density is taken into account by modifying the density of the stainless steel that makes up the hollow cylinder. The material properties selected are listed in Table 35.

Table 35. Solid Material Properties

Material	E (GPa)	ρ (kg/m ³)	ν (-)
Aluminum	69	2740	0.33
Stainless Steel	193	33676.313	0.26

A structured mesh is created for the hollow cylinder while a tetrahedral mesh is created for the aluminum end plug. Mesh refinement is controlled for the hollow cylinder by prescribing the number of elements in the axial, radial and azimuthal directions while that for the end plug is controlled using the base size in the mesh generator. Several candidate meshes are considered as listed in Table 36. Tests were carried out in a sequential fashion to determine, in order, the number of elements in the axial, radial and circumferential direction for the hollow cylinder and finally the base size for the end plug. These are indicated by the colored blocks in Table 36.

The meshes are 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

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

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

Table 36. Solid Mesh Candidates

Mesh Level	Hollow Cylinder				End Plug	
	Axial	Circumferential	Radial	Cell Count	Base Size (mm)	Cell Count
H411A1	400	100	3	120000	5	18445
H311A1	600	100	3	180000	5	18445
H211A1	800	100	3	240000	5	18445
H111A1	1000	100	3	300000	5	18445
H011A1	1200	100	3	360000	5	18445
H210A1	800	100	4	320000	5	18445
H211A1	800	100	3	240000	5	18445
H212A1	800	100	2	160000	5	18445
H213A1	800	100	1	80000	5	18445
H202A1	800	120	2	192000	5	18445
H212A1	800	100	2	160000	5	18445
H222A1	800	80	2	128000	5	18445
H232A1	800	60	2	96000	5	18445
H212A0	800	100	2	160000	3.5	39960
H212A1	800	100	2	160000	5	18445
H212A2	800	100	2	160000	6	10756
H212A3	800	100	2	160000	7	6805

To incorporate the natural damping of the rod in air, the Rayleigh damping model of STAR-CCM+ is employed using a mass damping coefficient of 0 Hz and a stiffness damping coefficient of 6.987s which is a value that targets the desired experimental damping ratio for an expected vibrational frequency. The 2nd order Newmark integration scheme is used with a model constant of 0.5.

For this test, the linear geometry model and linear finite elements are employed and the simulation is run, starting at rest conditions, for 10s using a time step size of 2.4ms and 5 iterations per time step. A sample rod tip displacement time history so obtained for mesh H212A1 is shown in Figure 145 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 37 for all meshes. Based on the computational effort and the accuracy of the natural frequency and damping ratio, mesh H212A1 is finalized for the solid and the approach for selecting the Rayleigh damping model parameters is used for the following tests. Figure 146 shows the finalized mesh.

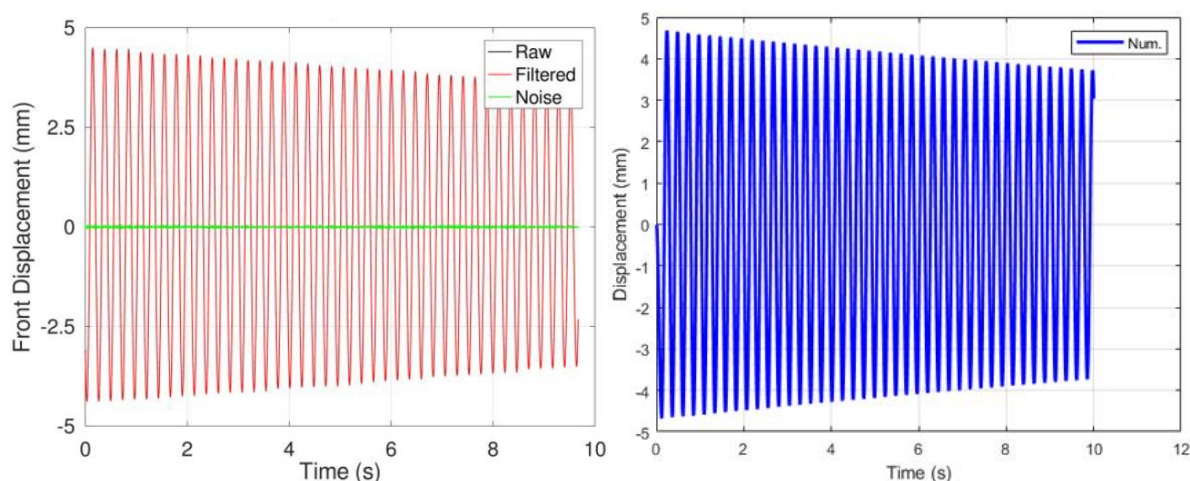


Figure 145. Time History of Vibrations for the Experiment (Cioncolini, et al., 2022) (left) and Mesh H212A1 (right)

Table 37. Obtained Results: (Experiment (Cioncolini, et al., 2022): $f_{n,a} = 4.1\text{Hz}$, $\zeta_{n,a} = 0.9 \times 10^{-3}$)

Mesh Level	$f_{n,a}$ (Hz)	Error (%)	$\zeta_{n,a} (\times 10^{-3})$	Error (%)	Total CPU Time ($\times 10^5$ s)
H411A1	4.1061	0.149	0.9100	1.115	3.820
H311A1	4.1061	0.149	0.9090	1.005	6.334
H211A1	4.1275	0.670	0.9156	1.735	10.408
H111A1	4.1275	0.670	0.9157	1.747	15.310
H011A1	4.1275	0.670	0.9159	1.762	21.278
H210A1	4.1254	0.620	0.9161	1.789	14.889
H211A1	4.1275	0.670	0.9156	1.735	10.408
H212A1	4.1316	0.770	0.9166	1.843	7.209
H213A1	4.1428	1.045	0.9189	2.097	3.896
H202A1	4.1408	0.995	0.9191	2.126	8.833
H212A1	4.1316	0.770	0.9166	1.843	7.209
H222A1	4.1051	0.124	0.9090	0.995	5.081
H232A1	4.1031	0.075	0.9099	1.096	3.381
H212A0	4.1051	0.124	0.9103	1.147	8.866
H212A1	4.1316	0.770	0.9166	1.843	7.209
H212A2	4.1604	1.474	0.9220	2.444	6.048
H212A3	4.2244	3.033	0.9329	3.659	5.470

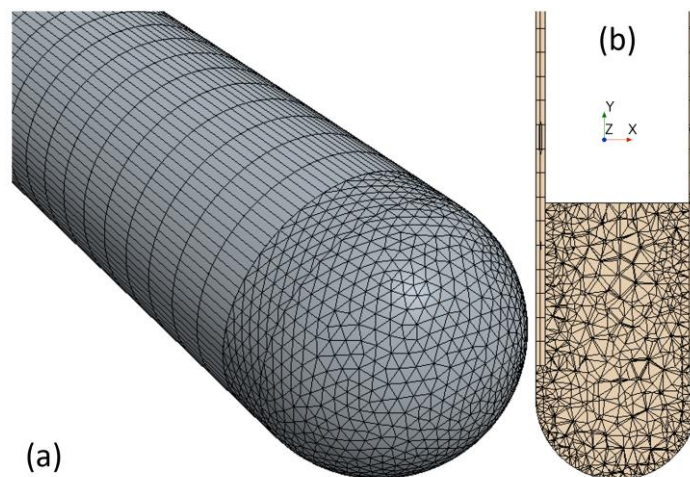


Figure 146. Solid Mesh H212A1: a) Closeup of the Rod Tip, b) Sectional Front View

8.2.2 Fluid Domain

Only the test section's fluid volume is considered for the analysis. The properties of water used in this study are provided in Table 38.

Table 38. Fluid Material Properties

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

A hexahedral mesh is created for the fluid domain using the automatic mesher of STAR-CCM+. Mesh refinement is controlled using the base size. A volumetric refinement is provided such that the bulk cell size is halved starting at about 12 rod diameters upstream of the rod free end. Three candidate meshes are considered as listed in Table 39. To test its suitability for the FSI forced flow test cases, a vibration test similar to that carried out for the solid domain is carried out. For this test, the fluid mesh uses 5 prism layers on all walls with a stretching factor of 1.24. This is an FSI calculation for the vibration of the rod in still water.

Table 39. Fluid Mesh Candidates

Mesh Level	Base Size (mm)	Cell Count (M)
F3	7.5	1.688
F2	5.0	4.115
F1	3.5	7.247
F2_ID	5.0	4.115

For the solid, the linear geometry model and linear finite elements are used. Tests are carried out using the Rayleigh damping model using the same model constants as the test in vacuum. An additional test labelled F2_ID is carried out where the mass damping and stiffness damping coefficients of 0Hz and 7.701s are used respectively, targeting the aforementioned structural damping ratio in vacuum but at the expected vibrational frequency in water. The 2nd order Newmark integration scheme is used with a model constant of 0.5.

For the fluid, an implicit unsteady SIMPLE scheme laminar flow solver is selected with a 2nd order upwind convection scheme. The inlet of the fluid domain is set to a velocity inlet boundary condition with an inflow of 0m/s while the two outlets are set to pressure outlets with 0Pa gauge pressure. All walls are prescribed with no-slip boundary conditions. A reference pressure of 1atm and an acceleration due to gravity of $[0, -9.81, 0]$ m/s² is applied for the fluid domain.

The FSI coupling is chosen to be strong coupling where the complete fluid loading is applied to the wetted faces of the structure and the structural displacement is fed back to the fluid via a mesh morpher step at every iteration of every time step. The fluid mesh is moved using the B-Spline mesh morphing technology of STAR-CCM+.

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

$$a = \begin{cases} [0, -9.81, -0.135] \text{ m/s}^2, & t < 0.1326\text{s} \\ [0, -9.81, 0] \text{ m/s}^2, & t \geq 0.1326\text{s} \end{cases}$$

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 5s using a 2nd order temporal discretization scheme with a time step size of 2.6ms and 25 iterations per time step.

A sample rod tip displacement time history so obtained for mesh F2_ID is shown in Figure 147 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 40 for all meshes. Based on the computational effort and the accuracy of the natural frequency and damping ratio, the meshing strategy of mesh F2_ID is finalized for the fluid and the choice of Rayleigh damping model parameters is used for the FSI tests.

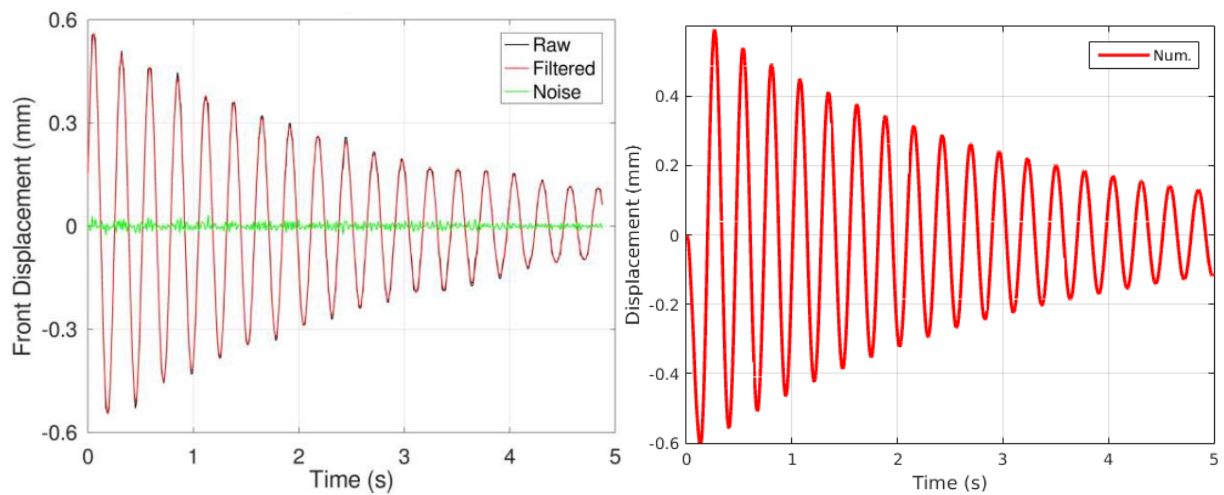


Figure 147. Time History of Vibrations for the Experiment (Cioncolini, et al., 2022) (left) and Mesh F2_ID (right)

Table 40. Obtained Results: (Experiment (Cioncolini, et al., 2022): $f_{n,a} = 3.72\text{Hz}$, $\zeta_{n,a} = 15 \times 10^{-3}$)

Mesh Level	$f_{n,a}$ (Hz)	Error (%)	$\zeta_{n,a} (\times 10^{-3})$	Error (%)	Total CPU Time ($\times 10^6$ s)
F3	3.7114	-0.232	14.58	-2.800	2.884
F2	3.7088	-0.301	14.27	-4.893	6.537
F1	3.7062	-0.370	14.17	-5.561	12.33
F2_ID	3.7088	-0.301	14.35	-4.354	4.285

8.2.3 Forced Flow FSI Study

With the numerical modelling strategy validated, the experiment with forced flow at $Re=21.2k$ and $40k$ are considered. The numerical model described in sections 8.2.1 and 8.2.2 is reused. Before tackling these cases, a few additional considerations are made for this model. Given the complex nature of rod vibrations, for improved accuracy, the non-linear geometry model is selected for the solid, allowing for updating the stiffness matrix in each iteration. The rod surface roughness is included here. The roughness height is set to be $0.45 \mu\text{m}$ for the aluminum plug surface and is set to be $2.5 \mu\text{m}$ for the stainless steel surface as described in the experiment (See section 3.2).

The fluid mesh is slightly modified with it having 15 prism layers with a stretching factor of 1.39 while having the same bulk mesh size, resulting in a mesh with 8.490M cells as shown in Figure 148. This is done targeting a wall-resolved mesh for a high flow rate of $Re=70k$, ensuring that the wall y^+ remains lower than 1 for both cases of interest. Given that the flow is turbulent, the Standard K-Epsilon Low Re model of STAR-CCM+ is employed with a 2nd order convection scheme, realizability, Yap correction and a cubic constitutive relationship to better account for the anisotropy of turbulence. Gravity is considered for both solid and fluid domains, and a reference fluid pressure of 1atm is used.

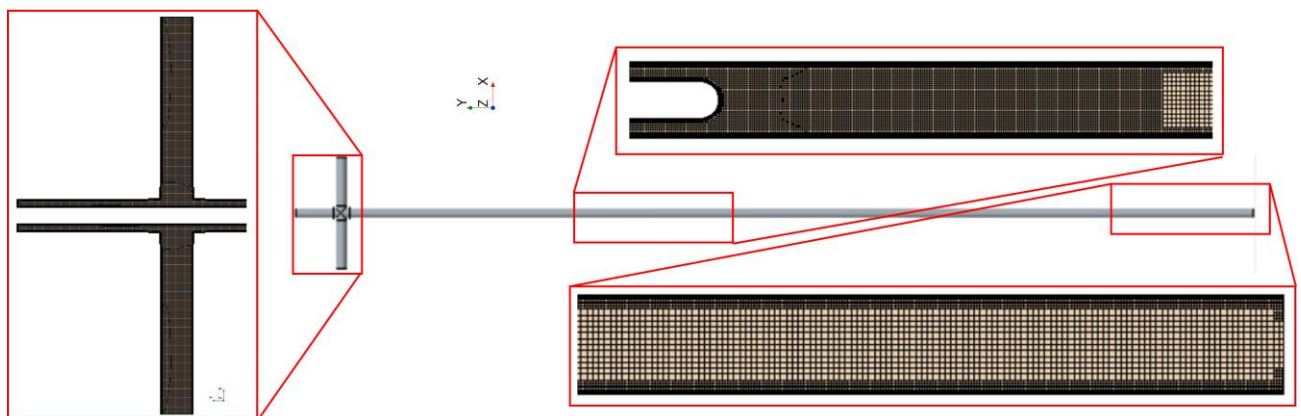


Figure 148. Fluid Mesh

The fluid inlet is provided with a common inflow turbulence of 5% and unique inflow velocities of 1.510m/s and 2.848m/s respectively for the two cases of $Re=21.2k$ and $40k$. The fluid outlets are defined as pressure outlet boundaries with 0Pa gauge pressure. The wetted surfaces of the rod are set to be FSI interfaces and the fluid mesh is allowed to morph.

The rod starts from rest. To generate good initial conditions for the fluid domain, for each inflow velocity, a CFD-only RANS simulation is carried out starting from rest conditions for 2000 iterations which is enough to establish a flow field around the fixed rod. From this point, the implicit unsteady SIMPLEC scheme flow solver is selected with a 2nd order upwind convection scheme.

The FSI calculation is thus run as a transient using a time step size of 0.1ms and 25 iterations per time step for both cases. The simulation is run for a total of 14.28FP (19.643s) and 17.70FP (12.9041s) for the two cases of $Re=21.2k$ and $40k$ respectively, where 1 'Flow Pass' (FP) is defined as time taken for the fluid to travel from the inlet to one of the outlets. To compare the vibrational statistics with the experiment, the first 3.5FP of the obtained solution for both cases are ignored. To compare the time-averaged contour plots with the experimental PIV results, the first 4.5FP and 5.1FP of the obtained solution are ignored for the two cases of $Re=21.2k$ and $40k$ respectively. This is done to forego the initial transient where the rod just begins to move.

Before diving into the results for each case, a check is made on the inflow conditions by comparing the time-averaged non-dimensional axial velocity profile 10D ahead of the rod tip as shown in Figure 149. What is seen from the experimental results is that the non-dimensional velocity profile is independent of the inflow velocity. This is also seen with the numerical results with nearly overlapping profiles. The experimental results show that the velocity profile is more curved at the center of the pipe as compared to the numerical results which show a lower and flatter profile at the center and slightly higher profile close to the walls. From experience (Hussain, et al., 2025), this is known to come from the choice of the turbulence model. The profile shape conforms to what is expected of a fully-developed pipe flow.

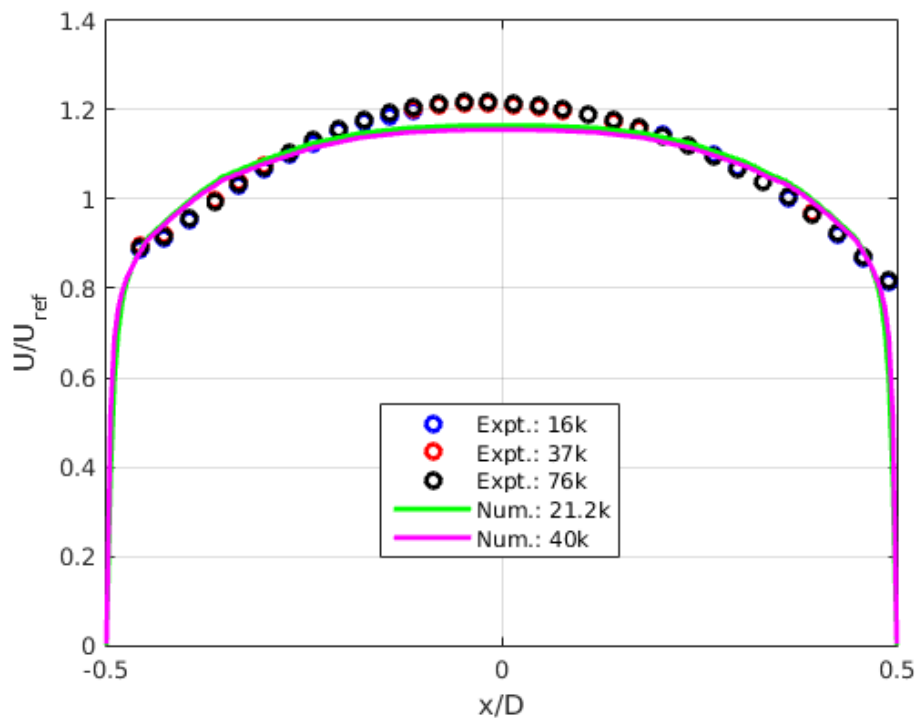


Figure 149. Time-Averaged Non-dimensional Axial Velocity 10D ahead of Rod Tip

8.2.3.1 Results: $Re=21.2k$

The time-averaged contour plots are shown in Figure 150. 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 numerical results are found to be in good agreement with the experimental results.

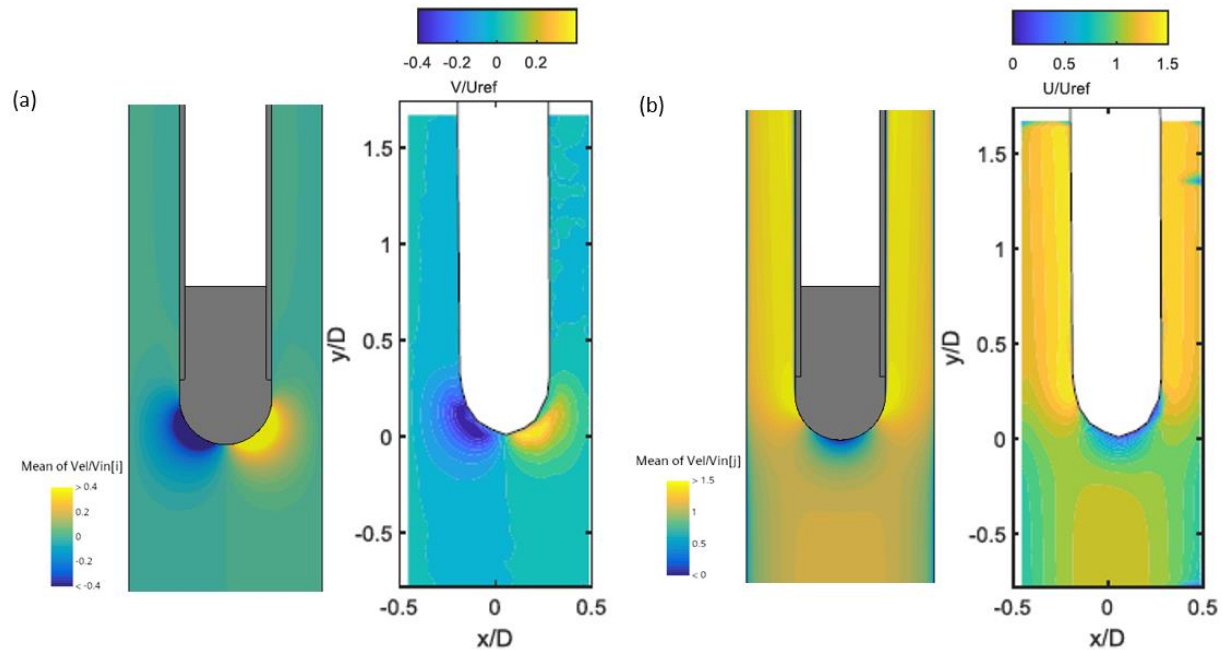


Figure 150. Comparison of Numerical (left) and Experimental (Cioncolini, et al., 2022) (right) Results at $Re\ 21.2k$ for (a) Non-dimensionalized In-plane Component of Velocity and (b) Non-dimensionalized Axial Component of Velocity

Several radial profiles at the PIV measurement locations are provided in Figure 151 to Figure 156. The PIV data inside of the beam are discarded owing to metallic reflections. The time-averaged axial and transverse velocity profiles are drawn. There is a good agreement for the axial velocity. Some discrepancy is noted for the transverse velocity. At a large distance upstream, this is noted close to the walls. This can be explained in part by the limitations of PIV close to walls and in part by the presence of turbulence/swirl in the flow. The agreement for the transverse velocity is better in the vicinity of the rod tip as the flow predominantly changes direction to accommodate in the narrower annular region between the rod and confining pipe. Moving further downstream, the numerical solution predicts flow with predominantly axial flow (or in other words, predicts decreasing transverse flow) but at a rate faster than the experiment. The plots also reveal the rod's misalignment via the asymmetry in the experimental solution.

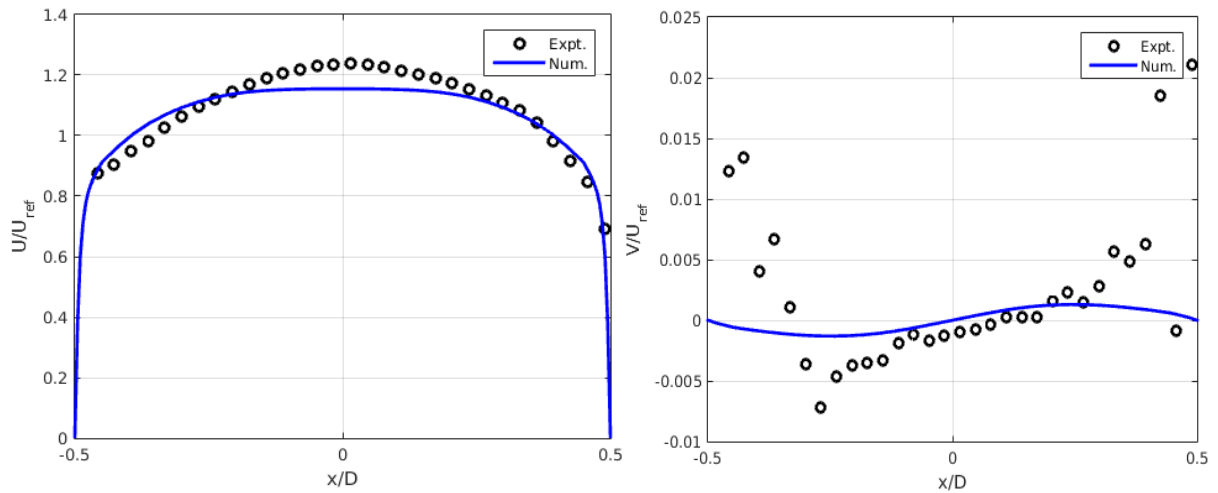


Figure 151. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 15.63786 mm Upstream of the Rod Tip

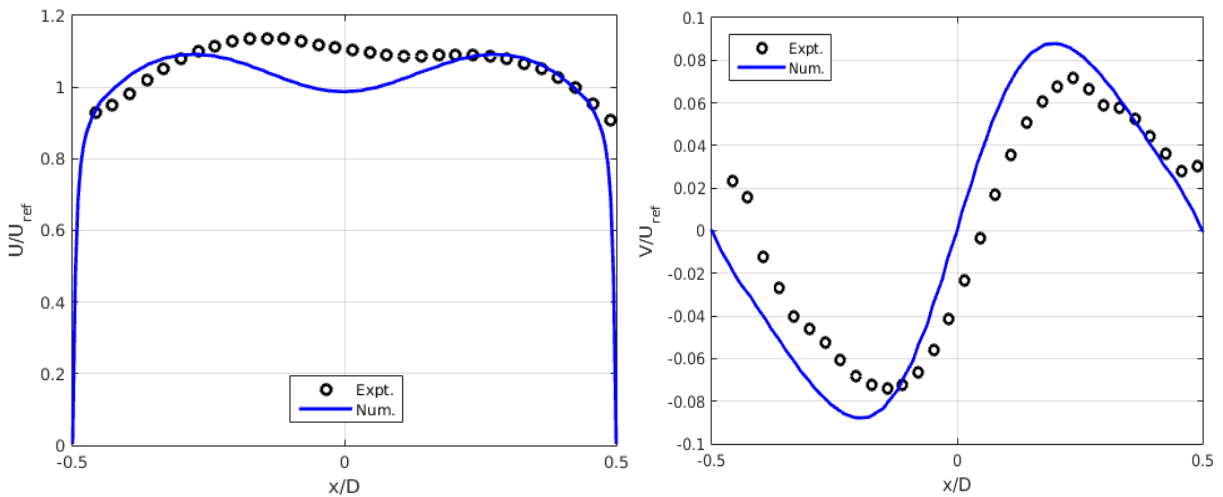


Figure 152. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 3.786006 mm Upstream of the Rod Tip

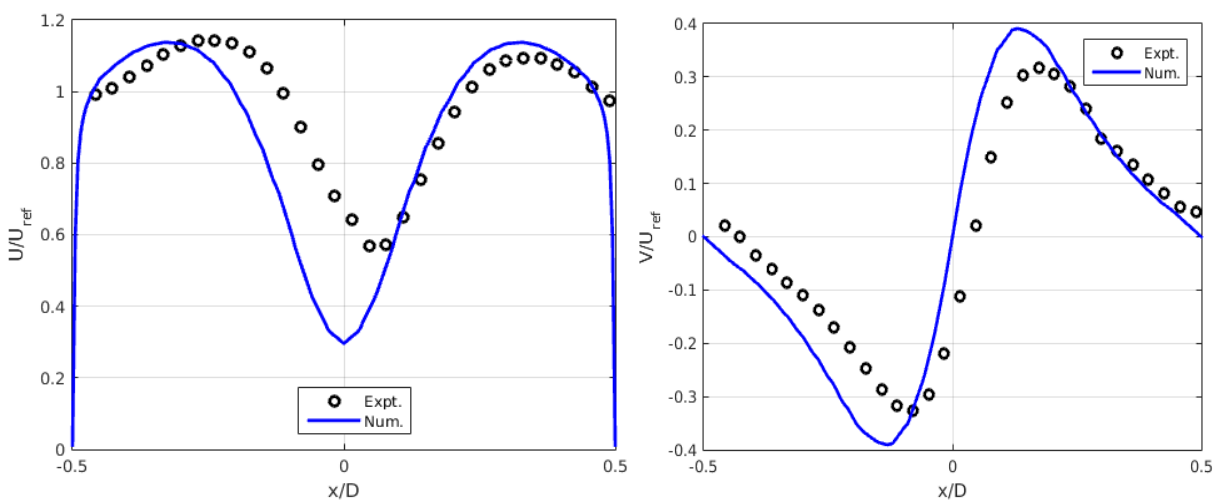


Figure 153. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip

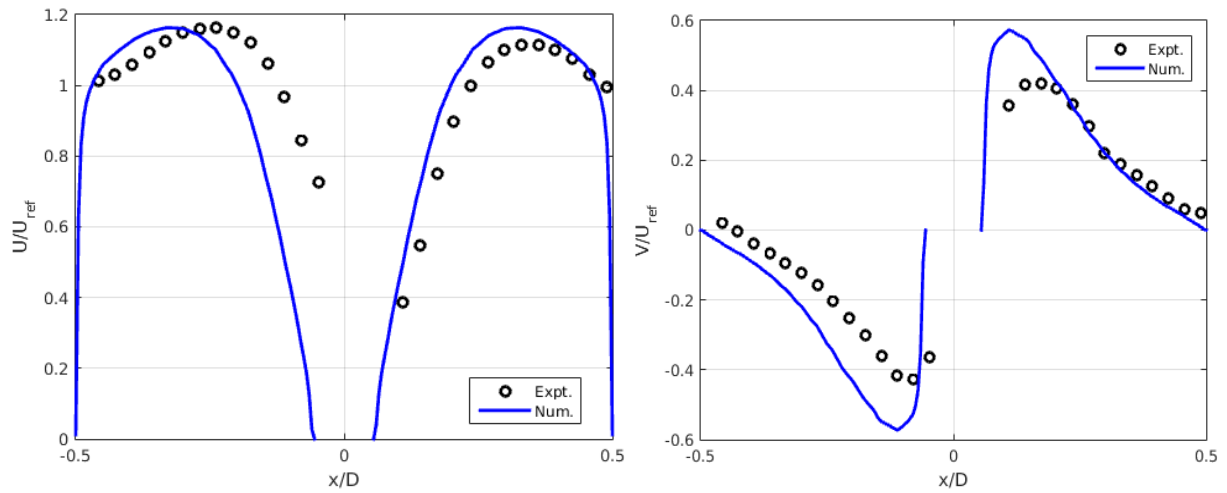


Figure 154. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619 mm Downstream of the Rod Tip

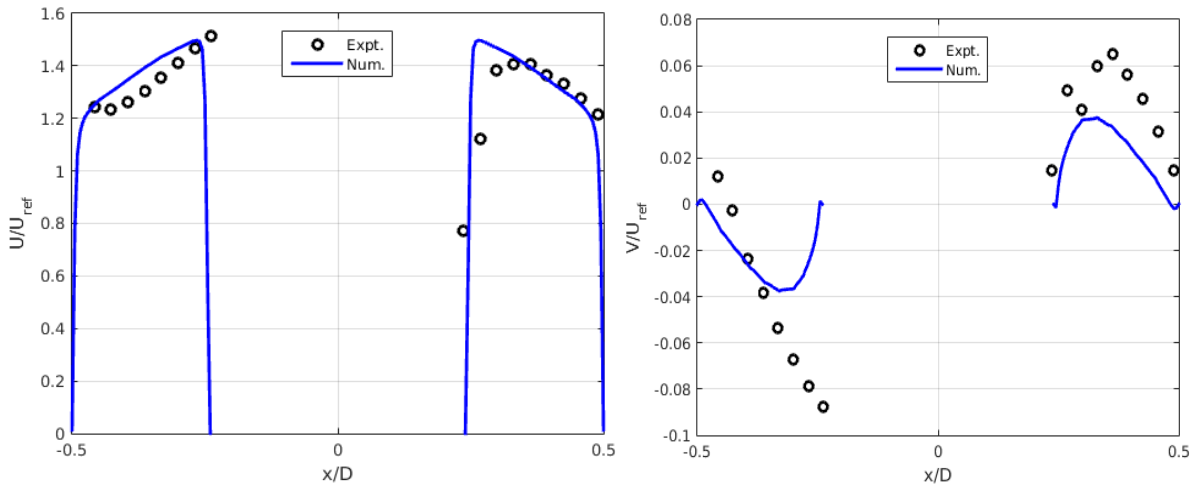


Figure 155. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip

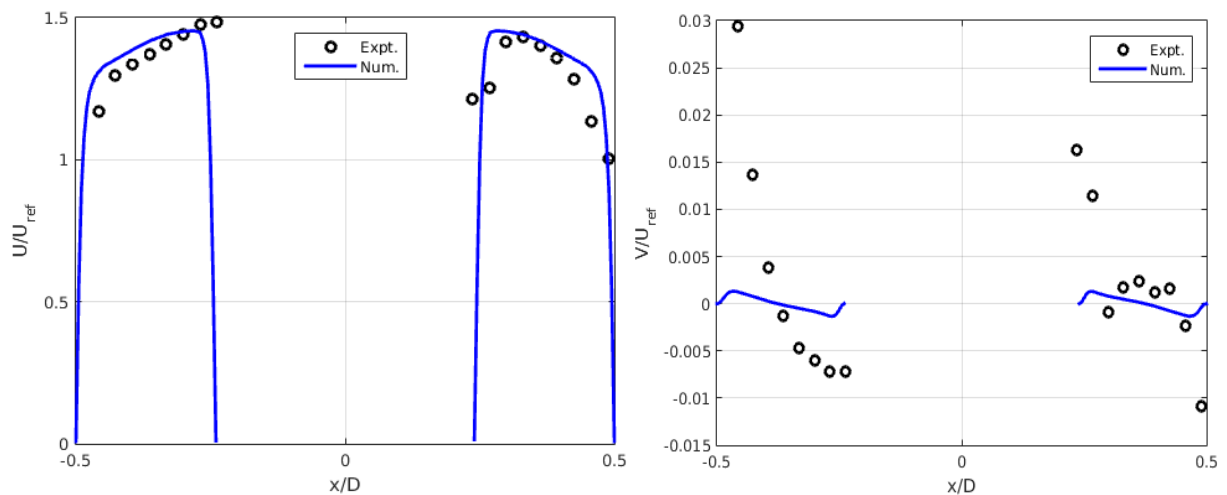


Figure 156. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip

Three vertical profiles at the PIV measurement locations moving upwards from the rod tip height (non-dimensionalized by the rod diameter) are provided in Figure 157. The time-averaged axial and transverse velocity profiles are drawn. For the axial velocity profiles, the experimental results show increasing axial velocities from the rod tip height at a high rate until a non-dimensional height of $\sim 0.7d$ beyond which there is a much slower increase in axial velocity. With increasing radial distance from the rod, the axial velocity is observed to decrease in magnitude. For the transverse velocity profiles, the experimental results show first an increasing trend moving up from the rod tip height to a non-dimensional height of $\sim 0.3d$ and then a decreasing trend until the transverse velocity takes a near-zero value from a non-dimensional height of $\sim 1.3d$.

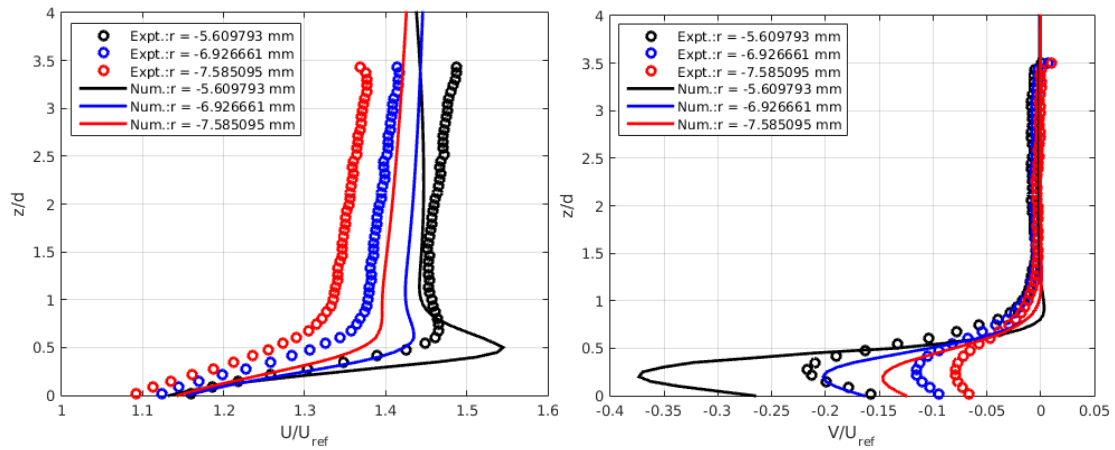


Figure 157. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.609793 mm, -6.926661 mm and -7.858095 mm

The numerical results also capture the aforementioned trends for the axial and transverse velocity profiles, but significantly differ in the magnitude. This can partly be explained by the fact that URANS is used for the fluid side and partly by the vibrating rod tip being off-centred, thereby creating unequal annular gaps on either side of the rod tip. This directly affects the velocity profiles. The asymmetry of the experimental solutions is captured in Figure 158. The solutions at nearly equal radial locations on either side of the rod show distinctly different solutions. In fact, at $r=+5.583627$ mm, the rod is unexpectedly present causing severe metallic reflections thereby corrupting the measured solution as seen in the axial velocity plot.

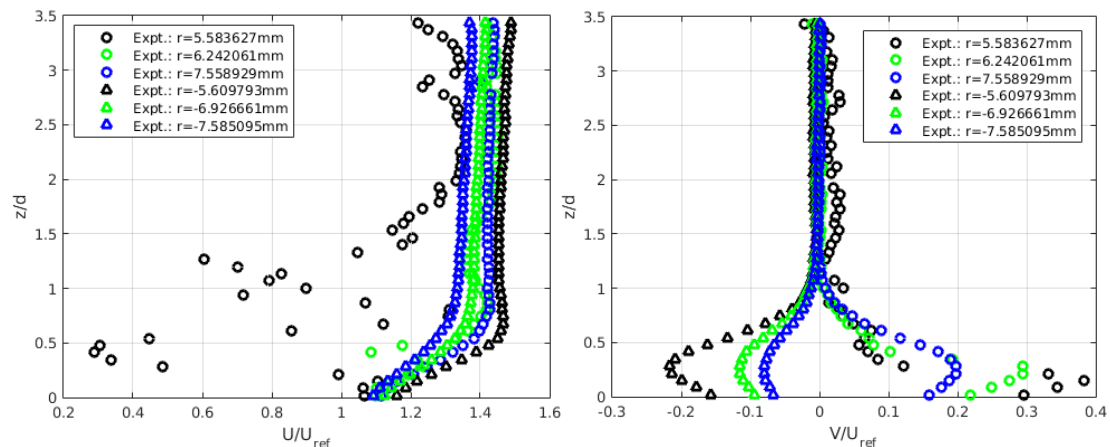


Figure 158. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations ± 5.609793 mm, ± 6.926661 mm and ± 7.858095 mm

Additional vertical profiles are presented for the numerical results in Figure 159. From the axial velocity profile, it is observed that near the rod wall ($r = -5.1$ mm), moving upwards, the axial velocity sharply changes twice unlike the other curves. Moving radially away from the rod, the second jump is no longer observed and the peak velocity consistently decreases. From the transverse velocity plot, it is observed that moving upwards for all curves, the velocity increases in magnitude and then decreases reaching and maintaining a near-zero value from a non-dimensional height of $\sim 1.3d$. Moving away from the rod, the peak velocity magnitude is also observed to consistently decrease. Both plots give insight on how the flow adjusts from pipe flow to annular flow.

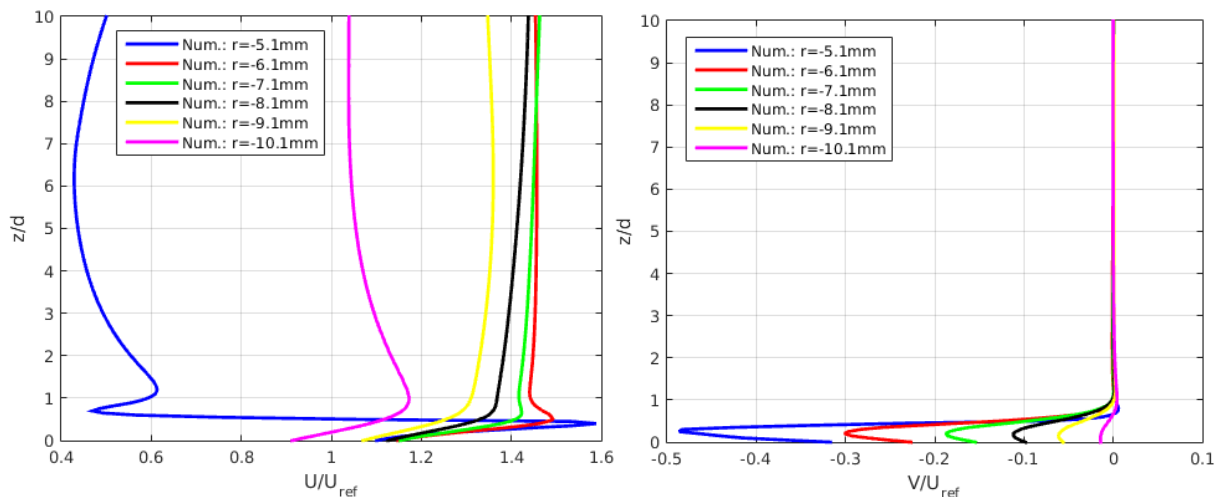


Figure 159. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.1 mm, -6.1 mm, -7.1 mm, -8.1 mm, -9.1 mm and -10.1 mm

The motion of the rod tip has been monitored during the simulation albeit not as long as the experiment owing to computational costs. The time history obtained from the simulation is compared against the experiment in Figure 160. The time units are non-dimensional in flow passes (FP) and 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 first 3.5FP for the numerical result). During this procedure, it was found that the rod tip was oscillating around [1.47675 mm, 0.02654 mm] in the experiment, giving a radial offset of 1.477 mm. Numerically, the corresponding mean location is [0.0019 mm, -0.0015 mm] giving a radial offset of only 2.4 μ m arising from a numerical artifact.

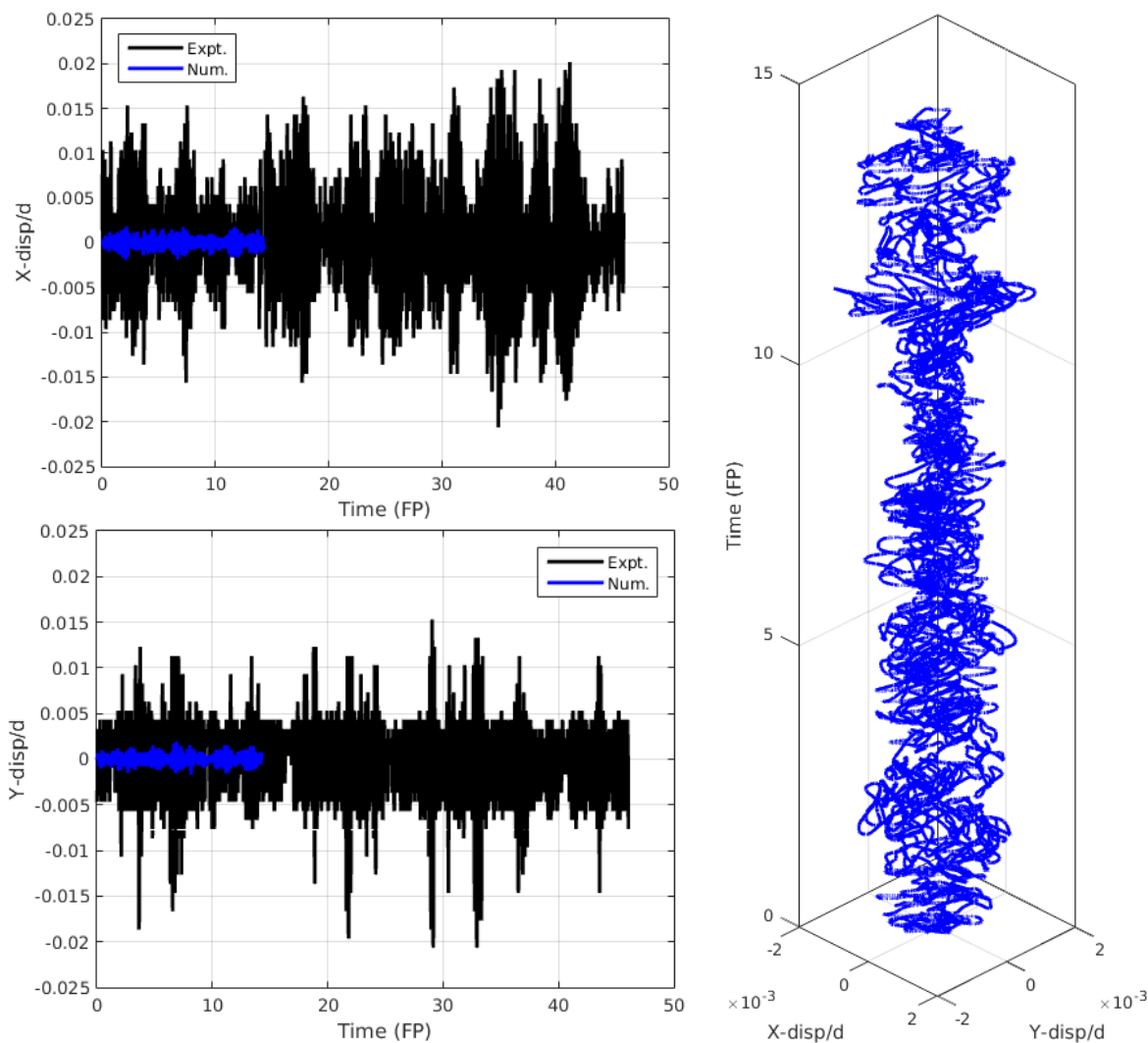


Figure 160. Non-dimensional Time History of the Non-dimensional Rod Tip Front (left top) and Side (left bottom) Displacements. 3D Time History of Rod Tip Displacements (right)

Key results obtained from the time history are presented in Table 41. The RMS polar displacement is numerically predicted to be an order of magnitude lower than the experiment. This is expected as URANS is incapable of predicting fluctuations in velocity and pressure necessary to achieve turbulence-induced vibrations. The RMS front (x-) and side (y-) displacements are found to be very dissimilar for the experiment and the numerical predictions. This is hypothesised to stem partly from the rod tip offset and partly from the asymmetry of the test section.

Table 41. Obtained Displacement Results:

Property	Expt.	Num.
RMS Polar Disp. (μm)	80.688	8.199
RMS X-Disp. (μm)	64.756	6.121
RMS Y-Disp. (μm)	48.138	5.455

The Power Spectral Density (PSD) of the rod tip front (x-) displacement is compared to the experimental data in Figure 161. The trends of the PSD are in good agreement with the experiment. However, the numerical results are consistently lower in energy compared to the experiment. The frequencies of the peaks obtained numerically, experimentally and analytically via classical Euler-Bernoulli beam theory (refer section 6.2.1) are provided in

Table 42. The numerical results are in good agreement for predicting the first two peaks available in the experimental data. The peaks frequencies significantly differ from the analytical solution at higher modes of natural frequencies.

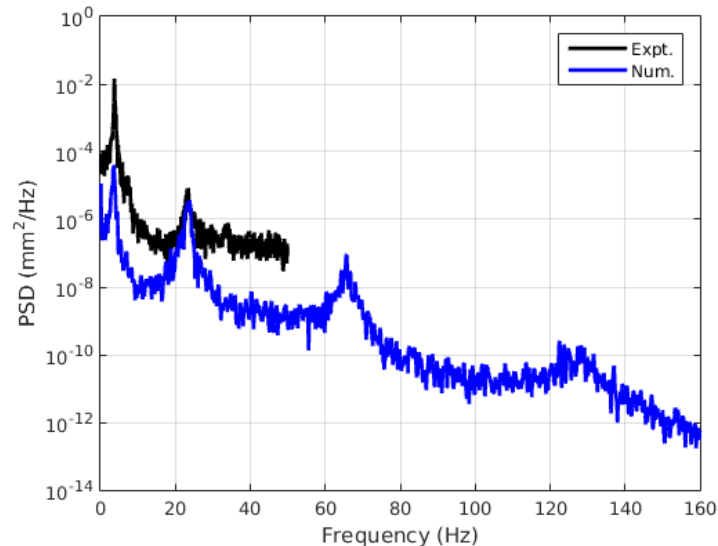


Figure 161. PSD of the Tip Displacement in the Front (x-) Direction

Table 42. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically

Approach	#1	#2	#3	#4
Num. (Hz)	3.64	23.81	65.68	128.2
Exp. (Hz)	3.76	23.47	N/A	N/A
Analytical (Hz)	3.71	25.6	71.8	141

A similar look is also taken for the rod tip side (y-) displacement as shown in Figure 162. 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 161). The first mode frequency for both directions is summarized in Table 43. The average frequency is observed to be in good agreement (within 2%) with the experiment and theory.

Table 43. Frequencies of the Numerical and Experimental Side Displacement PSD Peak #1 Compared to the Beam Natural Frequencies Obtained Analytically

Approach	Front	Side	Average
Num. (Hz)	3.64	3.71	3.68
Exp. (Hz)	3.76	3.71	3.73
Analytical (Hz)	3.71	3.71	3.71

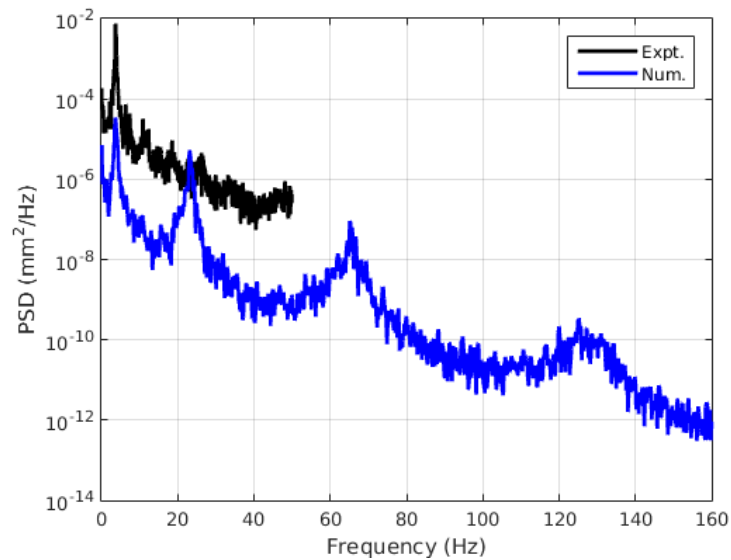


Figure 162. PSD of the Tip Displacement in the Side (y-) Direction

8.2.3.2 Results: Re=40k

Comparisons are made between the numerical and the experimental results. It is to be noted that the comparison is not as fair as that done for the Re=21.2k case. As mentioned in Section 8.2, the numerical results are at Re=40k but the experimental results are available at Re≈42.5k (PIV) and Re=42.06k (displacement), implying different U_{ref} values of 2.848 m/s for the numerical results and ~3.026 m/s (PIV) and 2.995 m/s (FSI) for the experiment (since the exact Re is not clarified for PIV, even this is uncertain). However, for the sake of completeness, the comparisons are still made.

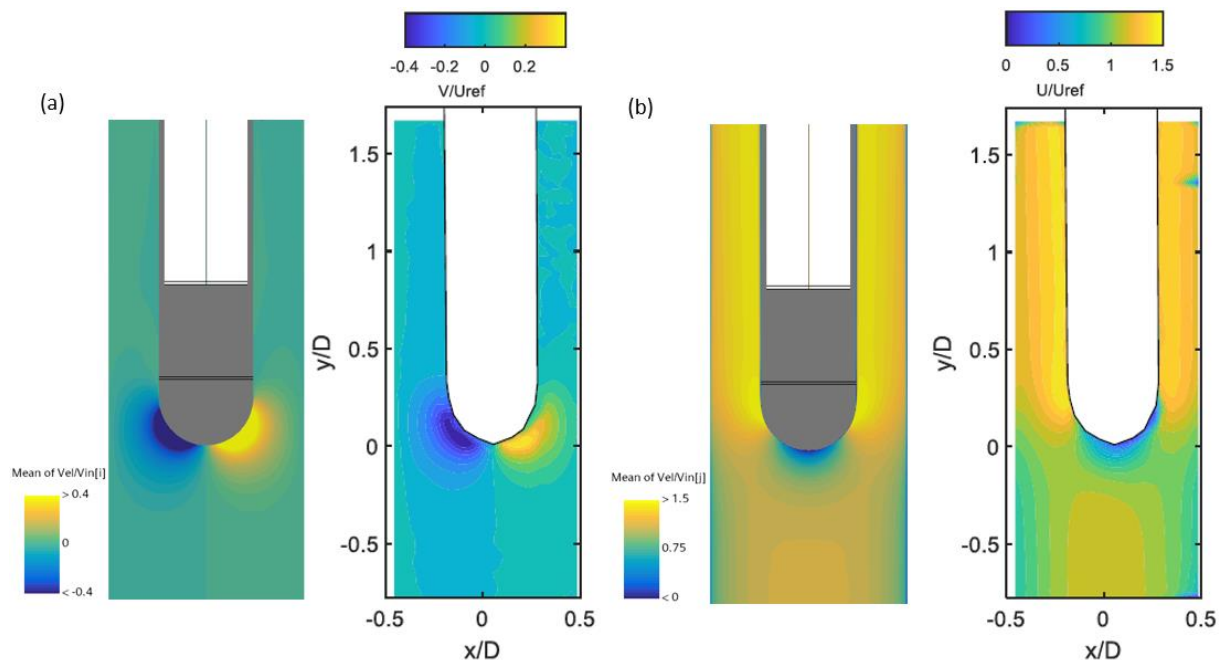


Figure 163. Comparison of Numerical (left) and Experimental (Cioncolini, et al., 2022) (right) Results at Re 40k for (a) Non-dimensionalized In-plane Component of Velocity and (b) Non-dimensionalized Axial Component of Velocity

The time-averaged contour plots are shown in Figure 163. Given that the contours are presented using non-dimensional quantities, the flow pattern observed in Figure 163 is nearly identical to that in Figure 150 for $Re=21.2$. Overall, the numerical results are found to be in good agreement with the experimental results.

Several radial profiles at the PIV measurement locations are provided in Figure 164 to Figure 169. The PIV data inside of the beam are discarded owing to metallic reflections. The time-averaged axial and transverse velocity profiles are drawn. The discussion of the corresponding results of the $Re=21.2k$ case (Figure 151 to Figure 156) also holds here. It is noted that on account of presenting the solutions using non-dimensional numbers, the profiles presented for $Re=40k$ are nearly identical to those of $Re=21.2k$. To check the Re dependence, the mean non-dimensional velocity profiles at the locations 0.493836 mm upstream and 6.74898 mm downstream of the rod tip are re-considered as presented in Figure 170 and Figure 171. The numerical results show overlapping profiles showing little-to-no sensitivity to Re , while the experimental appear to show slight sensitivity. This could be mainly due to the uncertainty in U_{ref} for the assumed $Re \approx 42.5k$ but also due to measurement errors and uncertainty, and possible change in rod tip misalignment during the Re sweep.

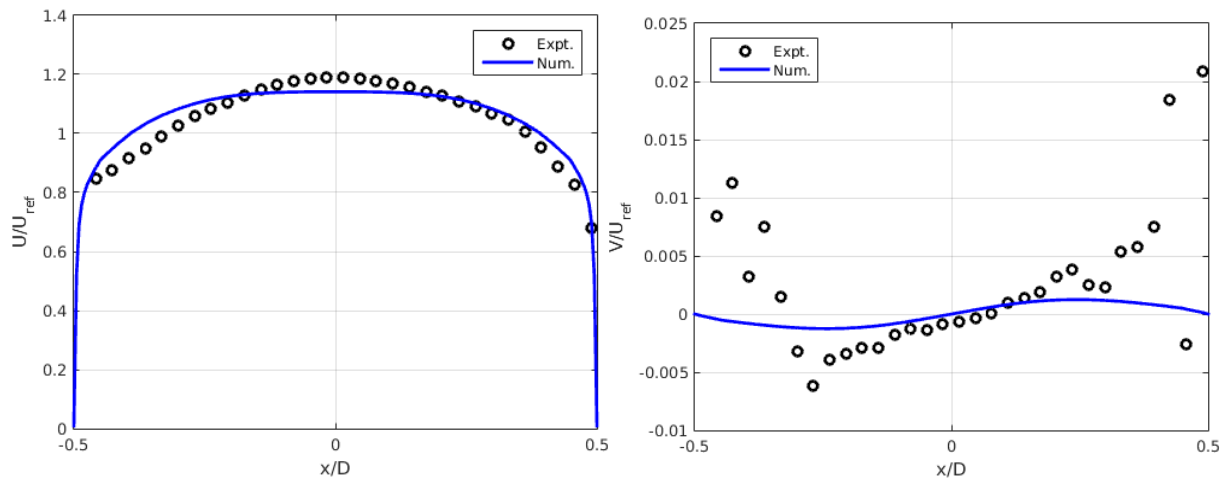


Figure 164. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 15.63786 mm Upstream of the Rod Tip

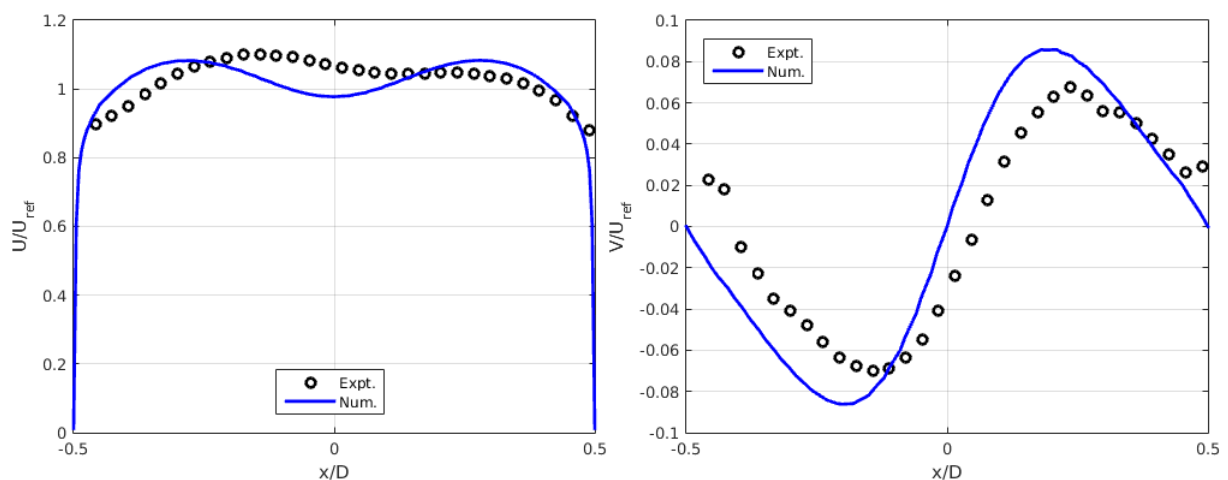


Figure 165. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 3.786006 mm Upstream of the Rod Tip

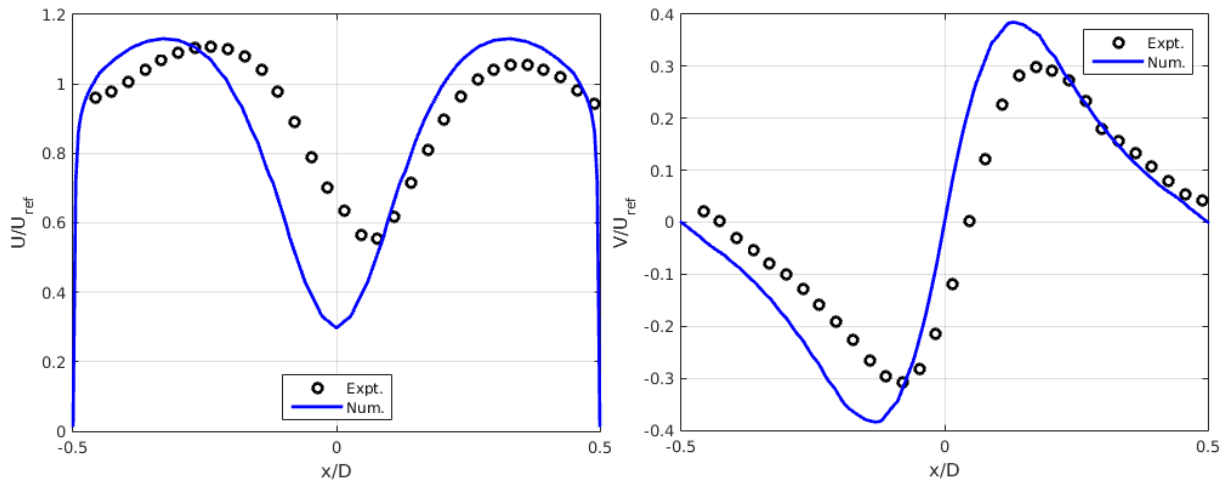


Figure 166. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip

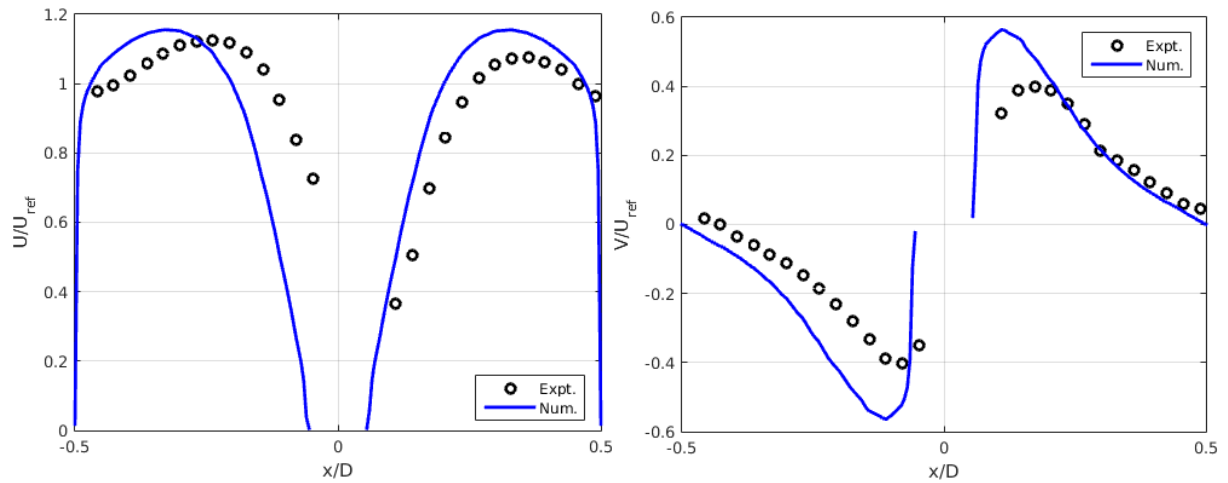


Figure 167. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619 mm Downstream of the Rod Tip

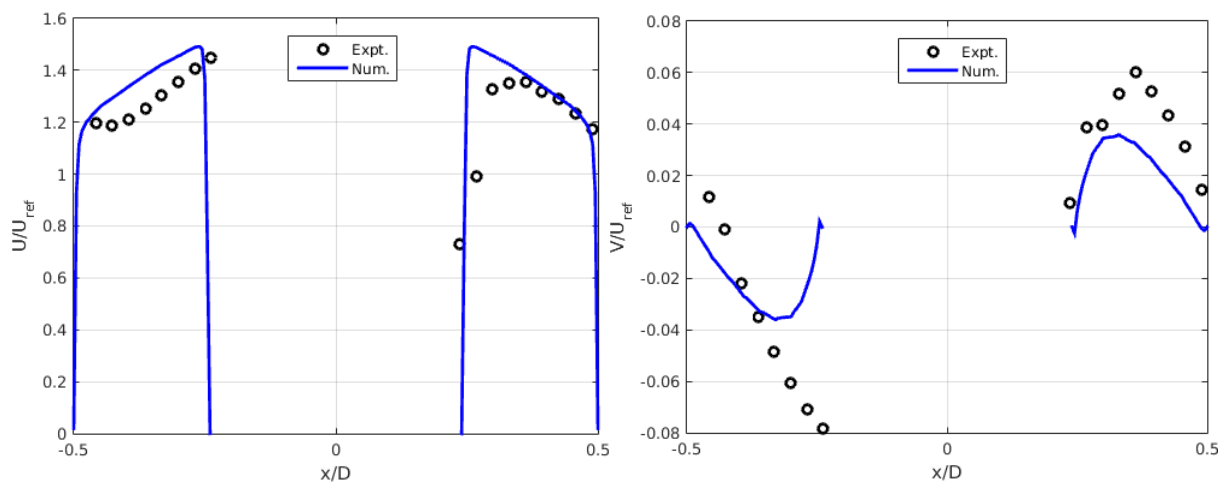


Figure 168. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip

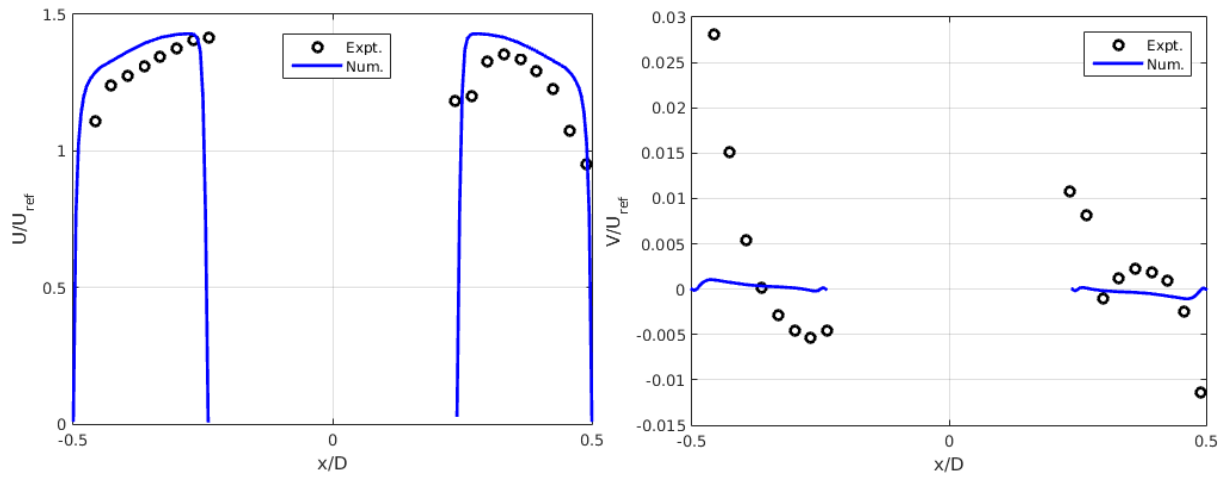


Figure 169. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip

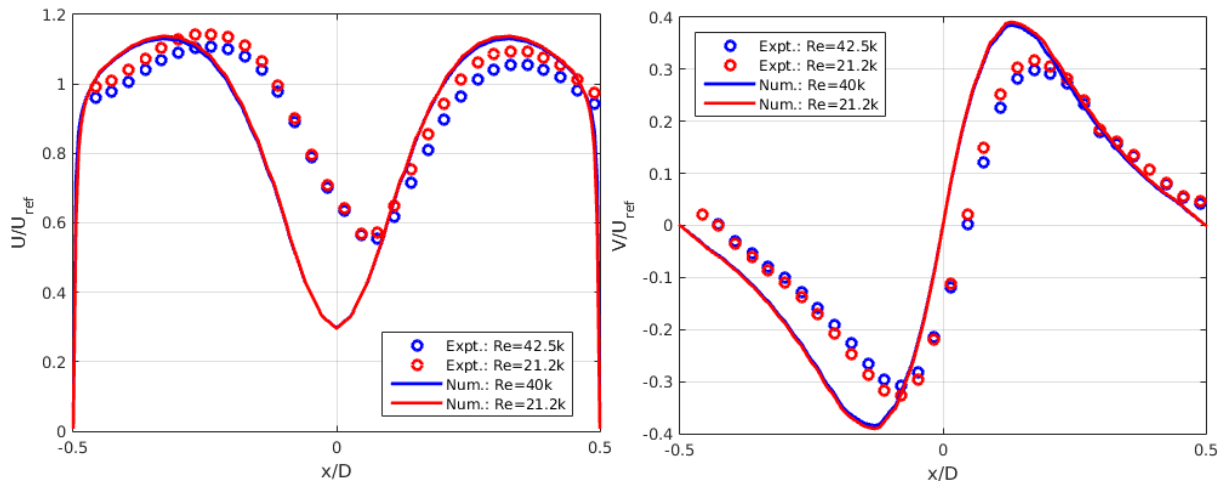


Figure 170. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip

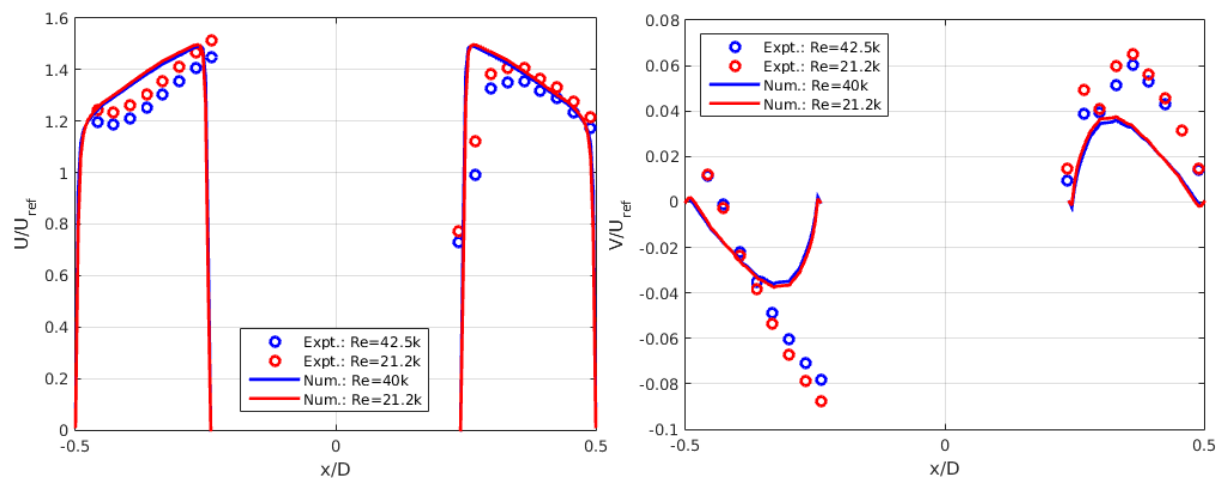


Figure 171. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip

Three vertical profiles at the PIV measurement locations moving upwards from the rod tip height (non-dimensionalized by the rod diameter) are provided in Figure 172. The time-averaged axial and transverse velocity profiles are drawn. The results are found to show similar trends as those for $Re=21.2k$ (Figure 157).

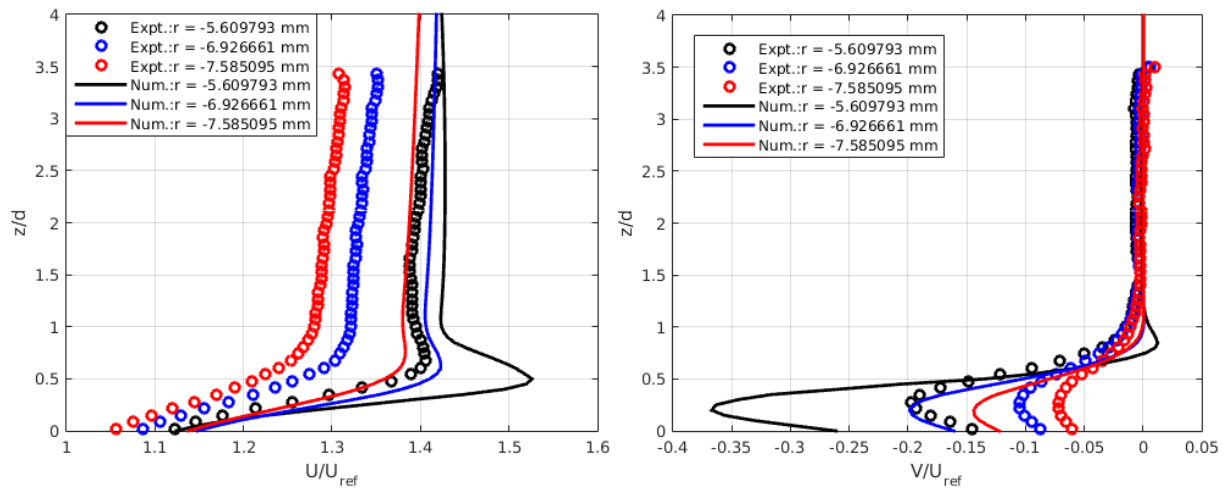


Figure 172. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.609793 mm, -6.926661 mm and -7.858095 mm

The difference in magnitude of the numerical predictions as compared to the experiment stems from the offset in the rod tip location as was also observed in the $Re=21.2k$ case. This asymmetry in the experiment is confirmed again through Figure 173 for $Re \approx 42.5k$. The observations are the similar to those for the $Re=21.2k$ case (Figure 158). At $r=+5.583627$ mm, the rod is again unexpectedly present causing severe metallic reflections thereby corrupting the measured solution as seen in the axial velocity plot.

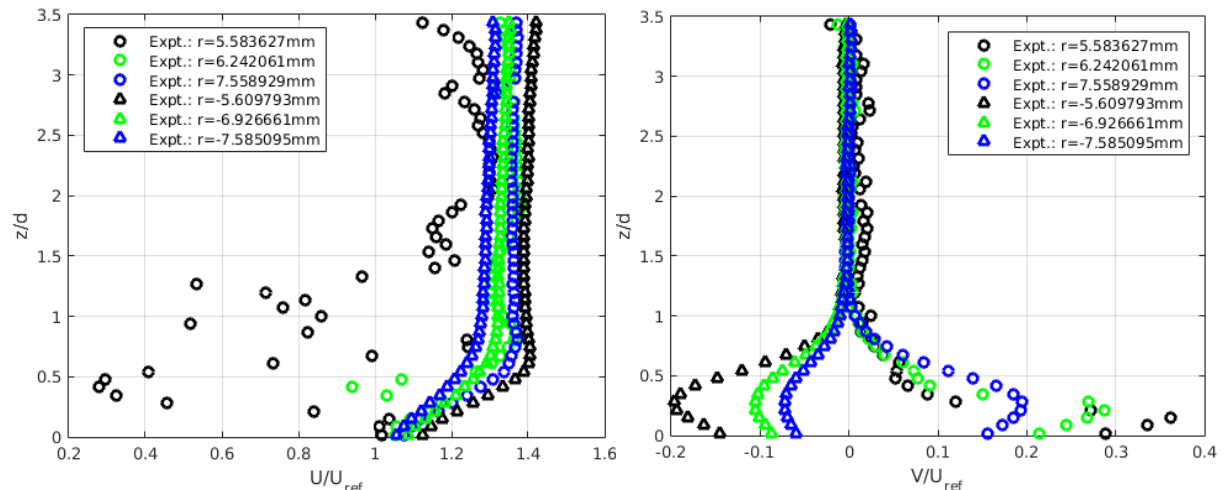


Figure 173. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations ± 5.609793 mm, ± 6.926661 mm and ± 7.858095 mm

Additional vertical profiles are presented for the numerical results in Figure 174 as solid lines along with the $Re=21.2k$ results of Figure 159 presented as dashed lines. For the transverse velocity profile, the change in the solution between the two Re cases are observed to be very minimal. However, the axial velocity profile clearly show significant differences in magnitude between the two solutions while displaying similar trends.

This is more so for the location close to the vibrating rod ($r=-5.1$ mm) and can be reasoned by the fact that the rod maximum displacements between the two cases are not the same, and perhaps even the dominating mode shape. This suggests that there is, in fact, sensitivity of the results to Re , seemingly contrary to what was thought to be observed in Figure 171. However, the location considered for that comparison was at $z/d \approx 0.67$, just shy of $z/d = 0.7$, the height above which the differences become more notable. The solutions at locations away from the rod and pipe walls show much lower sensitivity to the Re .

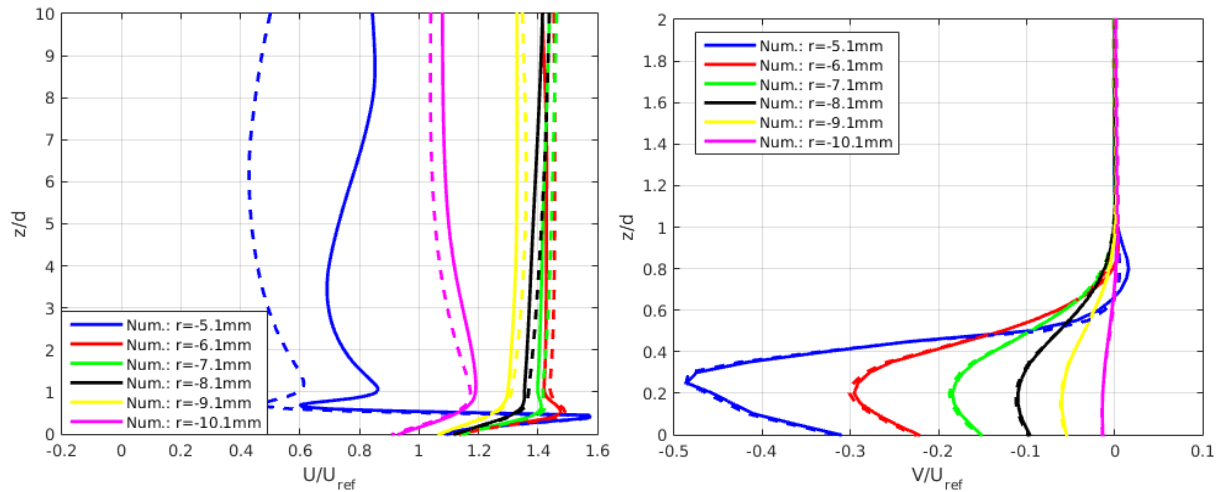


Figure 174. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.1 mm, -6.1 mm, -7.1 mm, -8.1 mm, -9.1 mm and -10.1 mm for $Re=21.2k$ (Dashed Lines) and $Re=40k$ (Solid Lines)

The motion of the rod tip has been monitored during the simulation albeit not as long as the experiment owing to computational costs. The time history obtained from the simulation is compared against the experiment ($Re=42.06k$) in Figure 175. The time units are non-dimensional in flow passes (FP) and 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. It was found that the rod tip was oscillating around $[0.73430$ mm, -0.27925 mm] in the experiment, giving a radial offset of 0.7856 mm. This is about half the value found for $Re=21.2k$ and could explain why the experimental results of $Re=21.2k$ and $Re=40k$ did not nearly collapse on each other in Figure 170 and Figure 171. Numerically, the corresponding mean location is $[0.0036$ mm, -0.00279 mm] giving a radial offset of only 4.6 μ m arising from a numerical artifact.

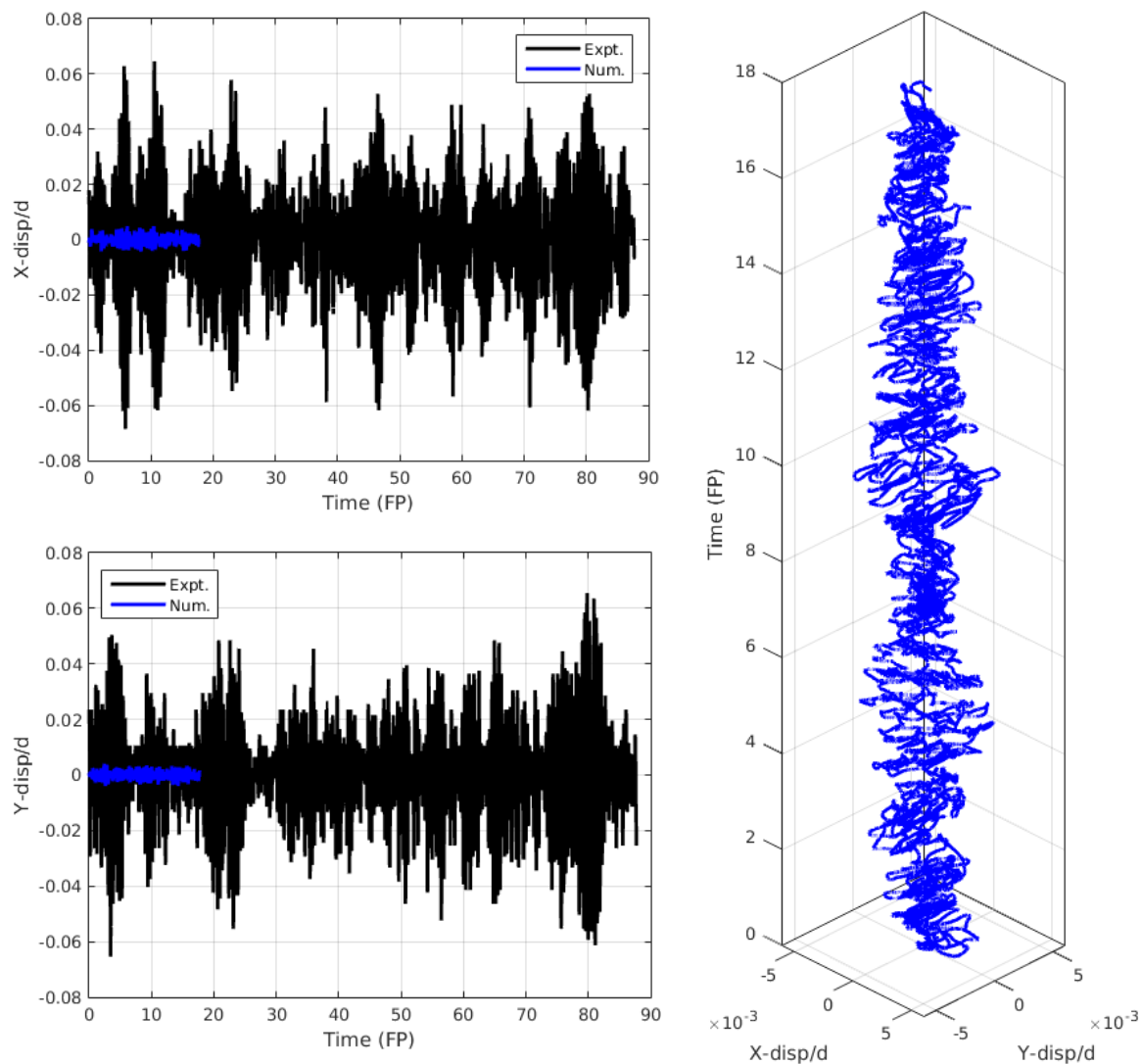


Figure 175. Non-dimensional Time History of the Non-dimensional Rod Tip Front (left top) and Side (left bottom) Displacements. 3D Time History of Rod Tip Displacements (right)

Key results obtained from the time history are presented in Table 44. The RMS polar displacement is numerically predicted to be an order of magnitude lower than the experiment as was also observed for the $Re=21.2k$ case. The RMS front (x-) and side (y-) displacements are also found to be very dissimilar for the experiment and the numerical predictions. This is hypothesised to stem partly from the rod tip offset and partly from the asymmetry of the test section. Another possibility is the asymmetry of the mesh giving rise to numerical errors.

Table 44. Obtained Displacement Results:

Property	Expt.	Num.
RMS Polar Disp. (μm)	271.737	21.277
RMS X-Disp. (μm)	200.879	17.285
RMS Y-Disp. (μm)	182.999	12.408

The Power Spectral Density (PSD) of the rod tip front (x-) displacement is compared to the experimental data in Figure 176. The trends of the PSD are in good agreement with the experiment. However, the numerical results are consistently lower in energy compared to the experiment. The frequencies of the peaks obtained numerically, experimentally and analytically are provided in Table 45. The numerical results slightly underpredict the first two peaks available in the experimental data. The peaks frequencies significantly differ from the analytical solution at higher modes of natural frequencies. An additional peak at 33.05Hz is observed in the experimental data that is not observed in the numerical results. This may be related to the pump frequency, but it needs to be clarified by the experimentalists.

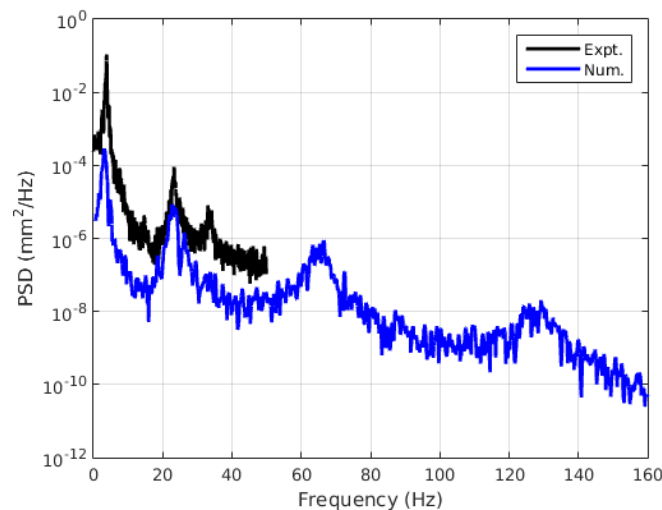


Figure 176. PSD of the Tip Displacement in the Front (x-) Direction

Table 45. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically

Approach	#1	#2	#3	#4
Num. (Hz)	3.19	22.61	66.60	129.2
Exp. (Hz)	3.78	23.30	N/A	N/A
Analytical (Hz)	3.71	25.6	71.8	141

A similar look is also taken for the rod tip side (y-) displacement as shown in Figure 177. 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 176). The first mode frequency for both directions is summarized in Table 46. The average frequency is observed to be underpredicted by ~14% compared to the experiment and theory. The frequency prediction ideally should not change with change in inflow velocity as showcased by the experimental results. The reason for this change in numerical results is unclear and requires further investigation.

Table 46. Frequencies of the Numerical and Experimental Side Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically

Approach	Front	Side	Average
Num. (Hz)	3.19	3.29	3.24
Exp. (Hz)	3.78	3.78	3.78
Analytical (Hz)	3.71	3.71	3.71

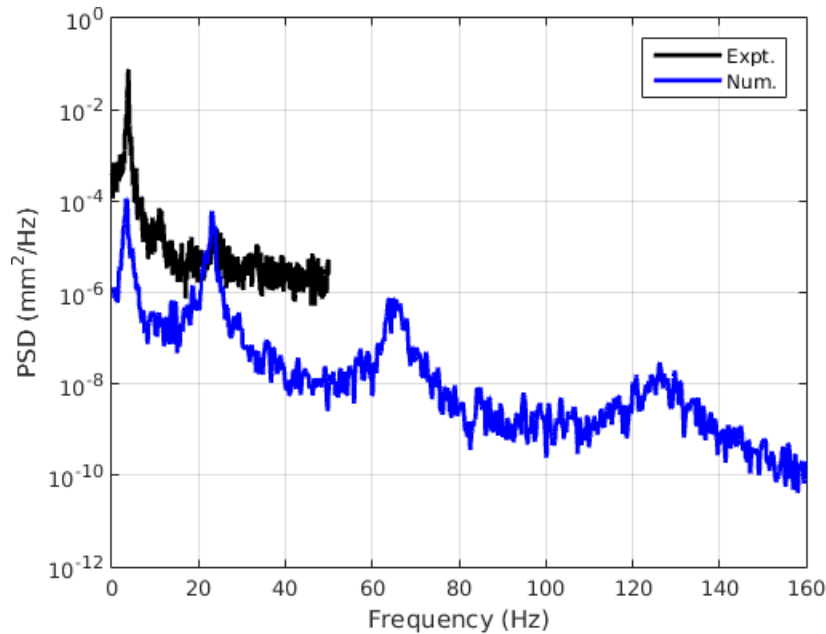


Figure 177. PSD of the Tip Displacement in the Side (y-) Direction

8.3 Hemispherical Tip - Off-centered Rod

As discussed in section 8.2.3, the experimental results indicated that the rod was not perfectly centered. Although the rod tip displacement time history indicated a misalignment of about 1.48 mm, the experimentalists relayed a possible offset of 0.3 mm during progress meetings. It was thus decided to re-run the Re=21.2k case for the rod with a hemispherical tip including the proposed misalignment. More specifically, the entire rod is assumed to be inserted with its axis parallel to that of the confining pipe. The entire rod is shifted 0.3mm at an angle of 45° with respect to the front (x-) and side (y-) directions as shown by the schematic in Figure 178. The efforts undertaken towards this additional case are described in this section.

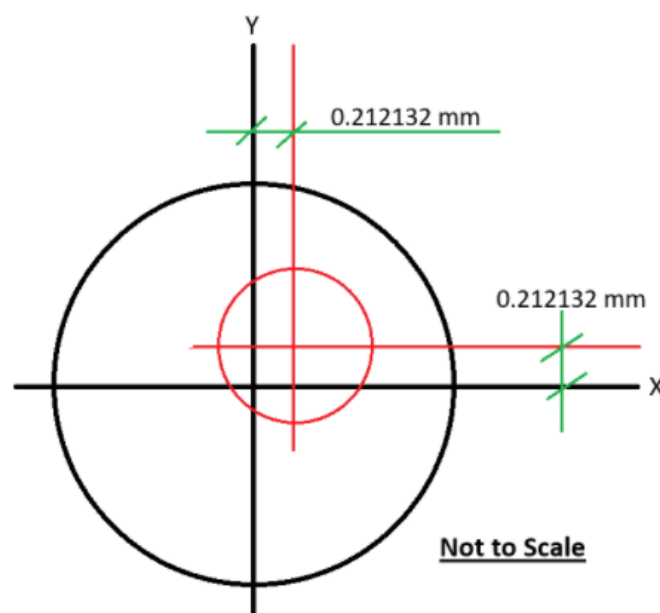


Figure 178. Schematic of the Rod Misalignment

8.3.1 Solid Domain

As with the centered rod scenario, only the hollow cylinder and the aluminum plug are considered. The effect of the lead shots on the linear mass density is taken into account by modifying the density of the stainless steel that makes up the hollow cylinder. The material properties listed in Table 35 are reused. The meshing strategy of the H212A1 mesh (Table 36) reported in section 8.2.1 was also reused. This lead to a structured hexahedral mesh for the hollow cylinder with 160k cells and a tetrahedral mesh for the aluminum plug with 18.421k cells.

8.3.2 Fluid Domain

As with the centered rod scenario, only the test section's fluid volume is considered for the analysis. The properties of water provided in Table 38 are reused. The fluid geometry is remade from scratch by removing the offset rod volume by a Boolean operation before meshing. A hexahedral mesh is created for this domain using the automatic mesher of STAR-CCM+ following the same meshing strategy as F2_ID (Table 39) described in section 8.2.2 leading to a hexahedral wall resolved ($y^+ < 1$) mesh with 8.574M cells.

8.3.3 Forced Flow FSI Study:

For the solid, the non-linear geometry model is selected, allowing for updating the stiffness matrix in each iteration. The rod surface roughness is included here. The roughness height is set to be $0.45\ \mu\text{m}$ for the aluminum plug surface and is set to be $2.5\ \mu\text{m}$ for the stainless steel surface as described in the experiment (See section 3.2). To incorporate the natural damping of the rod in air, the Rayleigh damping model of STAR-CCM+ is employed using a mass damping coefficient of 0 Hz and a stiffness damping coefficient of 7.701s which is the same as was used for the centered rod case. The 2nd order Newmark integration scheme is used with a model constant of 0.5.

For the fluid, the inlet is provided with a inflow turbulence of 5% and inflow velocity of 1.510m/s. The fluid outlets are defined as pressure outlet boundaries with 0Pa gauge pressure. All walls are prescribed with no-slip boundary conditions. The wetted surfaces of the rod are set to be FSI interfaces and the fluid mesh is allowed to morph using the B-Spline mesh morphing technology of STAR-CCM+. The rod starts from rest. To generate good initial conditions for the fluid domain, a CFD-only RANS simulation is carried out starting from rest conditions for 2000 iterations which is enough to establish a flow field around the fixed rod. From this point, the implicit unsteady SIMPLEC scheme flow solver is selected with a 2nd order upwind convection scheme. Given that the flow is turbulent, the Standard K-Epsilon Low Re model of STAR-CCM+ is employed with a 2nd order convection scheme, realizability, Yap correction and a cubic constitutive relationship to better account for the anisotropy of turbulence. Gravity is considered for both solid and fluid domains, and a reference fluid pressure of 1 atm is used.

The FSI calculation is thus run with strong coupling as a transient using a time step size of 0.25 ms and 25 iterations per time step. The simulation is run for a total of 15.0FP (20.6335s). To compare the vibrational statistics with the experiment and the previously obtained numerical result, the first 3.5FP of the obtained solution are ignored. To compare the time-averaged contour plots with the experimental PIV results, the first 4.5FP of the obtained solution are. This is done to forego the initial transient where the rod just begins to move.

8.3.3.1 Results: Re=21.2k

First, a check is made on the inflow conditions by comparing the time-averaged non-dimensional axial velocity profile 10D ahead of the rod tip as shown in Figure 179. The effect of the offset is found to have no consequence on the profile 10D ahead of the rod tip with results overlapping those of the centered case.

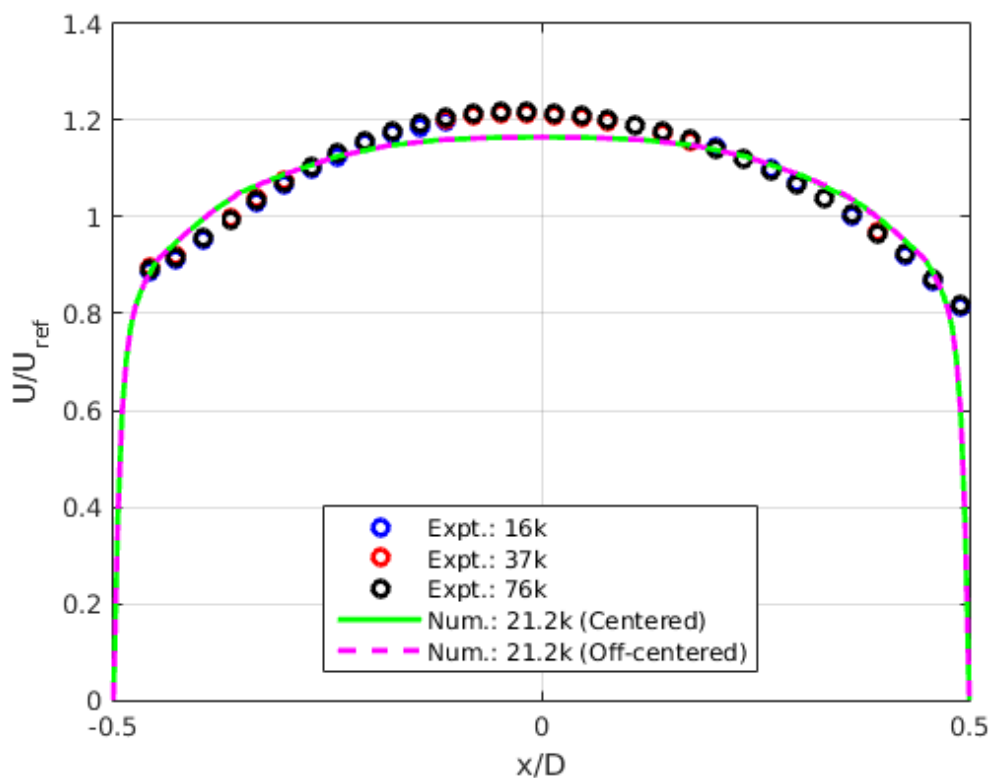


Figure 179. Time-Averaged Non-dimensional Axial Velocity 10D ahead of Rod Tip

The time-averaged contour plots are shown in Figure 180. From these results, no visible changes appear in the solution, which can be expected since the applied offset is 0.3 mm which is only about 1.4% of the confining pipe diameter.

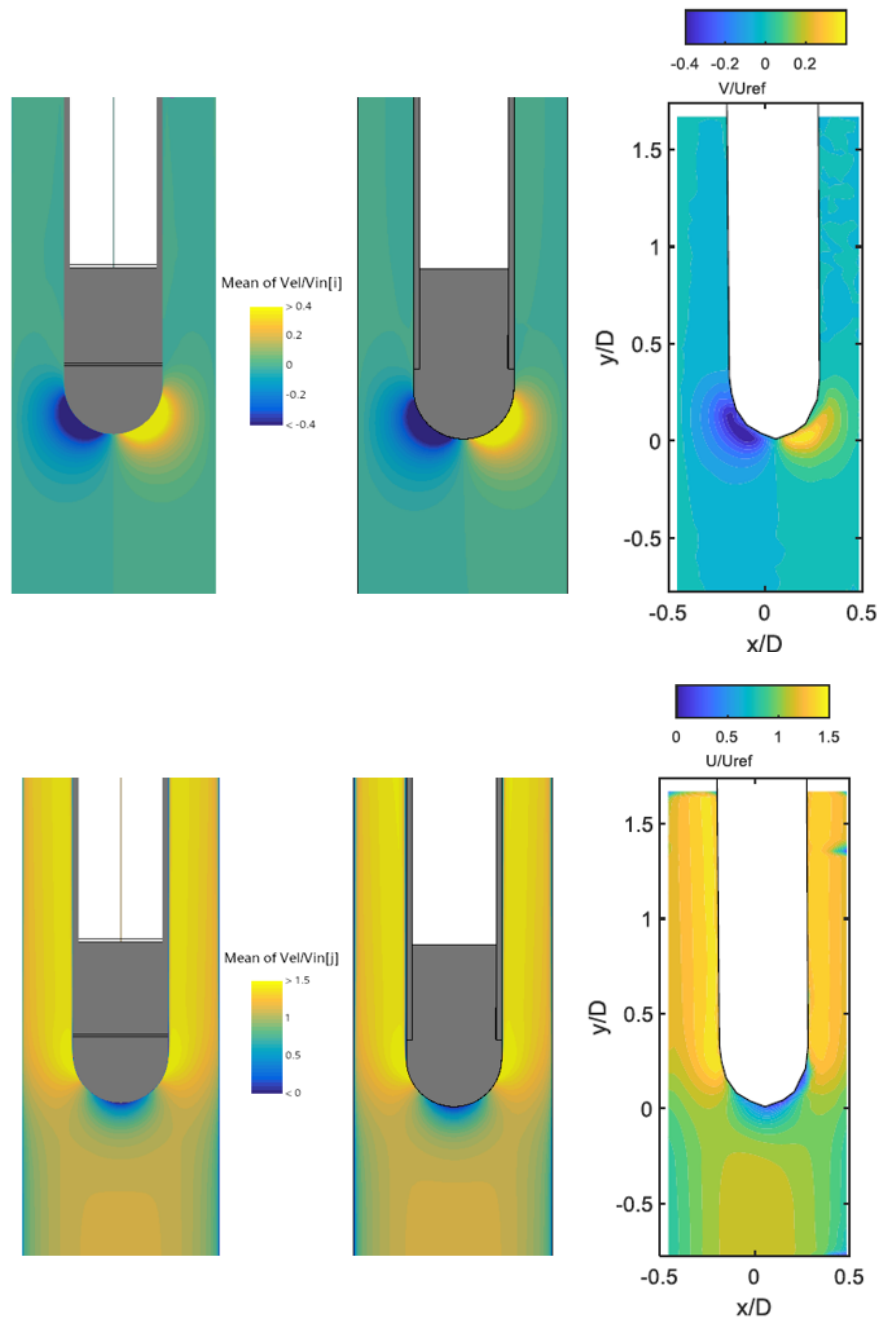


Figure 180. Comparison of Numerical Off-centered (left), Centered (middle) and Experimental (Cioncolini, et al., 2022) (right) Results at Re 21.2k for Non-dimensionalized In-plane Component of Velocity (top) and Non-dimensionalized Axial Component of Velocity (bottom)

Radial profiles at the same PIV measurement locations as those presented in section 8.2.3.1 are re-drawn and provided in Figure 181 to Figure 186. The effect of the rod offset on the profiles appear to be minimal and are still a far cry from the experimental results.

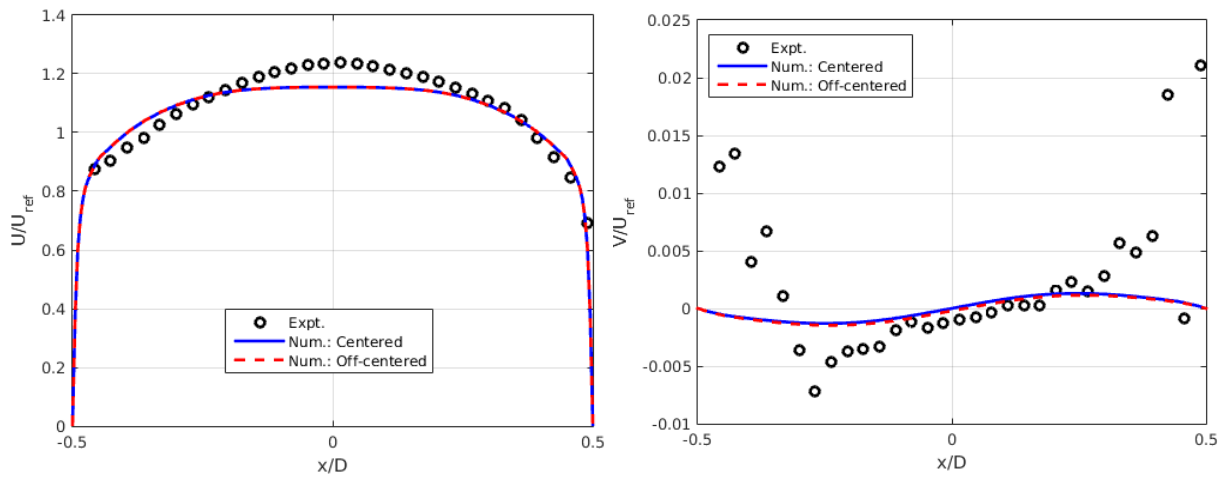


Figure 181. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 15.63786 mm Upstream of the Rod Tip

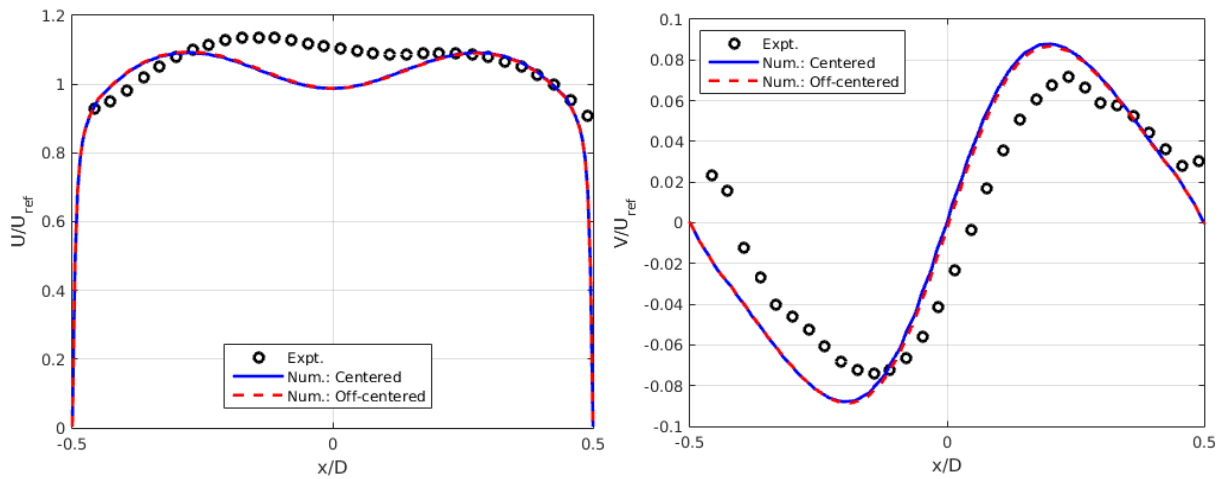


Figure 182. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 3.786006 mm Upstream of the Rod Tip

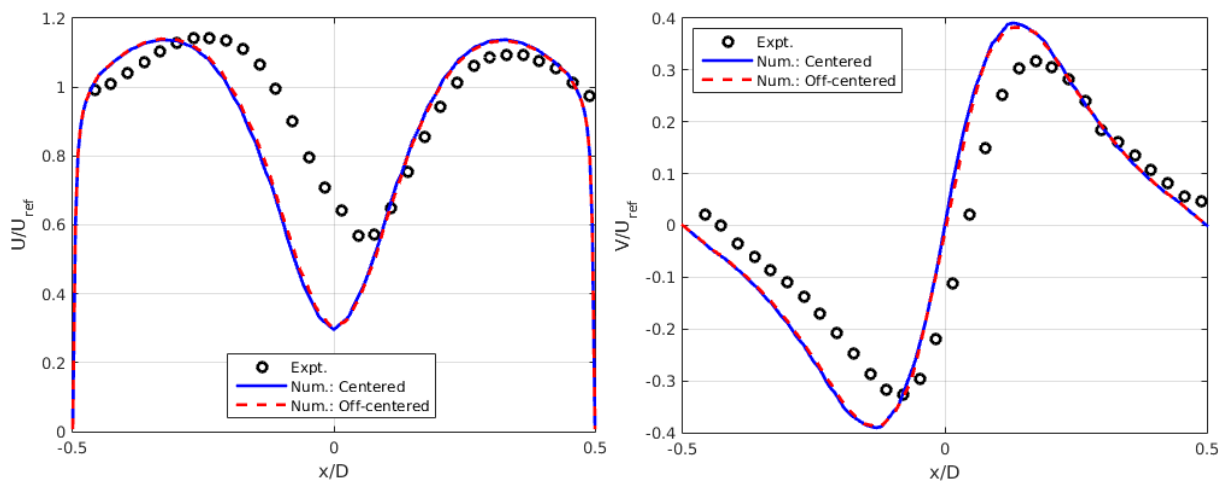


Figure 183. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.493836 mm Upstream of the Rod Tip

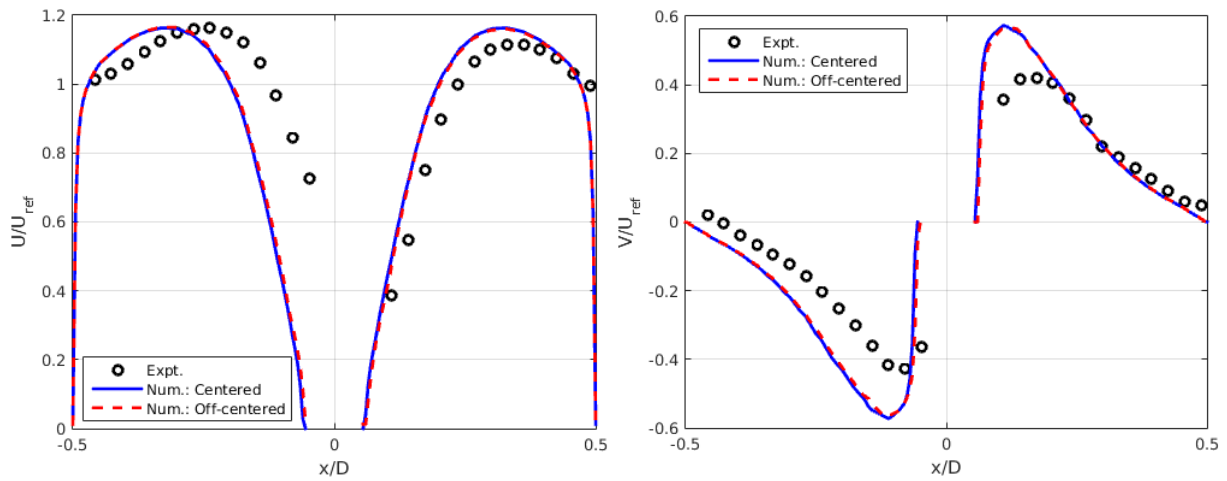


Figure 184. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619 mm Downstream of the Rod Tip

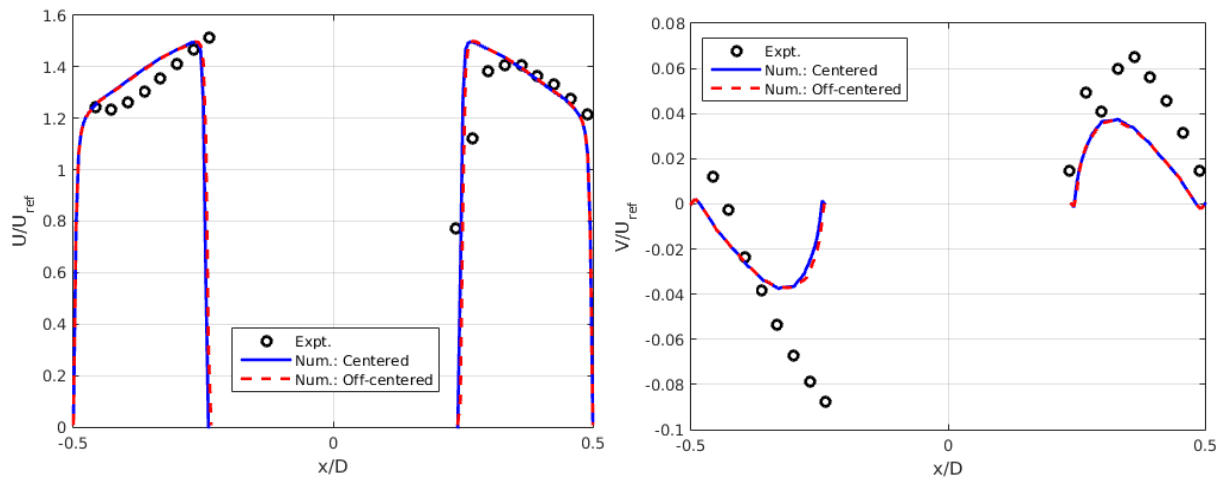


Figure 185. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip

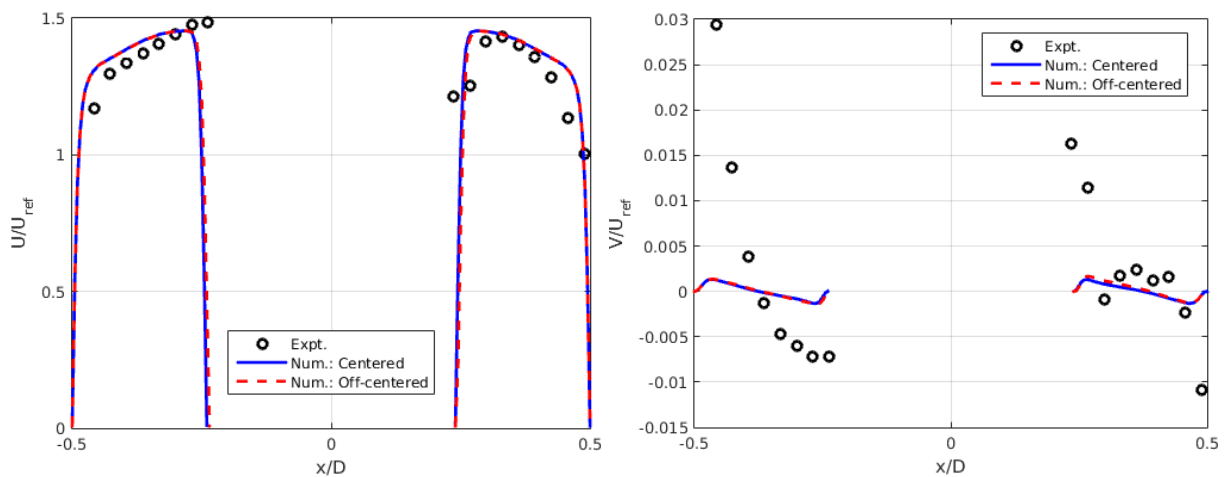


Figure 186. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip

The same vertical profiles are considered again for comparison with the centered case (section 8.2.3.1) as presented in Figure 187. As was observed with the horizontal profiles, the applied offset does not change the results to the extent that the solutions match the experiment.

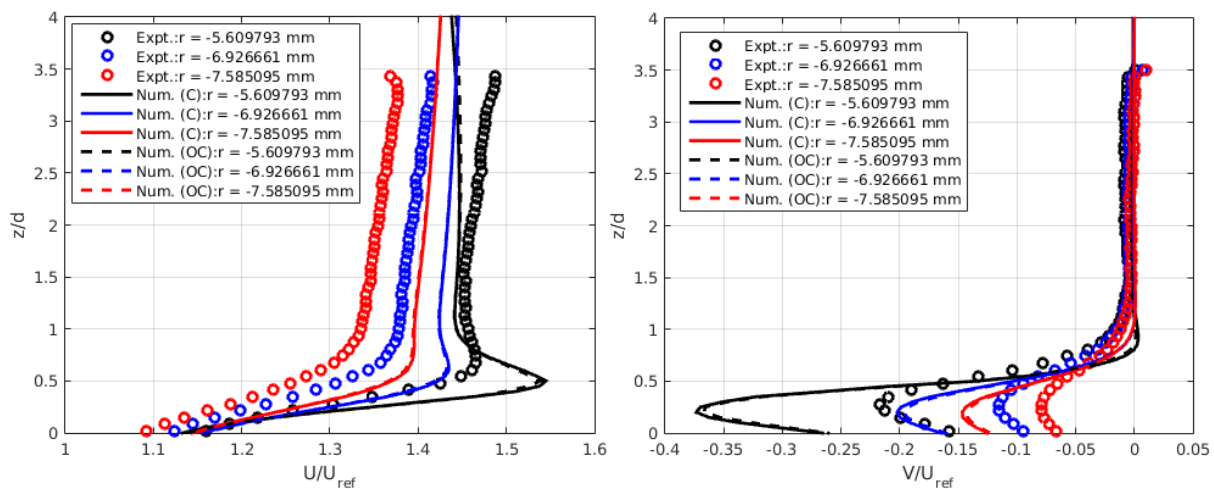


Figure 187. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.609793 mm, -6.926661 mm and -7.858095 mm

To appreciate the effect of the applied offset, additional vertical profiles are considered as shown in Figure 188. It is observed that only in the vicinity of the wall, does the velocity profile show a significant change while for the locations away from the vibrating rod, the effect is minimal.

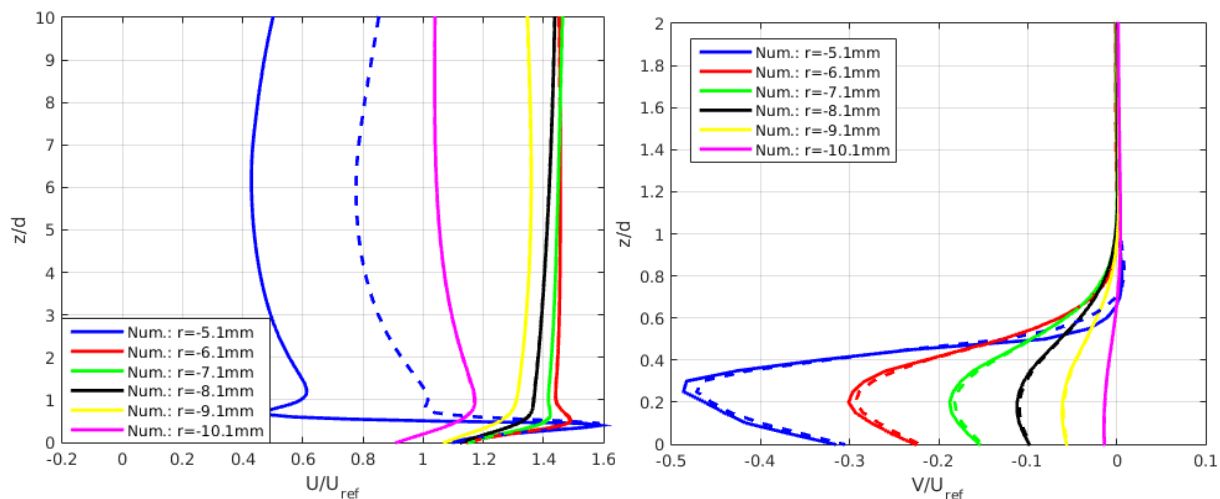


Figure 188. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.1 mm, -6.1 mm, -7.1 mm, -8.1 mm, -9.1 mm and -10.1 mm for the Centered (solid lines) and Off-Centered (dashed lines) Cases

The motion of the rod tip has been monitored during the simulation for a duration similar to the centered case. The time history obtained from the simulation is compared against the centered case and the experiment in Figure 189. The time units are non-dimensional in flow passes (FP) and the displacements are non-dimensionalized using the rod diameter. All results were centered about the 0 axis by removing the mean of the data (excluding the first 3.5FP for the numerical results).

During this procedure, it was found that the rod tip was oscillating around [1.47675 mm, 0.02654 mm] in the experiment, giving a radial offset of 1.477 mm. Numerically, the corresponding mean location is [0.0019 mm, -0.0015 mm] for the centered rod and [0.0639 mm, 0.0716 mm] for the off-centered rod, giving corresponding radial offsets of 2.4 μm and 95.98 μm respectively.

It is interesting to observe that the radial offset for the off-centered rod is much lower than what was prescribed at rest (0.3 mm or 300 μm). It is confirmed that the fixation point of the rod remains at a radial offset of 0.3 mm throughout the simulation. This implies that the slender rod was, in fact, pushed by the flow in such a way that the rod tip was pushed closer to the axis of the confining pipe. It is thus understood that the type of offset observed in the experiment is not achieved by simply moving the fixation point. It is thus hypothesized that the rod was not inserted perfectly vertical, but was installed at an angle in the experiment. This would allow the rod tip to be vibrating about a radial offset of 1.477 mm as obtained from the time history.

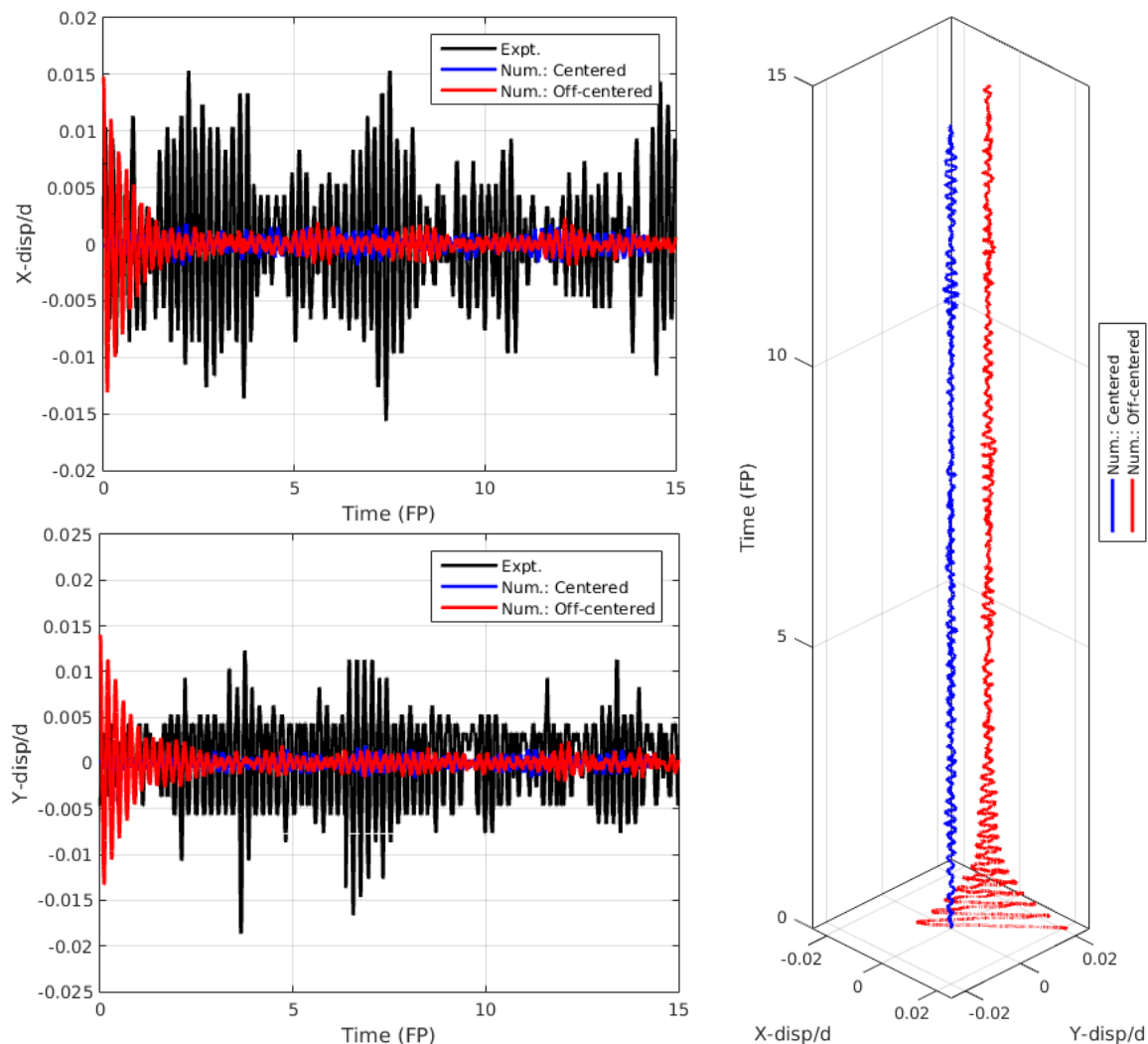


Figure 189. Non-dimensional Time History of the Non-dimensional Rod Tip Front (left top) and Side (left bottom) Displacements. 3D Time History of Rod Tip Displacements (right)

Key results obtained from the time history are presented in Table 47. The RMS polar displacement is still numerically predicted to be about an order of magnitude lower than the experiment for the off-centered case albeit showing slight improvements compared to the centered case. Unlike the experiment, where the RMS front (x-) and side (y-) displacements are very dissimilar, the numerical results show similar results with the difference between the two directions reducing for the off-centered case. It is hypothesized that it could partly be due to the applied offset with equal initial offsets in both directions.

Table 47. Obtained Displacement Results:

Property	Expt.	Num. (C)	Num. (OC)
RMS Polar Disp. (μm)	80.688	8.199	8.764
RMS X-Disp. (μm)	64.756	6.121	6.024
RMS Y-Disp. (μm)	48.138	5.455	6.365

The Power Spectral Density (PSD) of the rod tip front (x-) displacement is compared to the experimental data in Figure 190. The trends of the PSD are in good agreement with the experiment. The numerical results are found to agree well with each other showing minimal difference arising on account of the applied offset. The numerical results are observed to be consistently lower in energy compared to the experiment. The frequencies of the peaks obtained numerically, experimentally and analytically are provided in Table 48. The numerical results are in good agreement for predicting the first two peaks available in the experimental data with the off-centered case showing a better agreement. The peaks frequencies still significantly differ from the analytical solution at higher modes of natural frequencies.

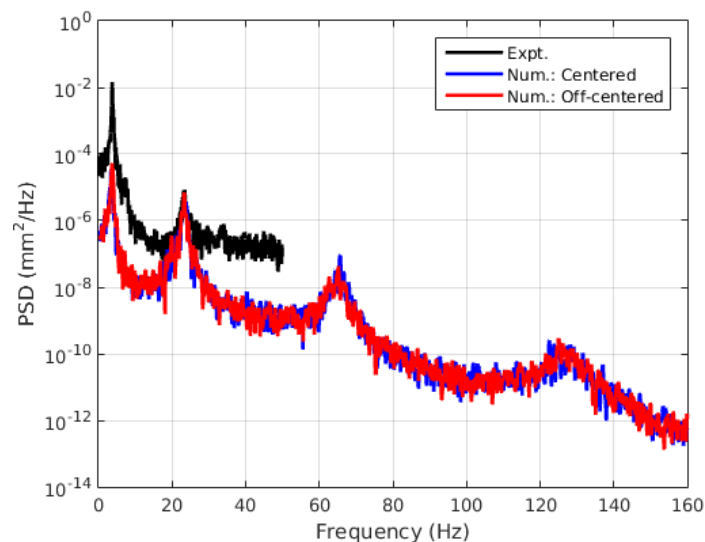


Figure 190. PSD of the Tip Displacement in the Front (x-) Direction

Table 48. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically

Approach	#1	#2	#3	#4
Num.: C (Hz)	3.64	23.81	65.68	128.2
Num.: OC (Hz)	3.73	23.33	65.17	125.0
Exp. (Hz)	3.76	23.47	N/A	N/A

Analytical (Hz)	3.71	25.6	71.8	141
------------------------	-------------	-------------	-------------	------------

A similar look is also taken for the rod tip side (y-) displacement as shown in Figure 191. 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 190). The first mode frequency for both directions is summarized in Table 49. The numerical average frequencies are observed to be in good agreement with each other, the experiment and theory.

Table 49. Frequencies of the Numerical and Experimental Side Displacement PSD Peak #1 Compared to the Beam Natural Frequencies Obtained Analytically

Approach	Front	Side	Average
Num.: C (Hz)	3.64	3.71	3.68
Num.: OC (Hz)	3.73	3.79	3.76
Exp. (Hz)	3.76	3.71	3.73
Analytical (Hz)	3.71	3.71	3.71

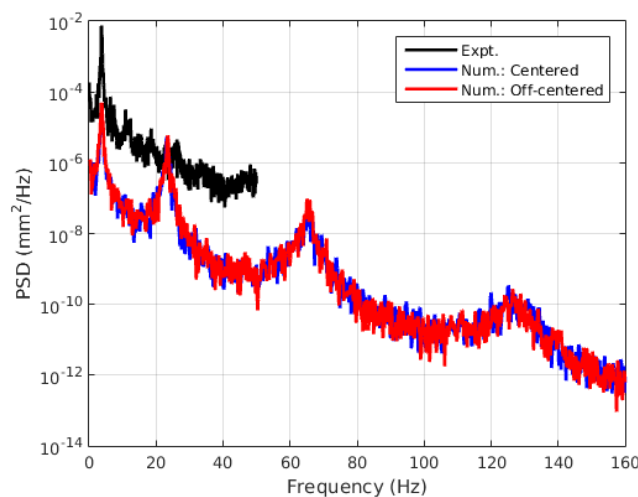


Figure 191. PSD of the Tip Displacement in the Side (y-) Direction

8.4 Blunt Tip

Simulations are carried out for the free-clamped flow at $Re=23.33k$ for the rod with a blunt tip. The Re is again defined using the velocity and distance in the annular gap between the rod and the confining pipe. Simulations were run with the rod being positioned exactly in the center of the confining pipe.

The geometry is again created in STAR-CCM+. Only the experimental test section is considered (see Figure 2) including the union cross, part of the outlet pipes and excluding the curved inlet with flow straighteners of the experiment.

8.4.1 Solid Domain

Only the hollow cylinder and the aluminum plug are considered. The effect of the lead shots on the linear mass density is taken into account by modifying the density of the stainless steel that makes up the hollow cylinder. The material properties selected are listed in Table 50.

Table 50. Solid Material Properties

Material	E (GPa)	ρ (kg/m³)	ν (-)
Aluminum	69	2740	0.33
Stainless Steel	193	33625.47746	0.26

A structured hexahedral mesh is created for the hollow cylinder while a hexahedral mesh is created for the aluminum end plug. Mesh refinement is controlled for the hollow cylinder by prescribing the number of elements in the axial, radial and azimuthal directions and for the end plug by prescribing the base size, and number of elements in the axial and azimuthal directions. As the blunt end plug can be envisioned as being made up of two cylinders of different diameters, separate control is possible for them. Several candidate meshes are considered as listed in Table 51. Tests were carried out in a sequential fashion to determine, in order, the number of elements for the hollow cylinder and end plug cylinders. These are indicated by the colored blocks in Table 51.

Table 51. Solid Mesh Candidates (A=Axial, C=Circumferential, R=Radial, CC=Cell Count)

Mesh Level	Hollow Cylinder				End Plug-1			End Plug-2		
	A	C	R	CC	C	A	CC	C	A	CC
H011A 21-31	1200	100	3	360000	100	8	8608	100	2	2108
H111A 21-31	1000	100	3	300000	100	8	8608	100	2	2108
H211A 21-31	800	100	3	240000	100	8	8608	100	2	2108
H311A 21-31	600	100	3	180000	100	8	8608	100	2	2108
H411A 21-31	400	100	3	120000	100	8	8608	100	2	2108
H210A 21-31	800	100	4	320000	100	8	8608	100	2	2108
H211A 21-31	800	100	3	240000	100	8	8608	100	2	2108
H212A 21-31	800	100	2	160000	100	8	8608	100	2	2108
H213A 21-31	800	100	1	80000	100	8	8608	100	2	2108
H202A 21-31	800	120	2	192000	100	8	8608	100	2	2108
H212A 21-31	800	100	2	160000	100	8	8608	100	2	2108
H222A 21-31	800	80	2	128000	100	8	8608	100	2	2108

H232A 21-31	800	60	2	96000	100	8	8608	100	2	2108
H212A 01-31	800	100	2	160000	100	12	12912	100	2	2108
H212A 11-31	800	100	2	160000	100	10	10760	100	2	2108
H212A 21-31	800	100	2	160000	100	8	8608	100	2	2108
H212A 31-31	800	100	2	160000	100	6	6456	100	2	2108
H212A 41-31	800	100	2	160000	100	4	4304	100	2	2108
H212A 51-31	800	100	2	160000	100	2	2152	100	2	2108
H212A 61-31	800	100	2	160000	100	1	1076	100	2	2108
H212A 10-31	800	100	2	160000	120	10	14650	100	2	2108
H212A 11-31	800	100	2	160000	100	10	10760	100	2	2108
H212A 12-31	800	100	2	160000	80	10	7340	100	2	2108
H212A 13-31	800	100	2	160000	60	10	4030	100	2	2108
H212A 10-01	800	100	2	160000	120	10	14650	100	8	5270
H212A 10-11	800	100	2	160000	120	10	14650	100	6	4216
H212A 10-21	800	100	2	160000	120	10	14650	100	4	3162
H212A 10-31	800	100	2	160000	120	10	14650	100	2	2108
H212A 10-41	800	100	2	160000	120	10	14650	100	1	1054
H212A 10-40	800	100	2	160000	120	10	14650	120	1	1437
H212A 10-41	800	100	2	160000	120	10	14650	100	1	1054
H212A 10-42	800	100	2	160000	120	10	14650	80	1	684
H212A 10-43	800	100	2	160000	120	10	14650	60	1	407



The meshes are 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

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

where gravity is also accounted for. The time instant of 0.12s, when the body force is removed, is carefully chosen to match the time instant the maximum rod displacement is achieved. To incorporate the natural damping of the rod in air, the Rayleigh damping model of STAR-CCM+ is employed using a mass damping coefficient of 0 Hz and a stiffness damping coefficient of 6.987s which is a value that targets the desired experimental damping ratio for an expected vibrational frequency. The 2nd order Newmark integration scheme is used with a model constant of 0.5.

For this test, the linear geometry model and linear finite elements are employed and the simulation is run, starting at rest conditions, for 10 s using a time step size of 2.4 ms and 5 iterations per time step. A sample rod tip displacement time history so obtained for mesh H212A10-41 is shown in Figure 192 along. No experimental data is available exactly for this rod. However, given the same mass density and very similar length of rod (1045 mm vs 1043 mm), the values of the hemispherical tip are recalled. The natural frequency and damping ratio extracted from the time history are compared to the assumed values in Table 52 for all meshes. Based on the computational effort and the accuracy of the natural frequency and damping ratio, mesh H212A10-41 is finalized for the solid and the approach for selecting the Rayleigh damping model parameters is used for the following tests. Figure 193 shows the finalized mesh.

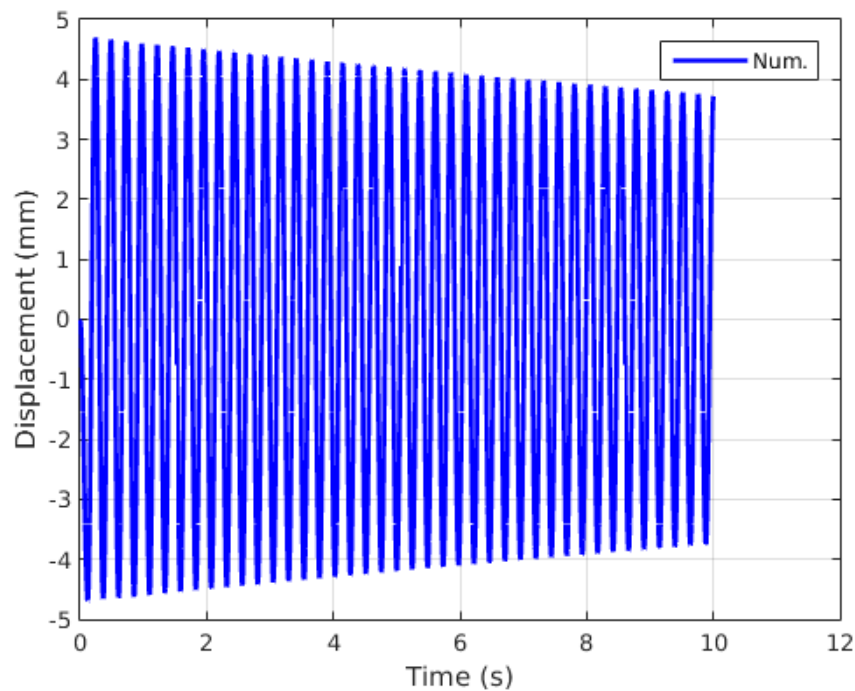


Figure 192. Time History of Vibrations for the Mesh H212A10-41

Table 52. Obtained Results: (Assumed Experimental: $f_{n,a} = 4.10\text{Hz}$, $\zeta_{n,a} = 0.9 \times 10^{-3}$)

Mesh Level	$f_{n,a}$ (Hz)	Error (%)	$\zeta_{n,a} (\times 10^{-3})$	Error (%)	Total CPU Time ($\times 10^5$ s)
H011A2 1-31	4.0984	-0.0399	0.908	0.8911	21.278
H111A2 1-31	4.0984	-0.0399	0.908	0.9058	7.017
H211A2 1-31	4.0984	-0.0399	0.908	0.9355	5.873
H311A2 1-31	4.0984	-0.0399	0.909	1.0108	4.310
H411A2 1-31	4.0994	-0.0147	0.909	0.9891	4.440
H210A2 1-31	4.0984	-0.0399	0.908	0.9345	7.509
H211A2 1-31	4.0984	-0.0399	0.908	0.9355	5.873
H212A2 1-31	4.0984	-0.0399	0.908	0.9358	4.261
H213A2 1-31	4.1025	0.0610	0.909	1.0214	3.979
H202A2 1-31	4.0994	-0.0147	0.909	0.9707	5.498
H212A2 1-31	4.0984	-0.0399	0.908	0.9358	4.261
H222A2 1-31	4.0984	-0.0399	0.909	1.0063	3.197
H232A2 1-31	4.0984	-0.0399	0.908	0.8737	2.473
H212A0 1-31	4.0984	-0.0399	0.908	0.9358	3.922
H212A1 1-31	4.0984	-0.0399	0.908	0.9358	3.857
H212A2 1-31	4.0984	-0.0399	0.908	0.9358	4.261
H212A3 1-31	4.0984	-0.0399	0.908	0.9358	4.189
H212A4 1-31	4.0984	-0.0399	0.908	0.9358	3.991
H212A5 1-31	4.0984	-0.0399	0.908	0.9358	4.225
H212A6 1-31	4.0984	-0.0399	0.908	0.9358	4.221

H212A1 0-31	4.0984	-0.0399	0.909	0.9473	4.055
H212A1 1-31	4.0984	-0.0399	0.908	0.9358	3.857
H212A1 2-31	4.0984	-0.0399	0.909	1.0166	4.200
H212A1 3-31	4.1061	0.14886	0.91	1.1263	4.037
H212A1 0-01	4.0984	-0.0399	0.909	0.9483	4.190
H212A1 0-11	4.0984	-0.0399	0.909	0.9483	4.111
H212A1 0-21	4.0984	-0.0399	0.909	0.9473	4.072
H212A1 0-31	4.0984	-0.0399	0.909	0.9473	4.055
H212A1 0-41	4.0984	-0.0399	0.909	0.9473	3.935
H212A1 0-40	4.0984	-0.0399	0.909	0.9473	4.103
H212A1 0-41	4.0984	-0.0399	0.909	0.9473	3.935
H212A1 0-42	4.0984	-0.0399	0.909	0.9512	3.939
H212A1 0-43	4.0984	-0.0399	0.909	0.9512	3.887

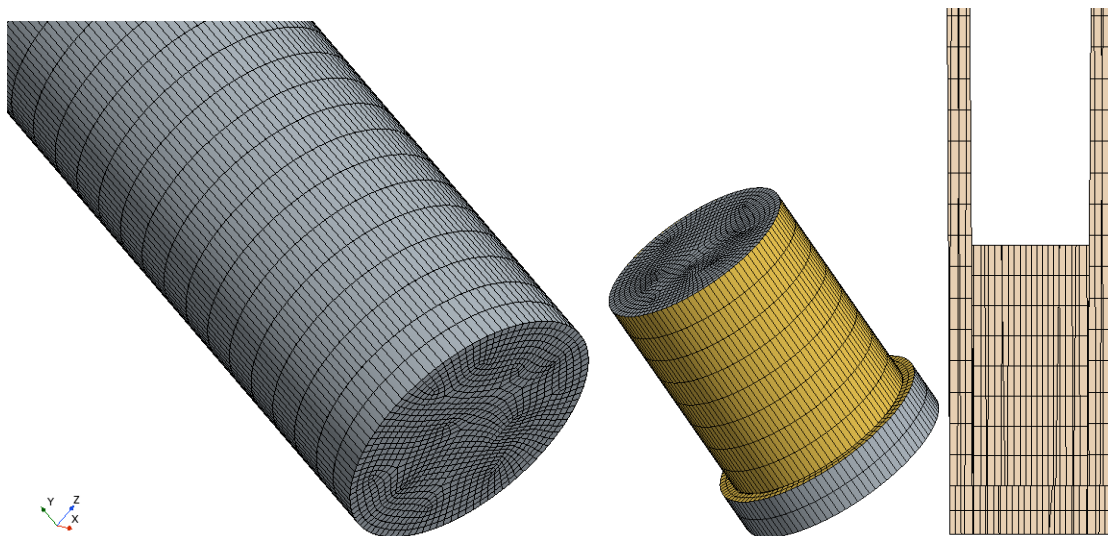


Figure 193. Solid Mesh H212A10-41: Closeup of the Rod Tip (left), View of the End Plug (middle) and Sectional Front View (right)

8.4.2 Fluid Domain

Only the test section's fluid volume is considered for the analysis. The properties of water used in this study are provided in Table 53.

Table 53. Fluid Material Properties

Material	$\mu (\times 10^{-3})$ (kg/(ms))	ρ (kg/m³)
Water	1.002	998.23

A hexahedral mesh is created for the fluid domain using the automatic mesher of STAR-CCM+. Mesh refinement is controlled using the base size. A volumetric refinement is provided such that the bulk cell size is halved starting at about 12 rod diameters upstream of the rod free end. 3 candidate meshes are considered as listed in Table 54. To test its suitability for the FSI forced flow test cases, a vibration test similar to that carried out for the solid domain is carried out. For this test, the fluid mesh uses 5 prism layers on all walls with a stretching factor of 1.24. This is an FSI calculation for the vibration of the rod in still water.

Table 54. Fluid Mesh Candidates

Mesh Level	Base Size (mm)	Cell Count (M)
F3	7.5	6.244
F2	5.0	6.796
F1	3.5	9.304
F2_ID	5.0	6.796

For the solid, the linear geometry model and linear finite elements are used. Tests are carried out using the Rayleigh damping model using the same model constants as the test in vacuum. An additional test labelled F2_ID is carried out where the mass damping and stiffness damping coefficients of 0Hz and 7.789s are used respectively, targeting the aforementioned structural damping ratio in vacuum but at the expected vibrational frequency in water. The 2nd order Newmark integration scheme is used with a model constant of 0.5.

For the fluid, an implicit unsteady SIMPLE scheme laminar flow solver is selected with a 2nd order upwind convection scheme. The inlet of the fluid domain is set to a velocity inlet boundary condition with an inflow of 0m/s while the two outlets are set to pressure outlets with 0Pa gauge pressure. All walls are prescribed with no-slip boundary conditions. A reference pressure of 1atm and an acceleration due to gravity of [0,-9.81,0] m/s² is applied for the fluid domain.

The FSI coupling is chosen to be strong coupling where the complete fluid loading is applied to the wetted faces of the structure and the structural displacement is fed back to the fluid via a mesh morpher step at every iteration of every time step. The fluid mesh is moved using the B-Spline mesh morphing technology of STAR-CCM+.

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

$$a = \begin{cases} [0, -9.81, -0.135] \text{ m/s}^2, & t < 0.1326\text{s} \\ [0, -9.81, 0] \text{ m/s}^2, & t \geq 0.1326\text{s} \end{cases}$$

The time instant of 0.1326 s, 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 2nd order temporal discretization scheme with a time step size of 2.6 ms and 25 iterations per time step.

A sample rod tip displacement time history so obtained for mesh F2_ID is shown in Figure 194. The natural frequency and damping ratio extracted from the time history can be compared to the experiment and is done so in Table 55 for all meshes. Based on the computational effort and the accuracy of the natural frequency and damping ratio, the meshing strategy of mesh F2_ID is finalized for the fluid and the choice of Rayleigh damping model parameters is used for the FSI tests.

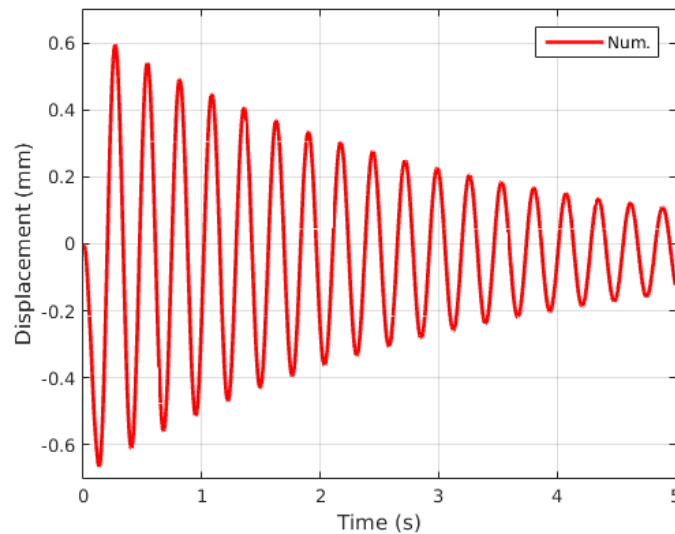


Figure 194. Time History of Vibrations for the Mesh F2_ID

Table 55. Obtained Results: (Experiment (Cioncolini, et al., 2022): $f_{n,a} = 3.72\text{Hz}$, $\zeta_{n,a} = 15 \times 10^{-3}$)

Mesh Level	$f_{n,a}$ (Hz)	Error (%)	$\zeta_{n,a} (\times 10^{-3})$	Error (%)	Total CPU Time ($\times 10^6$ s)
F3	3.6809	-1.050	14.66	-2.254	2.40
F2	3.6784	-1.119	14.25	-5.024	3.56
F1	3.6784	-1.119	14.14	-5.755	3.77
F2_ID	3.6784	-1.119	14.27	-4.888	2.70

8.4.3 Forced Flow FSI Study

With the numerical modelling strategy validated, the experiment with forced flow at $Re=23.33k$ is considered. The numerical model described in sections 8.4.1 and 8.4.2 is reused. Before tackling these cases, a few additional considerations are made for this model. Given the complex nature of rod vibrations, for improved accuracy, the non-linear geometry model is selected for the solid, allowing for updating the stiffness matrix in each iteration. The rod surface roughness is included here. The roughness height is set to be $0.45 \mu\text{m}$ for the aluminum plug surface and is set to be $2.5 \mu\text{m}$ for the stainless steel surface assuming values from the hemispherical rod end cases.

The fluid mesh is slightly modified with it having 15 prism layers with a stretching factor of 1.39 while having the same bulk mesh size, resulting in a mesh with 13.255M cells as shown in Figure 195. This is done targeting a wall-resolved mesh for a high flow rate of $Re=70k$, ensuring that the wall y^+ remains lower than 1 for both cases of interest. Given that the flow is turbulent, the Standard K-Epsilon Low Re model of STAR-CCM+ is employed with a 2nd order convection scheme, realizability, Yap correction and a cubic constitutive relationship to better account for the anisotropy of turbulence. Gravity is considered for both solid and fluid domains, and a reference fluid pressure of 1 atm is used.

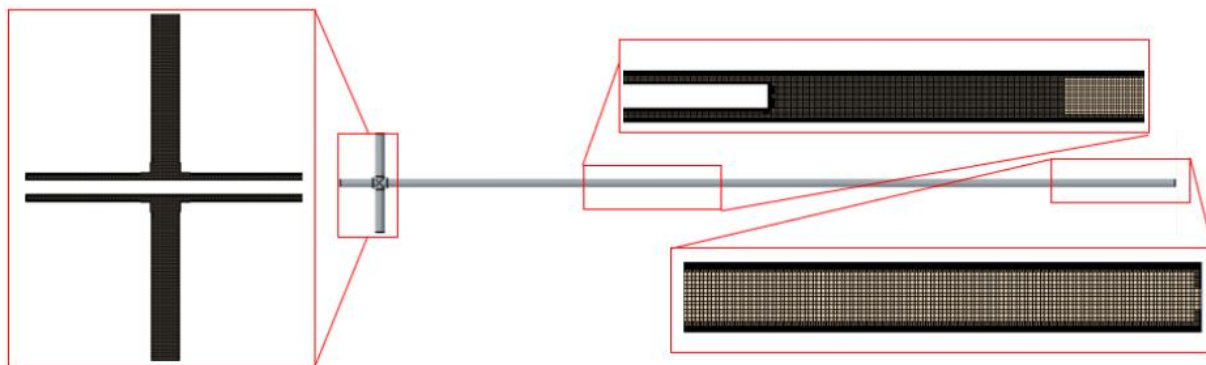


Figure 195. Fluid Mesh

The fluid inlet is provided with a common inflow turbulence of 5% and unique inflow velocity of 1.661m/s. The fluid outlets are defined as pressure outlet boundaries with 0 Pa gauge pressure. The wetted surfaces of the rod are set to be FSI interfaces and the fluid mesh is allowed to morph. The rod starts from rest. To generate good initial conditions for the fluid domain, for each inflow velocity, a CFD-only RANS simulation is carried out starting from rest conditions for 2000 iterations which is enough to establish a flow field around the fixed rod. From this point, the implicit unsteady SIMPLEC scheme flow solver is selected with a 2nd order upwind convection scheme.

The FSI calculation is thus run as a transient using a time step size of 0.1 ms and 25 iterations per time step. The simulation is run for a total of 12.50 FP (15.6246 s). To compare the vibrational statistics with the experiment, the first 3.5 FP of the obtained solution is ignored. To compare the time-averaged contour plots with the experimental PIV results, the first 4.5 FP of the obtained solution are ignored. This is done to forego the initial transient where the rod just begins to move.

8.4.3.1 Results: $Re=23.33k$

The contour plots are shown in Figure 196 and Figure 197. The experimental results are instantaneous and it is not known at which time instant it is. Two numerical results are presented: Instantaneous at the end of the simulation ($t=12.5FP$) and time-averaged. From these results, the flow in the vicinity of the rod tip is revealed. The flow path shows features similar to the curved end such as a stagnation point at the rod tip and flow speed up in the annular gap between the rod and the confining pipe, but also clearly shows different features such as vortex shedding at the blunt edge of the rod. This is characterized by a low vertical velocity zone and high vorticity zone in the rod tip vicinity. Qualitatively, there is good agreement with the experiment.

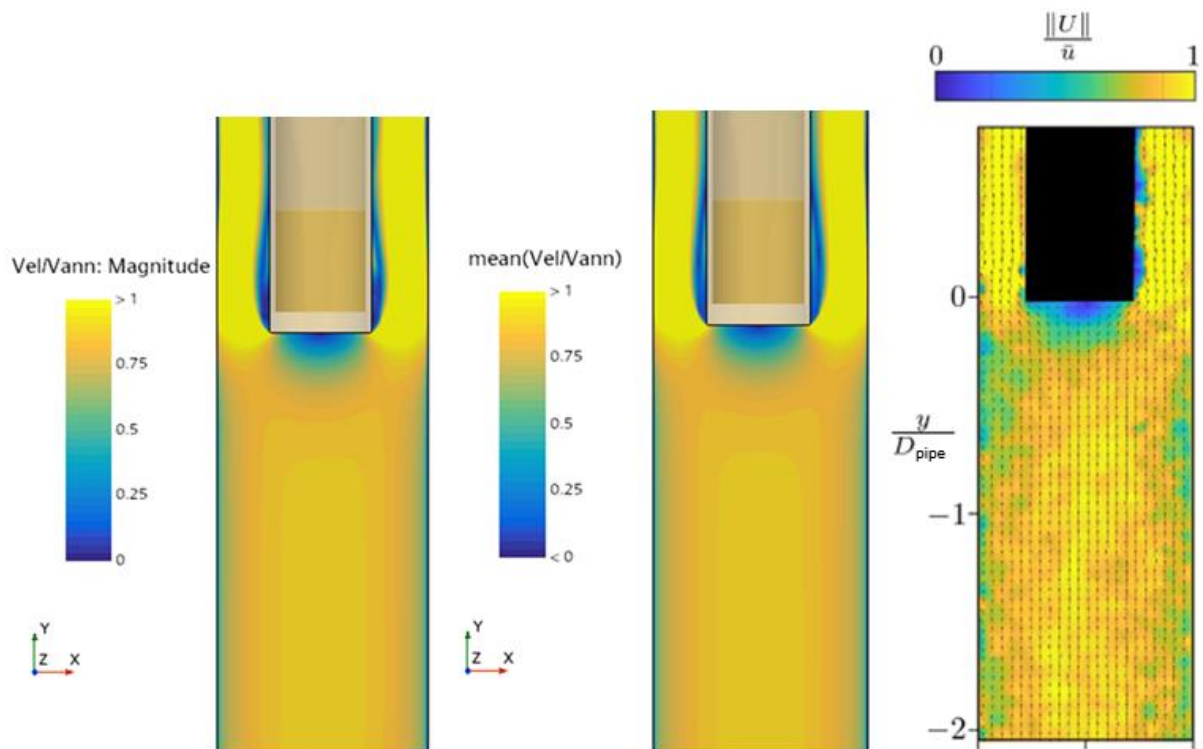


Figure 196. Comparison of Instantaneous Numerical ($t=12.5FP$, left), Time-averaged Numerical (middle) and Instantaneous Experimental (Cioncolini, et al., 2022) (right) Results at $Re\ 23.33k$ for Non-dimensionalized Axial Component of Velocity

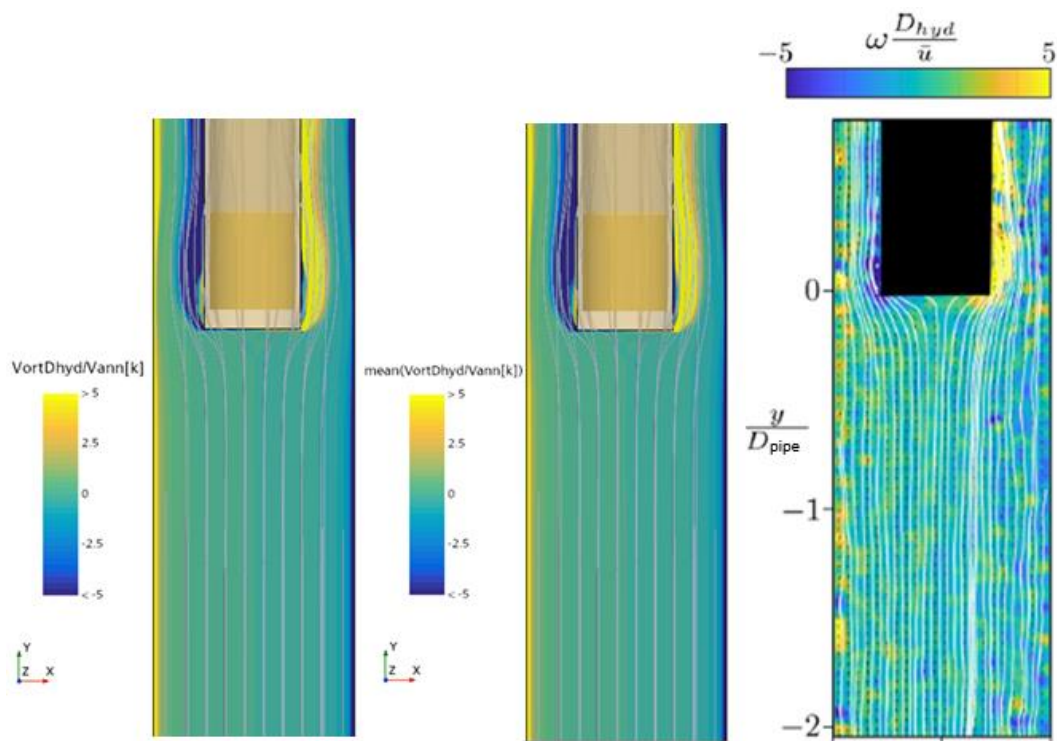


Figure 197. Comparison of Instantaneous Numerical ($t=12.5FP$, left), Time-averaged Numerical (middle) and Instantaneous Experimental (Cioncolini, et al., 2022) (right) Results at $Re\ 23.33k$ for Non-dimensionalized In-plane Component of Velocity

Several radial profiles at the PIV measurement locations are provided in Figure 198 to Figure 204. The PIV data inside of the beam are discarded owing to metallic reflections. The time-averaged numerical and instantaneous experimental axial and transverse velocity profiles are drawn. No time-averaged data was obtained from the experimentalists. There is good agreement for the axial velocity. Some discrepancy is noted for the transverse velocity, especially much upstream of the rod tip. The agreement for the transverse velocity is better in the vicinity of the rod tip as the flow predominantly changes direction to accommodate in the narrower annular region between the rod and confining pipe. Moving further downstream, the numerical solution predicts flow with predominantly axial flow (or in other words, predicts decreasing transverse flow) but at a rate faster than the experiment. Stronger conclusions cannot be drawn as the comparison is between instantaneous experimental data and time-averaged numerical data.

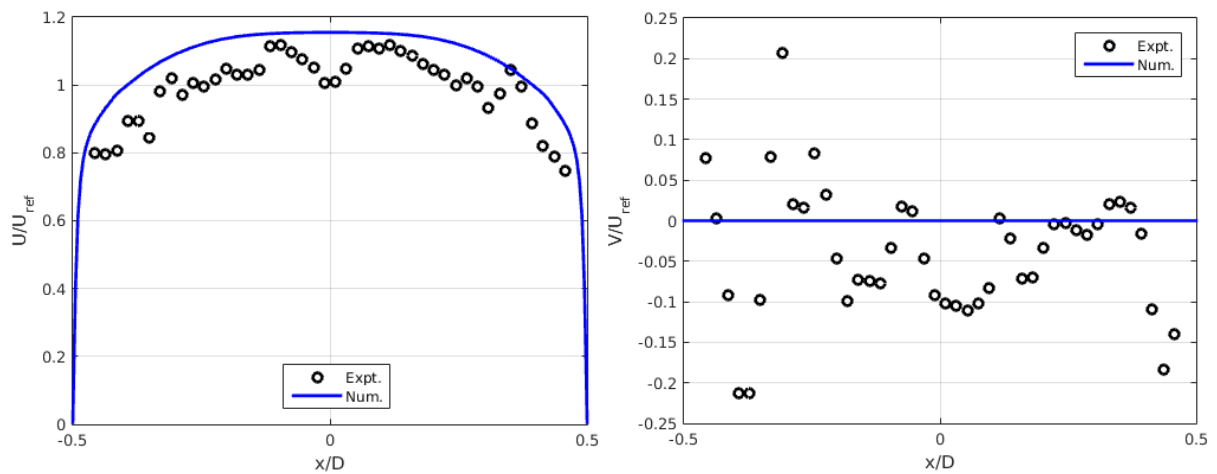


Figure 198. Axial (left) and Transverse (right) Velocity Profiles 45.20711503385 mm Upstream of the Rod Tip

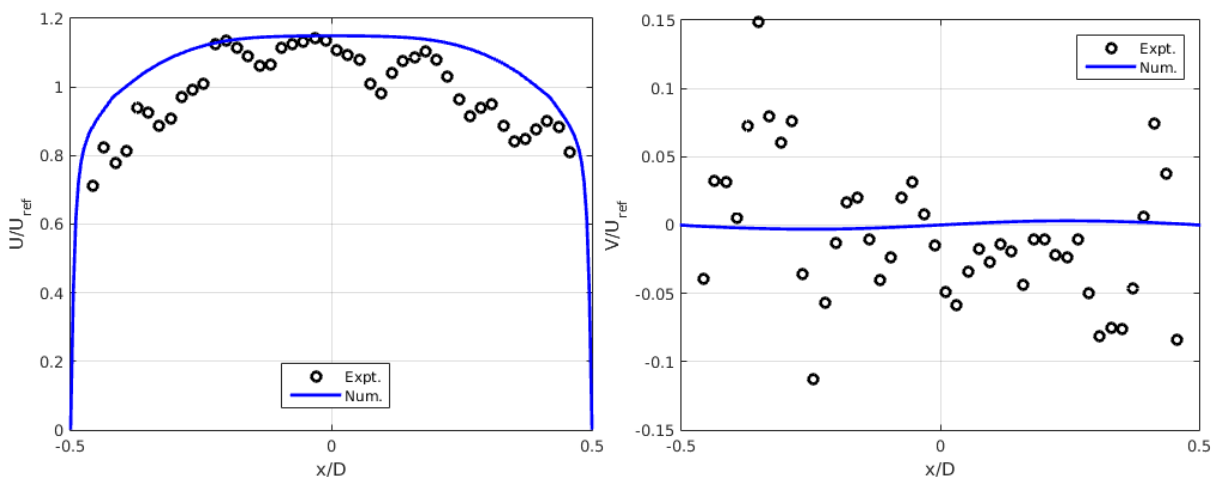


Figure 199. Axial (left) and Transverse (right) Velocity Profiles 15.927518269985 mm Upstream of the Rod Tip

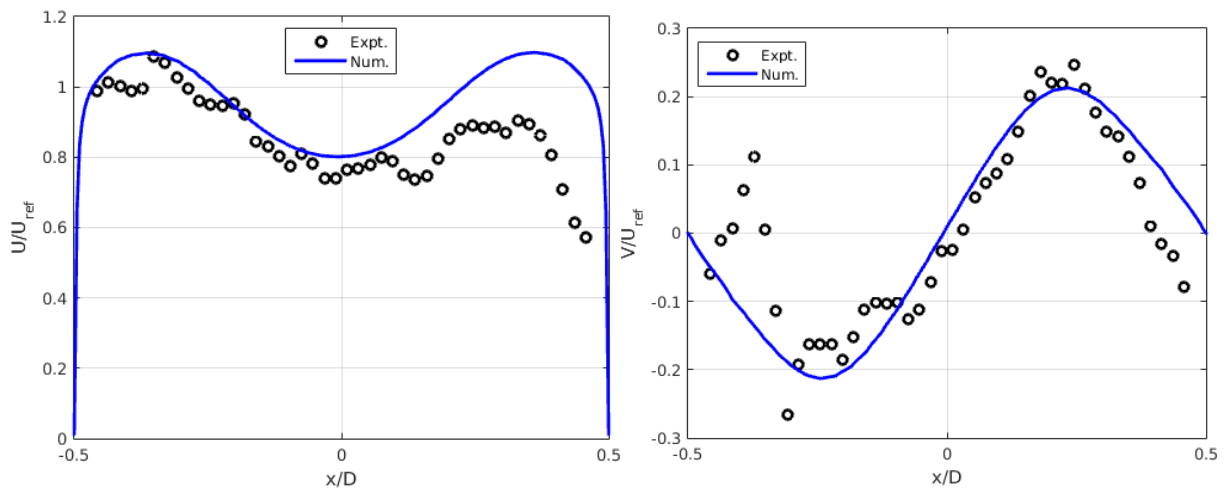


**Figure 200. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles
3.949501409525 mm Upstream of the Rod Tip**

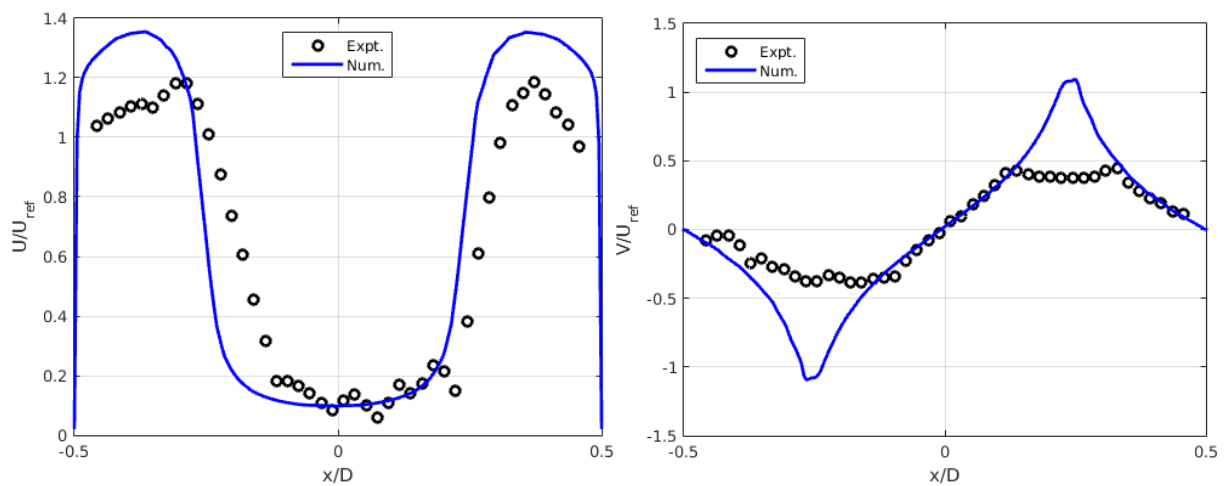


**Figure 201. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles
0.40045937981 mm Upstream of the Rod Tip**

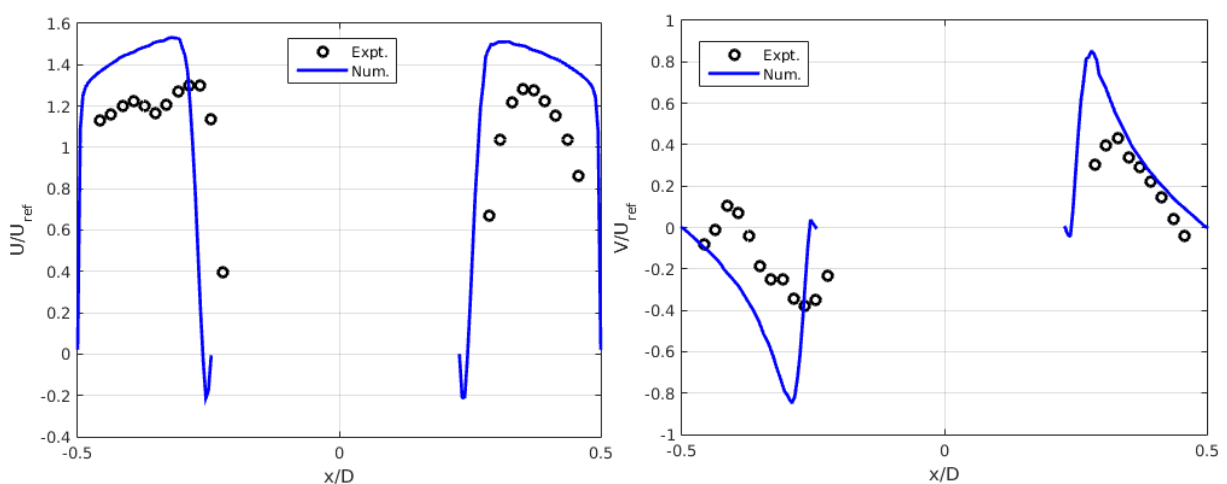


**Figure 202. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 0.164619
mm Downstream of the Rod Tip**

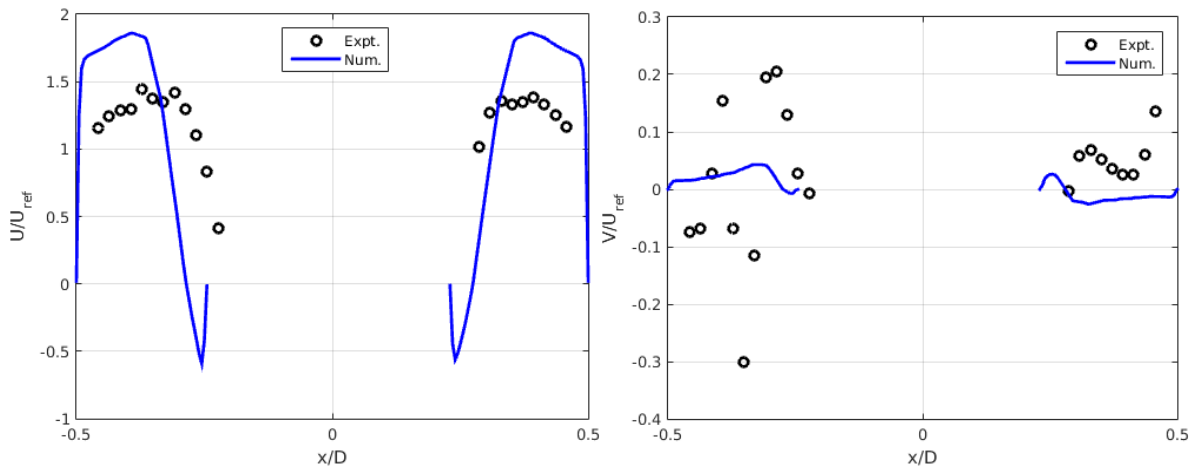


Figure 203. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 6.74898 mm Downstream of the Rod Tip

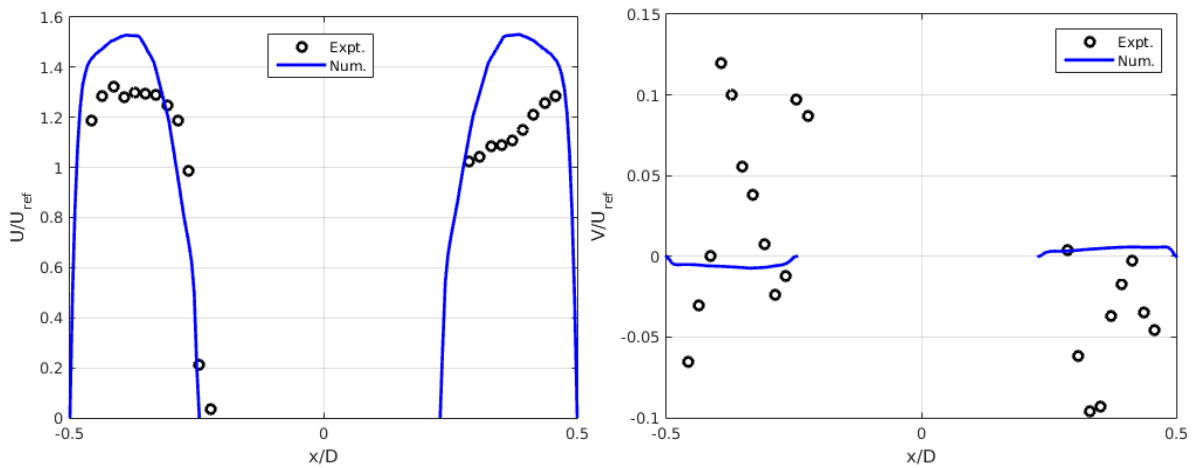


Figure 204. Non-dimensional Axial (left) and Transverse (right) Velocity Profiles 29.135799 mm Downstream of the Rod Tip

An attempt was made to compare the instantaneous experimental data against numerical results for the vertical probe lines. However, as can be seen from Figure 205, the instantaneous experimental data is too noisy to interpret meaningful trends. Thus only time-averaged numerical results are looked at.

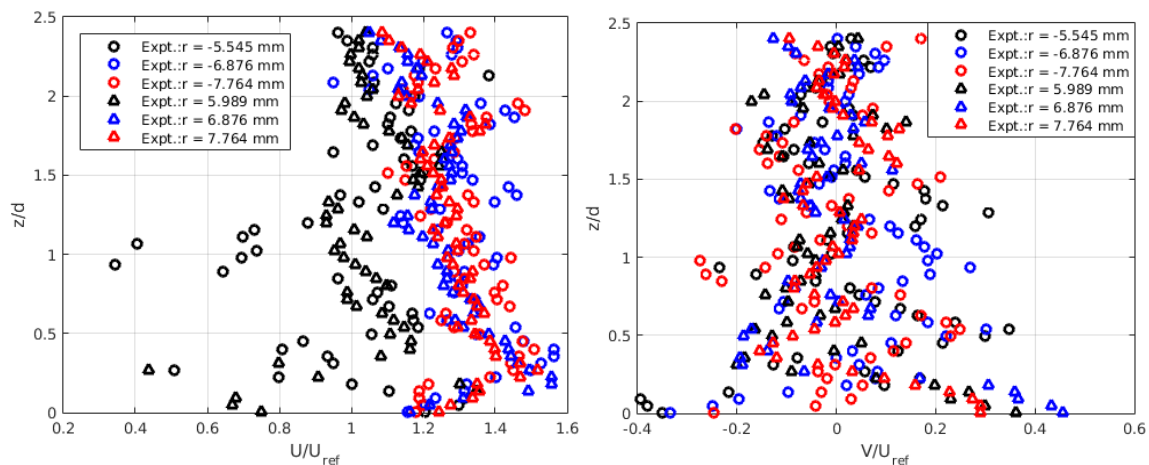


Figure 205. Instantaneous Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations ± 5.609793 mm, ± 6.926661 mm and ± 7.858095 mm

Vertical profiles are presented for the numerical results in Figure 206. From the axial velocity profile, it is observed that in the vicinity of the rod wall, moving upwards, the axial velocity decreases and then increases, while away from the rod wall, the velocity increases then decreases. Noticing similar results for the black, green and yellow curves, it is understood that there is less sensitivity to the solution away from the rod and confining pipe walls.

From the transverse velocity plot, it is observed that moving upwards for most curves, the velocity increases in magnitude and then decreases reaching and maintaining a near-zero value from a non-dimensional height of $\sim 2d$. Moving away from the rod, the peak velocity magnitude is also observed to first increase and then decrease when approaching the confining pipe wall. Both plots give insight on how the flow adjusts from pipe flow to annular flow. Given the change of transverse flow direction, flow separation occurring due to the blunt edge is indicated.

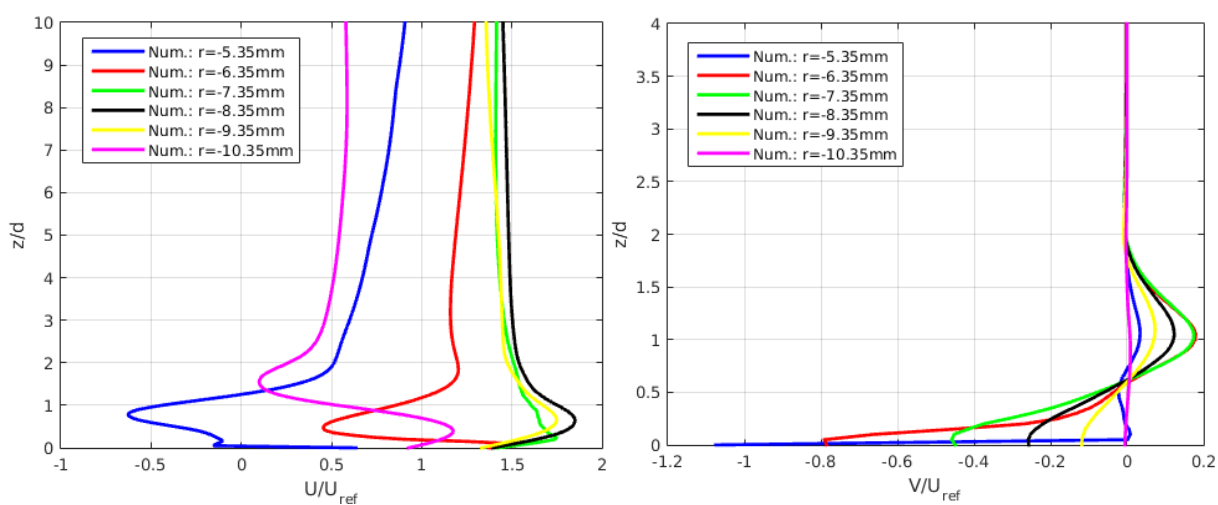


Figure 206. Time-Averaged Non-dimensional Axial (left) and Transverse (right) Velocity Profiles at Radial Locations -5.1 mm, -6.1 mm, -7.1 mm, -8.1 mm, -9.1 mm and -10.1 mm

The motion of the rod tip has been monitored during the simulation albeit not as long as the experiment owing to computational costs. The time history obtained from the simulation is compared against the experiment in Figure 207. The time units are non-dimensional in flow passes (FP) and 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 first 3.5FP for the numerical result). During the procedure, the rod tip was found to oscillate around a mean location of $[-0.02004 \text{ mm}, 0.03625 \text{ mm}]$ in the simulation, giving a radial offset of 0.0414 mm arising from a numerical artifact. For the experiment, there also exists an offset, however, since the origin of the coordinate system in the experimental data is unclear, the exact offset cannot be determined. An attempt was made assuming the same origin as the curved end and this gave a radial offset of 7.484 mm which is physically impossible.

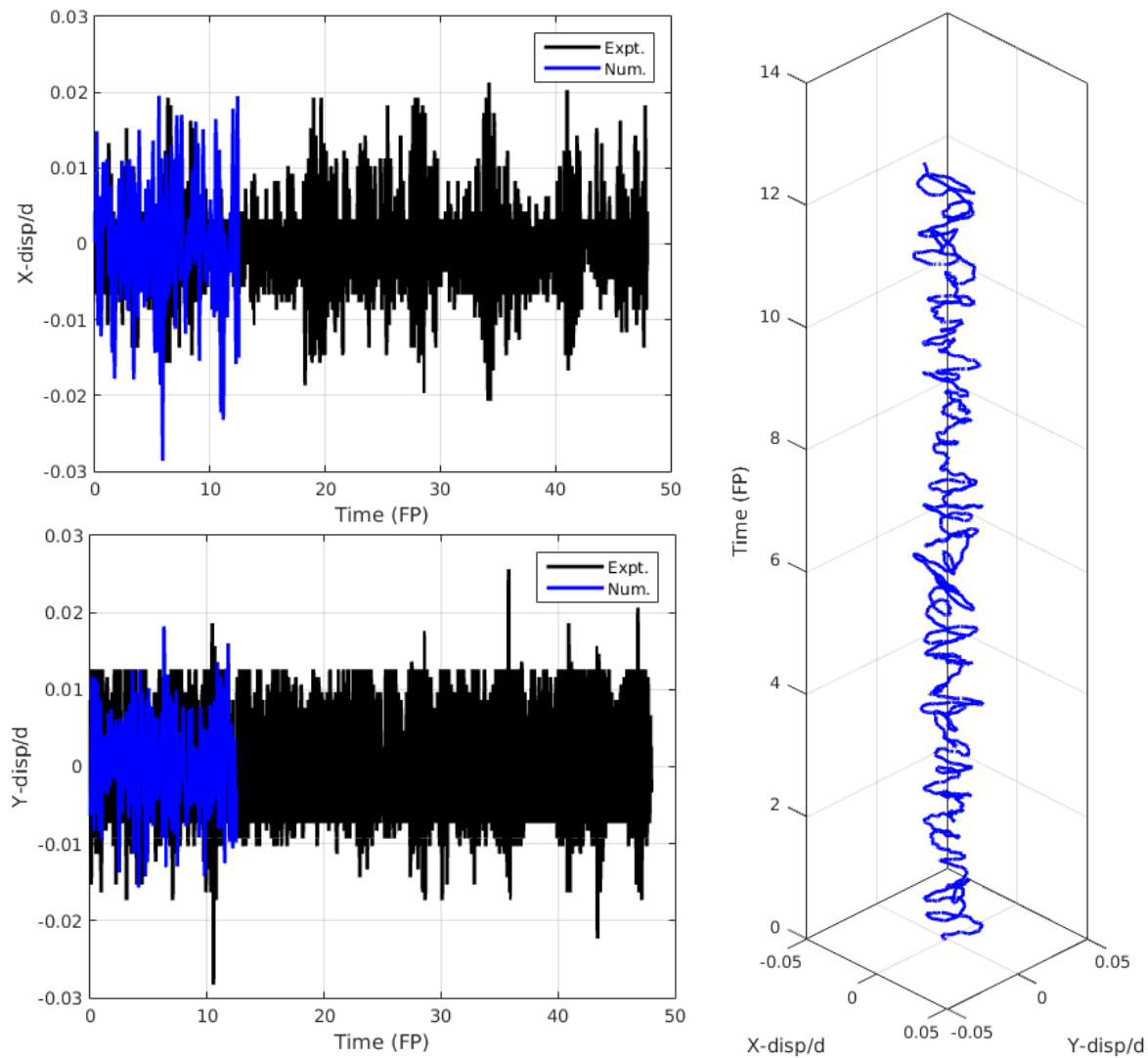


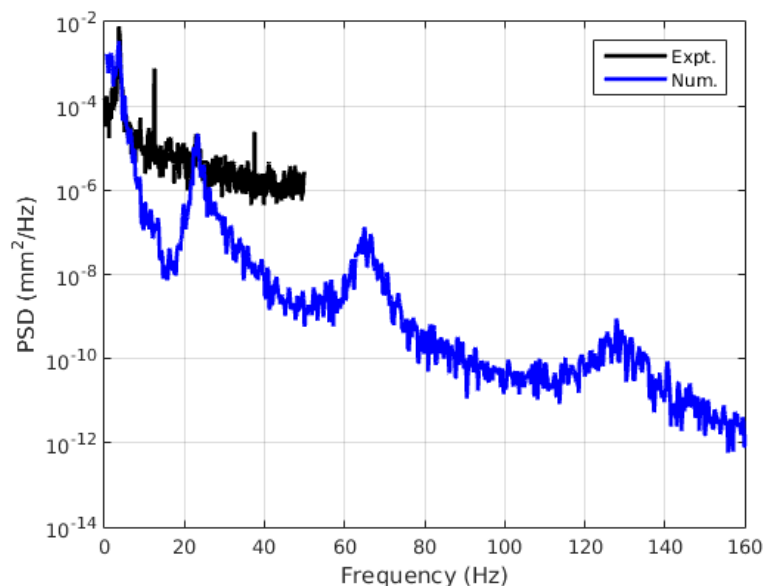
Figure 207. Non-dimensional Time History of the Non-dimensional Rod Tip Front (left top) and Side (left bottom) Displacements. 3D Time History of Rod Tip Displacements (right)

Key results obtained from the time history are presented in Table 56. The RMS polar displacement is numerically predicted to be only ~ 1.06 times lower than the experiment. This is much better than the order of magnitude difference observed in the hemispherical end simulations. Since there is flow separation and recirculation zone created by the blunt edge of the rod tip, URANS is able to capture these large structures well and thus make better predictions for such vibrations that are partly vortex-induced. The remaining underprediction is attributed to URANS being incapable of predicting fluctuations in velocity and pressure necessary to achieve turbulence-induced vibrations. The RMS front (x-) and side (y-) displacements are found to be very dissimilar for the experiment and the numerical predictions. This is hypothesised to stem partly from the rod tip offset and partly from the asymmetry of the test section.

Table 56. Obtained Displacement Results:

Property	Expt.	Num.
RMS Polar Disp. (μm)	106.808	100.987
RMS X-Disp. (μm)	59.811	80.543
RMS Y-Disp. (μm)	88.491	60.920

The Power Spectral Density (PSD) of the rod tip front (x-) displacement is compared to the experimental data in Figure 208. The trends of the PSD are in good agreement with the experiment. However, the numerical results are consistently lower in energy compared to the experiment. The frequencies of the peaks obtained numerically, experimentally and analytically are provided in Table 57. The numerical results are in good agreement for predicting the first two peaks available in the experimental data. The peaks frequencies significantly differ from the analytical solution at higher modes of natural frequencies. The experiment shows additional peaks at 12.5 Hz and 37.5 Hz. The source of these peaks are unclear.

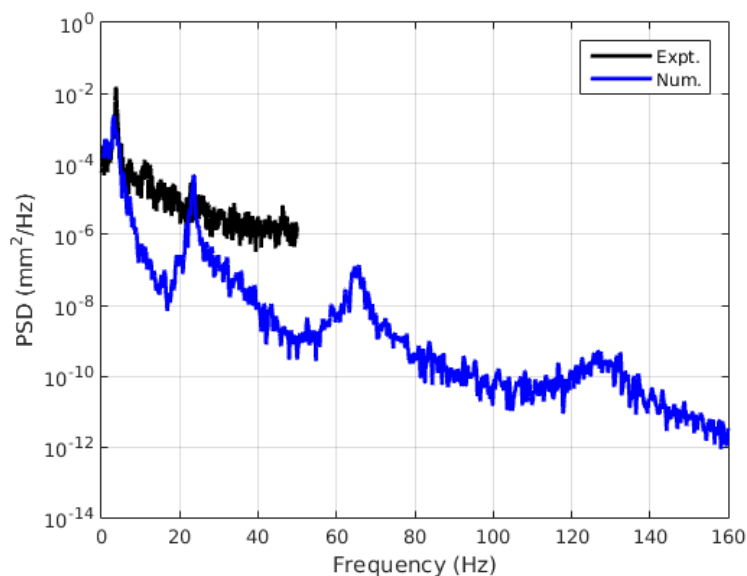
**Figure 208. PSD of the Tip Displacement in the Front (x-) Direction****Table 57. Frequencies of the Numerical and Experimental Front Displacement PSD Peaks Compared to the Beam Natural Frequencies Obtained Analytically**

Approach	#1	#2	#3	#4
Num. (Hz)	3.65	23.38	64.98	127.90
Exp. (Hz)	3.68	23.30	N/A	N/A
Analytical (Hz)	3.71	25.6	71.8	141

A similar look is also taken for the rod tip side (y-) displacement as shown in Figure 209. 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 208). The first mode frequency for both directions is summarized in Table 58. The average frequency is observed to be underpredicted by ~8% compared to the experiment and theory. The difference between the front and side frequency prediction is also concerning and requires further investigation.

Table 58. Frequencies of the Numerical and Experimental Side Displacement PSD Peak #1 Compared to the Beam Natural Frequencies Obtained Analytically

Approach	Front	Side	Average
Num. (Hz)	3.65	3.20	3.43
Exp. (Hz)	3.68	3.78	3.73
Analytical (Hz)	3.71	3.71	3.71

**Figure 209. PSD of the Tip Displacement in the Side (y-) Direction**

8.5 Conclusions

Simulations were performed by NRG PALLAS for the cantilevered rod configuration. These were performed for the hemispherical end and blunt end configurations using URANS for the fluid solver. Key findings indicated that good qualitative results and vibration frequency predictions are made for all cases. For the hemispherical end, the vibration amplitude is underpredicted by an order of magnitude. For the blunt end, the agreement is quite good to the experiment and is off by a factor of only ~ 1.05 . This is attributed to the presence of clear flow separation and vortex generation which can be captured well with URANS.

During the qualitative comparisons, it was discovered that the rod was actually placed at an offset. A value of 0.3 mm was provided by the experimentalists. To test the effect of this offset, the simulation for the curved end at $Re=21.2k$ was repeated. It was found that the provided offset did not influence the outcome of the calculation. Later, it was discovered that the offset in the experiment was actually about 1.5 mm, provided the origin of the coordinate system in the experimental data was correctly adopted. The latter constraint was a strong concern for the blunt end where the estimated radial offset was impossible with the assumed origin. Owing to resource constraints, another calculation could not be performed.

Effectively, the current work establishes the baseline accuracy that can be expected by the current state-of-the-art URANS. Future work will be performed to enhance the prediction capabilities via the inclusion of stochastic models for velocity and pressure fluctuations.

9. Simulations performed by IPP

9.1 Physics and numerical modelling

The ANSYS CFX code is applied for CFD calculations. To organize FSI analysis, the user FORTRAN subroutine of the beam equation solver has been implemented and coupled with ANSYS CFX. Thus, the beam motion is accounted in fluid domain at every time step by mesh displacement. Further sections describe the CFD model and appropriate mechanical model used for FSI, as well as their coupling details.

9.1.1 Fluid modelling

CFD model of the Cantilever Rod experiment (see Figure 210) represents the coolant domain (1), enclosed in a confining tube and flowing around the rod (2) fixed at the top. The upward flow is considered, which represents the "free-clamped" flow scheme according to the benchmark description. The model boundaries are chosen as follows – "inlet" boundary condition is placed in such a way, to consider the upstream flow part starting from the flow straightener location in the experiment. Further downstream, the flow part is modelled quite accurately up to the outlet sections located after the flow separation and direction change in union cross (4).

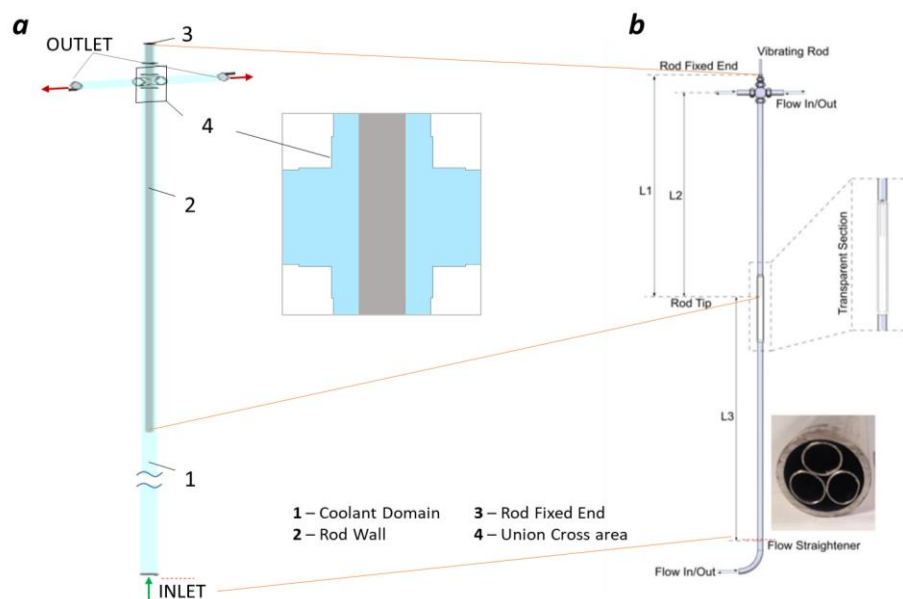


Figure 210. General view of the CFD model (a) according to the schematic representation of the Cantilever Rod test section (b) from benchmark setup (Deliverable 2.1)

The problem is isothermal. The mass flow rate at the inlet and reference pressure at the outlet are used as boundary conditions. The mass flow rate is determined based on the flow cross-sectional area, the coolant density and the Reynolds number specified in the experiment - $Re = 21200$ for the annular channel between the rod and the confining tube. The medium is water at a temperature of $18.2\text{ }^{\circ}\text{C}$, the values of dynamic viscosity and density of which are determined from the IAPWS library and are equal to: $\mu = 1.04737 \cdot 10^{-3} \text{ Pa} \cdot \text{s}$ and $\rho = 998.573 \text{ kg} \cdot \text{m}^{-3}$ respectively. The kinematic viscosity value is $\nu = 1.049 \cdot 10^{-6} \text{ m}^2 \cdot \text{s}^{-1}$.

The SST turbulence model is applied for calculations. Three different computational meshes were generated using ICEM CFD. All meshes are hexahedral with different discretization both in axial/radial directions and near the walls (see Figure 211). Further in the simulations Mesh#2 is used.

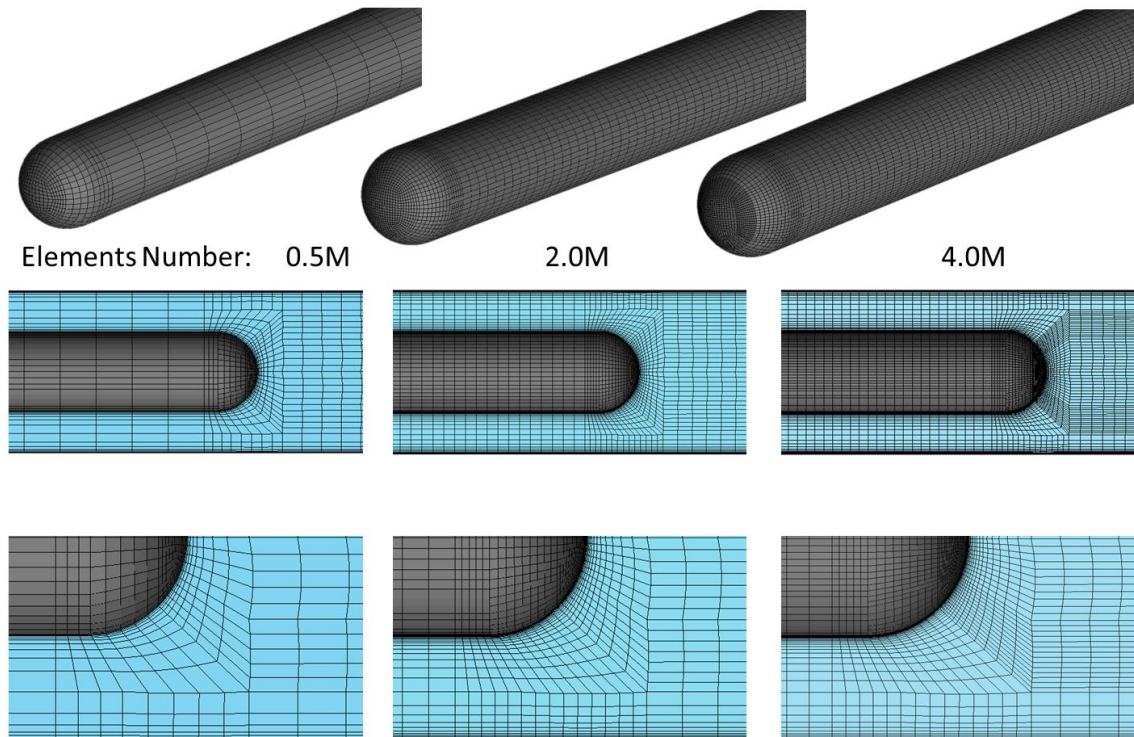


Figure 211. General view of the computational meshes

9.1.2 Mechanical modeling

9.1.2.1 Governing equations

The transversely vibrating beam problem is formulated in terms of the partial differential equations of motion, external excitation function, boundary and initial conditions. The four well-known beam theories (see Table 59) were all introduced before 1921. Analyzing Table 59 we can see that Euler-Bernoulli is the simplest and Timoshenko beam theory the most comprehensive.

Table 59. Four beam theories

Beam Models	Bending Moment	Lateral Displacement	Shear Deformation	Rotary Inertia
Euler-Bernoulli	+	+	-	-
Rayleigh	+	+	-	+
Shear	+	+	+	-
Timoshenko	+	+	+	+

Despite some efforts to come up with a better beam theory, the Euler-Bernoulli and Timoshenko beam theories are the most widely used. Governing equations for Euler-Bernoulli beam theory, as well as for Timoshenko beam theory are summarized in Table 60. A crucial parameter in Timoshenko beam theory is the shape factor k' , which is a function of Poisson's ratio and the frequency of vibration as well as the shape of cross-section.

Table 60. Governing equations for the most widely used theories

Equation	Euler-Bernoulli	Timoshenko
Equilibrium force	$-\frac{\partial Q}{\partial x} dx - \rho A \frac{\partial^2 y}{\partial t^2} dx = 0$	$-\frac{\partial Q}{\partial x} dx - \rho A \frac{\partial^2 y}{\partial t^2} dx = 0$
Equilibrium moment	$Q dx + \frac{\partial M}{\partial x} dx = 0$	$Q dx + \frac{\partial M}{\partial x} dx - \rho I \frac{\partial^3 y}{\partial x \partial t^2} dx = 0$
Geometry	$\frac{\partial y}{\partial x} = \psi$	$\frac{\partial y}{\partial x} = \psi + \tau$
Material	$M = -EI \frac{\partial \psi}{\partial x}$	$M = -EI \frac{\partial \psi}{\partial x}$ $Q = -k' \tau AG = -\left(\frac{\partial y}{\partial x} - \psi\right) AG$

For Euler-Bernoulli beam the final 4th order equation:

$$EI \frac{\partial^4 y}{\partial x^4} + \rho A \frac{\partial^2 y}{\partial t^2} = 0$$

For Timoshenko beam the final 4th order equation:

$$EI \frac{\partial^4 y}{\partial x^4} + \rho A \frac{\partial^2 y}{\partial t^2} - \rho I \left(1 + \frac{E}{k'G}\right) \frac{\partial^4 y}{\partial x^2 \partial t^2} + \frac{\rho^2 I}{k'G} \frac{\partial^4 y}{\partial t^4} = 0$$

If we apply decomposition $y(x, t) = Y(x) \cdot [\sin(\omega t) + \cos(\omega t)]$, solution of the equations:

$$Y(x) = C_1 Kr_1(\beta x) + C_2 Kr_2(\beta x) + C_3 Kr_3(\beta x) + C_4 Kr_4(\beta x)$$

Here the Kr – are so-called Krylov functions, summarized in Table 61. They are very convenient to use, since they satisfy the Cauchy boundary conditions with the identity matrix:

$$\begin{aligned}
Kr_1(0) &= 1, Kr_2(0) = 0, Kr_3(0) = 0, Kr_4(0) = 0 \\
Kr_1'(0) &= 0, Kr_2'(0) = 1, Kr_3'(0) = 0, Kr_4'(0) = 0 \\
Kr_1''(0) &= 0, Kr_2''(0) = 0, Kr_3''(0) = 1, Kr_4''(0) = 0 \\
Kr_1'''(0) &= 0, Kr_2'''(0) = 0, Kr_3'''(0) = 0, Kr_4'''(0) = 1
\end{aligned}$$

Table 61. Krylov or ‘beam’ functions

Function	Euler-Bernoulli	Timoshenko
Kr_1	$\frac{1}{2}(\cosh \beta x + \cos \beta x)$	$\frac{1}{2\nu}(\beta_2^2 \cosh \beta_1 x + \beta_1^2 \cos \beta_2 x)$
Kr_2	$\frac{1}{2}(\sinh \beta x + \sin \beta x)$	$\frac{1}{2\nu}\left(\frac{\beta_2^2}{\beta_1} \sinh \beta_1 x + \frac{\beta_1^2}{\beta_2} \sin \beta_2 x\right)$
Kr_3	$\frac{1}{2}(\cosh \beta x - \cos \beta x)$	$\frac{1}{2\nu}(\cosh \beta_1 x - \cos \beta_2 x)$
Kr_4	$\frac{1}{2}(\sinh \beta x - \sin \beta x)$	$\frac{1}{2\nu}\left(\frac{1}{\beta_1} \sinh \beta_1 x - \frac{1}{\beta_2} \sin \beta_2 x\right)$

Here for the Euler-Bernoulli beam:

$$\beta = \sqrt[4]{\frac{\rho A \omega^2}{EI}}$$

Here for the Timoshenko beam:

$$\beta_{1,2} = \frac{1}{2} \sqrt{\pm \frac{2L^2 \left(1 + \frac{E}{k'G}\right) \rho \omega^2}{E} + 2 \sqrt{4 \frac{\left(1 - \frac{I \rho \omega^2}{A k'G}\right) \rho \omega^2 L^4 A}{EI} + \frac{\left(1 + \frac{E}{k'G}\right)^2 \rho^2 \omega^4 L^4}{E^2}}}$$

$$\nu = \frac{\beta_1^2 + \beta_2^2}{2}$$

By adding the boundary conditions (see Table 62), we fully define our problem.

Table 62. Boundary conditions formulation

End support	1st	2nd
Free	$Q = 0$	$M = 0$
Supported	$y = 0$	$M = 0$
Clamped	$y = 0$	$\psi = 0$

9.1.2.2 Natural frequencies and mode shapes

The rod outer and inner diameters are $D_o = 10.01 \text{ mm}$ and $D_i = 8.83 \text{ mm}$, respectively. The beam length (including the end-piece) is $L = 1.045 \text{ m}$. The rod is inserted in the experimental loop, with a confining tube inner diameter $D_c = 21.0 \text{ mm}$. The beam is cylindrical and filled with lead shots in the experiment, however, to simplify the model, they are neglected but the density is modified to $\rho_m = 33676.31312 \text{ Kg} \cdot \text{m}^{-3}$ to account for it. The Young's modulus and second moment of area are $E = 1.93 \times 10^{11} \text{ Pa}$ and $I = 1.94 \times 10^{-10} \text{ m}^4$, respectively. The inlet length, upstream of the beam, is $58 D_c$ long, from the flow straightener (located

downstream of an elbow) to the tip of the end-piece. The origin of the domain is taken at the tip of the end-piece.

The added mass for a beam in a cylinder is estimated as follows (Blevins, 2001):

$$m_{add} = \frac{\pi}{4} \rho_f D_o^2 \frac{1 + \left(\frac{D_o}{D_c}\right)^2}{1 - \left(\frac{D_o}{D_c}\right)^2},$$

where $\rho_f = 997.567 \text{ Kg} \cdot \text{m}^{-3}$ denotes the fluid density. The characteristic equation for Euler-Bernoulli:

$$\cosh(\beta_n L) \cos(\beta_n L) + 1 = 0.$$

The characteristic equation for Timoshenko:

$$\tanh^2 \beta_1 - \frac{\left(1 + \frac{\beta_2^2 - \kappa \cos \beta_2}{\beta_1^2 + \kappa \cosh \beta_1}\right) \left(\frac{\beta_1^2}{\beta_2^2} + \frac{\cos \beta_2}{\cosh \beta_1}\right)}{\left(\frac{\beta_1}{\beta_2} - \frac{\sin \beta_2}{\sinh \beta_1}\right) \left(\frac{\beta_1}{\beta_2} + \frac{\beta_2^2 - \kappa \sin \beta_2}{\beta_1^2 + \kappa \sinh \beta_1}\right)} = 0$$

$$\kappa = A \frac{\rho^2 \omega^2 L^2}{k' G EI}$$

The resulting natural frequencies according to both theories are summarized in Table 63 in Air and in Water conditions, i.e. in those calculations only the density is different.

Table 63. Natural frequencies in water

Mode	In Air		In Water	
	Euler-Bernoulli	Timoshenko	Euler-Bernoulli	Timoshenko
1	4.09	4.09	3.72	3.72
2	25.65	25.61	23.30	23.26
3	71.83	71.61	65.25	65.05
4	140.76	140.05	127.86	127.21
5	232.69	230.90	211.36	209.74
6	347.60	343.83	315.74	312.31
7	485.50	478.44	440.99	434.58
8	646.37	572.72	587.12	520.22
9	830.23	634.27	754.12	576.12
10	1037.07	810.82	942.00	736.50

From Table 63 it can be seen that a simple Euler-Bernoulli beam model provides good results, only for the high frequencies $n > 7$ we can see the difference with more accurate Timoshenko beam theory. This is somewhat expected, since the cantilever rod is a slender structure. Also we can clearly see the effect of frequency shift to a lower values, when the surrounding fluid is accounted for. Results in Table 63 are in excellent agreement with the experimental ones, where the first natural frequency in water is equal to 3.72 Hz and 4.1 Hz in air.

The mode shapes of the displacement of the cantilever beam according to Euler-Bernoulli are:

$$y_i = (\sinh \beta_i L + \sin \beta_i L)(\cosh \beta_i x - \cos \beta_i x) - (\cosh \beta_i L + \cos \beta_i L)(\cosh \beta_i x + \cos \beta_i x)$$

The illustration of these modes is presented in Figure 212.

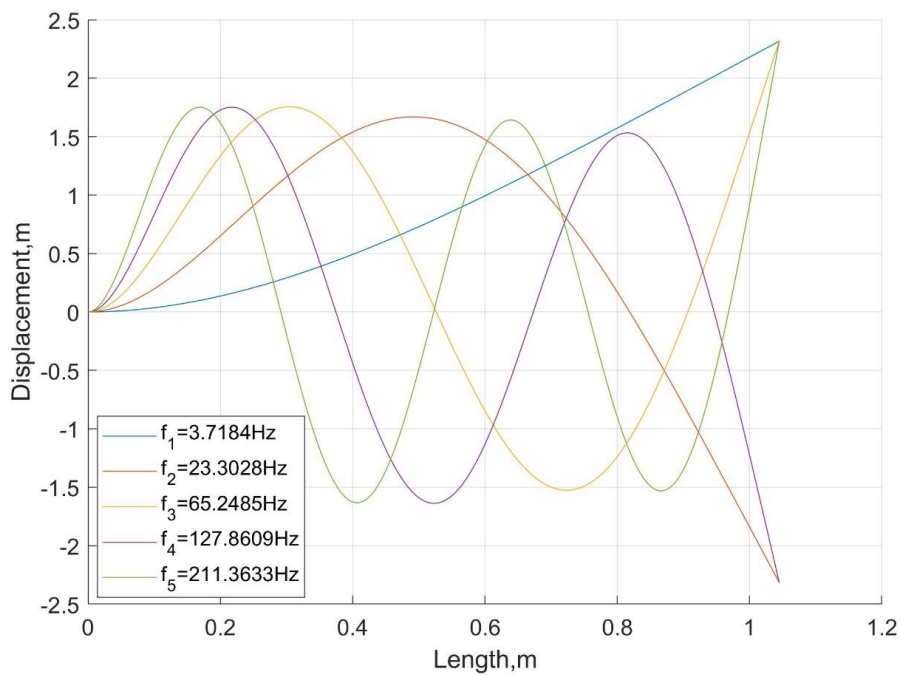


Figure 212. Mode shapes of cantilever rod (Euler-Bernoulli)

Up-flow and down-flow will cause not only lateral forces (and movement), but also some axial forces and vibrations. Although, those frequencies are quite big (the lowest axial frequency is 520Hz), the axial force itself will affect the lateral vibration. For the clamped-free beam the axial force influence can be calculated using simple formula:

$$\omega^* = \omega_1 \sqrt{1 + \frac{NL^2}{14EI}}$$

Here N —denotes the axial force, [N]. In Figure 213 we can see the influence of the axial force on the first natural frequency. Also, from this equation a critical or buckling force can be easily calculated. Compressive force (in case of up-flow) will lower the frequencies, tensile force will magnify the frequency (down-flow case). This effect needs to be estimated after the CFD calculation, and in case of significant axial force, it should be taken into account.

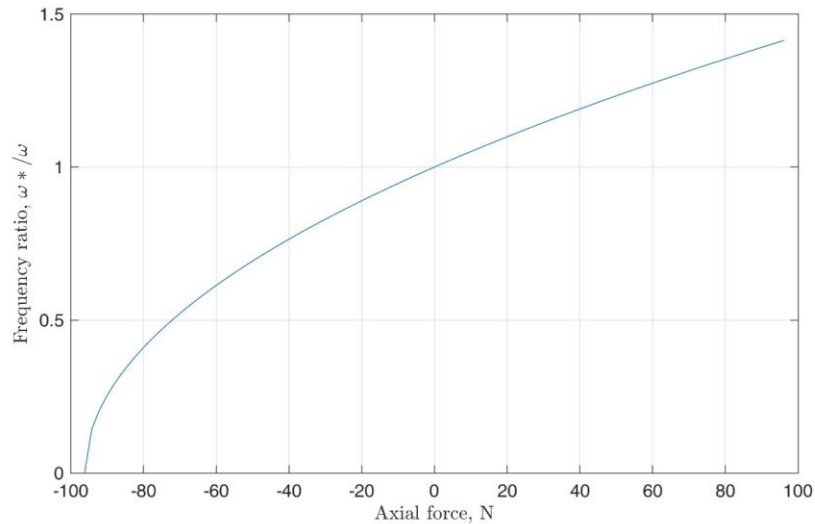


Figure 213. Influence of axial force to the first natural frequency

9.1.2.3 Free vibrations

To analyse the free and forced vibration we will use normal mode superposition method. It is based on the well-known property of self-adjoint systems, orthogonality conditions:

$$\int_0^l Y_i Y_j dx = 0 \text{ if } i \neq j$$

If the modes are also normalized in the following way:

$$\int_0^l Y_i^2 dx = 1$$

It greatly simplifies the linear system analysis, here the resulting displacement is obtained as:

$$y(x, t) = \sum_{i=1}^{\infty} Y_i(x) \eta_i(t)$$

In case of free vibration, the dynamic response, using normal mode superposition method

$$\eta_i(t) = \phi_{0i} \cos \omega_i t + \frac{\psi_{0i}}{\omega_i} \sin \omega_i t$$

Here ϕ_{0i} and ψ_{0i} represent the initial displacement and velocities in normal coordinates:

$$\phi_{0i} = \int_0^l f_1(x) Y_i(x) dx$$

$$\psi_{0i} = \int_0^l f_2(x) Y_i(x) dx$$

Here the $f_1(x)$ and $f_2(x)$ are the initial displacements and velocities over the rod length. Damping can also be accounted for, using the proportional damping assumption. For this, it is necessary to introduce the damping frequency.

$$\omega_1 = \sqrt{\omega_1^2 - n^2}$$

$$n = \frac{\omega \delta}{\sqrt{4\pi^2 + \delta}}$$

Here δ –logarithmic decrement. Then:

$$\eta_i(t) = e^{-nt} \left(\phi_{0i} \cos \omega_1 t + \frac{\phi_{0i} n + \psi_{0i}}{\omega_1} \sin \omega_1 t \right)$$

$$\eta_i(t) = e^{-nt} \left(-\phi_{0i} \omega_1 \sin \omega_1 t + (\phi_{0i} n + \psi_{0i}) \cos \omega_1 t \right)$$

$$- n e^{-nt} \left(\phi_{0i} \cos \omega_1 t + \frac{\phi_{0i} n + \psi_{0i}}{\omega_1} \sin \omega_1 t \right)$$

To verify free vibration approach a free cantilever rod vibration in air and in water was tested, the results are presented in Figure 214 and Figure 215

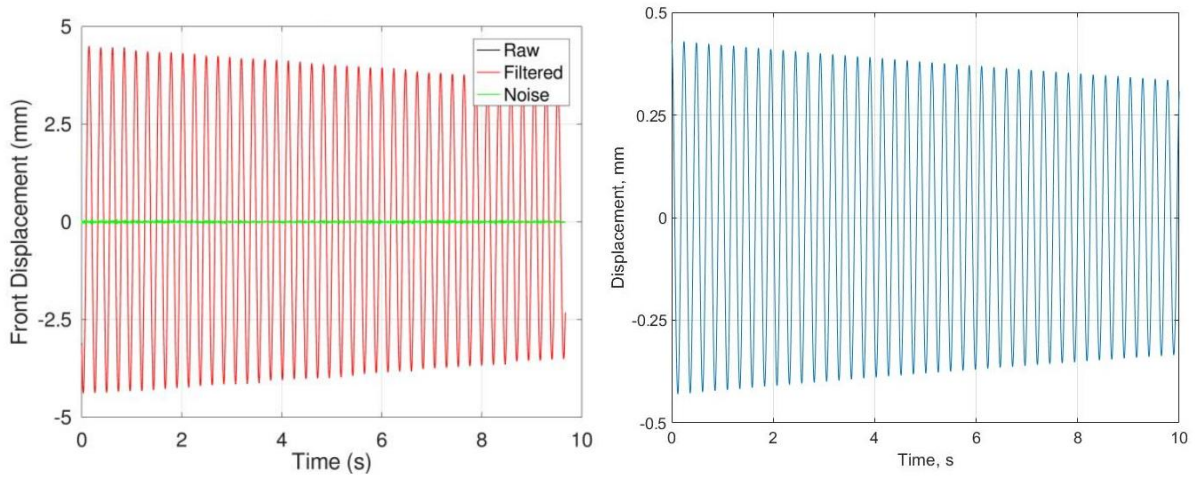


Figure 214. Free vibration in air, damping ratio 0.001

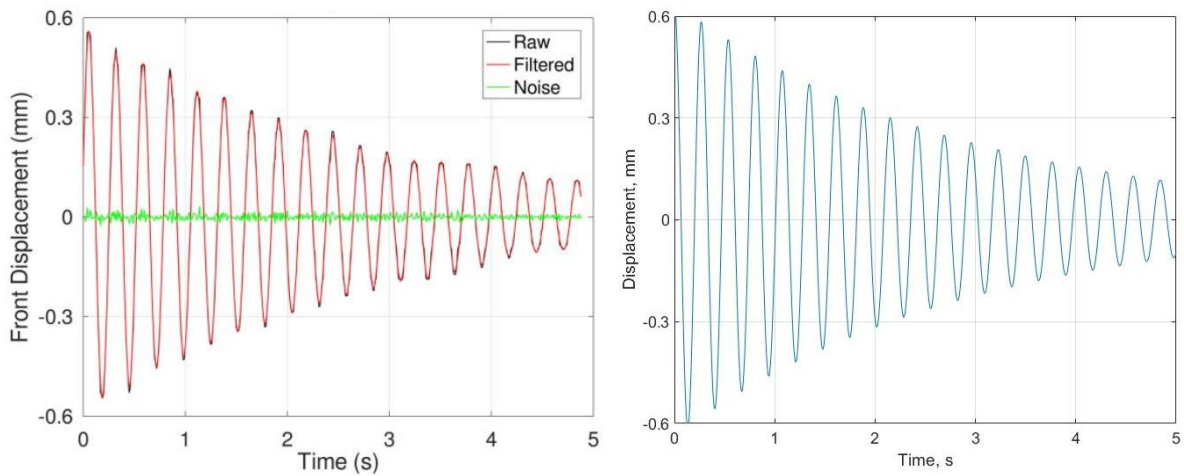


Figure 215. Free vibration in water, damping ratio 0.015

9.1.2.4 Dynamic response

When we have determined the natural frequencies and forms, force response:

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

This equation represents the equation of motion in normal coordinates, the integral on the right is external loading, which corresponds to the $i - th$ mode of vibration.

The normal mode decomposition method is very convenient to use together with Duhamel integral.

$$\eta_i(t) = \frac{1}{\omega_i} \int_0^l Y_i \int_0^t q(x, \tau) \sin \omega_i(t - \tau) d\tau dx$$

Thus the final dynamic displacement can be obtained:

$$y(x, t) = \sum_{i=1}^{\infty} \frac{Y_i}{\omega_i} \int_0^l Y_i \int_0^t q(x, \tau) \sin \omega_i(t - \tau) d\tau dx$$

If the loading $q(x, t)$ is a concentrated force $q_1(t)$ applied in the location x_1 , then we do not have to integrate over the entire rod:

$$y(x, t) = \sum_{i=1}^{\infty} \frac{Y_i Y_{i1}}{\omega_i} \int_0^t q_1(\tau) \sin \omega_i(t - \tau) d\tau$$

Here the Y_{i1} — is a normal mode value at the location x_1 .

The Duhamel integral can be easily used in the case of an external force of arbitrary shape, then the integration step must be divided into small intervals (Figure 216), and the dynamic response for each of them must be sequentially searched. If the intervals are chosen sufficiently small, then the force can be divided with sufficient accuracy into a constant q_{const} and a linearly variable one q_{linear} :

$$y(x, t) = \sum_{i=1}^{\infty} Y_i \left(\phi_{0i} \cos \omega_i t + \frac{\psi_{0i}}{\omega_i} \sin \omega_i t + \frac{1}{\omega_i^2} (1 - \cos \omega_i t) \int_0^l Y_i q_{const} dx + \frac{1}{\omega_i^3} (\omega_i t - \sin \omega_i t) \int_0^l Y_i q_{linear} dx \right)$$

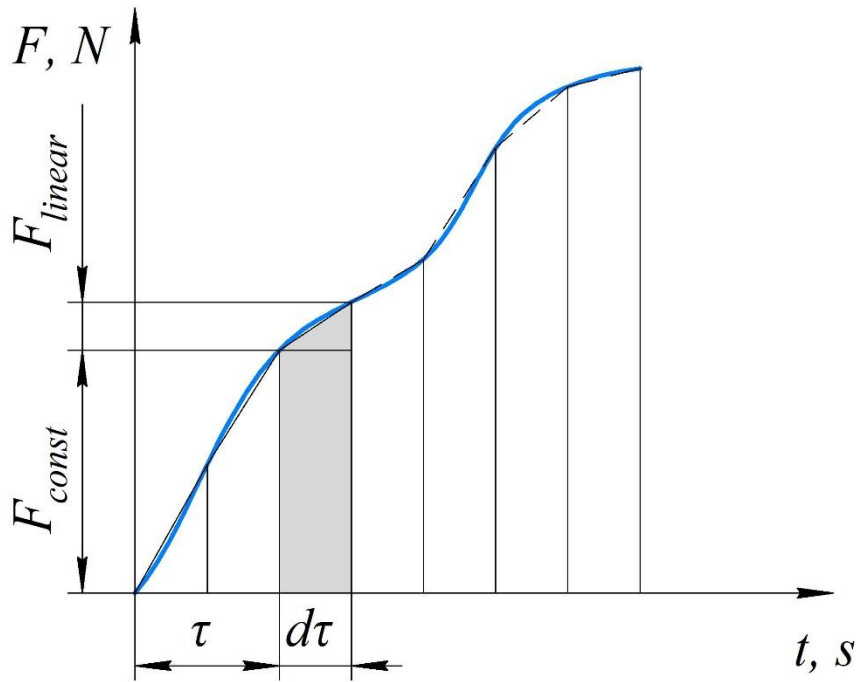


Figure 216. Arbitrary force loading

The Duhamel integral can be modified to take damping by eigenmodes into account. This approach is widely used in seismic analysis, where the input data is given in the form of accelerograms and therefore consideration of damping is natural.

In case of damping the displacements and velocities, for the each timestep can be determined as:

$$\begin{aligned} \eta_i(t) = e^{-nt} & \left(\phi_{0i} \cos \omega_{1i} t + \frac{\phi_{0i} n + \psi_{0i}}{\omega_{1i}} \sin \omega_{1i} t \right) \\ & + \frac{Y_i q_{const}}{\omega_{1i} (n^2 + \omega_{1i}^2)} [\omega_{1i} - e^{-nt} (\omega_{1i} \cos \omega_{1i} t + n \sin \omega_{1i} t)] \\ & + \frac{Y_i q_{linear}}{\omega_{1i} (n^2 + \omega_{1i}^2)^2} [n^2 \omega_{1i} + \omega_{1i}^3 t - 2n \omega_{1i} \\ & - e^{-nt} (2n \omega_{1i} \cos \omega_{1i} t + (n^2 - \omega_{1i}^2) \sin \omega_{1i} t)] \end{aligned}$$

Using this approach, it is possible to approximately model the effect of friction in dynamic analysis.

9.1.3 Coupling details

The mechanical model of beam vibrations is implemented as a user library in FORTRAN77/90. The actual calculation of displacements is implemented using Junction Box, and the return of displacements to CFX is performed using User CEL (data is interpolated from the discrete values of the beam to the nodes of the rod surface mesh). Each of the directions (X and Y) is calculated separately from the mechanical point of view. The library contains two solvers, one of which is based on the solution of the Euler beam by a two-parameter FDM, in which both the velocity and acceleration of the beam are distinguished, which allows obtaining a weakly

dissipative solution with optimal discretization parameters. The second solver is based on the normal mode method described in section 9.2.2. FDM can be used to organize “truly” coupled oscillations, which in the limiting case allows frequencies to be reconstructed without pre-set eigenvalues and without accounting for the frequency shifts due to the influence of the liquid mass. Usually, FDM requires stricter discretization parameters.

The library is developed in such a way that it can be used in parallel mode, where the direct calculation of the Duhamel integral is performed on the master core. All preliminary preparation (calculation of the corresponding coefficients, forms) is carried out at the moment of parallelization. Also at this moment, zero distributions of displacements are formed for each of the coordinates. The discretization parameters, as well as the required number of calculation modes and the damping coefficient, are available for editing by the user using the User Input Parameters option. Specific forces q_x and q_y are calculated as averages over the entire surface (normal and shear stresses are accounted), which is a fairly rough approximation.

Depending on the point of calling the calculation procedure (after each time step or after each spatial iteration), it is possible to organize explicit or implicit coupling. The general scheme of the calculations is shown in Figure 217.

When parallelizing, it is recommended to select a special division of the computational domain in such a way that the rod surface is entirely within one kernel. Numerous tests have shown that the results of q_x and q_y calculations on the same mesh are sensitive to the parallelization parameters.

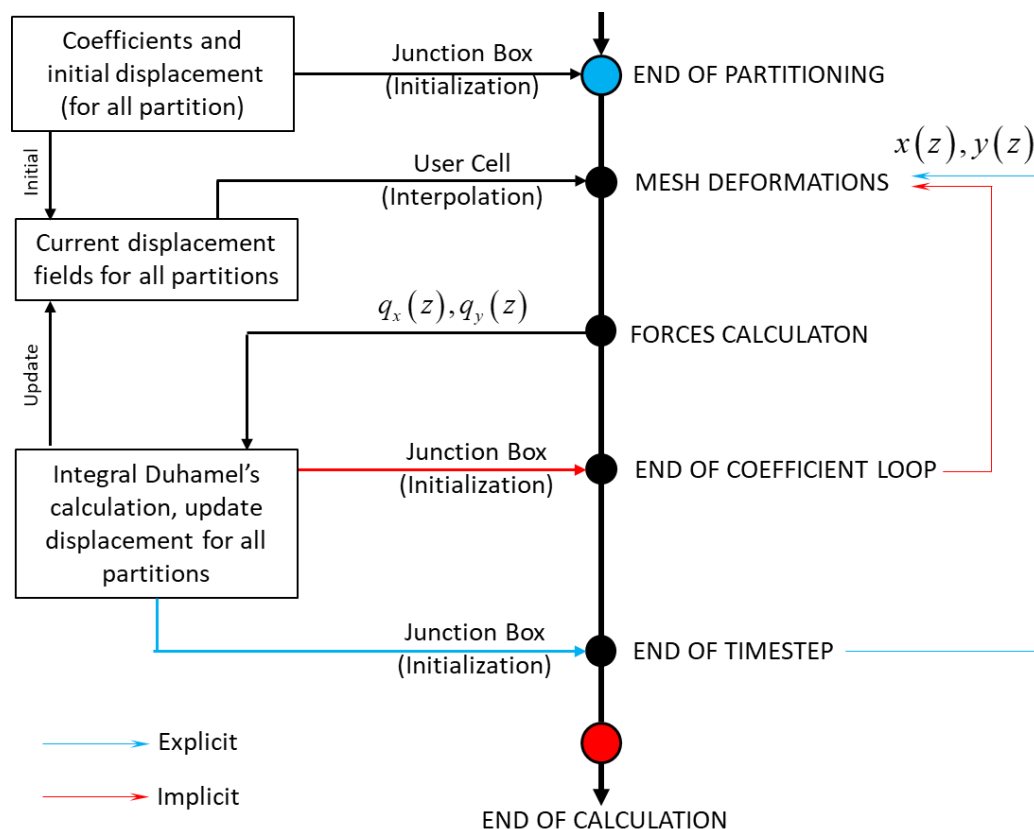


Figure 217. General scheme of the coupling

9.2 Results of the numerical FSI simulations of the cantilevered beam experiment at $Re=21200$

The numerical simulations of the cantilevered beam is conducted using the CFD model described in section 9.1.2. The computational analysis is performed considering FSI, for that a custom FORTRAN library for mechanical modelling of rod displacements is implemented in ANSYS CFX, a more detailed description of which is given in section 9.1.1.

As a calculation case, the option with upward flow and $Re=21200$ in annular cross section is considered. The simulation time is 30s with the adaptive time step size of 0.01-0.025s. Preliminary, a simulation of 10 s with the rod fixed was performed.

9.2.1 Analysis of the flow

Figure 218 and Figure 219 represent the time-averaged axial and transverse velocity fields (respectively) in the area near the rod tip compared with the experimental data. Quite good agreement in provided velocity fields is observed. Figure 220 – Figure 224 demonstrate several radial profiles of axial and transverse velocity components compared with the experimental data. The shape of analyzed profiles recovered quite well (including the amplitude values).

Additionally, Figure 225 provides the axial velocity profiles (for $z=0.0$ mm to 100mm from the rod tip) at different radial locations from $r=0.01$ mm to $r=5.0$ mm (where $r=0.0$ mm - is the rod tip position).

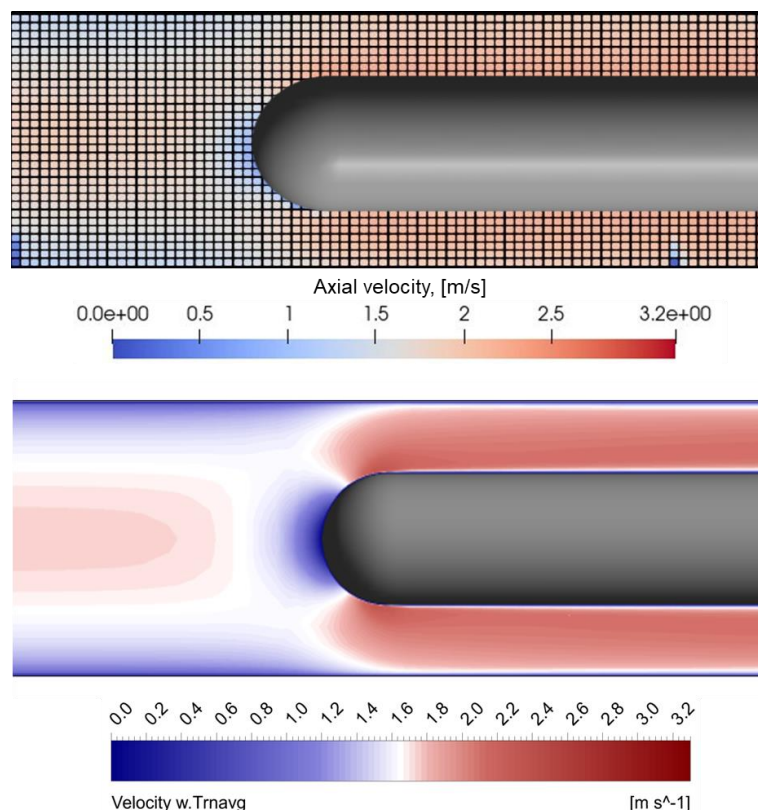


Figure 218. Cut view of the time-averaged axial velocity field (bottom) compared with averaged experimental results for $Re=21200$ (top)

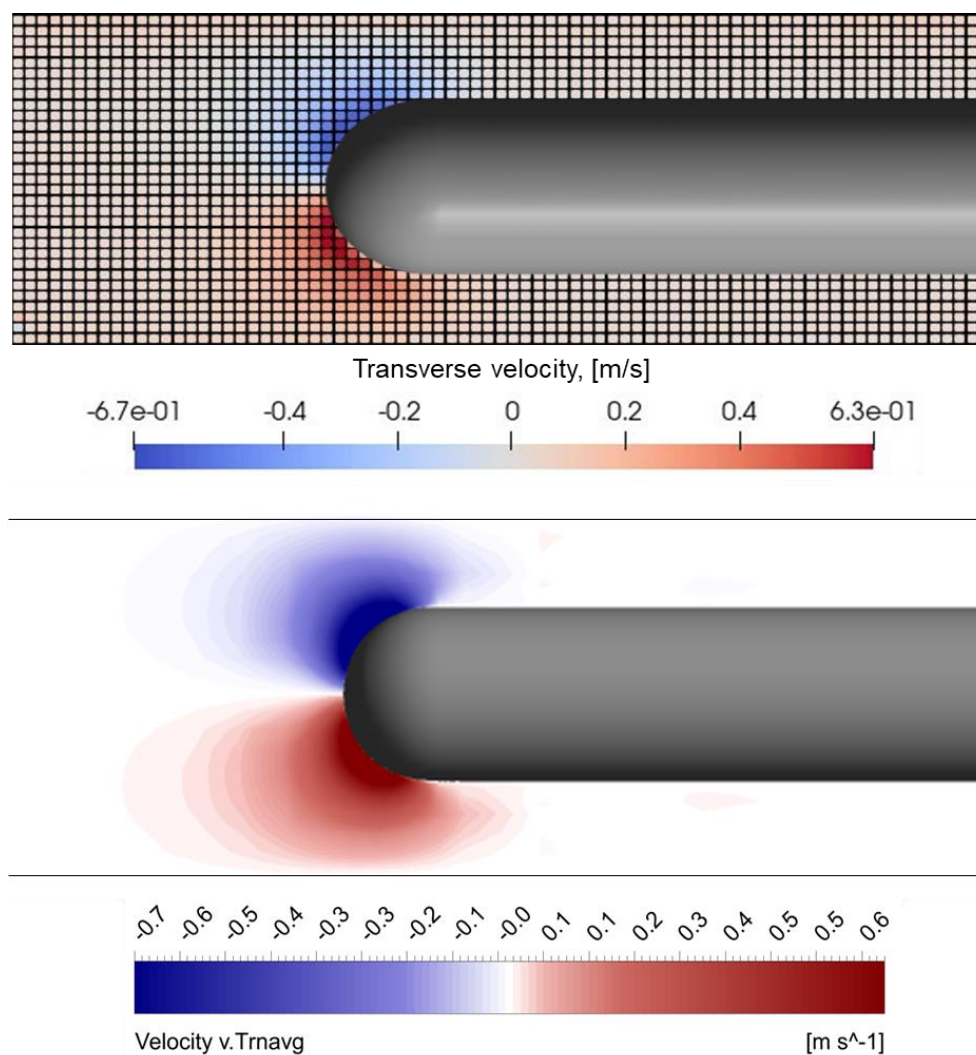


Figure 219. Cut view of the time-averaged transverse velocity field (bottom) compared with averaged experimental results for $Re=21200$ (top)

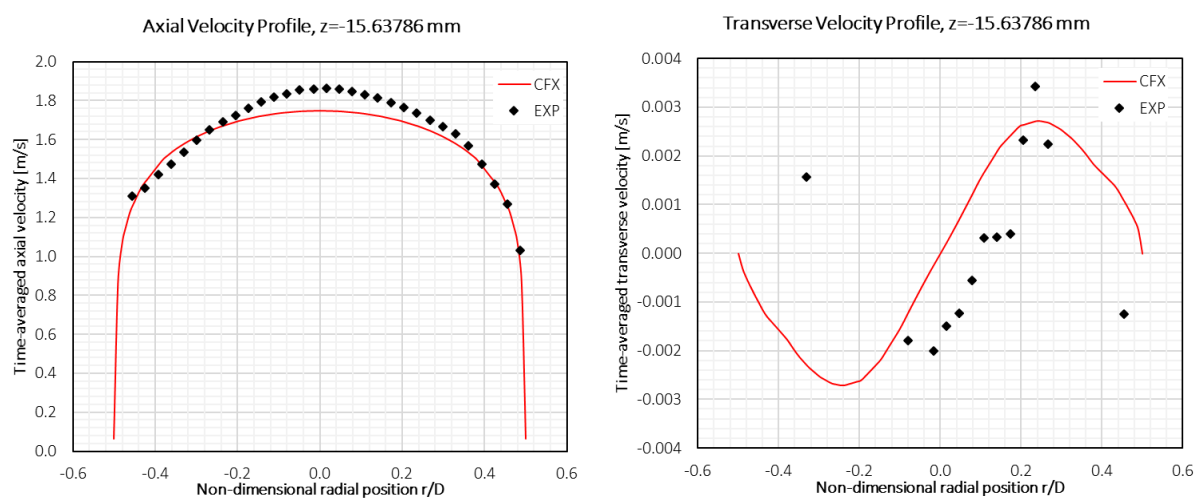


Figure 220. Time-averaged axial and transverse velocity profiles 15.63786 mm upstream of the tip compared with experimental results

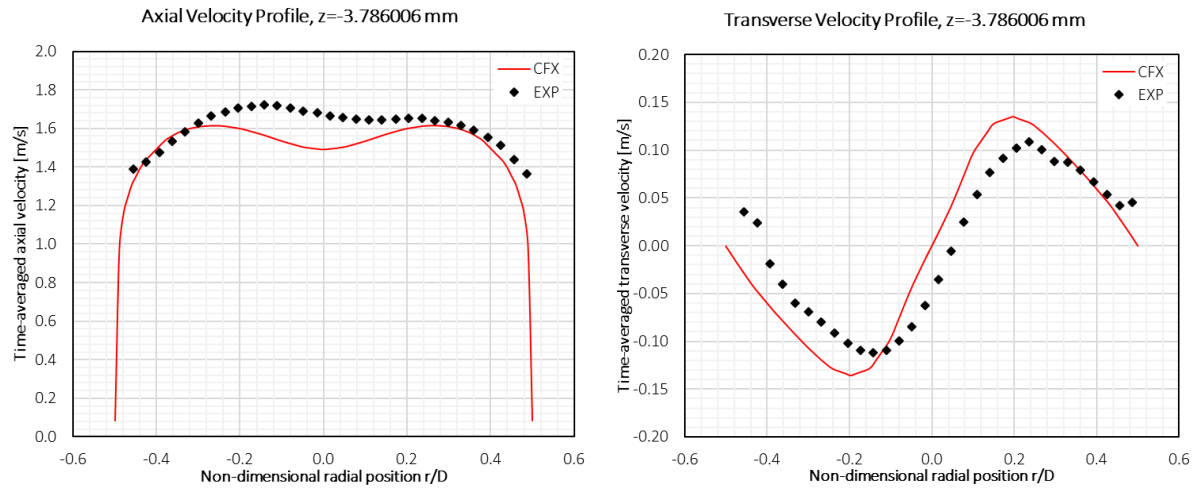


Figure 221. Time-averaged axial and transverse velocity profiles 3.786006 mm upstream of the tip compared with experimental results

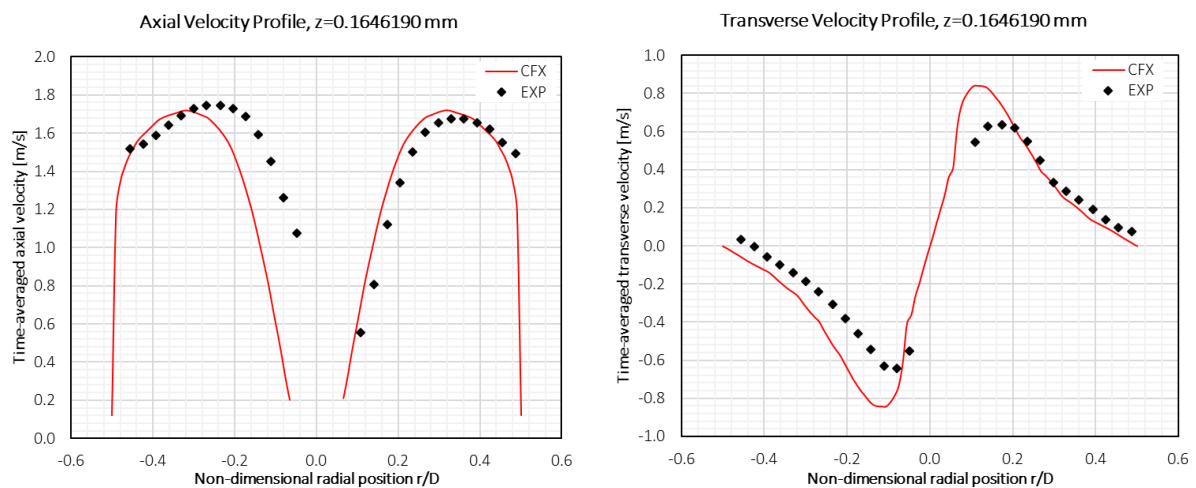


Figure 222. Time-averaged axial and transverse velocity profiles 0.1646190 mm downstream of the tip compared with experimental results

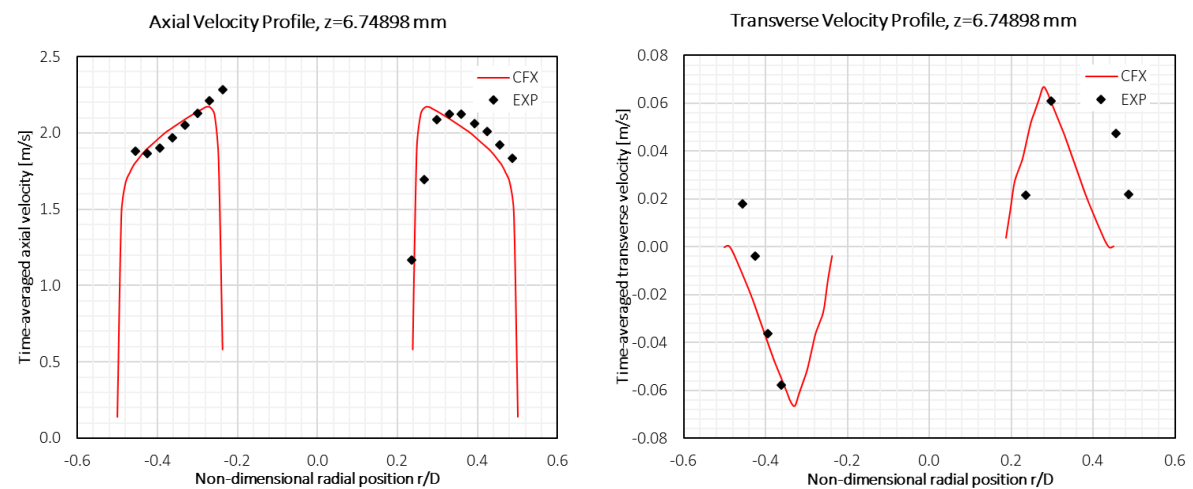


Figure 223. Time-averaged axial and transverse velocity profiles 6.74898 mm downstream of the tip compared with experimental results

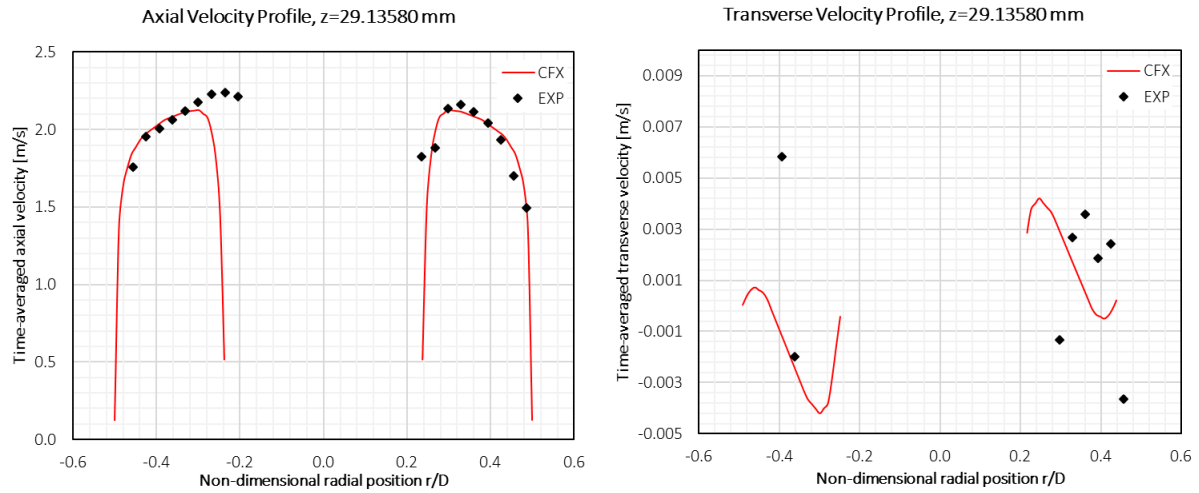


Figure 224. Time-averaged axial and transverse velocity profiles 29.13580 mm downstream of the tip compared with experimental results

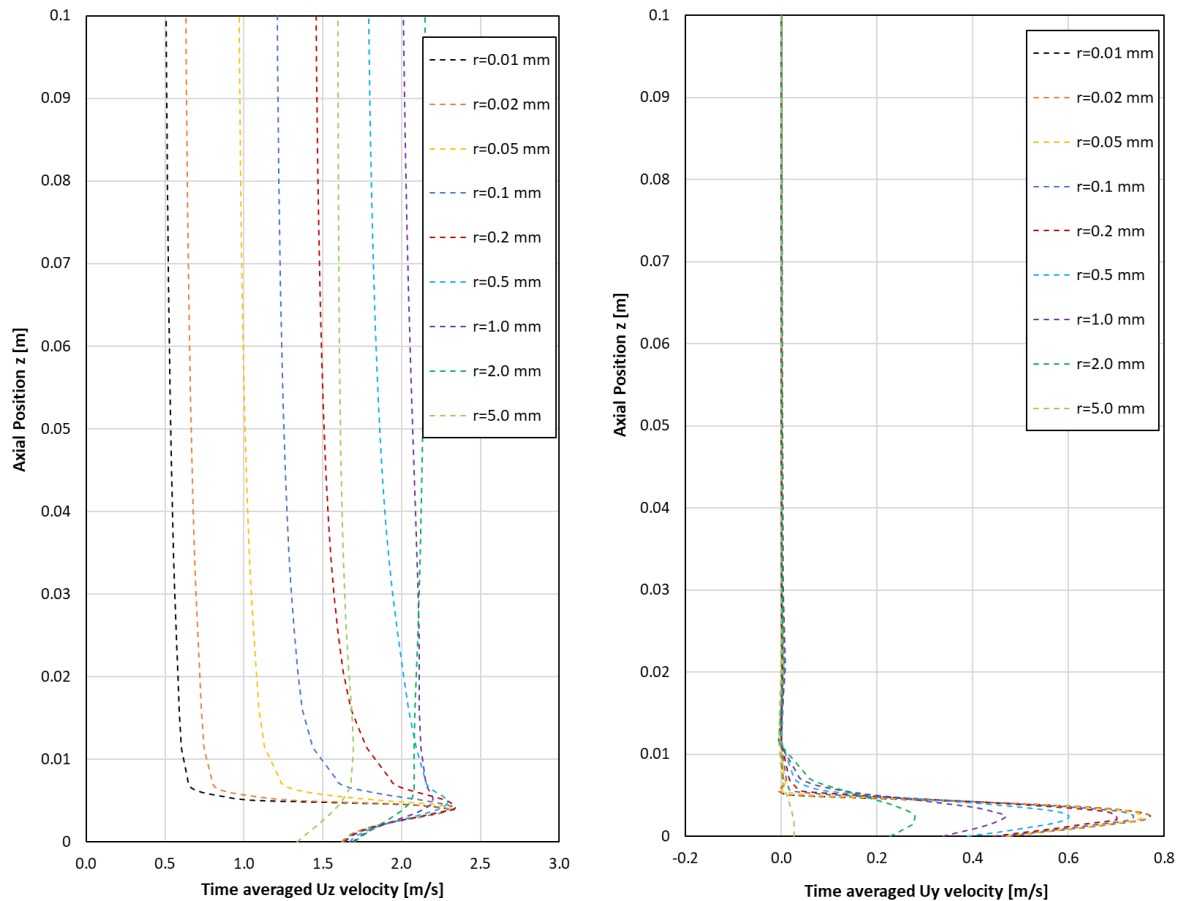


Figure 225. Axial profiles – Time-averaged axial Uz velocity (left) and time-averaged transverse Uy velocity (right)

9.2.2 FIV and rod displacement analysis

During the simulation the rod motion was monitored. Figure 226 and Figure 227 represent the rod tip displacements obtained in the front and side directions, compared with the experimental data. The maximum amplitude value of tip displacement obtained in CFD is 0.055mm. In both directions, the amplitude of vibrations remains lower than in the experiment. The RMS. values obtained in CFD – $RMS(U_{x,CFD}) = 0.016\text{ mm}$ and $RMS(U_{y,CFD}) = 0.018\text{ mm}$, which are lower than the appropriate values in the experiment: $RMS(U_{x,exp}) = 0.067\text{ mm}$ and $RMS(U_{y,exp}) = 0.048\text{ mm}$.

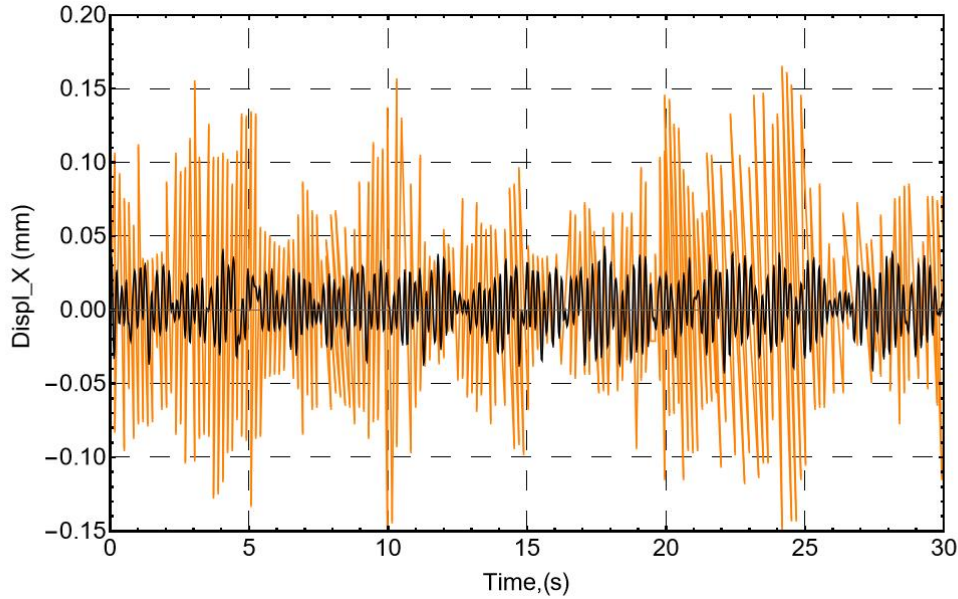


Figure 226. Comparison of the experimental and CFD displacements of the rod tip in the transverse e_x direction

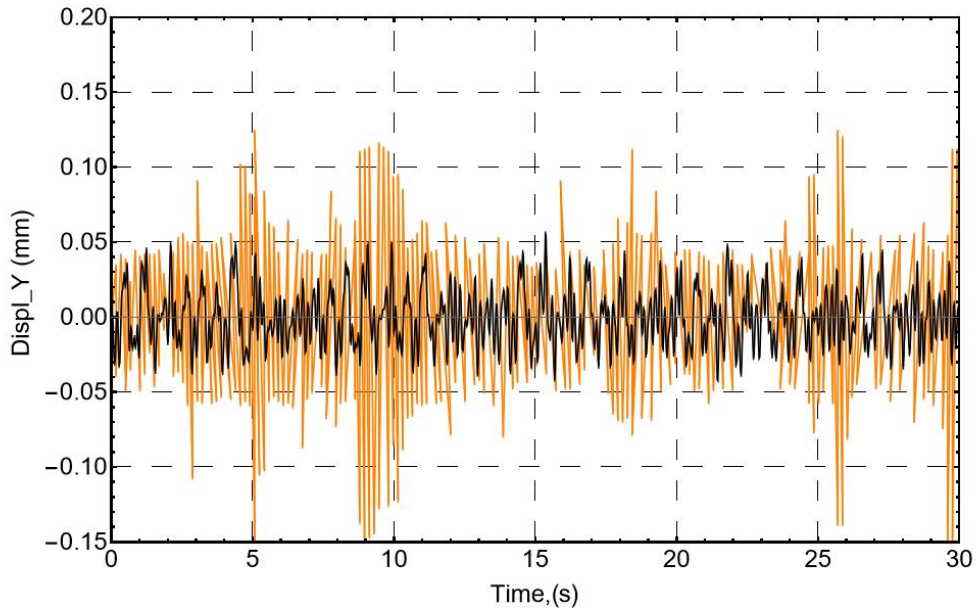


Figure 227. Comparison of the experimental and CFD displacements of the rod tip in the transverse e_y direction

The Power Spectral Density (PSD) of the tip displacements in X and Y directions are shown in Figure 228. The obtained in simulations first peaks frequencies are – $f_x=4.9$ Hz and $f_y=1.8$ Hz (their mean value – 3.35 Hz), while experimental and beam natural frequency first peak values are equal to 3.72 Hz.

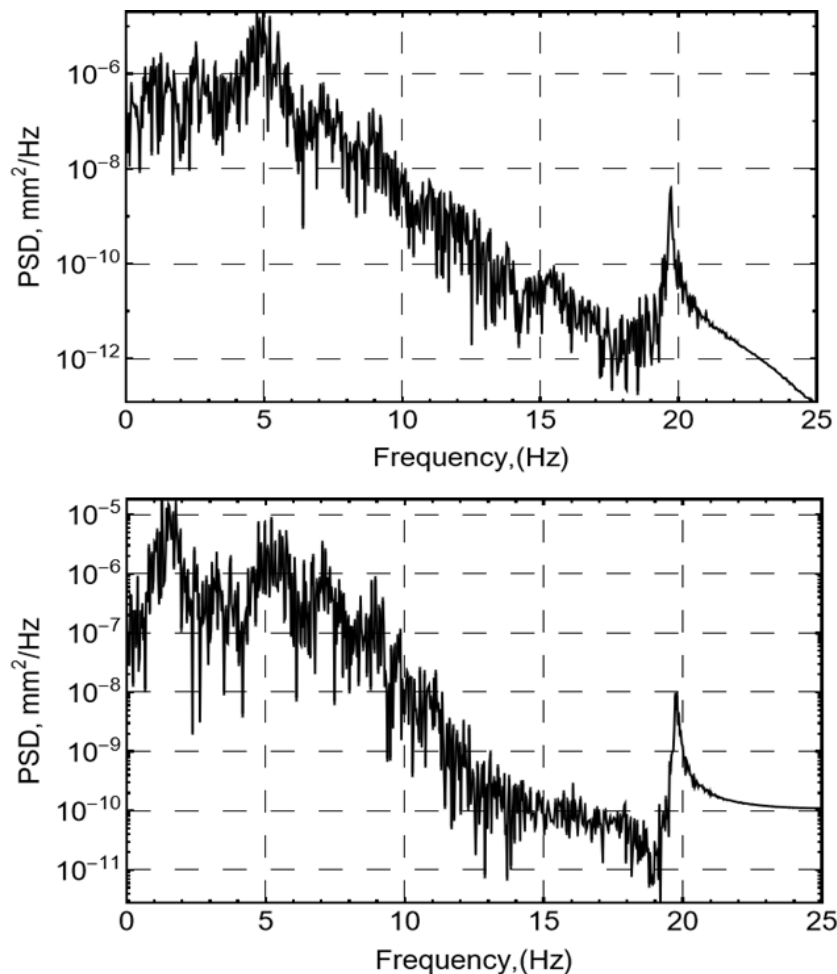


Figure 228. PSD of the Cantilever Rod Tip displacements in X (top) and Y (bottom) directions

9.3 Conclusions

To perform a benchmark with longitudinal flow around the Cantilever rod by a single-phase coolant flow, medium-resolution CFD simulation with the implementation of the Beam Equation Solver user library is performed.

Calculation case with the “free-clamped” flow scheme with $Re=21200$ in the annular cross section is considered. 30 seconds of transient with FIV is modelled with preliminary simulation of 10s with fixed rod. Obtained time-averaged velocity fields near the rod tip as well as radial velocity profiles for axial and transverse directions demonstrated quite good agreement with experimental data. However, the obtained dynamics of the rod tip displacements in both the front and side directions have significantly lower amplitudes and, accordingly, the RMS values. Also, in the obtained results, there is an asymmetry of oscillations along the X and Y coordinates - oscillations in the Y direction have a greater amplitude than in the X direction

throughout the entire simulation. Power Spectral Density of the tip displacements in both X and Y directions also have different first peak frequencies - $f_x=4.9$ Hz and $f_y=1.8$ Hz, while experimental value is 3.72Hz.

The main difficulty in obtaining results using simplified URANS formulations is primarily related to the ability of the model to qualitatively and quantitatively process the boundary layer. The physical difficulty is that, starting from the tip, vortex structures are generated in the boundary layer zone. Moreover, these vortex structures have not only a changing frequency, but also a phase. In general, one cannot expect a correct restoration of the force effect using an isotropic turbulence model, which is usually based on the isolation of a natural pulsation field. Although problems with a moving boundary usually require the separation of pulsations caused not by the mean flow, but by a compliant solid wall.

In general, the obtained results can be applied in engineering practice with some reservations related to safety factors, as well as the accuracy of the solid body model formulation.

10. Overall Conclusions

Deliverable D2.2 successfully concludes the comprehensive numerical benchmarking campaign for the single-phase cantilever rod, definitively marking the achievement of Task 2.1 within the GO-VIKING project. This report not only evaluated the predictive capacity of the Fluid-Structure Interaction (FSI) methodologies but also stands as a testament to the effective collaborative efforts of the contributing partners: PSU, EDF, NRG PALLAS, EDF-Energy, and IPP.

The core objective—comparing the solutions obtained from different computational methods against the experimental data from D2.1—was achieved with the following takeaways from all the simulations performed by the participants including the benchmark case:

- **Experimental Data:** A large quantity of first and second order flow field statistics as well as vibration data are available for validation. While the experimental data provided valuable insight, they cannot be used directly for quantitative validation due to slight deviations in the installation of the long and slender rod – the rod appears to be misaligned relative to the confining pipe axis. NRG PALLAS investigated the effect of the rod offset, assuming the maximum offset of 0.3 mm reported by the experimentalists. At this offset, the effect on the flow field and vibrational characteristic of the rod was found to be negligible. This would suggest the actual misalignment to likely be larger. Further investigation is required to establish the actual misalignment.
- **High Resolution Data:**
 - Rich DNS data is made available by PSU through CFD investigations of flow around the rod with hemispherical end and blunt end. The Q-criterion and TKE budget was carefully examined. The evolution of statistics is remarkably different near the rod tip and for a considerable distance along the rod wall, for each case, before returning to expected annular flow statistic behaviors. These statistics provide high fidelity DNS data for benchmarking and validating future, lower costing, turbulence modeling tools.
 - EDF performed simulations using WR-LES on the case with the rod having a hemispherical tip, investigating the effect of the rod vibrations by simulating the fixed rod scenario and the FSI scenario. The costs were made manageable by utilizing the beam model for the rod. FSI and fixed rod simulations yielded very close velocity and Reynolds stress profiles, which means that the rod vibration does not have a strong impact on the flow. Results also showed good predictions of vibration frequencies but underpredictions in the vibration amplitude compared to the experiment.
- **Medium Resolution Data:**
 - EDF-Energy performed FSI simulations of the blunt end and rounded end configurations using the SSG RSM model and a wall modelled mesh. The key vibrations results showed that the approach worked well in predicting vibrations for the blunt end configuration, however with differences in the prediction of the amplitude (underpredicting for $Re=16k$ by a factor of ~ 2 , overpredicting for $Re=46k$



by a factor of ~ 3). On the other hand, it completely failed to predict any significant vibration for the rounded end configuration at $Re=21.2k$.

- NRG PALLAS performed similar FSI simulations on both configurations using the K-Epsilon (Cubic) model and a wall resolved mesh. The findings of the work showed good predictions of the flow field and vibration frequencies. For the vibration amplitude, a very good match was observed for the blunt end at $Re=23.33k$ with a slight underprediction by a factor of ~ 1.05 while it was more severe for the curved end at $Re=21.2k$, $40k$ by an order of magnitude.
- IPP also performed FSI simulations solely for the rounded end configuration at $Re=21.2k$ using the K-Omega SST model and a wall resolved mesh, and modelled the rod with a beam model. The flow field data showed good agreement with experimental data. However, the obtained dynamics of the rod tip displacements in both the front and side directions were peculiar. The vibration amplitude was found to be of the same order as the prediction of EDF who used LES which is quite good. On the other hand, the frequency of vibration was found to be very dissimilar in the two directions of measurement – more than a factor of 2 between them.

Finally, within the benchmark study (curved end rod, $Re=21.2k$), a stronger focus was made on the modelling choices, simulation strategies and computational costs along with the comparison of obtained flow field and vibration results. A few key takeaways of the benchmark are mentioned below. The reader is referred to section 4.4 for a more detailed summary of the outcomes of the benchmark.

- The choice of turbulence model has the most impact on the accuracy of the solution. The impact of this choice overshadowed any potential gain in accuracy of using CSM over beam model, and respecting the 2 outlet configuration of the test section over a simpler single outlet configuration.
 - As expected, the high resolution approaches offer the highest accuracy in results in the ranked order of DNS followed by LES.
 - However, URANS is also shown to be able to make good predictions for first order flow field statistics and decent FSI predictions. The latter appears to be connected to having a wall resolved mesh as results showed a combination of eddy viscosity models with a wall resolved mesh to predict significant vibrations, as opposed to the better SSG RSM turbulence model in combination with a wall modelled mesh.
- The ranked accuracy has an inversed ranking when it comes to computational cost. This is precisely the reason that URANS-based models remain the working horse of the industry today. The gap in accuracy from URANS can be envisioned to be closed with better scale resolving approaches as was established in the current study. Hybrid models which use RANS near the wall may not be suitable for TIV but could be investigated. Another approach could be using synthetic turbulence such as the Anisotropic Fluctuation Model which is investigated in another WP of the project.

11. Bibliography

- Archambeau, F., Méchitoua, N. & Sakiz, M., 2004. Code_Saturne: a Finite Volume Code for the Computation of Turbulent Incompressible Flows. *International Journal on Finite Volumes*, Volume 1.
- Bakosi, J. et al., 2013. Large-eddy simulations of turbulent flow for grid-to-rod fretting in nuclear reactors. *Nuclear Engineering and Design*, Volume 262, pp. 544-561.
- Benhamadouche, S., Moussou, P. & Le Maitre, C., 2009. *CFD Estimation of the Flow-Induced Vibrations of a Fuel Rod Downstream a Mixing Grid*. Prague, Czech Republic, s.n., pp. 489-497.
- Bistany, S., 2024. *Direct Numerical Simulation (DNS) of Flow Past a Cantilevered Rod - Q Criterion*, s.l.: s.n.
- Blevins, R., 2001. *Flow-Induced Vibration*, Second ed. Malabar, USA: Kreiger Publishing Company.
- CEA, C.-S., 2004. *CATHARE 2 V2.5 : Fluid and material constitutive relationships*, s.l.: s.n.
- Cioncolini, A. et al., 2018. Axial-flow-induced vibration experiments on cantilevered rods for nuclear reactor applications. *Nuclear Engineering and Design*, Volume 238, pp. 102-118.
- Cioncolini, A. et al., 2018. Axial-flow-induced vibration experiments on cantilevered rods for nuclear reactor applications. *Nuclear Engineering and Design*, pp. 102-118.
- Cioncolini, A. et al., 2022. *Deliverable 2.1 of the GO-VIKING project (101060826) Single-phase cantilever rod experiment and benchmark setup*, Manchester, UK: s.n.
- Cioncolini, A. et al., 2023. Experiments on axial-flow-induced vibration of a free-clamped/clamped-free rod for light-water nuclear reactor applications. *Annals of Nuclear Energy*, Issue 190.
- Cioncolini, A. et al., 2023. Experiments on axial-flow-induced vibration of a free-clamped/clamped-free rod for light-water nuclear reactor applications. *Annals of Nuclear Energy*.
- De Santis, D. & Shams, A., 2017. Numerical modeling of flow induced vibration of nuclear fuel rods. *Nuclear Engineering and Design*, Volume 320, pp. 44-56.
- Elmahdi, A. M. et al., 2011. *Flow induced vibration forces on a fuel rod by les cfd analysis*. Toronto, Ontario, Canada, s.n.
- Fischer, P. et al., 2022. NekRS, a GPU-accelerated spectral element Navier–Stokes solver.. *Parallel Computing*, p. 102982.
- Geuzaine, C. & Remacle, J.-F., 2009. Gmsh: a three-dimensional finite element mesh generator with built-in pre- and post-processing facilities.. *International Journal for Numerical Methods in Engineering*, 79(11), pp. 1309-1331.
- Harris, E. D., 2023. *Progress Report of the Single Phase Cantilever Rod Flow-Induced Vibration Case*, London: EDF Energy R&D.



- Hussain, M., Zwijsen, K. & van Zuijlen, A., 2025. Vortex-Induced Vibrations of in-line cantilevered cylinders. *Computers and Fluids*, Volume 297, p. 106647.
- Khoury, G. et al., 2013. Direct Numerical Simulation of Turbulent Pipe Flow at Moderately High Reynolds Numbers.. *Flow Turbulence Combust.*, pp. 475-495.
- Lavoie, P. et al., 2007. Spatial resolution of PIV for the measurement of turbulence. *Experiments in Fluids*, Volume 43, pp. 39-51.
- Li, H. et al., 2024. ip shape effects on the axial-flow-induced vibration of a cantilever rod. *Journal of Fluids and Structures*, Volume 127, p. 104132.
- Liu, Z. G., Liu, Y. & Lu, J., 2013. Numerical simulation of the fluid–structure interaction for two simple fuel assemblies. *Nuclear Engineering and Design*, Volume 258, pp. 1-12.
- Norddine, T. & Benhamadouche, S., 2023. Wall-Resolved LES and URANS Simulations of an Axial Flow on a Cantilevered Rod at a Moderate Reynolds Number. *Conference: 20th International Topical Meeting on Nuclear Reactor Thermal Hydraulics (NURETH-20)*.
- Rogers, R., Taylor, C. & Pettigrew, M., 1984. *Fluid effects on multi-span heat exchanger*, San Antonio-Tx, USA: s.n.
- Salachna, J., Cioncolini, A. & Iacovides, H., 2023. Benchmark simulation of the flow-induced vibrations for nuclear applications. *Annals of Nuclear Energy*, Volume 180.
- Sarno, L. et al., 2019. Challenges and improvements in applying a particle image velocimetry (PIV) approach to granular flows. *Journal of Physics: Conference Series*, Volume 1249, p. 012011.
- Speziale, C., Sarkar, S. & Gatski, T., 1991. Modeling the Pressure-Strain Correlation of Turbulence: an Invariant Dynamical Systems Approach. *Journal of Fluid Mechanics*, Volume 227, pp. 245-272.
- Toonder, J. & Nieuwstadt, F., 1997. Reynolds number effects in a turbulent pipe flow for low to moderate Re.. *Physics of Fluids.*, pp. 3398-3409.

