



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



GOKSTAD model preparation and simulation results

Authors : Kevin ZWIJSEN (NRG), Kevin Zwijsen (NRG PALLAS), Trevor Franklin (VCU), Lane Carasik (VCU), Sofiane Benhamadouche (EDF)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	GOKSTAD model preparation and simulation results
Author(s)	Kevin ZWIJSEN, Kevin Zwijsen (NRG PALLAS), Trevor Franklin (VCU), Lane Carasik (VCU), Sofiane Benhamadouche (EDF)
Number of pages	115
Document type	Deliverable
Work Package	WP3
Document number	D3.4
Issued by	NRG
Date of completion	2026-05-18 17:46:23
Dissemination level	Public

Summary

The GO-VIKING deliverable D3.4 is concerned with providing the results of the numerical simulations for the GOKSTAD benchmark of Work Package 3 (WP3). This involves making predictions of the Flow Induced Vibrations (FIV) of a tube bundle of circular rods in a square lattice shape in a cross-flow single phase. The experimental facility is that of the von Karman Institute in Belgium, which has been described in detail along with obtained experimental data in the GO-VIKING deliverable D3.3. The area of interest for the numerical simulations is the test section which comprises of a single or two centrally located tubes inside the tube bundle Three contributing partners: VCU, EDF and NRG PALLAS, performed numerical simulations of the tube bundle subjected to various inlet velocities. The modelling efforts and obtained results by each institution are documented in dedicated chapters. One of these simulation cases, with an inlet velocity of 0.125 m/s, 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 a considerable impact on the accuracy and involved computational cost of the simulations; high-resolution methods are generally more accurate for flow field predictions, but also require considerably more computational costs. Medium-resolution approaches could be considered instead, at slightly reduced accuracy and significantly reduced computational costs. Looking at displacement statistics, all used approaches give similar results, as long as the relevant turbulence scales are resolved to a reasonable level.

Approval

Date	By
2026-05-18 17:51:17	Kevin ZWIJSEN (NRG)
2026-05-18 17:54:50	Papukchiev ANGEL (GRS)



D3.4 GOKSTAD model preparation and simulation results

Version N°2
May / 2026

Kevin Zwijsen¹, Trevor Franklin², Lane Carasik², Sofiane Benhamadouche³

¹NRG PALLAS, Westerduinweg 3, 1755 LE Petten, The Netherlands

²Virginia Commonwealth University (VCU), Richmond, Virginia, United States

³Electricité De France (EDF) R&D, Fluid Mechanic division, 6 Quai Watier 78400 Chatou, France



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 Coordinator	Angel Papukchiev, angel.papukchiev@grs.de , GRS Kevin Zwijsen, kevin.zwijsen@nrgpallas.com, NRG PALLAS
Project starting date (duration)	1st June 2022 – 31st May 2026 (48 Months)
Related Work Package	WP3
Related Task(s)	T3.2 Analysis of the GOKSTAD cross flow tube bundle
Lead Organisation	NRG PALLAS
Contributing Partner(s)	NRG PALLAS, VCU, EDF
Due Date	30-11-2025
Submission Date	18-05-2026
Dissemination level	PU

History

Date	Version	Submitted by	Reviewed by	Comments
11-05-2026	1.0	K. Zwijsen	A. Papukchiev, S. Benhamadouche	
18-05-2026	2.0	K. Zwijsen		

Table of contents

1.	Introduction	17
2.	GOKSTAD benchmark.....	18
2.1	Experiment description	18
2.2	Available measurement data	20
3.	Benchmark comparison	24
3.1	Introduction & considered benchmark case	24
3.2	Simulation approaches.....	24
3.2.1	Geometry & Mesh	24
3.2.1.1	Solid Domain	24
3.2.1.2	Fluid Domain	25
3.2.2	Material properties	25
3.2.2.1	Solid properties	25
3.2.2.2	Fluid properties	26
3.2.3	Boundary conditions	26
3.2.3.1	Solid domain.....	26
3.2.3.2	Fluid domain.....	27
3.2.4	Numerical settings.....	27
3.2.5	Computational resources	28
3.2.6	Simulation Strategy	28
3.2.6.1	VCU	28
3.2.6.2	EDF.....	29
3.2.6.3	NRG PALLAS.....	29
3.3	Results comparison	29
3.3.1	Flow data comparison	29
3.3.2	Vibration data comparison.....	37
3.4	Benchmark conclusion	39
4.	Simulations performed by VCU.....	41
4.1	Method description.....	41
4.1.1	Modeling approach	41
4.1.2	Numerical simulation approach	47
4.2	Flow-only simulation results for 0.125 m/s bulk velocity	48
4.3	FSI simulation results for 0.125 m/s bulk velocity	59
4.4	Conclusions.....	60
5.	Simulations performed by EDF	61

5.1	Method description.....	61
5.2	Rigid tube bundle	62
5.2.1	Fluid flow prediction at a bulk velocity of 0.5 m/s.....	62
5.2.2	Fluid flow prediction at a bulk velocity of 1 m/s.....	67
5.3	Simulations with one moving cylinder in a tube bundle.....	70
5.4	Conclusions.....	70
6.	Simulations performed by NRG PALLAS	72
6.1	Introduction.....	72
6.2	CAD & Mesh	72
6.3	Numerical Model.....	74
6.4	Flow-only simulation results	75
6.4.1	Inlet velocity of 1.0 m/s – inter-comparison of models.....	76
6.4.2	Inlet velocity of 0.125 m/s – comparison with experiment	89
6.4.3	Inlet velocity of 0.5 m/s – comparison with experiment	99
6.5	FSI simulation results	107
6.5.1	Inlet velocity of 0.125 m/s.....	107
6.5.2	Inlet velocity of 0.5 m/s.....	109
7.	Overall conclusions	113
	Bibliography.....	114

List of figures

Figure 1: Side view (left) and isometric 3D representation (right) of the experimental facility.	18
Figure 2: The actual test section (left) and the various PIV measurement viewing windows (right).....	19
Figure 3: Schematic of the spring–cylinder mounting system.	20
Figure 4: Additional PIV measurements performed to get line profiles of several flow quantities.	22
Figure 5: Schematic representation of the three vibrating-rod configurations investigated with the GOKSTAD facility, with, left: single moving tube, (middle) vertical (in-line) configuration, and (right) horizontal (side-by-side) configuration.....	22

Figure 6: Comparison of the time-averaged velocity in the inlet region just upstream the tube bundle.	30
Figure 7: Comparison of the time-averaged normalized turbulent kinetic energy in the inlet region just upstream the tube bundle.	31
Figure 8: Comparison of the time-averaged velocity on a streamwise plane inside the test section between experiment (left), VCU (middle-left), EDF (middle-right) and NRG PALLAS (right), for $u = 0.125$ m/s.	31
Figure 9: Comparison of the time-averaged velocity on a streamwise plane inside the bundle regio between experiment (upper left), VCU (upper right), EDF (lower left) and NRG PALLAS (lower right), for $u = 0.125$ m/s.	33
Figure 10: Comparison of the time-averaged normalized TKE on a streamwise plane inside the test section between experiment (left), VCU (middle-left), EDF (middle-right) and NRG PALLAS (right), for $u = 0.125$ m/s.	33
Figure 11: Comparison of the time-averaged TKE on a streamwise plane inside the bundle regio between experiment (upper left), VCU (upper right), EDF (lower left) and NRG PALLAS (lower right), for $u = 0.125$ m/s.	34
Figure 12: Comparison between experiment and simulations of the velocity inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right).	35
Figure 13: Comparison between experiment and simulations of the TKE inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right).	35
Figure 14: Comparison between experiment and simulations of the velocity in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right).	36
Figure 15: Comparison between experiment and simulations of the TKE in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right).	36
Figure 16: Lift (up) and drag (bottom) forces (left) and displacements (right) as function of time for VCU (blue), EDF (orange) and NRG PALLAS (green).	37
Figure 17: PSD of the lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for VCU (blue), EDF (orange) and NRG PALLAS (green).	38
Figure 18: Schematic of the computational domain used to simulate the GOKSTAD experimental tube bundle in this work. Relevant dimensions and direction of flow are highlighted	41
Figure 19: Schematic of the boundary conditions used in the simulation	42
Figure 20: Spanwise (top) and streamwise (bottom left) slice of GMSH mesh. Zoomed in picture of the mesh around a tube from the GMSH mesh (bottom middle) and mesh from Nek5000 (bottom right).	43

Figure 21: Y plus histogram for the simulation with average, maximum, and minimum values provided.	43
Figure 22: Contours of the LES GLL spacing (top left) and Taylor microscale (bottom left). The plots on the right show line comparisons of the GLL spacing and Taylor microscale at three different line locations depicted in the figure	44
Figure 23: Location of history points in the bundle on the symmetry domain. The streamwise velocity signal are provided for the history points around the middle rod on the last row of tubes. The turbulent spectra is also shown for the velocity signal.	45
Figure 24: Instantaneous LES (top) and Averaged LES (bottom) velocity magnitude contour map.	46
Figure 25: Schematic depicting the different data sampling regions and line locations taken throughout the tube bundle to be compared to against the experiment.	46
Figure 26: Instantaneous streamwise velocity shown as an isotropic view with streamwise, transverse and spanwise planes (left) and normal to the spanwise direction with a spanwise slice (right).....	49
Figure 27: Average velocity contour fields with streamlines for both the experiment and the LES results. Top two colormaps show v1 and the bottom two show v2.	49
Figure 28: Average TKE contour fields for both the experiment and the LES results from V1.	50
Figure 29: Average velocity contour plots at the outlet of the bundle for experiments (top) and LES (bottom).....	51
Figure 30: TKE contour plots at the outlet of the tube bundle for experiments (top) and LES (bottom).	51
Figure 31: Average velocity and normalized TKE for the LES and experimental results at the inlet of the bundle. Line 1, 2, and 3 correspond to $z=0.00$ m, 0.03 m, 0.065 m respectively	53
Figure 32: Average velocity and TKE for Lines 1-7 on the symmetry plane ($z=0$).....	54
Figure 33: Average velocity and TKE for Lines 1-7 on the $z=+2$ cm plane.....	55
Figure 34: Average Velocity and TKE for lines 1-7 on the $+4$ cm plane.	56
Figure 35: Average Velocity and TKE for lines 1-7 on the -4 cm plane.	57
Figure 36: Average velocity and TKE for lines 8-11 (outlet) on the symmetry plane ($z=0$)	58
Figure 37: Drag (left) and Lift (right) force history from LES on the center tube of the bundle.	59

Figure 38: Displacement in the X and Y direction using a one-way coupled spring mass solve from the time series force data in the LES.....	60
Figure 39: Mean stream-wise velocity and turbulent kinetic energy (TKE) at the inlet with the two turbulence models	62
Figure 40: Instantaneous non-dimensional distance to the wall for a bulk velocity of 0.5 m/s – view of all the walls. Left: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.....	63
Figure 41: Instantaneous CFL number at a bulk velocity of 0.5 m/s on the center plane. Left: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.....	63
Figure 42: Velocity magnitude on the experimental plane (x,y) for a bulk velocity of 0.5 m/s – view of the bundle and the outlet. Left: experiment, middle: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.	64
Figure 43: Velocity magnitude on the experimental plane (x,y) for a bulk velocity of 0.5 m/s – view of the bundle. Left: experiment, middle: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.....	64
Figure 44: TKE on the experimental plane (x,y) for a bulk velocity of 0.5 m/s – view of the bundle. Left: experiment, middle: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.....	64
Figure 45: Time evolution of the lift force applied to cylinders (4,3) (the central one) and (7,3) (at the outlet) for a bulk velocity of 0.5 m/s. Left: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.....	65
Figure 46: Velocity magnitude on the center plane for a bulk velocity of 0.5 m/s – comparisons of profiles along the lines P2 (top right), P4 (bottom left) and P6 (bottom right), see top left for profile locations.	66
Figure 47: TKE on the center plane for a bulk velocity of 0.5 m/s – comparisons of profiles along the lines P2 (top right), P4 (bottom left) and P6 (bottom right), see top left for profile locations.....	67
Figure 48: Modeled and resolved TKE in the center plane along the line p6 for a bulk velocity of 0.5 m/s. Left: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.	67
Figure 49: Time evolution of the lift force applied to cylinders (4,3) (the central one) and (7,3) (at the outlet) for a bulk velocity of 1.0 m/s. Left: EB-RSM, right: k- ω SST.....	68
Figure 50: Velocity magnitude on the experimental plane (x,y) for an approximate bulk velocity of 1.0 m/s – view of the bundle and the outlet. Left: experiment, middle: EB-RSM, right: k- ω SST.	68
Figure 51: Velocity magnitude on the experimental plane (x,y) for an approximate bulk velocity of 1.0 m/s – view of the bundle and the outlet. Left: experiment, middle: EB-RSM, right: k- ω SST.	69

Figure 52: Velocity magnitude on the experimental plane (x,y) for an approximate bulk velocity of 1.0 m/s – view of the bundle. Left: experiment, middle: EB-RSM, right: k- ω SST.	69
Figure 53: TKE on the experimental plane (x,y) for an approximate bulk velocity of 1.0 m/s – view of the bundle. Left: experiment, middle: EB-RSM, right: k- ω SST.....	69
Figure 54: Displacement spectra for the single moving cylinder case at a bulk velocity of 0.25 m/s	70
Figure 55: Geometry of GOKSTAD used by NRG PALLAS in its simulations.	73
Figure 56: Mesh used by NRG PALLAS for its simulations with, top: front view of the full domain, middle left: top view of bundle area, middle right: mesh inside bundle around cylinders, and bottom: front view zoomed on the bundle region.	74
Figure 57: Wall y^+ as obtained with the URANS (left) and LES (right) turbulence models.....	76
Figure 58: CFL values on the central plane, zoomed in around the tube bundle, obtained with URANS.	77
Figure 59: Histogram of the CFL values obtained with URANS, split over the various mesh refinement zones.	77
Figure 60: Time-averaged streamwise velocity, capped between -2.0 m/s and 5.0 m/s, for a) URANS, b) RSM, c) Lag-EB, d), IDDES and e) LES.....	78
Figure 61: Time-averaged horizontal velocity, capped between -2.0 m/s and 2.0 m/s, for a) URANS, b) RSM, c) Lag-EB, d), IDDES and e) LES.....	79
Figure 62: Time-averaged spanwise velocity, capped between -0.5 m/s and 0.5 m/s, for a) URANS, b) RSM, c) Lag-EB, d), IDDES and e) LES.....	80
Figure 63: Instantaneous vorticity magnitude, capped at $1.0e+3$ /s, for a) URANS, b) RSM, c) Lag-EB, d), IDDES and e) LES	81
Figure 64: Time-averaged lift force coefficient on the central cylinder of row 1 (upper left), row 2 (upper right), row 3 (lower left) and row 4 (lower right) for URANS (yellow), RSM (grey), Lag-EB (orange), IDDES (blue) and LES (red).....	82
Figure 65: Time-averaged total drag and lift force coefficients, and the resulting contributions from the pressure (normal) and shear (tangential) stresses, on the central cylinder in the tube bundle as predicted by URANS.....	83
Figure 66: Pearson correlation coefficient between the lift forces along the central tube obtained with URANS. Different lines indicate different locations used as reference location.	83
Figure 67: Pearson correlation coefficient between the lift forces along the central tube, for URANS (upper left), RSM (upper right), Lag-EB (middle left), IDDES (middle right) and LES (bottom)	84

Figure 68: Autocorrelation of the lift forces at three different locations along the central tube, for URANS (upper left), RSM (upper right), Lag-EB (middle left), IDDES (middle right) and LES (bottom).....	85
Figure 69: Autocorrelation drop times of the lift forces as function of the location along the central tube, for URANS (upper left), RSM (upper right), Lag-EB (middle left), IDDES (middle right) and LES (bottom).	86
Figure 70: Time-averaged pressure coefficient on the central row of tubes for URANS (upper left), RSM (upper right), Lag-EB (middle left), IDDES (middle right) and LES (bottom)....	87
Figure 71: Time-averaged streamwise velocity, capped between -2.0 m/s and 5.0 m/s, for a) URANS + walls, b) URANS + periodic, c) LES + walls, d) LES + periodic.....	88
Figure 72: Autocorrelation drop times of the lift forces as function of the location along the central tube, for URANS (up), and LES (bottom), with side walls (left) and periodic sides (right).....	89
Figure 73: Time-averaged pressure coefficient on the central row of tubes for URANS (up), and LES (bottom), with side walls (left) and periodic sides (right).	89
Figure 74: Comparison between experiment and simulations of the average velocity as a function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region, for $u = 0.125$ m/s.	90
Figure 75: Comparison between experiment and simulations of the average velocity as a function of the x-coordinate on the near wall $z = 0.065$ m plane in the inlet region, for $u = 0.125$ m/s.	91
Figure 76: Comparison between experiment and simulations of the normalized TKE as a function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region, for $u = 0.125$ m/s.	92
Figure 77: Modelled (blue), resolved (yellow) and total (green) TKE profiles as function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region for IDDES (upper left), URANS (upper right) and LES (bottom), for $u = 0.125$ m/s.	93
Figure 78: Comparison of the time-averaged velocity on a streamwise plane inside the test section between experiment (left) and simulation, for URANS (middle-left), IDDES (middle-right) and LES (right), for $u = 0.125$ m/s.	94
Figure 79: Comparison of the time-averaged normalized TKE on a streamwise plane inside the test section between experiment (left) and simulation, for URANS (middle-left), IDDES (middle-right) and LES (right), for $u = 0.125$ m/s.	94
Figure 80: Comparison between experiment and simulations of the velocity inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right), for $u = 0.125$ m/s.	95

Figure 81: Comparison between experiment and simulations of the TKE inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right), for $u = 0.125$ m/s.	96
Figure 82: Comparison between experiment and simulations of the velocity in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right), for $u = 0.125$ m/s.	97
Figure 83: Comparison between experiment and simulations of the TKE in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right), for $u = 0.125$ m/s.	97
Figure 84: PSD of the lift force coefficient on the most upstream cylinder (4.1), central cylinder (4.4) and most downstream cylinder (4.7) of the central column of cylinders obtained by the three different turbulent models, for $u = 0.125$ m/s.	98
Figure 85: PSD of the drag force coefficient on the most upstream cylinder (4.1), central cylinder (4.4) and most downstream cylinder (4.7) of the central column of cylinders obtained by the three different turbulent models, for $u = 0.125$ m/s.	98
Figure 86: Comparison between experiment and simulations of the average velocity as a function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region, for $u = 0.5$ m/s.	100
Figure 87: Comparison between experiment and simulations of the normalized TKE as a function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region, for $u = 0.5$ m/s.	100
Figure 88: Modelled (blue), resolved (yellow) and total (green) TKE profiles as function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region for IDDES (upper left), URANS (upper right) and LES (bottom), for $u = 0.5$ m/s.	101
Figure 89: Comparison of the time-averaged velocity on a streamwise plane inside the test section between experiment (left) and simulation, for URANS (middle-left), IDDES (middle-right) and LES (right), for $u = 0.5$ m/s.	102
Figure 90: Comparison of the time-averaged normalized TKE on a streamwise plane inside the test section between experiment (left) and simulation, for URANS (middle-left), IDDES (middle-right) and LES (right), for $u = 0.5$ m/s.	102
Figure 91: Comparison between experiment and simulations of the velocity inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right), for $u = 0.5$ m/s.	103
Figure 92: Comparison between experiment and simulations of the TKE inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right), for $u = 0.5$ m/s.	104

Figure 93: Comparison between experiment and simulations of the velocity in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right), for $u = 0.5$ m/s.	105
Figure 94: Comparison between experiment and simulations of the TKE in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right), for $u = 0.5$ m/s.	105
Figure 95: PSD of the lift force coefficient on the most upstream cylinder (4.1), central cylinder (4.4) and most downstream cylinder (4.7) of the central column of cylinders obtained by the three different turbulent models, for $u = 0.5$ m/s.	106
Figure 96: PSD of the drag force coefficient on the most upstream cylinder (4.1), central cylinder (4.4) and most downstream cylinder (4.7) of the central column of cylinders obtained by the three different turbulent models, for $u = 0.5$ m/s.	107
Figure 97: Lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for IDDES (blue), URANS (orange) and LES (green), for $u = 0.125$ m/s.	108
Figure 98: PSD of the lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for IDDES (blue), URANS (orange) and LES (green), for $u = 0.125$ m/s.	109
Figure 99: Lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for IDDES (blue), URANS (orange) and LES (green), for $u = 0.5$ m/s.	110
Figure 100: PSD of the lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for IDDES (blue), URANS (orange) and LES (green), for $u = 0.5$ m/s.	111

List of tables

Table 1: Dynamic parameters of the cylinder–spring system under air conditions	20
Table 2: Summary of PIV Test Matrix	21
Table 3: General Information of the Submissions	24
Table 4: Fluid domain details	25
Table 5: Solid properties	26
Table 6: Fluid properties	26
Table 7: Solid boundary conditions	26
Table 8: Fluid boundary conditions	27

Table 9: Numerical settings	27
Table 10: Computational resources	28
Table 11: Comparison between experiment and simulations of relevant FSI statistics for the GOKSTAD benchmark.....	39
Table 12: Measurements for different regions of the computational tube bundle domain. Dimensions are given using tube diameter, nondimensional, and dimensional units. ...	42
Table 13: Mean and RMS displacement of the center tube in the Lift (x) and Drag (y) direction between LES and the experiment.	59
Table 14: Comparison between experiment and simulations of relevant FSI statistics for $u = 0.125$ m/s	109
Table 15: Comparison between experiment and simulations of relevant FSI statistics for $u = 0.5$ m/s	111

Abbreviations and Acronyms

Acronym	description
ALE	Arbitrary Lagrangian–Eulerian
BDF2	Second-Order Backward Differentiation Formula
CFL	Courant–Friedrichs–Lewy number
DFBI	Dynamic Fluid-Body Interaction
DFC	Drag Force Coefficient
DNS	Direct Numerical Simulations
DOF	Degree-Of-Freedom
(lag)EB	(Lagrangian) Elliptic-Blending
FIV	Flow-Induced Vibrations
FP	Flow Passage
FSI	Fluid-Structure Interaction
GOKSTAD	GO-VIKING experimental SeTup to Assess FIV by cross-flow in tube bunDles)
GO-VIKING	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
IDDES	Improved Delayed Detached Eddy Simulation
LES	Large-Eddy Simulations
LFC	Lift Force Coefficient
PSD	Power Spectral Density
PWR	Pressurized Water Reactor

(U)RANS	(Unsteady) Reynolds-Averaged Navier Stokes
RMS	Root-Mean Square
RSM	Reynolds-Stress Model
TIV	Turbulence-Induced Vibrations
TKE	Turbulent Kinetic Energy
VIV	Vortex-Induced Vibrations
WALE	Wall-Adapting Local Eddy-Viscosity
WP	Work Package

Summary

The GO-VIKING deliverable D3.4 is concerned with providing the results of the numerical simulations for the GOKSTAD benchmark of Work Package 3 (WP3). This involves making predictions of the Flow Induced Vibrations (FIV) of a tube bundle of circular rods in a square lattice shape in a cross-flow single phase. The experimental facility is that of the von Karman Institute in Belgium, which has been described in detail along with obtained experimental data in the GO-VIKING deliverable D3.3. The area of interest for the numerical simulations is the test section which comprises of a single or two centrally located tubes inside the tube bundle

Three contributing partners: VCU, EDF and NRG PALLAS, performed numerical simulations of the tube bundle subjected to various inlet velocities. The modelling efforts and obtained results by each institution are documented in dedicated chapters. One of these simulation cases, with an inlet velocity of 0.125 m/s, 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 a considerable impact on the accuracy and involved computational cost of the simulations; high-resolution methods are generally more accurate for flow field predictions, but also require considerably more computational costs. Medium-resolution approaches could be considered instead, at slightly reduced accuracy and significantly reduced computational costs. Looking at displacement statistics, all used approaches give similar results, as long as the relevant turbulence scales are resolved to a reasonable level.

Keywords

Flow-Induced Vibrations, GOKSTAD, benchmark, tube bundle, cross-flow, numerical simulations.

1. Introduction

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

The report is structured as follows: Chapter 2 provides the GOKSTAD experimental description with all available experimental tests. Chapter 3 presents the selected benchmark case along with a comparison of obtained numerical results against the experiment. Chapters 4-6 present all numerical efforts performed by the institutions VCU, EDF and NRG PALLAS, respectively. The overall summary and conclusions are provided in Chapter 7.

2. GOKSTAD benchmark

The experiment of interest is the GOKSTAD (GO-VIKING experimental SeTup to Assess FIV by cross-flow in tube bundles) facility. GOKSTAD is a new water loop experiment designed and built at the von Karman Institute in Belgium to study flow-induced vibration (FIV) of circular rods in a square lattice shape in a cross-flow single phase. This chapter provides key details of the experiment and available data. Readers interested in further details are referred to GO-VIKING D3.3 report (Can Akkurt, et al., 2026).

2.1 Experiment description

The GOKSTAD facility is a dedicated experimental water loop developed at the von Karman Institute for Fluid Dynamics to investigate flow-induced vibration (FIV) and turbulent cross-flow in tube bundles representative of configurations found in nuclear steam generators. The facility enables detailed experimental characterization of the flow field and fluid–structure interaction phenomena in a controlled laboratory environment using advanced optical measurement techniques.

The experimental loop is a closed hydraulic circuit designed for single-phase water flow. Its main objective is to produce a well-controlled cross-flow over a tube bundle while allowing high-resolution measurements of velocity fields and structural motion. A side view and a CAD representation of the facility are shown in Figure 1, which illustrate the overall configuration of the loop and its main components.

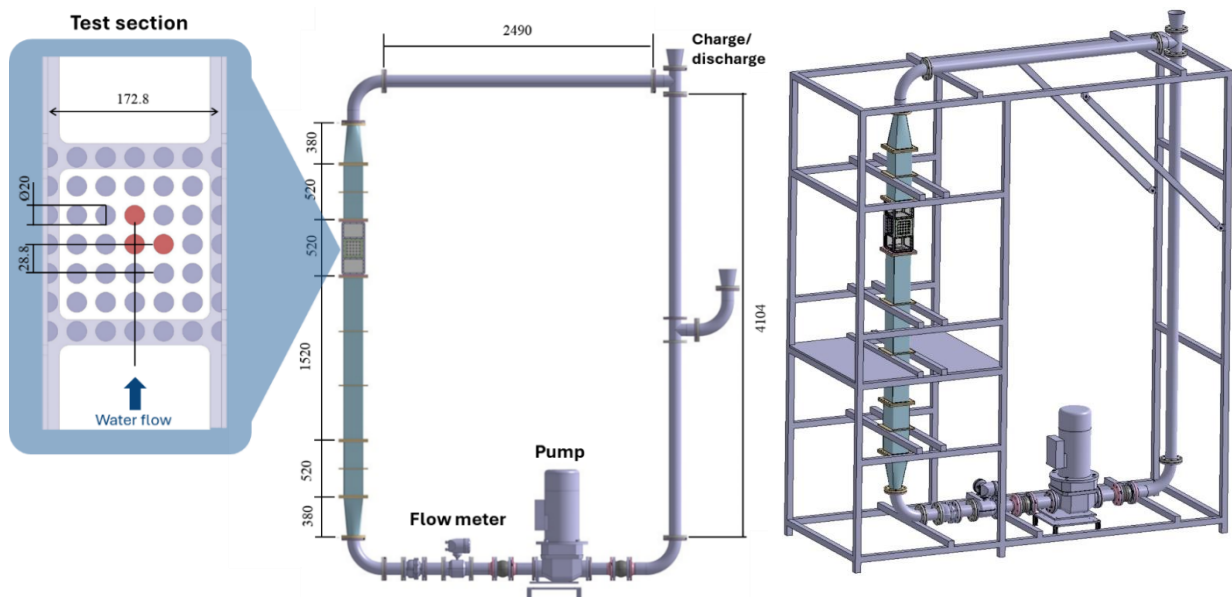


Figure 1: Side view (left) and isometric 3D representation (right) of the experimental facility.

Flow in the loop is generated using a centrifugal pump capable of delivering a maximum volumetric flow rate of approximately $187 \text{ m}^3/\text{h}$, which corresponds to a bulk velocity of about 1.5 m/s in the main channel. Flow rate control is achieved using a variable frequency drive, allowing precise adjustment of the pump speed. The working fluid in the loop is deionized water, maintained under pressure to prevent cavitation and minimize the formation of

bubbles that could interfere with optical measurements. The flow rate is measured with a Waterflux 3000 electromagnetic flowmeter, which provides high-accuracy measurements with an uncertainty of approximately $\pm 0.2\%$ of the measured flow rate. Additional monitoring is provided by Type-T thermocouples placed upstream and downstream of the test section to measure water temperature.

Before reaching the test section, the water passes through a flow-conditioning section designed to produce a stable and well-developed flow. The conditioning system includes a round-to-rectangular transition followed by a long rectangular channel with internal dimensions of 200×172.8 mm and a length of approximately 2040 mm, corresponding to about 11 hydraulic diameters. This channel allows the flow to develop before entering the measurement region.

The test section is the central part of the facility where all measurements are performed. It is a rectangular stainless-steel channel 520 mm long with the same cross-section as the upstream channel (200×172.8 mm). To enable optical measurements, the test section includes Plexiglass windows on each side, allowing laser access and camera visualization. The different observation regions—inlet, bundle, and outlet—are illustrated in Figure 2, which also shows the fields of view used for the optical measurements.

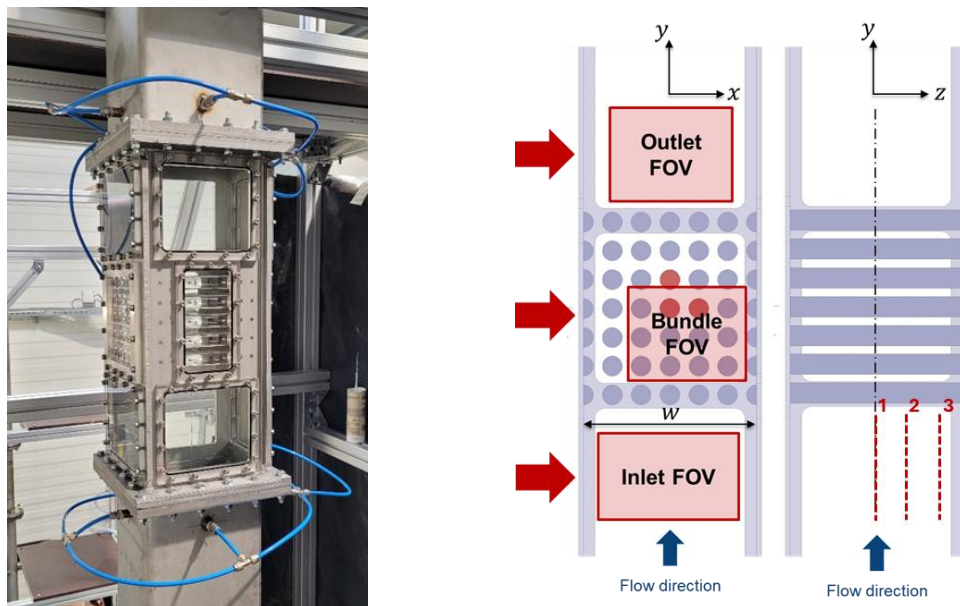


Figure 2: The actual test section (left) and the various PIV measurement viewing windows (right).

Inside the test section, the flow passes through a rod bundle composed of 49 cylinders arranged in a 7×7 square lattice, with five full cylinders in each row and two half-cylinders placed at the side walls. Each column of tubes has seven full rods. The rods are manufactured from PMMA (Plexiglass) to maintain optical transparency and reduce distortion during measurements. Each rod has a diameter $D = 20\text{mm}$ and a length $L = 200\text{mm}$. The pitch between rods $P = 28.8\text{mm}$, resulting in a $P/D = 1.44$.

One of the key capabilities of the GOKSTAD facility is the use of Particle Image Velocimetry (PIV) to measure detailed velocity fields in the flow. The PIV system uses laser illumination and tracer particles to visualize the fluid motion. The flow is seeded with hollow glass particles of

approximately 10 μm diameter, which are nearly neutrally buoyant in water and therefore closely follow the flow dynamics. Images of the particles are recorded using a dual-frame CCD camera equipped with a telecentric lens. This telecentric lens plays a critical role in the experiment because it eliminates perspective distortion and maintains constant magnification across the field of view. This property is particularly important for measurements inside densely packed rod bundles, where conventional lenses would introduce geometric distortions.

In addition to flow characterization, the facility can also study flow-induced vibration of cylinders. In this configuration, selected rods within the bundle are mounted on spring-based supports that allow motion in two orthogonal directions. The spring mounting system, shown in Figure 3, uses four coil springs placed at 90° intervals around the cylinder, allowing controlled displacement in both streamwise and transverse directions. Resulting properties of the tube are listed in Table 1.

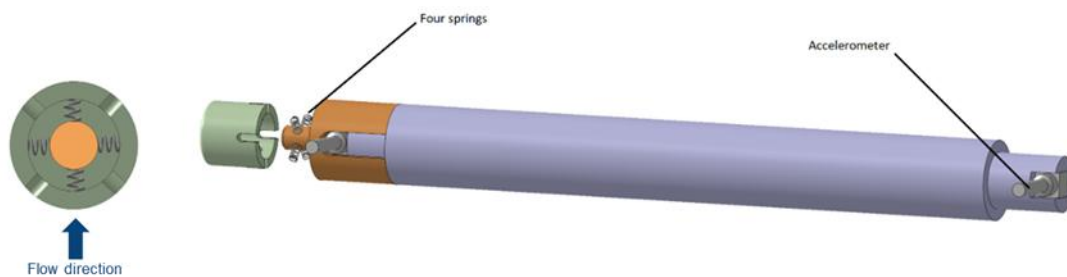


Figure 3: Schematic of the spring-cylinder mounting system.

Table 1: Dynamic parameters of the cylinder-spring system under air conditions

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

Three vibration configurations were investigated, viz. a single vibrating rod located at the center of the bundle, two vertically aligned rods (in-line configuration) and two horizontally aligned rods (side-by-side configuration). Cylinder motion is measured using high-speed cameras, operate at 300–500 frames per second with a resolution of 1280 × 800 pixels.

2.2 Available measurement data

As mentioned previously, to fully characterize the flow field and support CFD validation, PIV measurements were performed across three key regions of the test section: inlet, bundle, and outlet. Each region required different measurement strategies due to optical access constraints and specific flow features of interest. In total, more than 40 flow cases were investigated, varying both inlet velocity and laser sheet position along the z-axis as seen in the right of Figure 2 for different cross-sectional planes. Flow rates ranged from 15 m^3/h to 120

m³/h, corresponding to inlet bulk velocities ranging from 0.125 to 0.975 m/s. Experiments were performed at room temperature, with inlet temperatures ranging between 20°C and 21°C.

Table 2 summarizes the measurement conditions across the three regions. In addition to the primary dataset, repeatability tests were conducted at a fixed condition to assess measurement consistency. Per region, the following data is available:

- Inlet: profiles as function of transverse coordinate (across tube bundle column) of average velocity, root-mean-square (RMS) of streamwise velocity, RMS of transverse velocity, and mean turbulent kinetic energy (TKE).
- Bundle and outlet: 2D fields of mean velocity and TKE on different planes. Additionally, profiles along several transverse lines of various statistics are available, such as mean velocity, mean streamwise velocity, mean transverse velocity, standard deviations of these velocities, mean TKE, etc.

Table 2: Summary of PIV Test Matrix

Region	Bulk Velocities (m/s)	Measurement Planes (z)	Notes
Inlet	0.125, 0.250, 0.500, 0.750, 1.000	0 cm (symmetry), 3 cm, 6.5 cm	Cross-sectional inlet profiles
Bundle	0.125, 0.250, 0.500, 0.750, 1.000	0 cm (symmetry)	Main scan of rod-bundle at the symmetry (center) plane
Bundle	0.125	0 cm, +2 cm, ±4 cm	Additional scans at low velocity to investigate flow change across cross-sectional planes
Outlet	0.125, 0.187, 0.250, 0.315, 0.350, 0.410, 0.500, 0.750, 1.000	0 cm (symmetry)	Focus on the flow field at outlet
Repeatability	0.500	0 cm (symmetry, bundle region)	4 repeated runs; one case tested with 5 Hz acquisition frequency

In addition to the plane measurements listed in Table 2, additional PIV profile measurements were performed in the bundle and outlet region. These were performed for bulk velocities ranging from 0.125 m/s till 1.0 m/s (0.97 m/s to be exact). Figure 4 shows the location of the available line profiles. All profiles were obtained for the central plane, i.e., at $z = 0.0$ m. In addition, for bulk velocity $U_b = 0.125$ m/s, line profiles at $z = +0.02$ m, $z = +0.04$ m and $z = -0.04$ m are also available. The statistics available on (almost) all the line are averages of the following quantities: transverse velocity, streamwise velocity, 2D velocity, kinetic energy, standard deviation of the transverse velocity, standard deviation of the streamwise velocity,

standard deviation of the 2D velocity, turbulent kinetic energy, Reynolds stress R_{xy} , Reynolds stress R_{xx} , Reynolds stress R_{yy} and turbulent shear stress

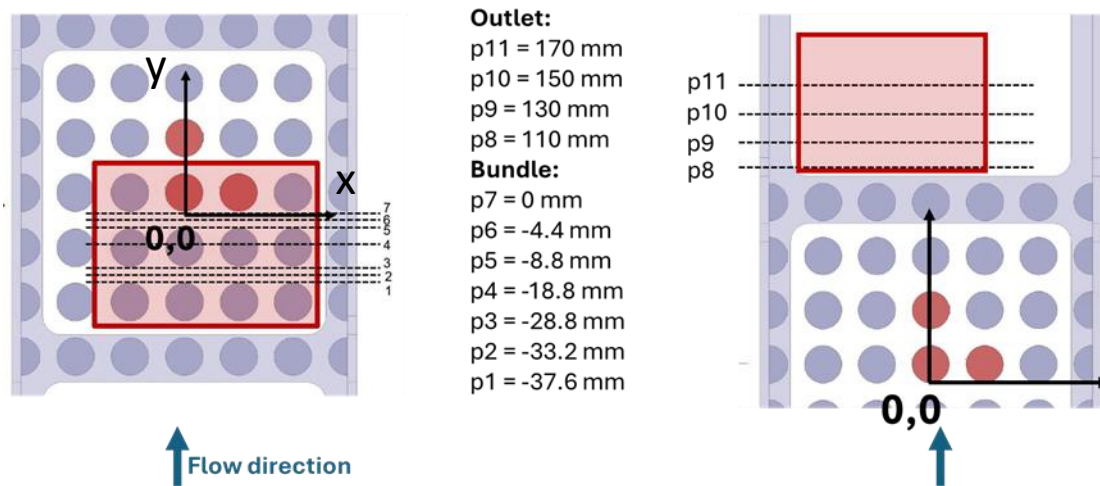


Figure 4: Additional PIV measurements performed to get line profiles of several flow quantities.

The second experimental campaign conducted with GOKSTAD focused on measuring the dynamic response of cylinders subjected to turbulent cross-flow. These experiments build upon the same hydraulic facility but incorporate the vibration-enabled rod mounting system described previously. Three different structural configurations were investigated, see Figure 5:

- Single vibrating cylinder located at the center of the bundle
- Two cylinders in vertical (in-line) arrangement, aligned with the flow direction
- Two cylinders in horizontal (side-by-side) arrangement, positioned perpendicular to the flow

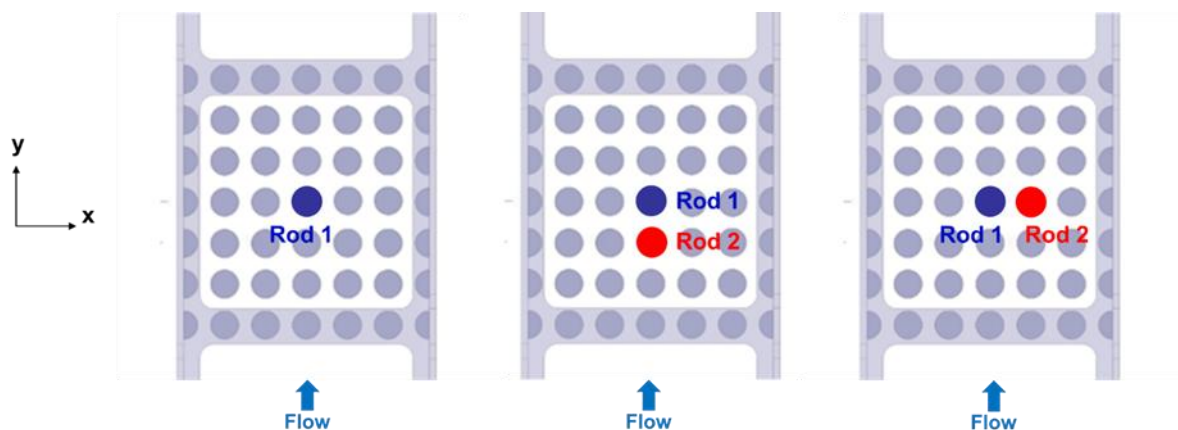


Figure 5: Schematic representation of the three vibrating-rod configurations investigated with the GOKSTAD facility, with, left: single moving tube, (middle) vertical (in-line) configuration, and (right) horizontal (side-by-side) configuration.

Measurements were performed for many different bulk velocities ranging from approximately 0.10 to 0.81 m/s. The motion of the rods was recorded using one or two high-speed cameras operating at 300–500 frames per second, with the recording durations ranging from about 14 to 23 seconds. The vibration experiments produced time-resolved datasets describing the motion of the cylinders. This allowed to extract among others:

- The Mean displacement of the cylinder center
- The Root-mean-square vibration amplitudes
- Dominating vibration frequency obtained using FFT analysis

3. Benchmark comparison

3.1 Introduction & considered benchmark case

The single vibrating tube configuration at a bulk velocity of 0.125 m/s was the chosen configuration for the benchmark. This was the lowest velocity at which experimental data is available, and corresponds to a Reynolds number based on the bulk velocity $Re_{U_{bulk}} \approx 23.125$, and that based on the gap velocity of $Re_{U_{gap}} \approx 6.875$. Hence the case is in the low turbulence regime. While at the lower range of tested velocities, it still represents a challenging case due to the possible combination of turbulence-induced vibrations (TIV) and vortex-induced vibrations (VIV). The lower bulk velocity is chosen so that high fidelity methods such as Direct Numerical Simulations (DNS) and Large-Eddy Simulations (LES) would still be computationally feasible in an Fluid-Structure Interaction (FSI) context.

A total of three institutions have participated in this benchmark: VCU, EDF, and NRG PALLAS. Key details of the numerical simulations of the participants are provided in Section 3.2, while results are compared to the experimental available data in Section 3.3. The benchmark findings are concluded in Section 3.4.

3.2 Simulation approaches

Table 3 summarizes the participants' individual submissions. VCU performed one-way high-resolution FSI simulations, while EDF and NRG PALLAS performed two-way medium-resolution FSI calculations. Although the focus of this chapter will be on the benchmark case with an inlet velocity of 0.125 m/s, EDF and NRG PALLAS additionally performed simulations for an inlet velocity of 0.5 m/s. Details on all the results achieved by each participating institution can be found in the dedicated Chapters 4 through 6.

Table 3: General Information of the Submissions

Institution	Approach	Turbulence Modelling	Solid Modelling	Software(s)
VCU	One-way FSI	Resolved LES	Rigid-body approach	Nek5000
EDF	Two-way FSI	Elliptic-Blending RSM	Rigid-body approach	Code_Saturne V8
NRG PALLAS	Two-way FSI	WR-LES WALE	Rigid-body approach	Simcenter STAR-CCM+ V2510

3.2.1 Geometry & Mesh

3.2.1.1 Solid Domain

All partners used a rigid-body approach to simulate the moving central cylinder, thereby representing it as a set of two mass-spring-damper systems. Hence, the cylinder is modelled as a 2-Degree-of-Freedom system, whereby it is allowed to translate only in lift (transverse) and drag (streamwise) direction. The cylinder's length and diameter are respected by all partners in the simulations.

3.2.1.2 Fluid Domain

As it is impossible to consider the full experimental loop in the simulations, the participants chose to limit the fluid domain to the region of interest, i.e., the test section containing the tube bundle, with added upstream and downstream sections. Table 4 summarizes the fluid domains used by the participants for their simulations. All participants included the full tube bundle, with different lengths for the inlet and outlet regions.

Table 4: Fluid domain details

Institution	Geometry Model	Mesh Cell Type	Mesh Cell Count (M Cells)	Wall Y+	Courant Number
VCU	Full bundle included. Inlet region is 10D, and outlet region 15D.	Hexahedral	1.02 (Polynomial Order=7)	Max. = 6.08 Avg. = 0.854	Avg. = 0.36
EDF	Full bundle included. Inlet region is 10D, and outlet region 50D.	Hexahedral	10	Max. = 2.0 Surf. Avg. = 0.26	Max. = 0.97 Vol. Avg. = 0.06
NRG PALLAS	Full bundle included. Upstream and downstream regions $\approx$ 0.9m.	Hexahedral	10.7	Max. = 0.207 Surf. Avg. = 0.0386 (Excl. front and back)	Max. = 2.99 Vol. Avg. = 0.0821

3.2.2 Material properties

3.2.2.1 Solid properties

The adopted solid properties are listed in Table 5 for all participants. Since all three partners used a similar rigid body approach based on the provided experimental values, the same values are used by all.

Table 5: Solid properties

Institut ion	Mass (kg)	Damping coefficient (Ns/m)	Spring stiffness (N/m)	Other Provided Data
VCU	0.1	1.14	7000.0	Used for each direction
EDF	0.1	1.14	7000.0	Used for each direction
NRG PALLAS	0.1	1.14	7000.0	Used for each direction

3.2.2.2 Fluid properties

In addition to the solid properties, fluid properties are needed in the simulations. Those used by the participants are listed in Table 6. All participants adopted properties at room temperature, in line with the experiment.

Table 6: Fluid properties

Institution	Density (kg/m ³)	Dynamic Viscosity (mPa·s)	Remarks
VCU	1000.0	1.0	Experiment at 20-21°C, which means $\rho \approx 998 \text{ kg/m}^3$ and $\mu \approx 1.0 \text{ mPa}$
EDF	1000.0	1.0	
NRG PALLAS	1000.0	1.0	

3.2.3 Boundary conditions

3.2.3.1 Solid domain

The boundary conditions adopted for the solid domain by all participants are listed in Table 7. As VCU used a one-way FSI approach, the moving cylinder is a fixed no-slip wall in the LES, and a moving no-slip wall in the stand-alone structural simulation.

Table 7: Solid boundary conditions

Institut ion	Moving cylinder	All other walls	Remarks
-----------------	--------------------	--------------------	---------

VCU	No-slip wall	No-slip	LES forces fed to cylinder in stand-alone simulation.
EDF	No-slip wall	No-slip	-
NRG PALLAS	No - slip + FSI interface	No-slip	Sides of moving cylinder slide over side walls.

3.2.3.2 Fluid domain

The boundary conditions used in the fluid domain by the different participants are listed in Table 8. VCU used a recycling inlet condition, using the profile at 5D downstream the inlet at the inlet.

Table 8: Fluid boundary conditions

Institut ion	Inlet BC	Outlet(s) BC	Remarks
VCU	Inlet velocity of 0.125 m/s	Dong outflow condition	Inflow velocity profile obtained by recycling over inlet section of length 5D _{pipe} .
EDF	Inlet velocity of 0.125 m/s	Free outlet	Fully developed profile using an embedded 2D simulation
NRG PALLAS	Uniform value of 0.125 m/s	0 Pa Gauge Pressure Outlet	At inlet and initial conditions, 5% turbulence intensity set with turbulent length scale of 0.13m

3.2.4 Numerical settings

An overview of important numerical settings used by the participants for the simulations is given in Table 9.

Table 9: Numerical settings

Institut ion	Time Step Size (ms)	Total Simulatio n Time (s)	Order of Discretization	Other Provided Data
-----------------	---------------------------	-------------------------------------	----------------------------	---------------------

VCU	0.25	9 FP for flow-only averaging and force data	Temporal = 2 nd (BDF2) Spatial = 7 th	For LES, only the explicit high-pass filter is used. No additional sub-grid scale model is included.
EDF	0.25	>361s	Temporal = 2 nd (Crank-Nicolson) Spatial = 2 nd	Maximum number of ALE outer iterations = 10.
NRG PALLAS	1.0	74.0 (40.0 s flow + 34.0s FSI)	Temporal = 2 nd (BDF2) Spatial = 2 nd	Number of inner iterations per time step = 25 SIMPLEC pressure-velocity coupling scheme

3.2.5 Computational resources

The computational resources consumed by each participant in this benchmark exercise 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
VCU	14d 22h 5m	61.9E5 node h	8.5 TB	17280 CPU cores	Number of nodes used = 90
EDF	44d 3h	6.77E5 CPU h	6 TB	640 CPU cores	10
NRG PALLAS	12d 5h 45m	4.22E5 CPU h	768 GB	144 CPU cores	Number of nodes used = 3

3.2.6 Simulation Strategy

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

3.2.6.1 VCU

LES was run until the flow was fully developed throughout the domain. Fully developed flow was determined by tracking history points. Averaging of the flow field was done over 9 flow through lengths. Force data was collected on the center tube during the flow averaging and used for the one-way coupling.

3.2.6.2 EDF

First a pure CFD simulation is performed with rigid, non-moving, tubes. This is run until a fully developed flow field is obtained. Subsequently, the two-way coupling with ALE is activated and run for more than 360.0 s to evaluate the emergence of distinct flow-field modes associated with the known bifurcation behavior in tube bundles (Zwijssen et al., 2025).

3.2.6.3 NRG PALLAS

NRG PALLAS performed a single simulation of 74.0s, consisting of the following phases:

- T = 0.0 – 16.0 s: Flow development
- T = 16.0 – 40.0s: Obtain flow statistics
- T = 40.0s: Release time of cylinder
- T = 40.0 – 42.0s: Ramp time of cylinder (slow increase of fluid forces)
- T = 42.0 – 50.0s: FSI adjustment
- T = 50.0 – 74.0s: Obtain FSI statistics

Lengths of these phases were chosen based on a single flow-passage taking about 8.0s for a domain length of 2.0m and an inlet velocity of 0.125 m/s.

3.3 Results comparison

The data obtained from the participants are consolidated into appropriate plots and are presented in this section. This will first be done for flow-only comparison for the inlet, bundle and outlet regions in Section 3.3.1, followed by a comparison for the vibration data in Section 3.3.2.

3.3.1 Flow data comparison

As outlined in Section 2.2, experimental data is available for the inlet, bundle and outlet regions. Hence simulation results for each of these regions will be compared to those available from the experiment. As a first comparison, Figure 6 compares the velocity profiles obtained by the participants inside the PIV measurement plane just upstream the tube bundle to that from the experiment. The velocity levels in the bulk region are quite well predicted by the simulations, as expected when the correct inlet velocity is prescribed. Also, the drop-off near the walls is reasonably captured by all participants. There are also some significant differences, however. The experiment predicts an asymmetric profile, while those from the simulations are all more or less symmetric. The asymmetry in the profile probably stems from the

upstream bend, which was not included in the simulations. This also indicates the flow is not fully developed yet near the bundle.

Examining the velocity profiles predicted by the simulations, the results from VCU and NRG PALLAS are in close agreement, both exhibiting a relatively flat core region with a thin boundary layer near the walls. In contrast, the EDF profile is more curved, with the maximum velocity occurring at the center of the duct. This profile more closely resembles that of a fully developed turbulent flow, whereas the VCU and NRG PALLAS profiles may indicate that the flow is not yet fully developed. Bulk velocities match quite well between all profiles.

For the same location, Figure 7 compares the turbulent kinetic energy, normalized by the bulk velocity. All models predict increased TKE near the walls, consistent with known theory of wall-bounded turbulent flows. This increase in TKE is caused by the shear layers present in the boundary layer region. Similar magnitudes of the peaks are predicted by the three participants. The TKE peaks at the wall are not seen in the experiment. This is most likely caused by the frame blocking the laser view in this region. Looking at the value of the normalized TKE in the bulk region, a decent match is found by EDF and NRG PALLAS. VCU predicts nearly zero TKE in that area. This most likely indicates their inlet region, with its recycling zone, was not long enough to develop sufficient turbulent fluctuations. Finally, in line with the more curved inlet velocity profile, giving more shear between the layers of the flow, EDF also finds a more curved TKE profile.

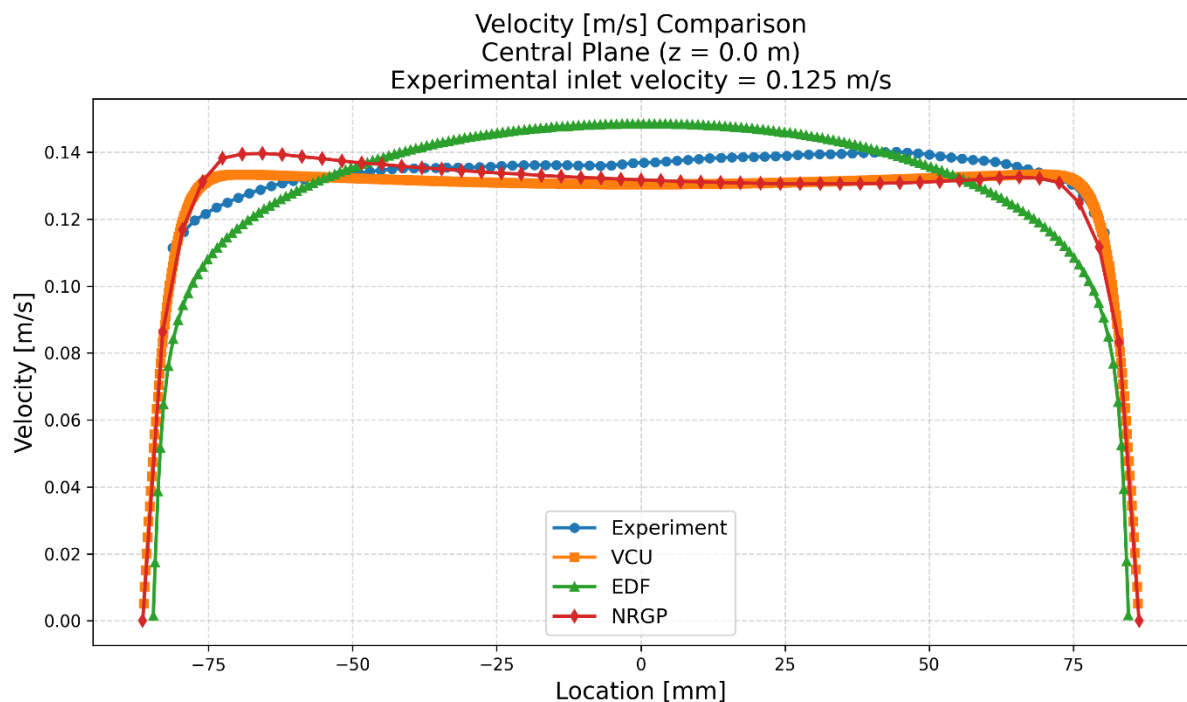


Figure 6: Comparison of the time-averaged velocity in the inlet region just upstream the tube bundle.

Having looked at the inlet region, attention is switched to the bundle and outlet region, for the same quantities, i.e., the average velocity and turbulent kinetic energy. As a first comparison for those regions, Figure 8 shows the time-averaged velocity on a streamwise central plane covering both (a part of) the bundle and outlet regions. The field of view is

depicted in the left of the figure. All plots use the same colour bar range and include streamlines, for easier comparison and visualisation. The plots show from left to right the results of the experiment, VCU, EDF and NRG PALLAS.

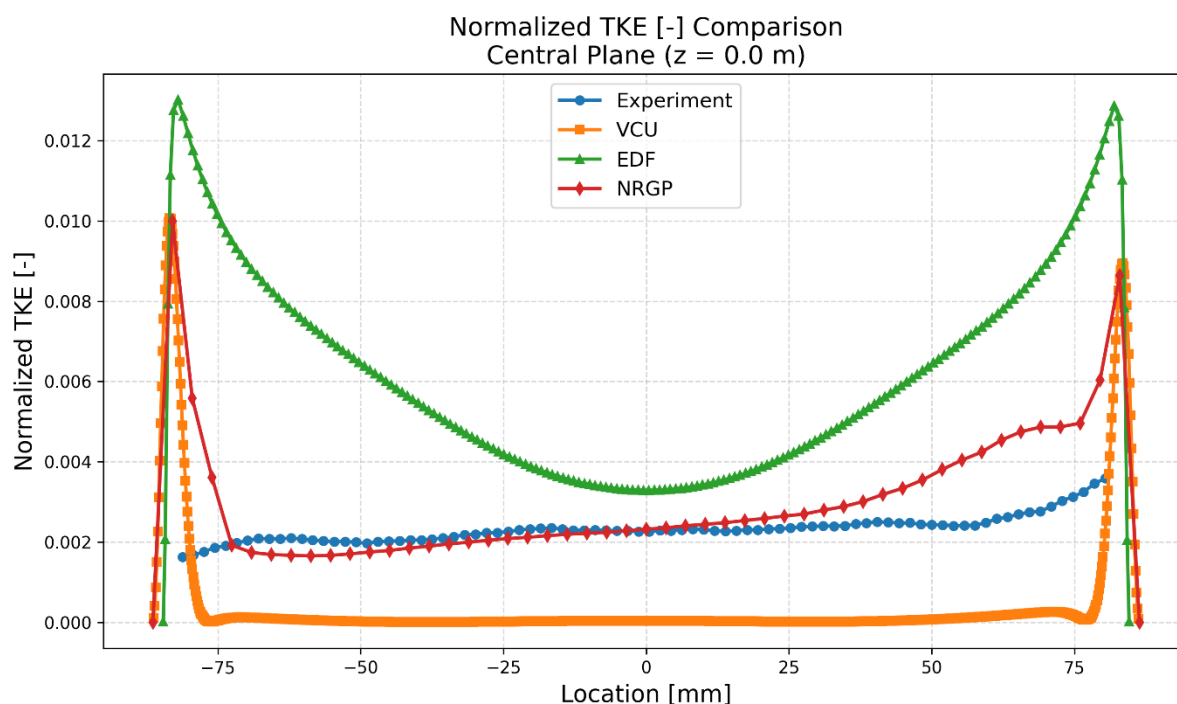


Figure 7: Comparison of the time-averaged normalized turbulent kinetic energy in the inlet region just upstream the tube bundle.

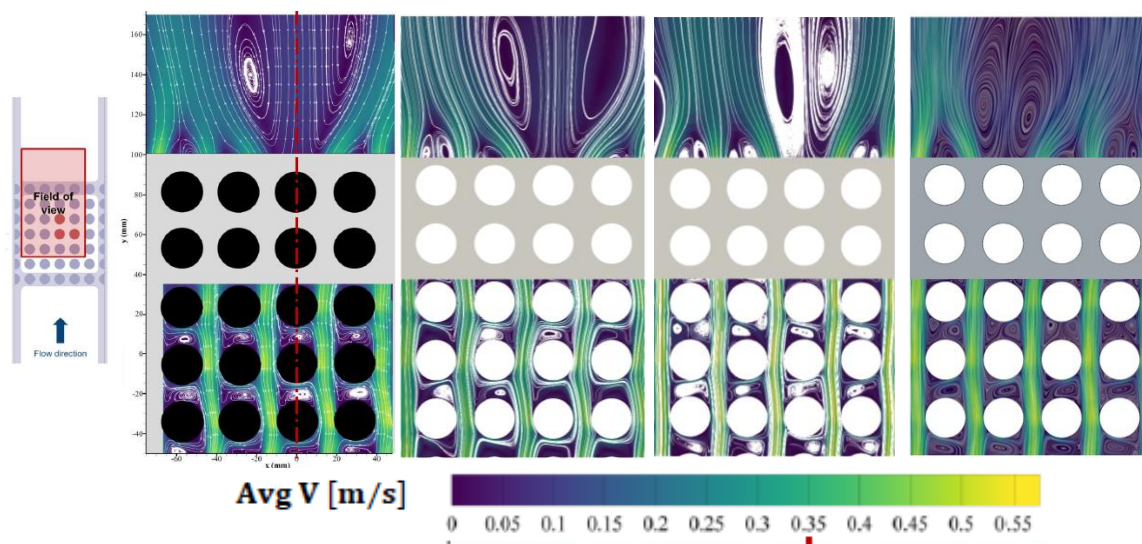


Figure 8: Comparison of the time-averaged velocity on a streamwise plane inside the test section between experiment (left), VCU (middle-left), EDF (middle-right) and NRG PALLAS (right), for $u = 0.125$ m/s.

Looking first at the outlet region, a similar behaviour is found in all simulations, and which match the experiment, viz. isolated jets emerging from the bundle flow lanes that merge into two larger jets directed towards the test section walls, with in between the two merged jets a

local recirculation zone. The colours are similar between all simulations and match the experiment well, indicating correct prediction of the outlet velocities. Also, all simulations recover the double vortices in the outlet recirculation zone, as well as some smaller recirculation zones between the last row of cylinders, in between the isolated jets.

In addition to the above, VCU and EDF correctly predict the location of the recirculation zone in the outlet region, located behind the central cylinder. For NRG PALLAS, this zone is located behind a cylinder one column closer to the wall. Looking at the width of the recirculation zone, VCU seems to match the experiment best, with EDF and NRG PALLAS predicting slightly narrower zones.

Looking at the bundle region, with Figure 9 showing the same quantity inside the bundle field of view as indicated in Figure 2, some interesting features can be observed. Strong jets form inside the flow channels bounded by the columns of tubes. These jets show a meandering pattern when passing through the tube bundle. This meandering pattern appears in both the experiment and all simulations. The jets do not necessarily go from left to right with each passing row. This is especially apparent in the experiment in Figure 8, where the jets remain attached to the left column for the last two rows visible in the field of view. Do note this field of view is not exactly the same as in Figure 9. The strong flow channels result in recirculation zones behind the cylinders, with the location of these zones dependent on the high-speed attachment areas of the jets; they tend to form to the lower velocity side of jets. This formation process of recirculation zones is seen in the experiment as well as in all simulations.

On the same planes at the velocity fields, Figure 10 and Figure 11 show the normalized TKE in the combined bundle and outlet region and the dimensional TKE in the bundle region, respectively. There is clearly a match between zones of high velocity and high TKE, especially in the outlet region. All simulations match the experimental picture quite well, with the note that the location of the recirculation zone is shifted one column to the right in the NRG PALLAS simulation. Inside the bundle, the TKE increases moving downstream the tube bundle. This is found by all participants and matches with what is observed in the experiment. In the bundle, zones of high TKE are especially seen everywhere at the detachment points next to the tubes. Looking at the levels of TKE, the simulations are reasonably close to the experiment, though VCU seems to over-estimate the TKE, while EDF and NRG PALLAS are on the lower side with their found values, especially in the bottom of the tube bundle.

To further compare the participants' results inside the tube bundle, profile plots of velocity and TKE are compared to the experimental data. This is done for the profiles P4 till P7, as defined in Figure 4. The resulting comparison is shown in Figure 12. As can be seen, the profiles are generally quite well predicated by all participants, for all the four P-lines. A good match is found in location and magnitude of the peak velocities and their shapes inside the flow channels in between successive tube columns. Additionally, the recirculation zones are also generally well captured by the participants, as evidenced by small intermediate peaks located in the wakes of the tubes. In particular VCU finds a good match for all profiles. Those of EDF and NRG PALLAS seem to overpredict the primary peaks slightly, though EDF gets a good match for the secondary peaks. NRG PALLAS' secondary peaks seem to be located mostly at the incorrect side of the wakes. This, however, is probably a result of the simulation being full axis-symmetric about a central plane. Hence, the transverse coordinate has no actual direction in the simulations, and all plots could just as well be mirrored.

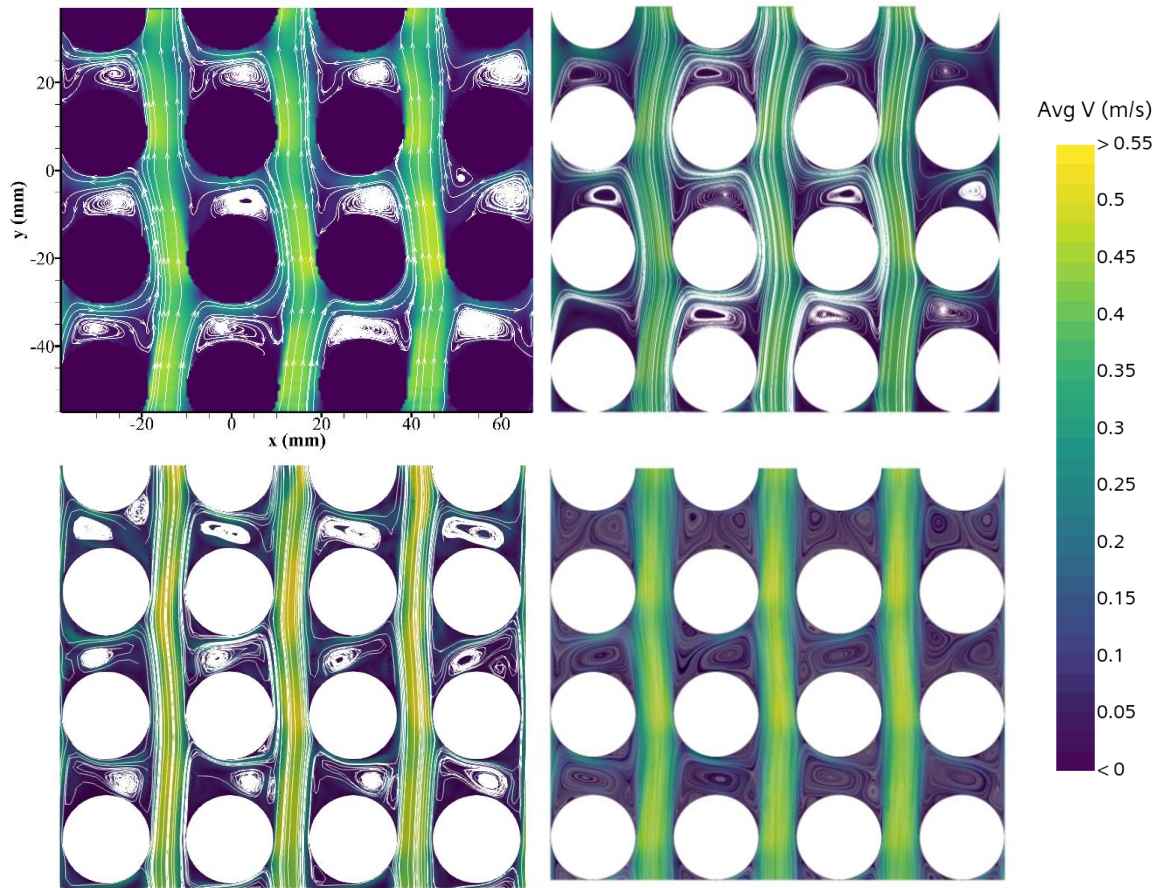


Figure 9: Comparison of the time-averaged velocity on a streamwise plane inside the bundle region between experiment (upper left), VCU (upper right), EDF (lower left) and NRG PALLAS (lower right), for $u = 0.125$ m/s.

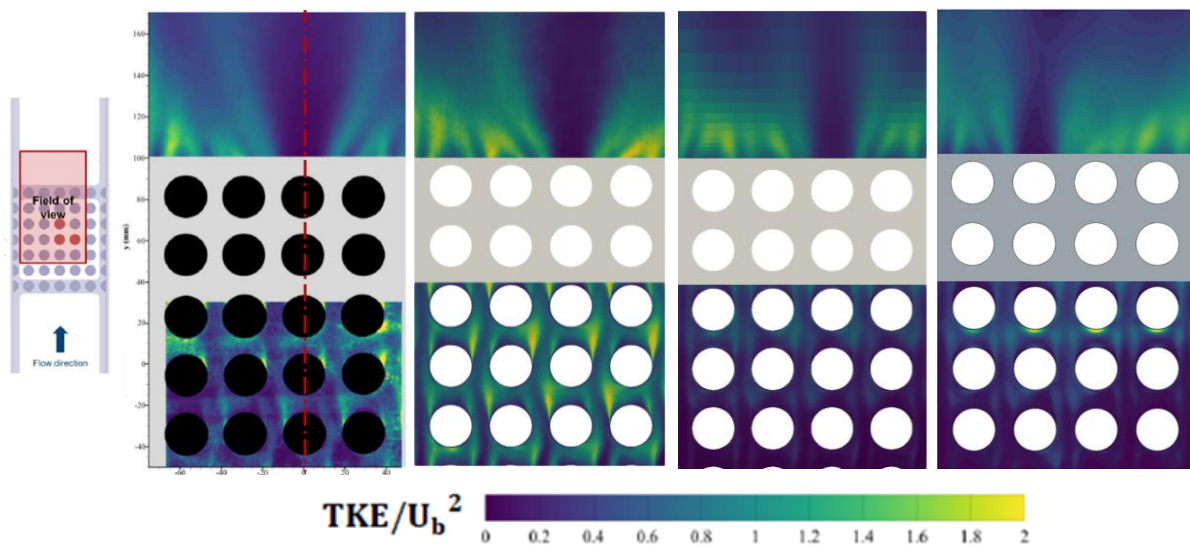


Figure 10: Comparison of the time-averaged normalized TKE on a streamwise plane inside the test section between experiment (left), VCU (middle-left), EDF (middle-right) and NRG PALLAS (right), for $u = 0.125$ m/s.

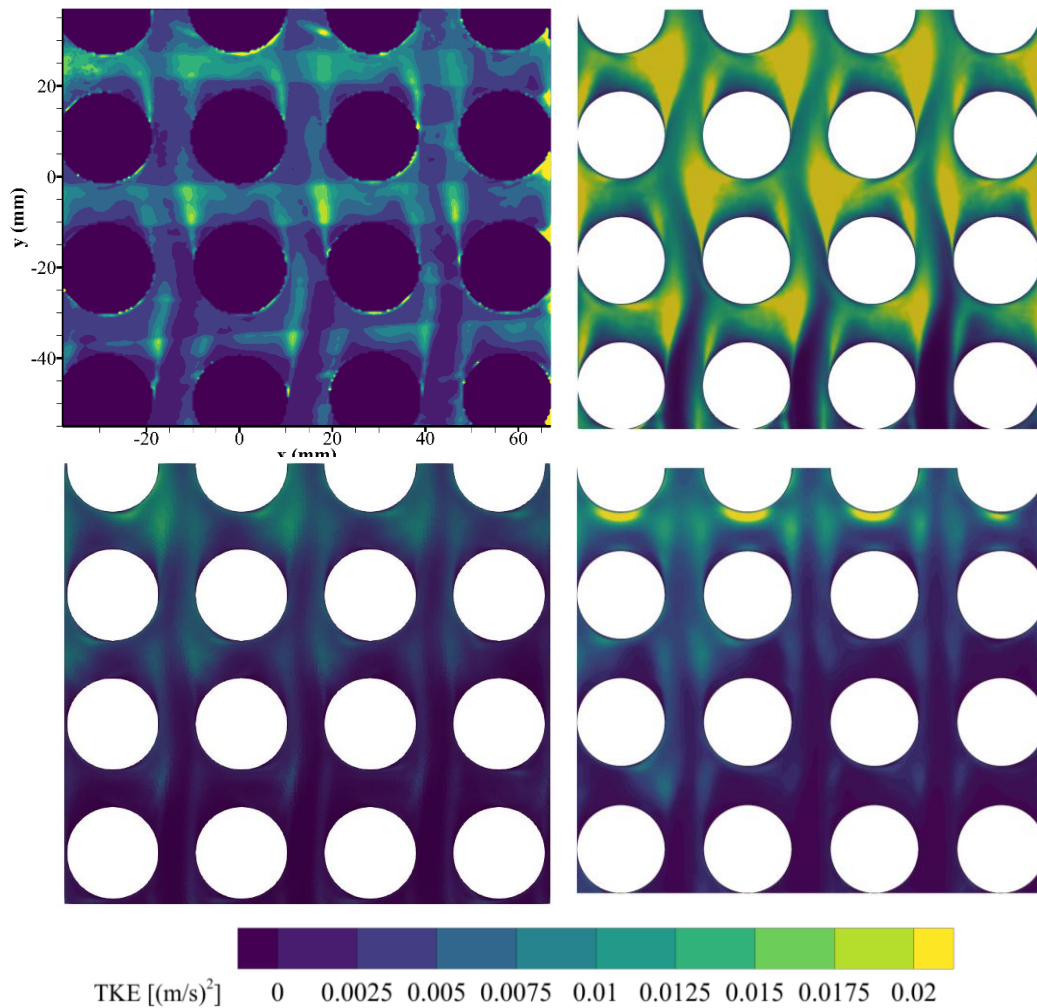


Figure 11: Comparison of the time-averaged TKE on a streamwise plane inside the bundle region between experiment (upper left), VCU (upper right), EDF (lower left) and NRG PALLAS (lower right), for $u = 0.125$ m/s.

Figure 13 gives a similar comparison as the previous figure, but now for the dimensional TKE. For this quantity, there are larger differences between the experimental results and those coming from the simulations. The differences are mainly in the found magnitudes, as the shapes of the profiles are decently captured, showing primary and secondary peaks again in all simulations, in line with what is observed in the experiment. Looking at the TKE levels, VCU mostly overpredicts the amount of TKE inside the bundle, while EDF and NRG PALLAS underestimate it. This is in line with the observation made earlier based on Figure 11.

Profiles of velocity and TKE on lines P8 to P11 in the outlet region are compared to the experiment in Figure 14 and Figure 15, respectively. Looking at the velocity profiles, a very good match is found by VCU, with EDF a good second. Those of NRG PALLAS are quite off, which is mainly a result of the recirculation zone being located to a column of cylinders shifted one to towards the wall, and off the center, as compared to the experiment. Shifting the profiles of NRG PALLAS by one pitch would result in a better match. All partners do predict quite well the transition of isolated jets towards two merged jets, with a recirculation zone in between, and how this transition takes place moving downstream the outlet region. This is apparent from the good match in profiles for each location.

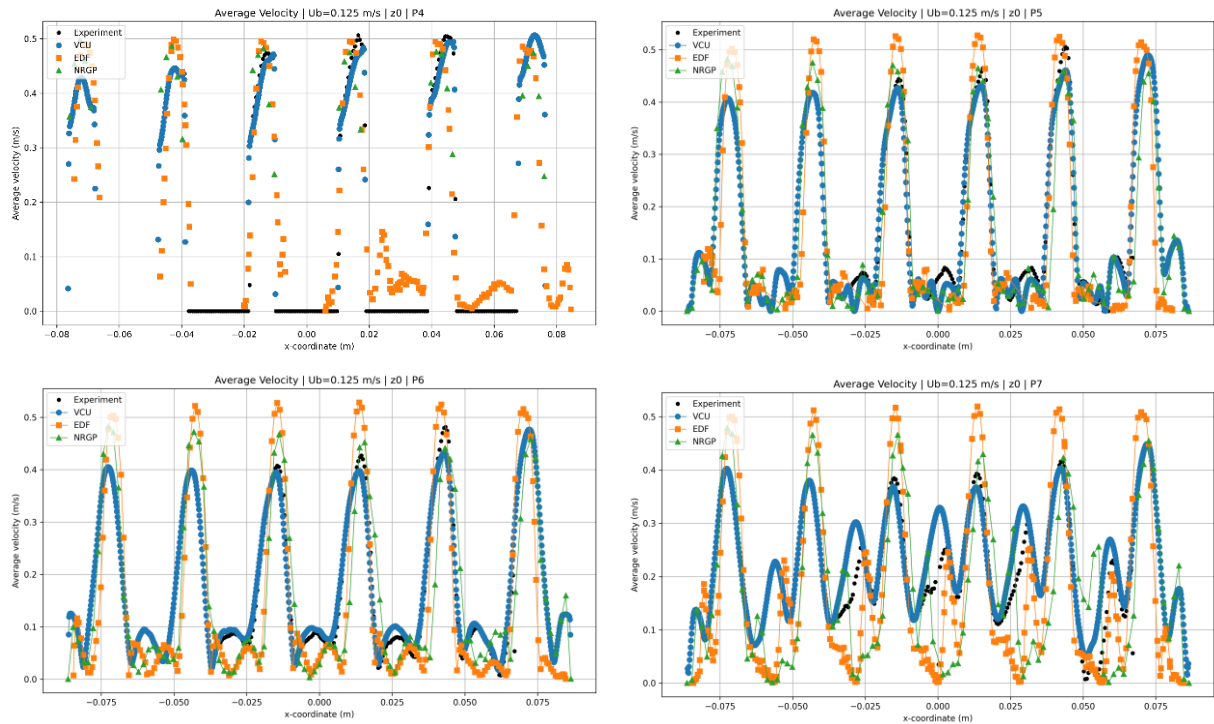


Figure 12: Comparison between experiment and simulations of the velocity inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right).

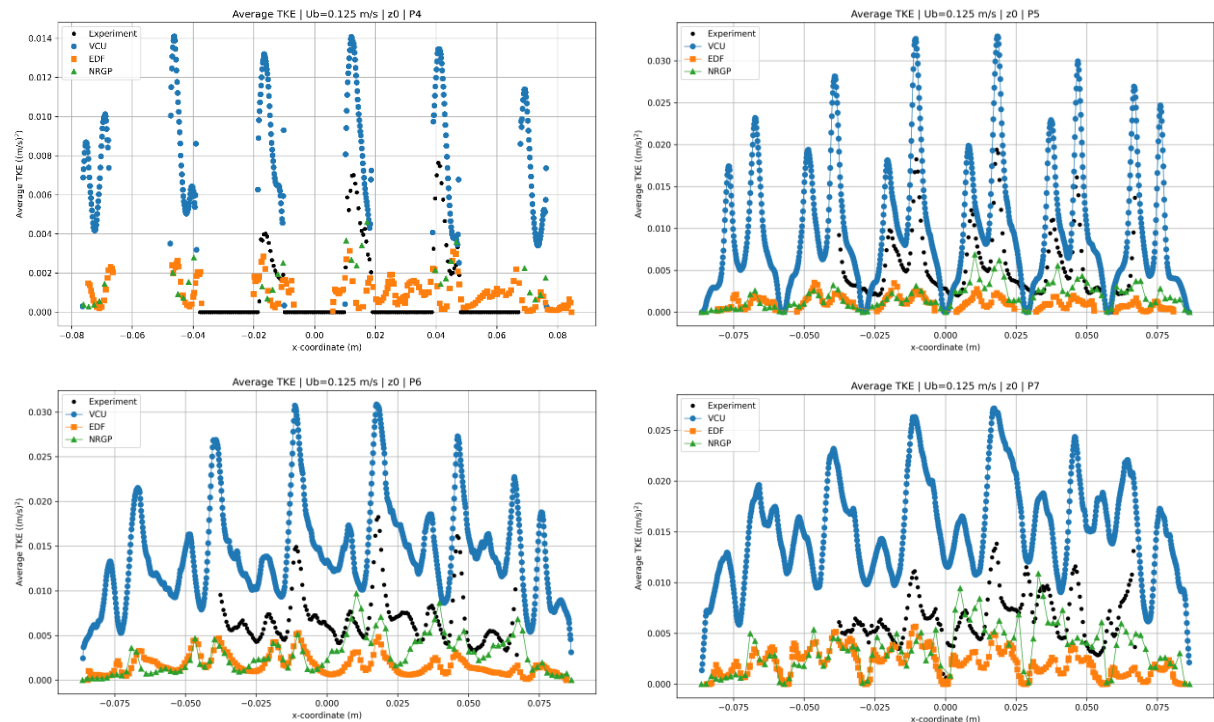


Figure 13: Comparison between experiment and simulations of the TKE inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right).

Magnitudes and profiles of the TKE are also reasonably well predicated by all participants, especially when accounting for the shift in recirculation zone found by NRG PALLAS. The

transition from isolated jets to a more uniform TKE profiles is well captured, and the corresponding levels of TKE are also recovered to a satisfactory degree.

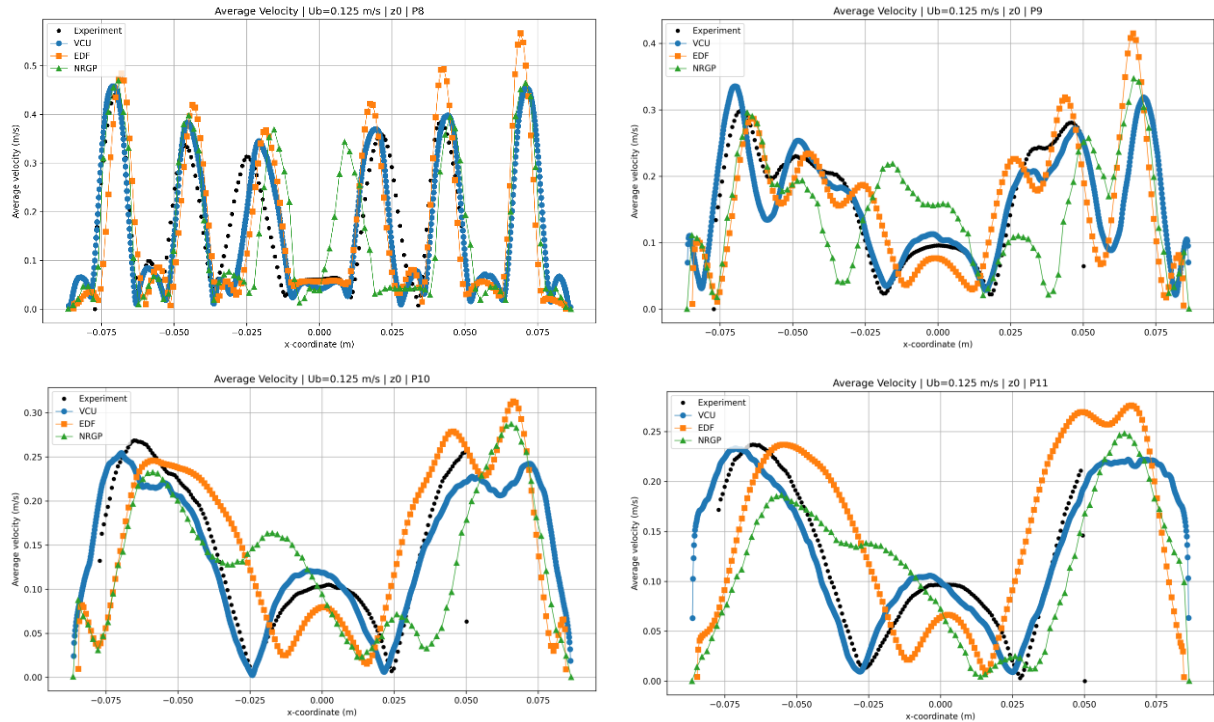


Figure 14: Comparison between experiment and simulations of the velocity in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right).

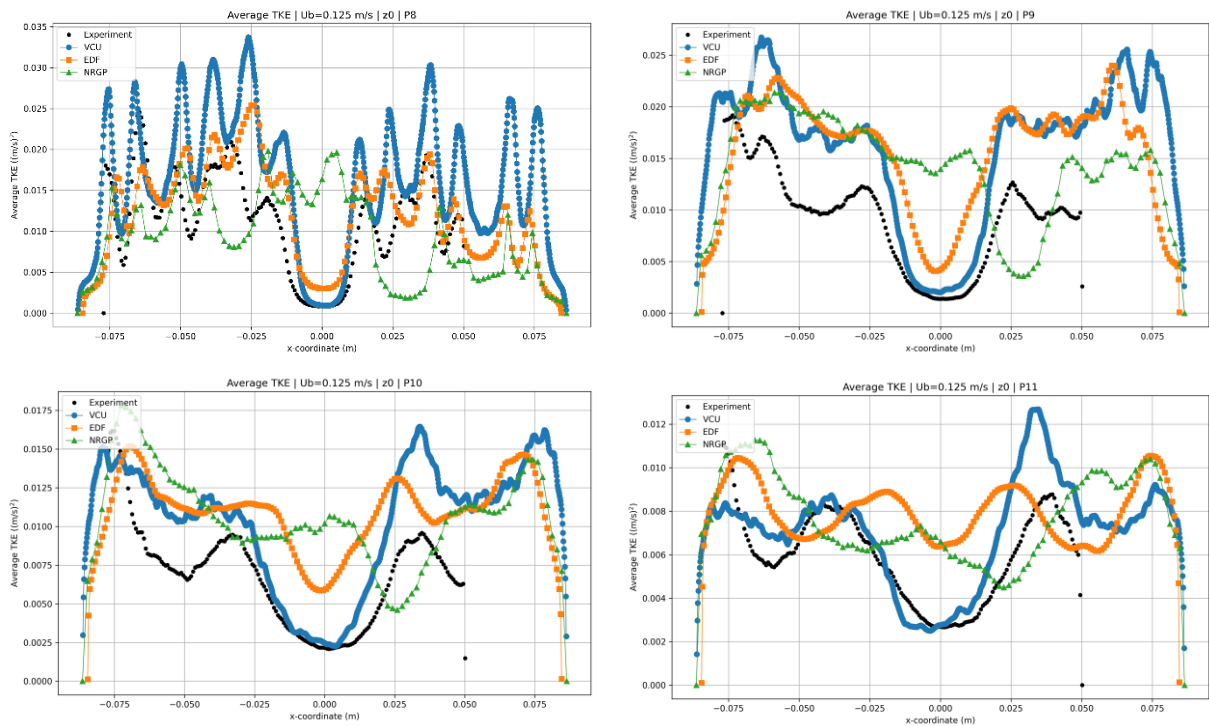


Figure 15: Comparison between experiment and simulations of the TKE in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right).

3.3.2 Vibration data comparison

Following comparison of flow field statistics, this section looks at FSI related quantities. These statistics were fully obtained in the two-way simulation, as performed by EDF and NRG PALLAS, during the phases of the simulations in which the central cylinder was allowed to vibrate. For VCU, which uses a one-way approach, the forces come from the stand-alone LES. These were subsequently prescribed to the structural only computation, in which the displacements were calculated.

Figure 16 shows the time series of the lift and drag forces and the resulting displacements as obtained by the partners. Unfortunately, these could not directly be compared to the experiment as these are not available. The EDF time series were truncated to 25.0 s, out of the total available data of over 360.0 s, to allow for better comparison. The other 335.0 s of data showed the same trend though as the 25.0 s currently shown. From the figure, it is interesting to see there are clear differences between the participants for the drag direction, while those for the lift direction are very similar. Regarding the latter, the signals from all three participants show very similar trends, with only NRG PALLAS predicting a slightly lower mean lift force and resulting displacement.

As said, differences are much larger for the drag direction. These differences mainly reflect the mean drag force and drag displacement, as the oscillations look quite similar in amplitudes. The means of the VCU and EDF time series are not very far off, with EDF predicting a slightly lower mean force, and resulting displacement. NRG PALLAS, on the other hands, predicts a larger mean force and, surprisingly, a lower, even negative, mean displacement.

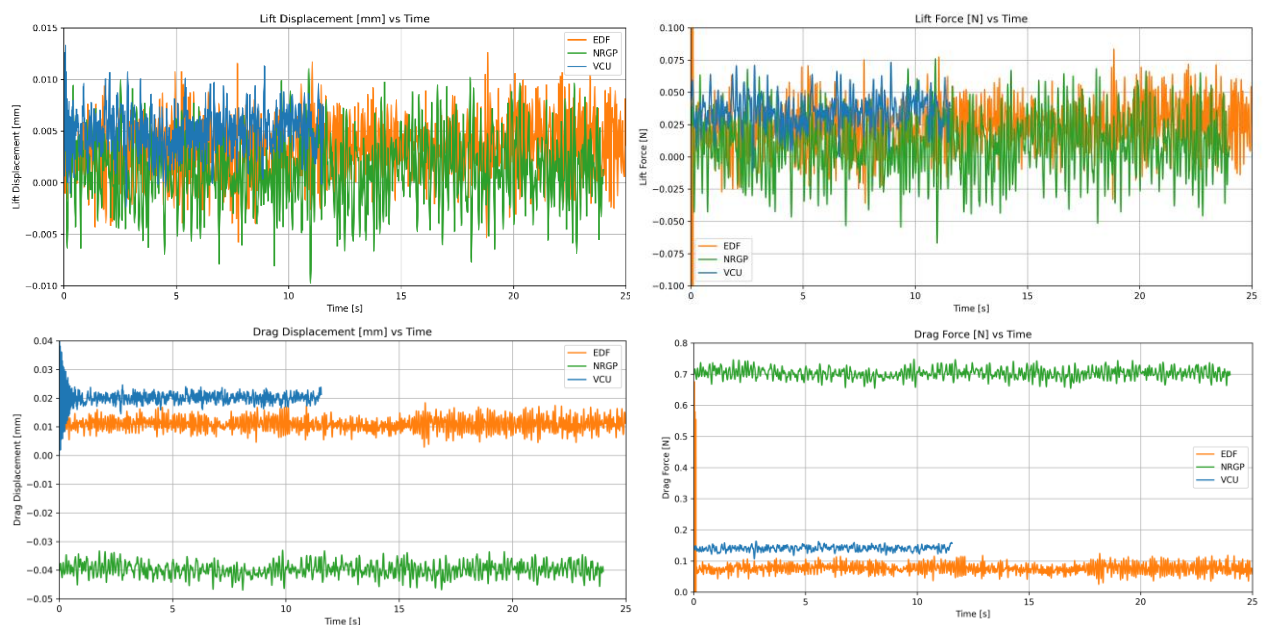


Figure 16: Lift (up) and drag (bottom) forces (left) and displacements (right) as function of time for VCU (blue), EDF (orange) and NRG PALLAS (green).

To analyse the frequency content of the time series, Power Spectral Density (PSD) plots were created of the time series. These are shown in Figure 17. Looking at the forces first, as these drive the displacement, it seems the level of forces are quite similar between all participants

up to about 30 Hz. From then on, there are some significant differences. The main difference is that EDF and NRG PALLAS predict a strong drop in the force spectra, while VCU finds a more gradual decrease consistent with normal turbulence spectra. This difference is most likely caused by the fact that VCU used a one-way FSI approach, while EDF and NRG PALLAS used a two-way FSI approach. For the latter, a dip in the force spectra is often observed at the structure's natural frequency in air (Zwijnsen et al., 2025). One other difference in the force spectra is that EDF finds multiple peaks at frequencies between 10Hz and 30 Hz, while VCU and NRG PALLAS mainly find a peak around 30 Hz.

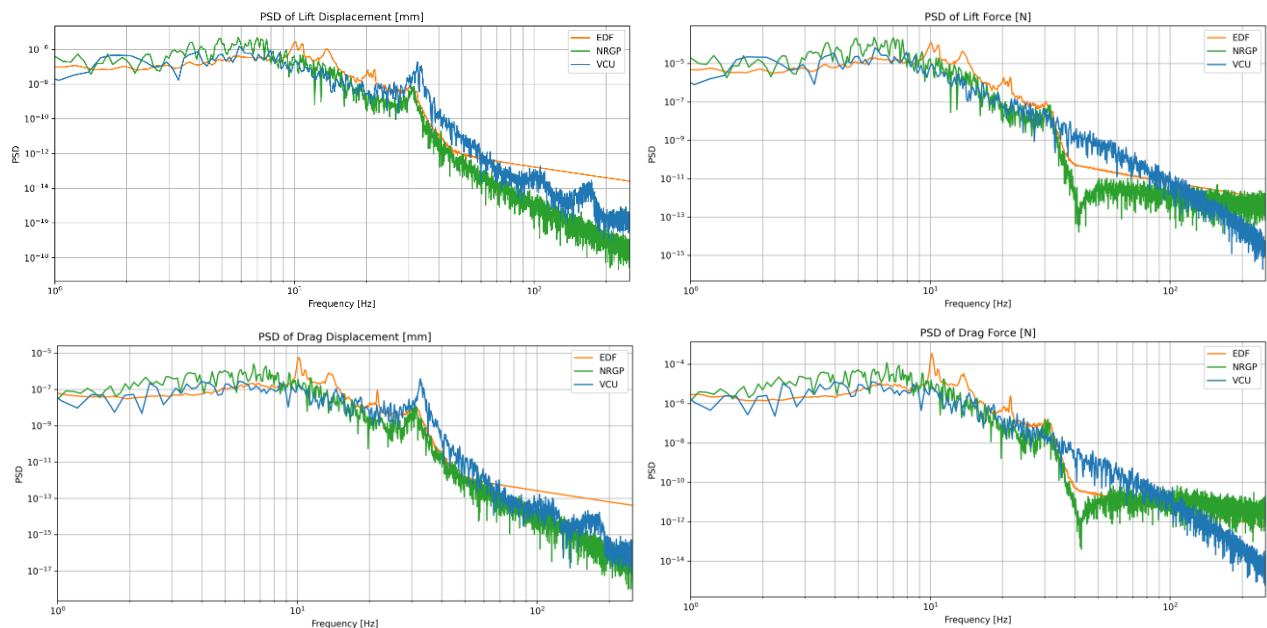


Figure 17: PSD of the lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for VCU (blue), EDF (orange) and NRG PALLAS (green).

Looking at the displacement spectra, more pronounced peaks are observed for all participants, at around 31-32 Hz. EDF again shows several peaks at lower frequencies. Overall, though, the spectra are similar, with VCU predicting higher energy content at the mid and high frequencies.

The time series and spectra were further analysed to obtain mean and root-mean-square (RMS) values, as well as predominant vibration frequencies. These are listed in Table 11, along with the experimental values, except for the frequencies, as it was difficult to discern any clear peak from the noise in the signals. Comparing the mean displacements, it is difficult to draw any conclusion. For the lift direction it is expected the mean displacement to be very close to 0.0. All partners find this, with mean values on the order of 1.0 micrometre. For the drag direction, as was already apparent from the time series plots, NRG PALLAS finds a negative mean value, while VCU and EDF find a positive value, in line with the experiment.

Looking at the RMS values, those from the participants for the drag direction are quite similar, and about a factor of 3.5-4.0 lower than in the experiment. There's a larger spread between lift RMS values from the participants, though all under-predict the experimental value.

Table 11: Comparison between experiment and simulations of relevant FSI statistics for the GOKSTAD benchmark.

U = 0.125 m/s	Frequency [Hz]		Mean displ. [mm]		RMS displ. [mm]	
	Lift	Drag	Lift	Drag	Lift	Drag
Experiment	N/A	N/A	-2.57E-2	6.43E-3	8.50E-3	8.32E-3
VCU	32	32	5.05E-3	2.01E-2	1.96E-3	2.18E-3
EDF	31	31	3.79E-3	1.10E-2	2.57E-3	2.34E-3
NRG PALLAS	31	31	1.25E-3	-3.98E-2	3.33E-3	2.30E-3

3.4 Benchmark conclusion

In this chapter, the GOKSTAD benchmark case was discussed. The selected benchmark case is that of the tube bundle with a central moving cylinder and an inlet velocity of 0.125 m/s. This was the lowest velocity at which experimental data is available, and corresponds to a Reynolds number based on the bulk velocity $Re_{U_{bulk}} \approx 23.125$, and that based on the gap velocity of $Re_{U_{gap}} \approx 6.875$. Hence the case is in the low turbulence regime.

Three participants provided their results for comparison: VCU, EDF and NRG PALLAS. Each participant used different modelling approaches, in particular for the flow field and coupling parts: one-way coupled with resolved LES for VCU, two-way coupled with an Elliptic Blending Reynolds Stress Model for EDF and two-way coupled with a wall-resolved LES for NRG PALLAS. All participants modelled the moving cylinder as a two degree-of-freedom mass-spring-damper system. Using different approaches makes the comparison interesting, but also at time difficult to draw clear conclusions. However, based on the found results the following can be said:

- Flow Field:
 - Using a high-resolution approach does improve predictions of the flow field, especially in regions of intense, locally generated, turbulence such as that inside a tube bundle. This comes at a significantly increased computation cost, however.
 - For high-resolution approaches, it is important to have a properly generated inlet turbulent velocity field.
 - Medium-resolution approaches can give quite accurate predictions, as long as the inlet turbulence is properly prescribed based on known, experimental, data. Especially if anisotropic turbulence models are used. Predictions are further improved if the relevant turbulent scales are resolved.
- Vibration data:
 - Using a simple 2DoF rigid-body approach seems to be sufficient when the motion of the structure is largely in a 2D-plane without any significant deformation.

- For cases with relatively low amplitudes, one-way approaches give similar predictions as two-way approaches, thereby avoiding the challenges of mesh morphing.
- General considerations:
 - For high-quality validation of numerical simulations, high-resolution, and CFD-grade, experimental data is needed. This means PIV measurements of the inlet region, to properly set the inlet conditions in the models, as well as in the regions of interest, to allow accurate comparison of the obtained results.
 - To properly assess FIV predictions, it is important to also have measurements of the forces on the moving cylinder.

4. Simulations performed by VCU

4.1 Method description

4.1.1 Modeling approach

The GOKSTAD experimental geometry simulated in this work consist of 5 rows of 7 full tubes and 2 rows of 7 half tubes. An inlet development region of 10 tube diameters and an outlet region of 15 tube diameters is simulated in the computational domain. The GOKSTAD experiment has a pitch-to-diameter ratio of 1.44 and tube diameter of 0.02 m. This corresponds to a total streamwise length of 35.08 tube diameters for the computational domain. The transverse and spanwise lengths of the computational domain are 8.64 and 10 tube diameters respectively. Figure 18 highlights the total computational domain with the streamwise, transverse, and spanwise dimensions.

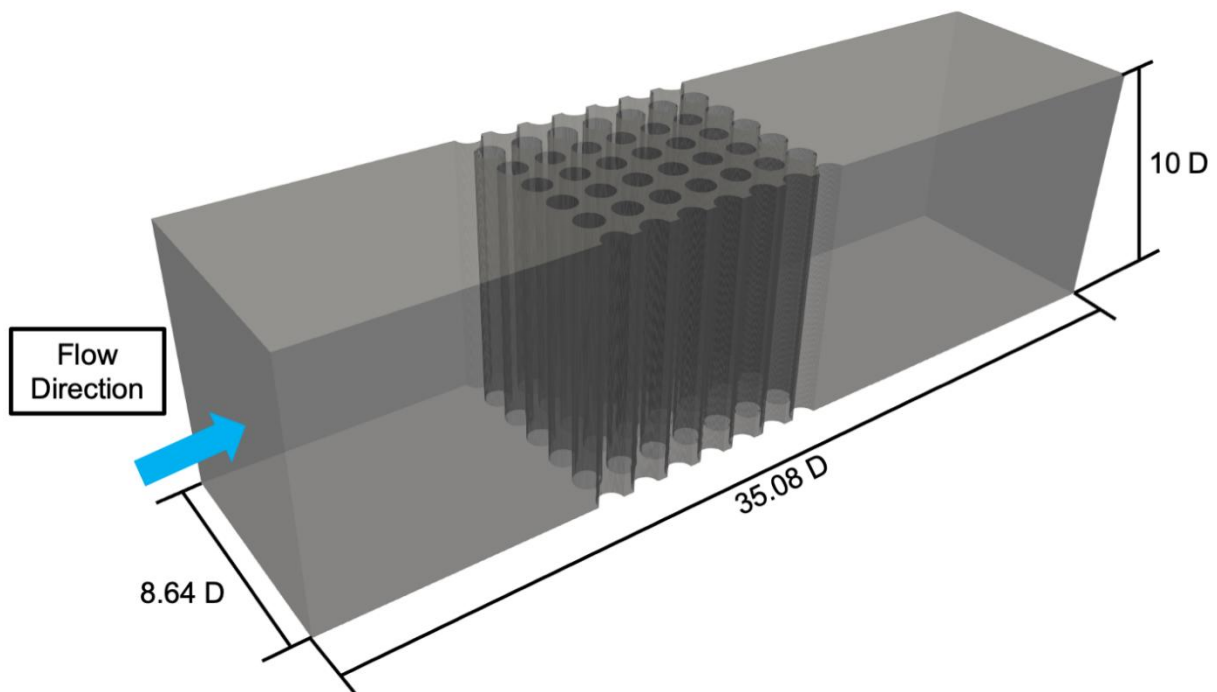


Figure 18: Schematic of the computational domain used to simulate the GOKSTAD experimental tube bundle in this work. Relevant dimensions and direction of flow are highlighted

The computational domain is nondimensionalized based on the hydraulic diameter of the tube gap ($D_h = 0.01686$ m). Table 12 shows the relevant dimensions of the computational domain both nondimensional and with dimensions. The simulations performed in this work aims to target the lowest flowrate measured in the GOKSTAD experimental tube bundle of $15.5 \text{ m}^3 \text{ h}^{-1}$. This corresponds to a bulk velocity of 0.125 m s^{-1} . The target gap Reynolds number for the lowest flow rate case is 6873.5.

Table 12: Measurements for different regions of the computational tube bundle domain.
Dimensions are given using tube diameter, nondimensional, and dimensional units.

	Tube Diameter	Nondimensional [-]	Dimensional [mm]
Tube Diameter (D)	D	1.186	20
Pitch	1.44D	1.708	28.8
Inlet Region	10D	11.862	200
Outlet Region	15D	17.794	300
Bundle Region	10.08D	11.957	201.6
Transverse	8.64D	10.249	172.8
Spanwise	10D	11.862	200

Figure 19 shows the boundary conditions that are applied to the computational domain. The inlet boundary condition uses recycling to ensure that the velocity field is fully developed before reaching the tube bundle. An average nondimensional velocity of 0.3556 is constrained to the inlet boundary determined from the bulk velocity mentioned before. The outlet boundary uses a Dong outflow boundary condition (Dong et al., 2014) to eliminate any backflow at the boundary. The remaining boundaries, including the tube walls and sides of the domain use a no slip boundary condition. It is important to state that nonslip boundary conditions are also used in-between the two rows of half tubes.

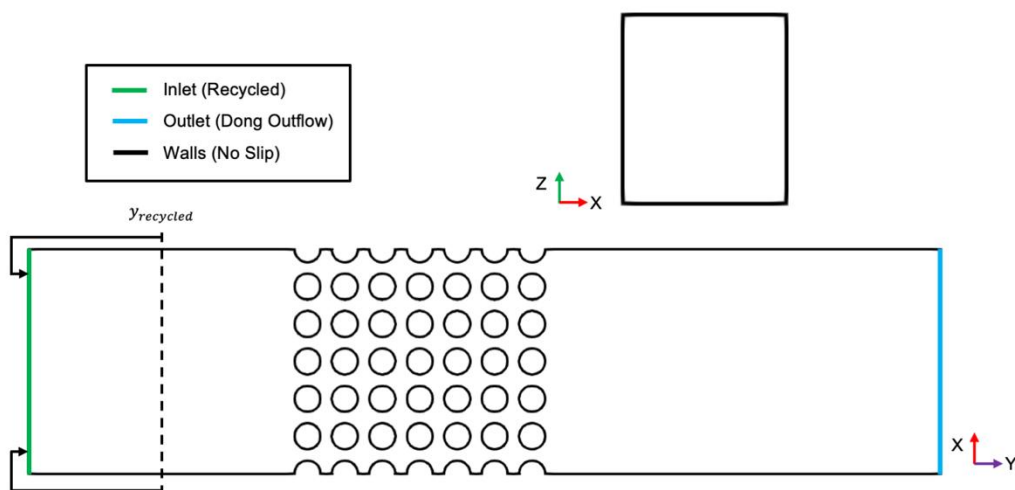


Figure 19: Schematic of the boundary conditions used in the simulation

The base mesh used in the computational domain was created in the open-source software GMSH (Geuzaine and Remacle, 2009). The mesh designed in GMSH uses second order

hexahedral elements. The base GMSH mesh consist of 1.02 Million elements. Nek5000 further discretizes the domain using the spectral element method (SEM). This work uses a 7th order polynomial which produces a total of 524 Million collocation points. Figure 20 shows the mesh produced from GMSH and from the further discretization that Nek5000 applies. Progressions are used towards all the walls in the domain to ensure that the boundary layer is fully resolved. A histogram of all the y^+ values in the domain is shown in Figure 21. The average y^+ in the domain is 0.85 with a maximum value of 6.08. Roughly 73% percent of all the y^+ values in the domain are less than 1 with 87% being less than 2.

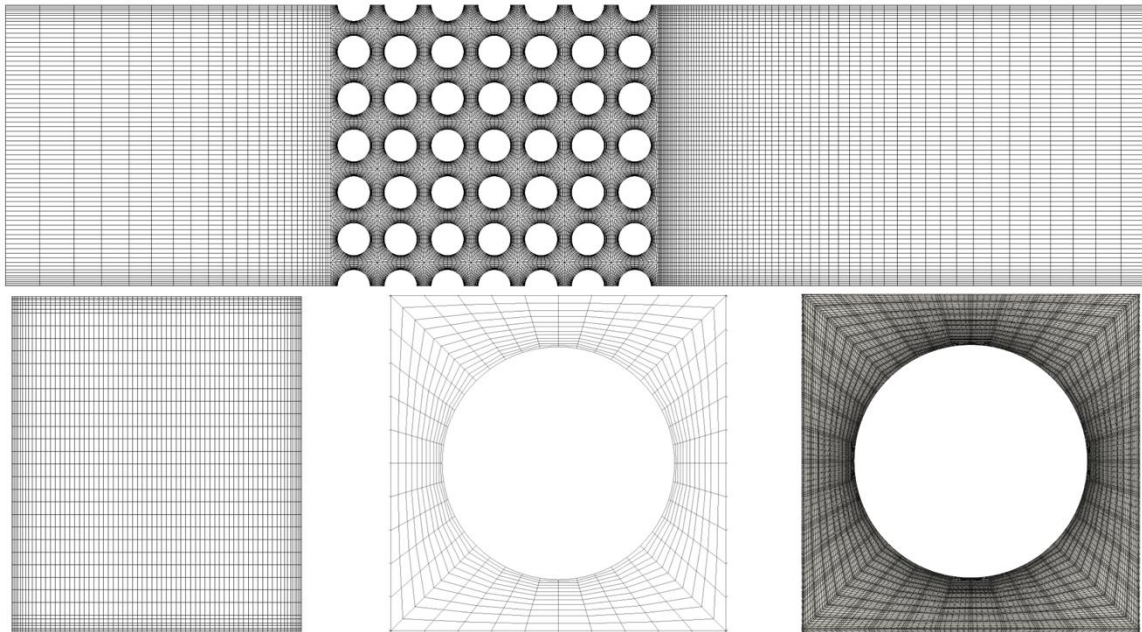


Figure 20: Spanwise (top) and streamwise (bottom left) slice of GMSH mesh. Zoomed in picture of the mesh around a tube from the GMSH mesh (bottom middle) and mesh from Nek5000 (bottom right).

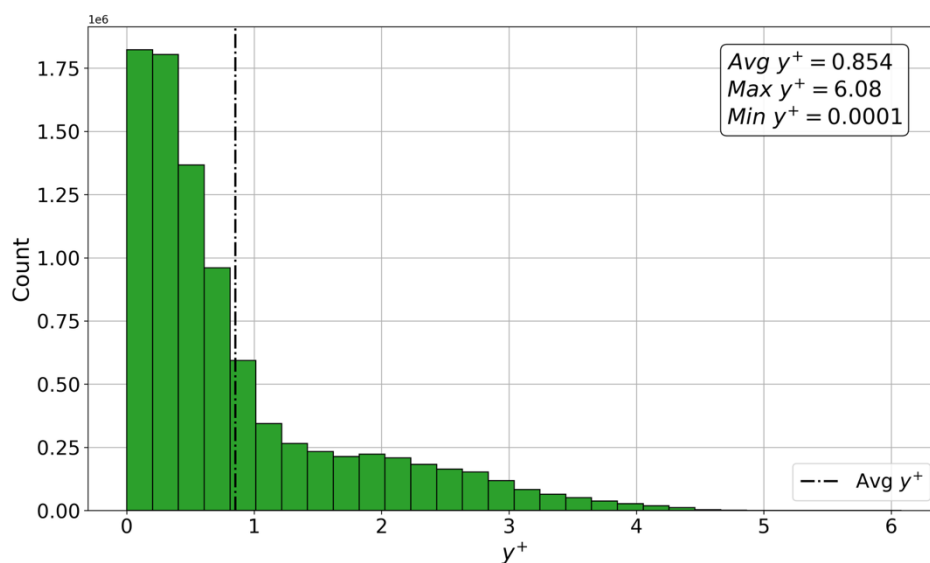


Figure 21: Y plus histogram for the simulation with average, maximum, and minimum values provided.

Prior to running the LES, a RANS simulation was run to determine the Taylor microscale. The Taylor microscale acts as an average of the dissipative scales at a location in the domain. Having a mesh size smaller than the Taylor microscale shows that the LES is in the dissipative region of the turbulent cascade. This metric provides a reasonable target for mesh sizing when conducting a LES (Novak et al., 2025). The RANS mesh used for estimating the Taylor microscale was created in GMSH and is similar to the LES mesh; however fewer elements were used. The base element count from GMSH for the RANS simulation was 251K elements. Nek5000 further discretizes the domain based on a 7th order polynomial resulting in a total of 128 Million collocation points. The $k - \tau$ RANS model was used in Nek5000 (Tomboulides et al., 2025).

The Taylor Microscale was computed using the simulated RANS results using Equation 1,

$$\lambda = \sqrt{10\nu\tau}, \quad (1)$$

where λ is the Taylor microscale, ν is the kinematic viscosity, and τ is the inverse of the turbulent dissipation rate ($1/\omega$). Figure 22 shows the comparison of the GLL spacing in the LES mesh and the Taylor microscale computed from the RANS results. Three different line locations were sampled and the GLL spacing and Taylor microscale were compared. The GLL spacing at the three separate line locations is smaller than the Taylor microscale. From this comparison, it can be concluded that the mesh used in the LES adequately resolves the Taylor microscale estimated from the RANS simulations.

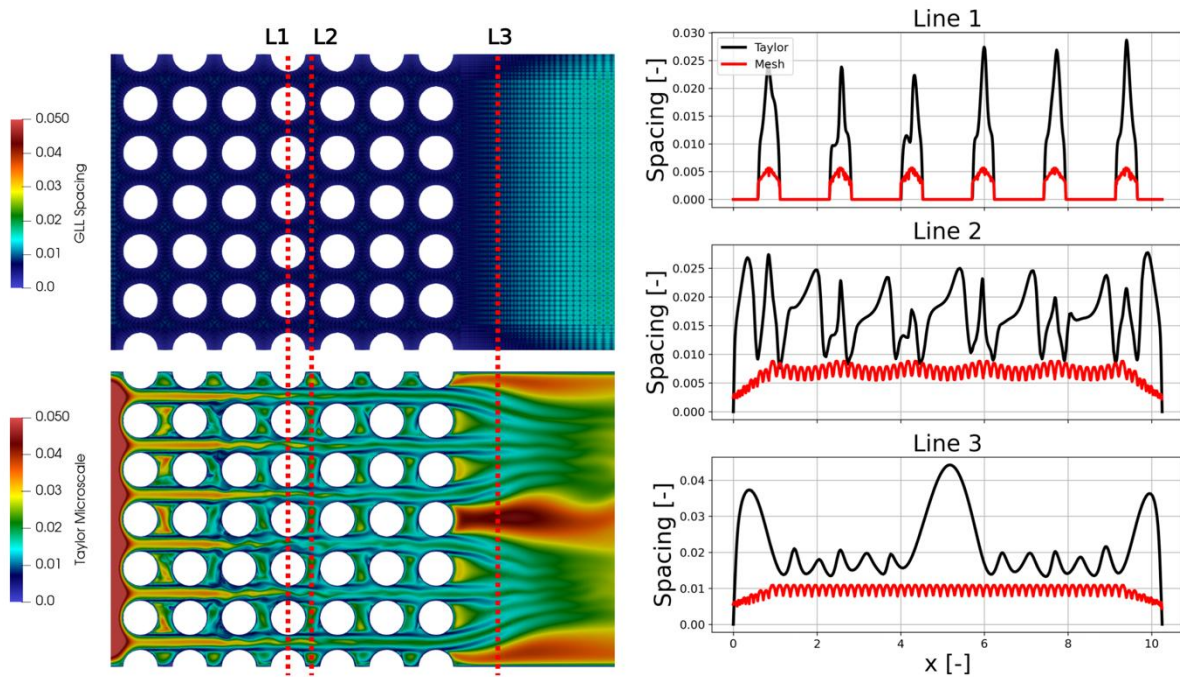


Figure 22: Contours of the LES GLL spacing (top left) and Taylor microscale (bottom left). The plots on the right show line comparisons of the GLL spacing and Taylor microscale at three different line locations depicted in the figure

To confirm turbulent behaviour for the LES of GOKSTAD, history points were used throughout the domain to capture the velocity signal and the turbulent spectra. Figure 23 shows the locations of the history points along with the velocity signal and turbulent spectra. The history

points are located around the middle tube on the first, middle, and last row of the tube bundle. The streamwise velocity signals shown in Figure 23 highlight high velocity regions as well as areas of recirculation in-between the tube rows. Finally, the turbulent spectra are obtained using the time series of the streamwise velocity. From the turbulent spectra, the fluid flow at the history point locations is deemed turbulent. Figure 24 shows an example of what the instantaneous and averaged velocity magnitude field look like throughout the domain.

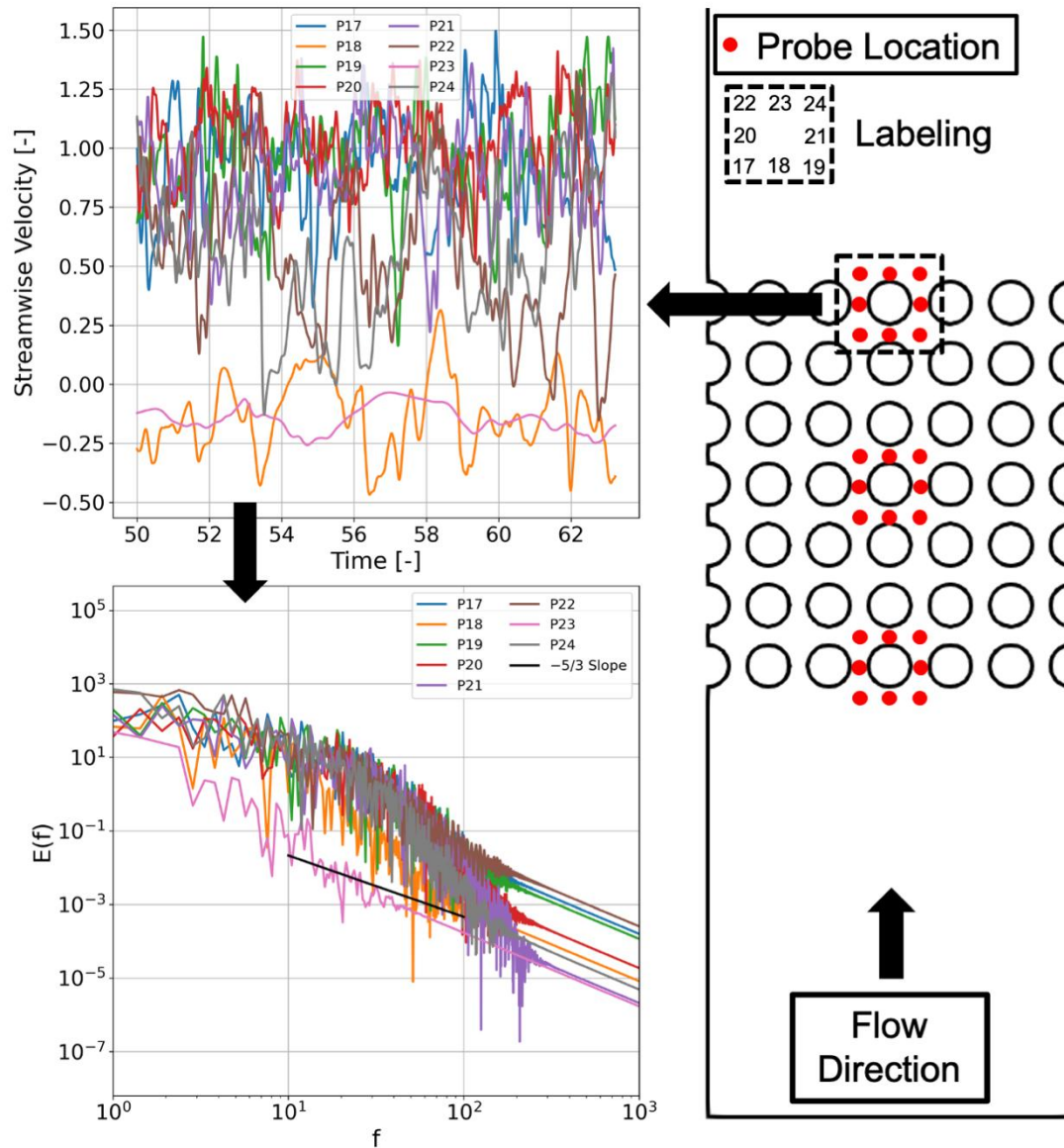


Figure 23: Location of history points in the bundle on the symmetry domain. The streamwise velocity signal are provided for the history points around the middle rod on the last row of tubes. The turbulent spectra is also shown for the velocity signal.

For benchmarking to the GOKSTAD experiment, different regions of the computational domain are extracted at the same locations as the data collected from the experiment. Furthermore, data lines are extracted to provide quantitative comparisons between the experiment and simulated results. Figure 25 provides the locations of the different regions and line extractions taken through both the experiment and simulated results. The v1, v2, and outlet regions are used to make qualitative comparisons using the velocity and turbulent

kinetic energy contour fields. Lines P1-P7 are used to extract the velocity and turbulent kinetic energy values within the tube bundle while Lines P8-P11 extract the data at the outlet region. Finally, the sampling locations are taken a planes $z=0$ (symmetry plane), 2 cm, 4 cm, and -4 cm. The outlet regions and lines (P8-P11) are only provided for the symmetry plane.

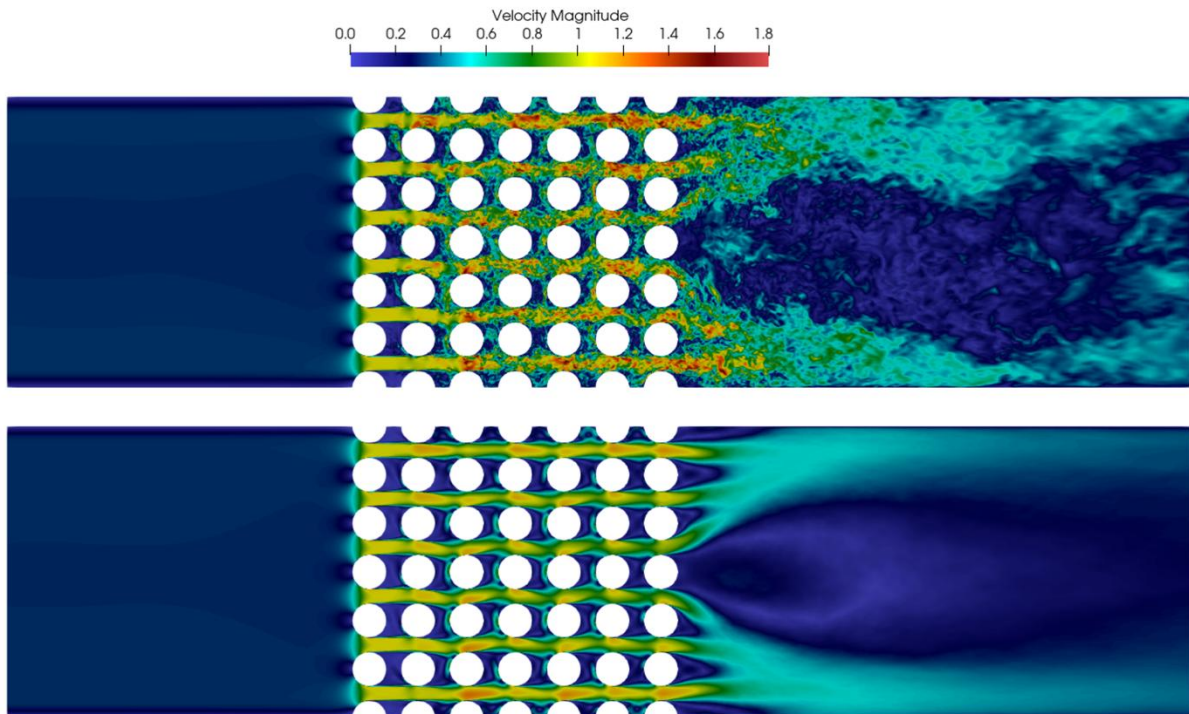


Figure 24: Instantaneous LES (top) and Averaged LES (bottom) velocity magnitude contour map.

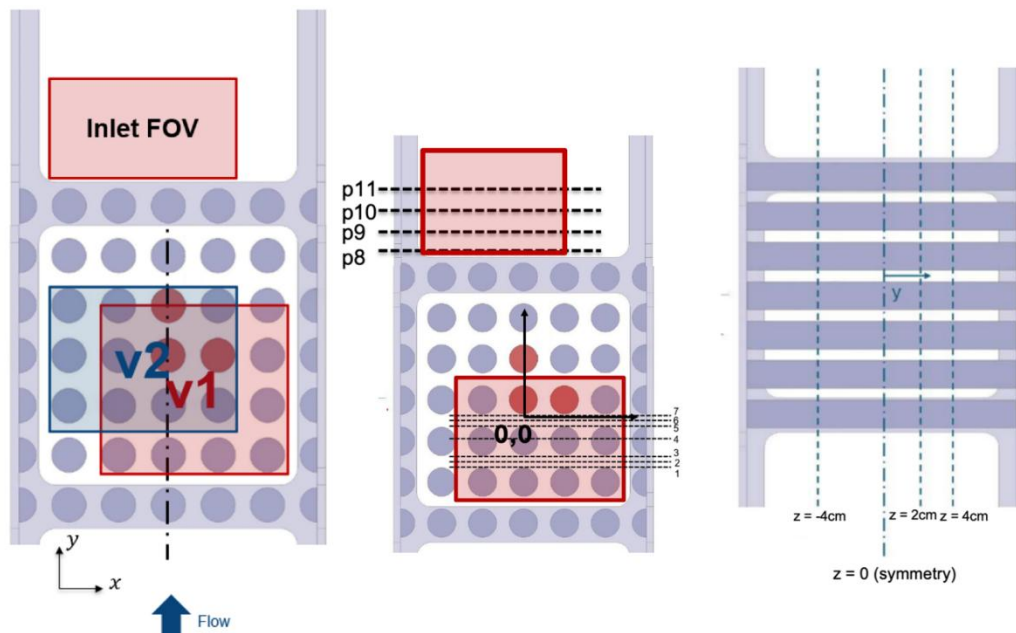


Figure 25: Schematic depicting the different data sampling regions and line locations taken throughout the tube bundle to be compared to against the experiment.

4.1.2 Numerical simulation approach

The CFD code used in this work is Nek5000. Nek5000 is a computational fluid dynamics (CFD) code developed at Argonne National Laboratory as part of the Nuclear Energy Advanced Modeling and Simulation (NEAMS) program (Nek5000, 2017). In this study, the case is nondimensionalized, and the non-dimensional Navier-Stokes equations are solved. The nondimensional Navier-Stokes equations are presented below in Equations 2 and Equation 3:

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

$$\nabla \cdot \mathbf{u}^* = 0. \quad (3)$$

Where $\mathbf{u}^*$ is the nondimensional velocity vector [-], t^* is nondimensional time [-], p^* is nondimensional pressure [-], Re is the Reynolds number, and $\mathbf{f}^*$ are nondimensional body forces [-]. The Reynolds number is defined in Equation 4.

$$Re = \frac{\rho UL}{\mu}. \quad (4)$$

Here ρ is the density [kg/m^3], U is the reference velocity [m/s], L is the reference length [m], and μ is the dynamic viscosity [$Pa \cdot s$]. In this work, a density of 1000 kg/m^3 , and a dynamic viscosity of $0.001 \text{ Pa} \cdot s$ was used. Finally, the relationship between the nondimensional variables and their dimensional counterparts are provided in the Equation 5.

$$x^* = \frac{x}{L}, \quad u^* = \frac{u}{U}, \quad t^* = \frac{tU}{L}, \quad f^* = \frac{fL}{U^2}, \quad p^* = \frac{p}{\rho U^2}. \quad (5)$$

Nek5000 uses the spectral element method and can perform Reynolds Averaged Navier Stokes (RANS) simulations, Large Eddy Simulations (LES), and Direct Numerical Simulations (DNS). Nek5000 uses a mixed implicit/explicit method. The diffusion, time integration, and linearized source terms uses an implicit backward difference formula, while the advection and forcing terms uses an Adams-Bashforth method. The LES capabilities of Nek5000 were leveraged in this work. The explicit low pass filter is used to provide an energy drain at the unresolved grid scale. No additional sub grid scale model is included.

A filter weight of 5% is used over 2 filter modes. The residuals were converged below 10^{-4} for pressure and 10^{-6} for velocity. For the LES, an adaptive time-stepper was used before averaging to achieve fully developed flow using a target CFL number of 0.5. Once averaging began, a time step of 2.5×10^{-4} was used resulting in a CFL of 0.36. Finally, the results were averaged over roughly 9 flow-through times.

A one-way fluid-structure interaction approach was used to estimate the dynamic response of the center tube subjected to fluid forces obtained from the LES. The structural response was modelled as a linear single degree of freedom mass-spring-damper system in both the streamwise and transverse directions. The governing equation of motion is given in Equation 6 and 7,

$$m\ddot{x}(t) + c\dot{x}(t) + kx(t) = F_x(t). \quad (6)$$

$$m\ddot{y}(t) + c\dot{y}(t) + ky(t) = F_y(t). \quad (7)$$

Where m is the effective mass, c is the damping coefficient, k is the stiffness, and $F_x(t)$ and $F_y(t)$ are the time-dependent fluid forces obtained from LES in the respective direction. In this work, $m = 0.1 \text{ kg}$, $c = 1.14 \text{ n} \cdot \text{s}/\text{m}$, $k = 7000 \text{ N}/\text{m}$.

The fluid force time series was imported from LES output files. To remove startup transients, the signal was truncated beginning at a specified time $t = t_0$, chosen as 5 seconds. The force signals $F_x(t)$ and $F_y(t)$ were interpolated using linear interpolation to allow for time integration. The second-order equations of motion were reconstructed as a system of first-order ordinary differential equations:

$$\frac{dx}{dt} = v \quad (8)$$

$$\frac{dy}{dt} = v \quad (9)$$

$$\frac{dv}{dt} = \frac{F_x(t) - cv - kx}{m} \quad (10)$$

$$\frac{dv}{dt} = \frac{F_y(t) - cv - ky}{m} \quad (11)$$

This system was solved independently in the x and y direction using the SciPy module in python. An Explicit Runge-Kutta method was used for the numerical integration. The initial conditions for the system were specified as 0 displacement and 0 velocity.

4.2 Flow-only simulation results for 0.125 m/s bulk velocity

The resulting instantaneous flow field for the LES of the GOKSTAD bundle are shown in Figure 26. Upstream of the bundle, the flow remains largely uniform; however, as it interacts with the tubes, shear layers develop and begin shedding vortices. Within and immediately downstream of the bundle, jet like flows form between the gaps of tubes, along with areas of low velocity recirculation flow. Further downstream, near the outlet of the bundle, these flow structures merge into a broad turbulent wake. A large recirculation region is observed between the high-velocity outlet jets from the tube bundle. In the outlet region, large scale mixing occurs as the unsteady flow exits the bundle.

Figure 27 shows the average velocity obtained from the experiment and the LES at view 1 (v1) and view 2 (v2). There is strong qualitative agreement between the experiments and simulations in the overall flow behaviour through the tube bundle. In both cases, high velocity jet-like structures form in the gap with similarity both spatially and in magnitude. Additionally, the averaged flow field for both results show similar recirculation zones in the wakes of the cylinders. However, some noticeable differences occur between the resulting datasets. The primary difference lies in the flow detachment behaviour from the jet-like structures in the downstream rows of tubes (i.e., the last rows in the positive y direction). In the simulation results, the flow detaches from the jets and oscillates between neighboring high-velocity streams, indicating a symmetric switching behavior. In the experimental results, this oscillatory behaviour is observed in the first two rows; however, in the downstream rows, the flow preferentially separates in a single direction, suggesting a more asymmetric pattern.

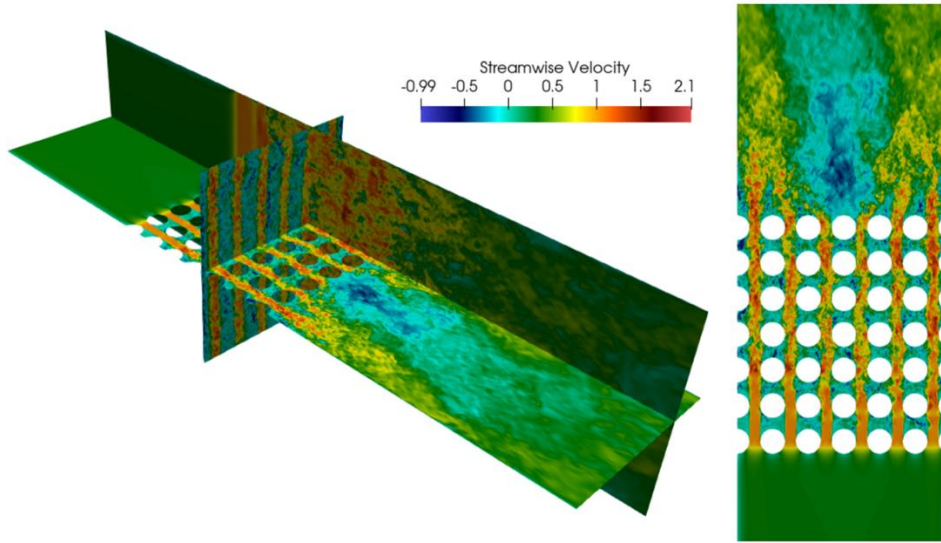


Figure 26: Instantaneous streamwise velocity shown as an isotropic view with streamwise, transverse and spanwise planes (left) and normal to the spanwise direction with a spanwise slice (right).

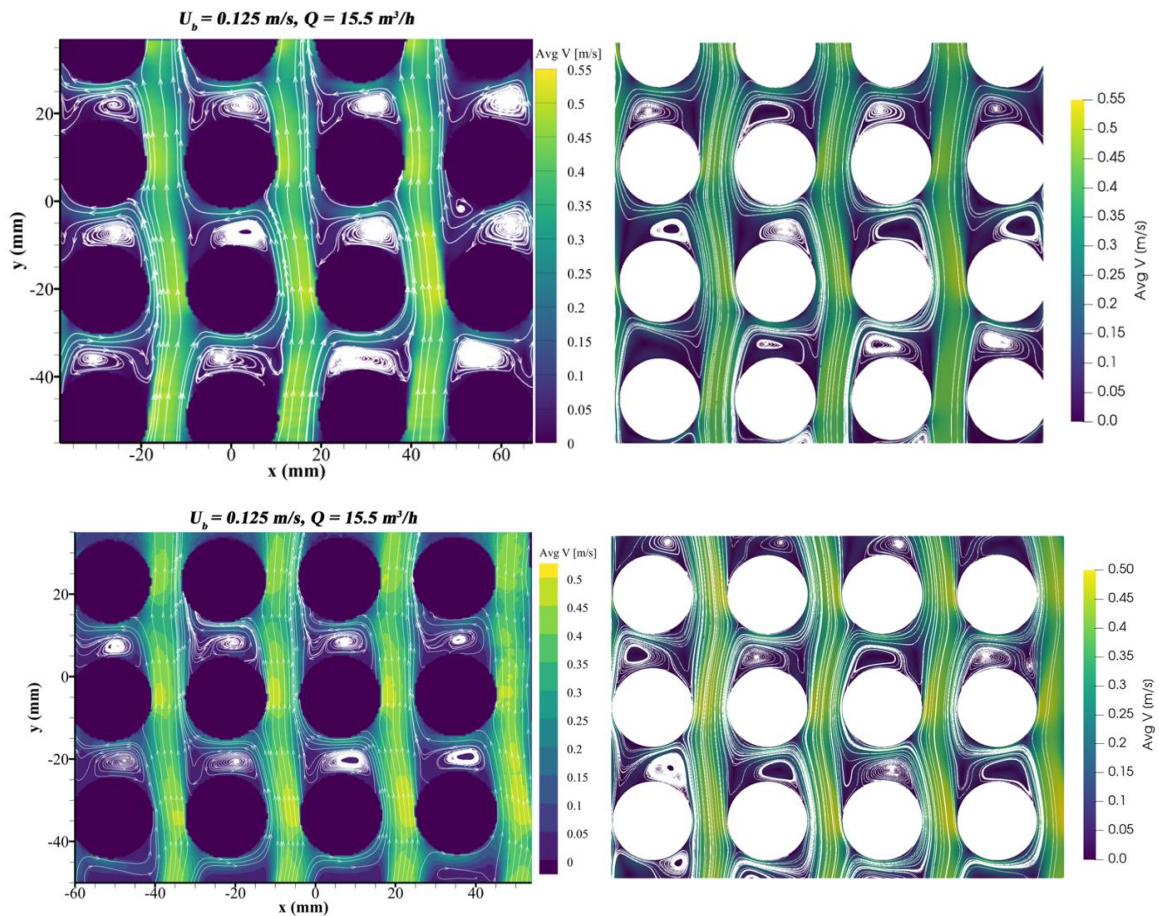


Figure 27: Average velocity contour fields with streamlines for both the experiment and the LES results. Top two colormaps show v_1 and the bottom two show v_2 .

Similar to the average velocity contour plots, qualitative comparisons of the Turbulent Kinetic Energy (TKE) between the simulations and the experiments are done. Figure 28 shows the time averaged TKE from V1. Both TKE contour plots show high magnitude peaks forming off the tube walls in the shear layer coming off the tube. These high TKE regions are likely due to flow accelerating through the tube gaps, forming high velocity jet-like flow structures. The LES captures the spatial distribution for TKE consistently for the first two rows of tubes. However, disagreements between the TKE profile after the last row of tubes are present. The experimental findings show a more diffusive and asymmetric profile when compared to the LES. Finally, the LES results overpredict the magnitude of TKE throughout the domain.

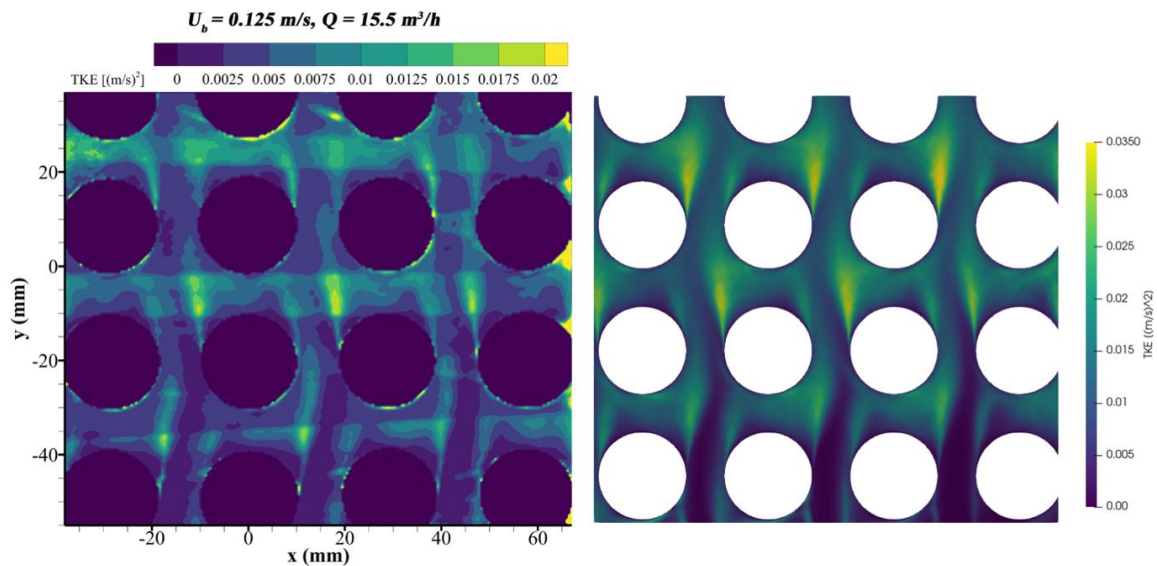


Figure 28: Average TKE contour fields for both the experiment and the LES results from V1. Flow from bottom to top.

Along with qualitative comparisons within the bundle, further comparisons can be made about the velocity and TKE at the outlet of the bundle. Figure 29 and Figure 30 show the velocity and TKE for the experiment and LES. For the average velocity both results show the presence of recirculation zones formed in the wake of the bundle. These recirculation zones are located in the mid-transverse region of the domain and are flanked on both sides by high velocity channels as the jet-like structures collide near the edges. The experimental and LES results generally show good agreement spatially and in magnitude. The location of the colliding jet-like structures and recirculation area are accurately predicted. However, the experimental results show that the two recirculation bubbles have slightly asymmetrical locations, whereas the recirculation bubbles in the LES show a more symmetrical behaviour.

The TKE at the outlet of the bundle show high magnitude in the area between the high velocity regions and the lower velocity recirculation regions. These are present near the outlet of the tube bundle, where the jet-like structures collide and recirculation pockets form between the jets. The high TKE regions are also present more downstream of the bundle, between the recirculation region in the middle of the domain and the high velocity region on the edges. Similar to the average velocity, there is good spatial agreement of the TKE profiles between the experiment and LES results. However, the LES overpredicts the magnitude of the TKE observed in the experiment. Furthermore, a larger averaging window is required for the LES to smoothen out the instantaneous effects.

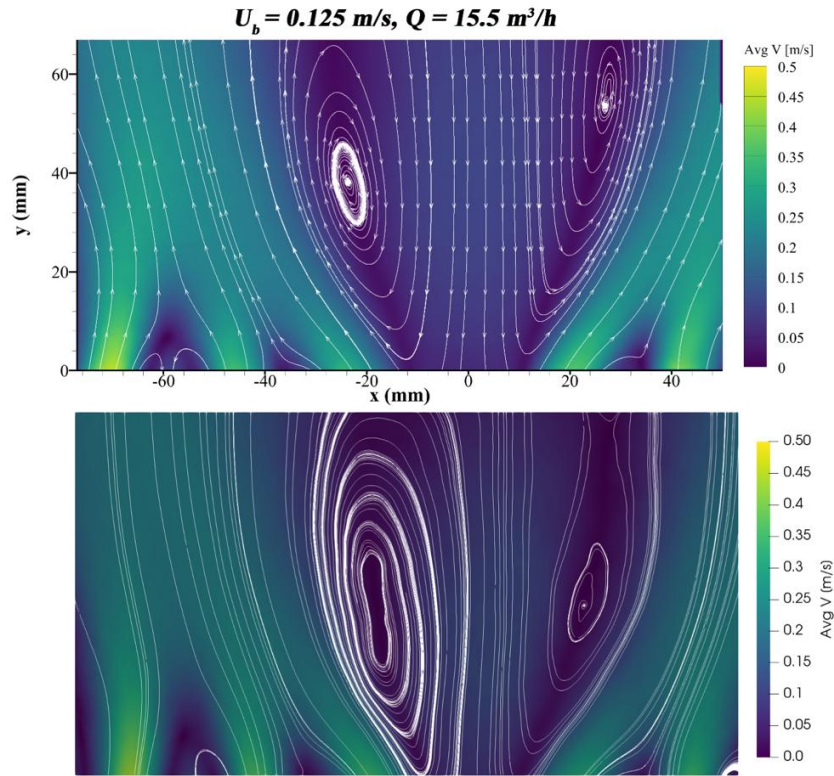


Figure 29: Average velocity contour plots at the outlet of the bundle for experiments (top) and LES (bottom).

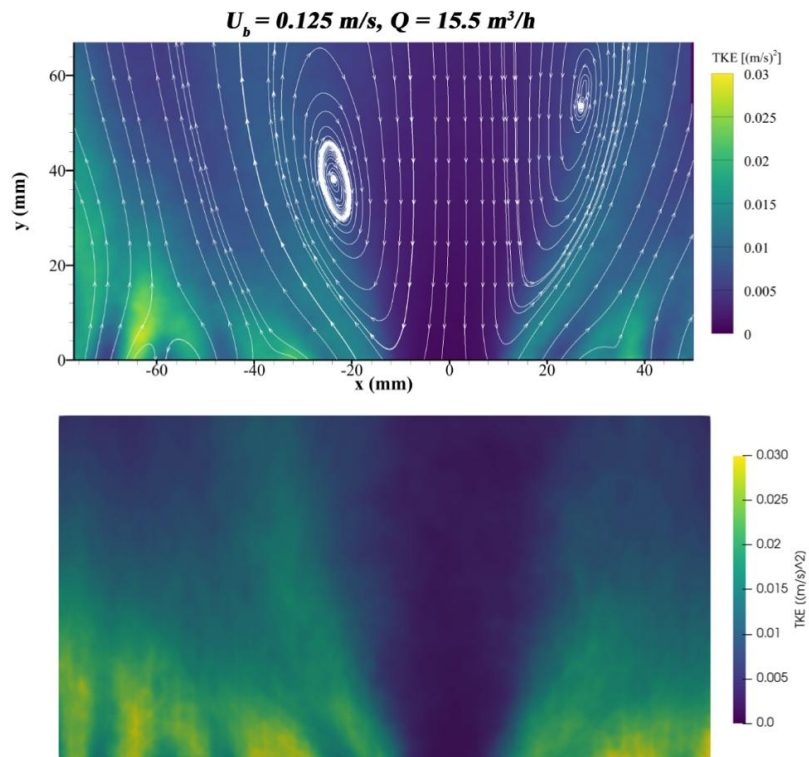


Figure 30: TKE contour plots at the outlet of the tube bundle for experiments (top) and LES (bottom).

Along with the qualitative comparisons using the inlet, v_1 , v_2 , and the outlet, quantitative comparisons can be made by extracting line data at different locations in the domain. Figure 31 shows the time averaged velocity and normalized TKE at the inlet plane. Three separate z planes are compared; $z = 0.0$ m (symmetry), $z = 0.3$ m (midplane), and $z = 0.65$ m (near wall). Figure 32 - Figure 35 show the line plot data for average velocity and TKE within the bundle region. These locations are taken at lines 1-7 in the domain at the $z = 0$ cm (Figure 32), $+2$ cm (Figure 33), $+4$ cm (Figure 34), and -4 cm (Figure 35) plane. Finally, Figure 36 shows average velocity and TKE at four separate lines at the outlet of the bundle on the symmetry plane.

There is good agreement between the LES and experimental results for all the lines and planes in the bundle when comparing average velocity. The spatial profile and magnitudes are accurately captured. Slight differences between LES and the experiment appear in the low velocity recirculation areas on Lines 1 and 5 as well as parts of the profile on Lines 3 and 7. These locations are captured directly off the tube wall. The LES results tend to slightly overpredict the velocity magnitude in this area. Similar to the average velocity, the TKE profile from the LES follows a similar spatial profile to the experimental results. However, LES overpredicts the TKE throughout the geometry. Overall, the LES captures the spatial distribution and magnitude of the mean velocity with good agreement to experiments, but tends to overpredict turbulent kinetic energy.

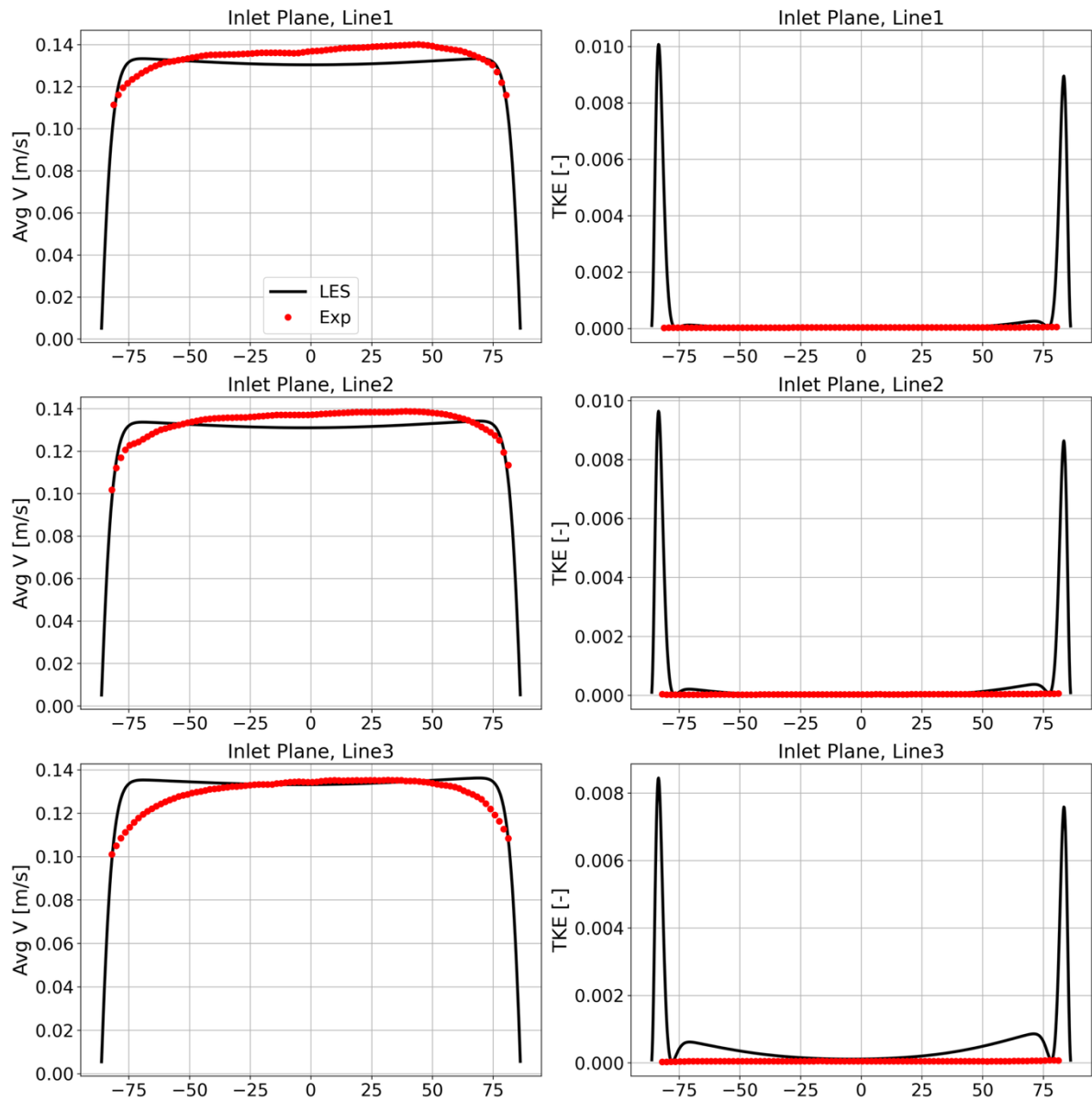


Figure 31: Average velocity and normalized TKE for the LES and experimental results at the inlet of the bundle. Line 1, 2, and 3 correspond to $z=0.00$ m, 0.03 m, 0.065 m respectively

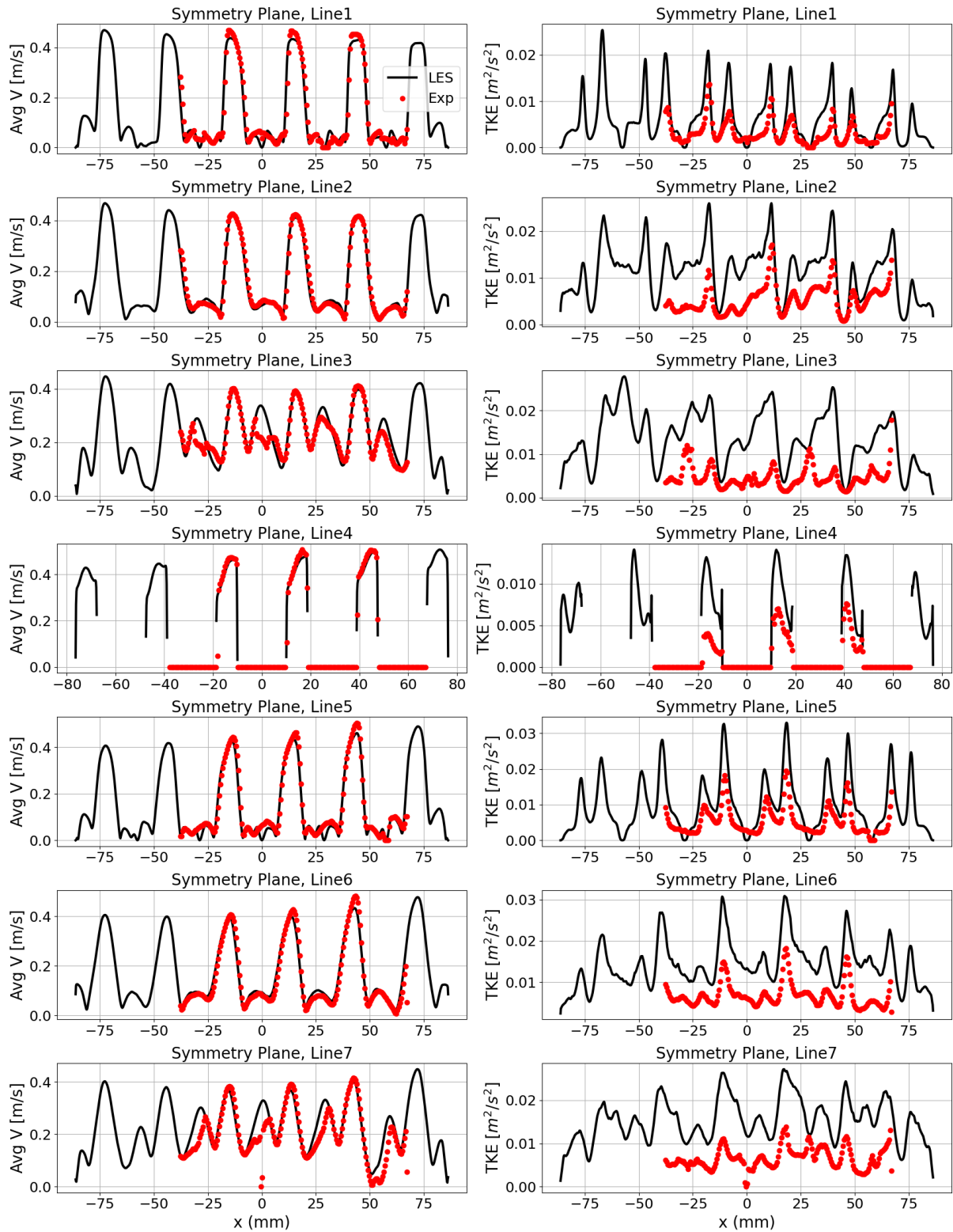


Figure 32: Average velocity and TKE for Lines 1-7 on the symmetry plane (z=0).

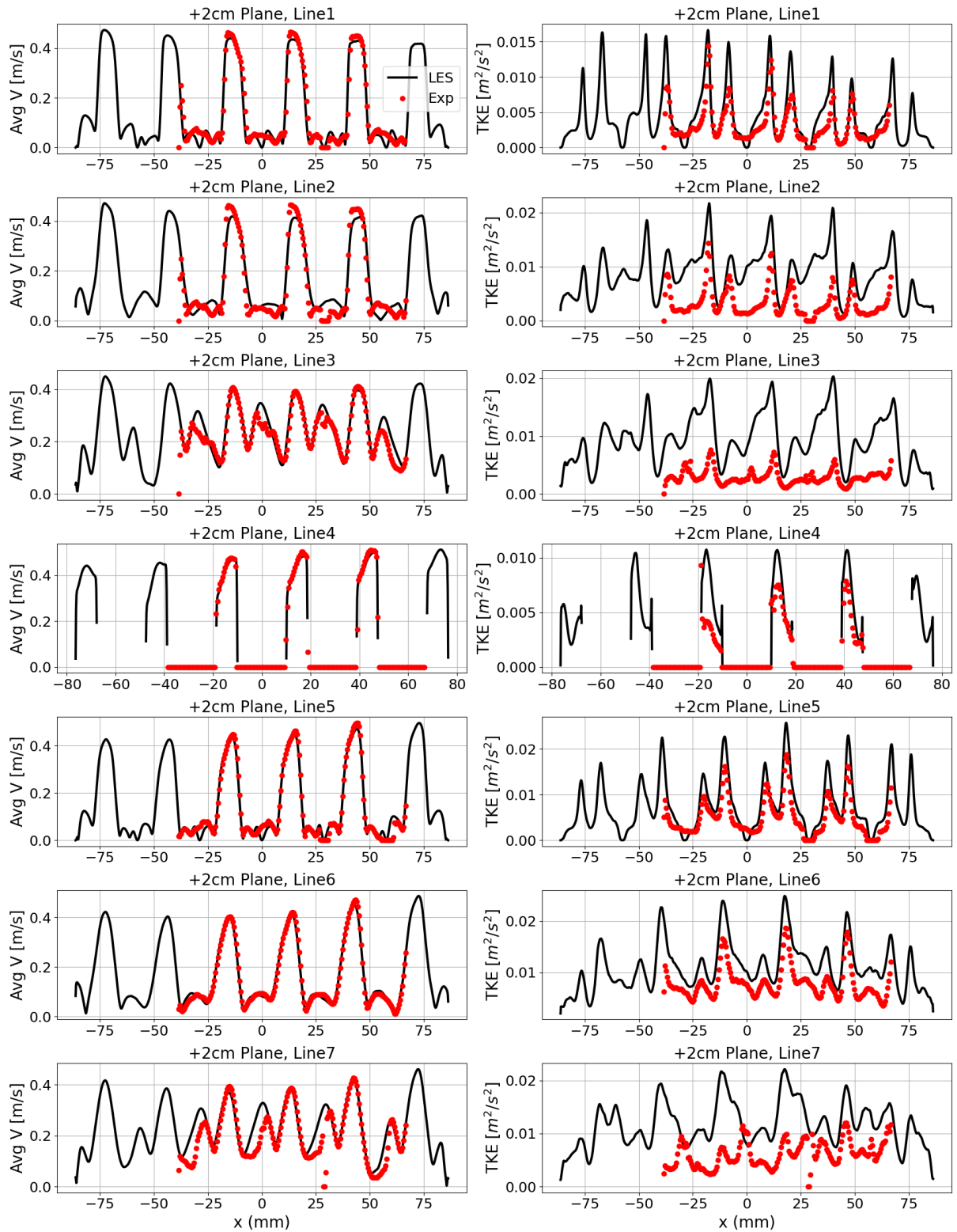


Figure 33: Average velocity and TKE for Lines 1-7 on the z=+2 cm plane.

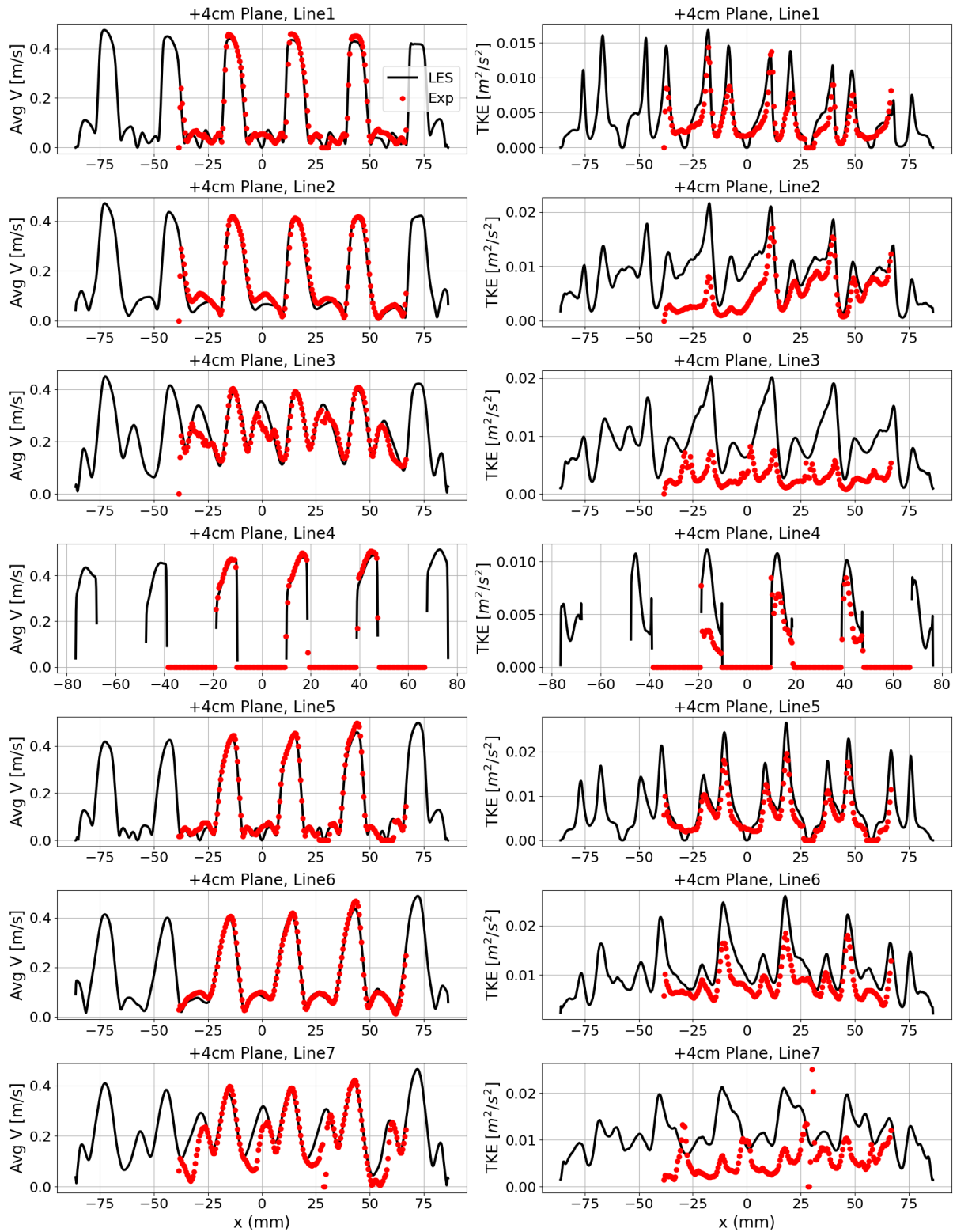


Figure 34: Average Velocity and TKE for lines 1-7 on the +4 cm plane.

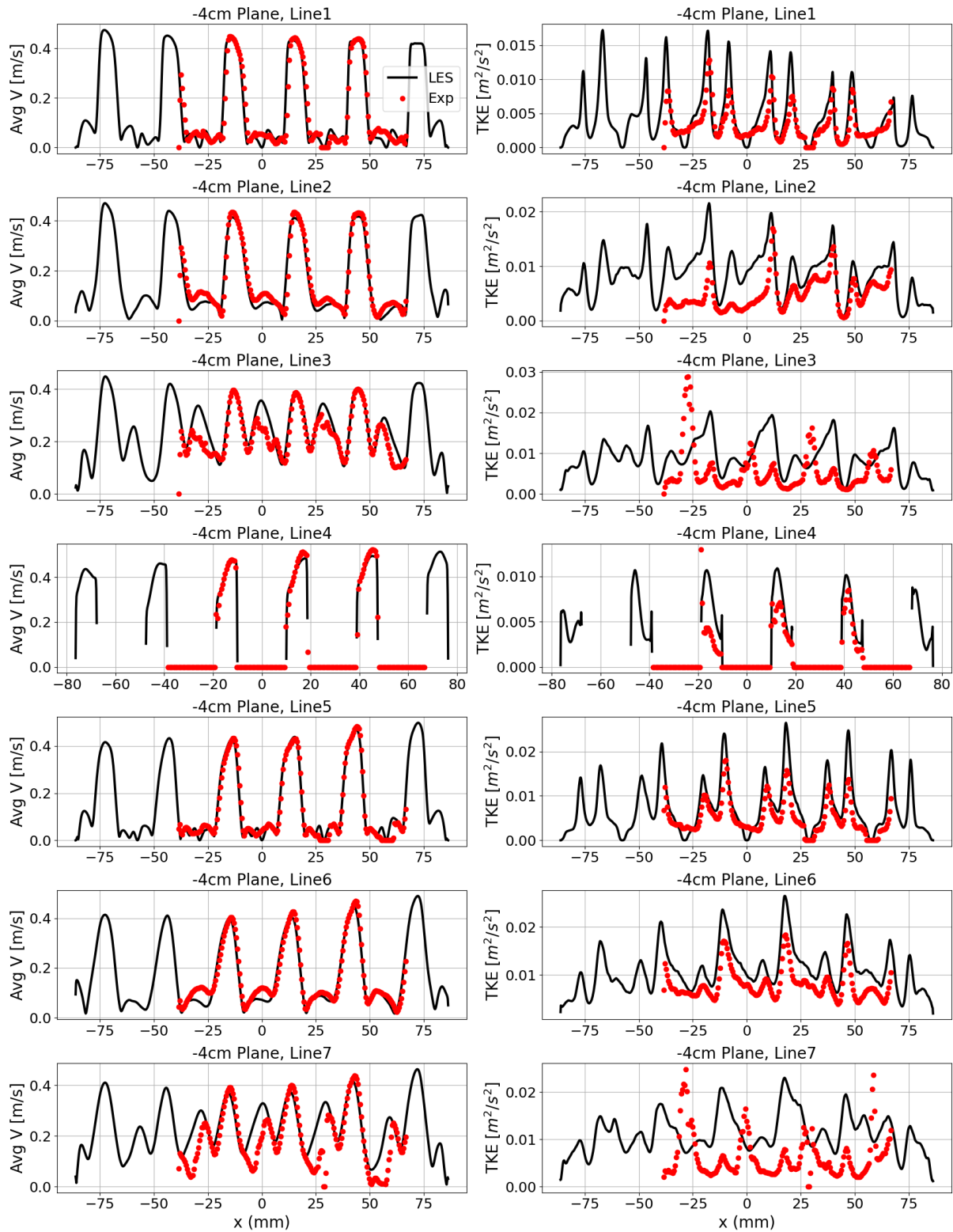


Figure 35: Average Velocity and TKE for lines 1-7 on the -4 cm plane.

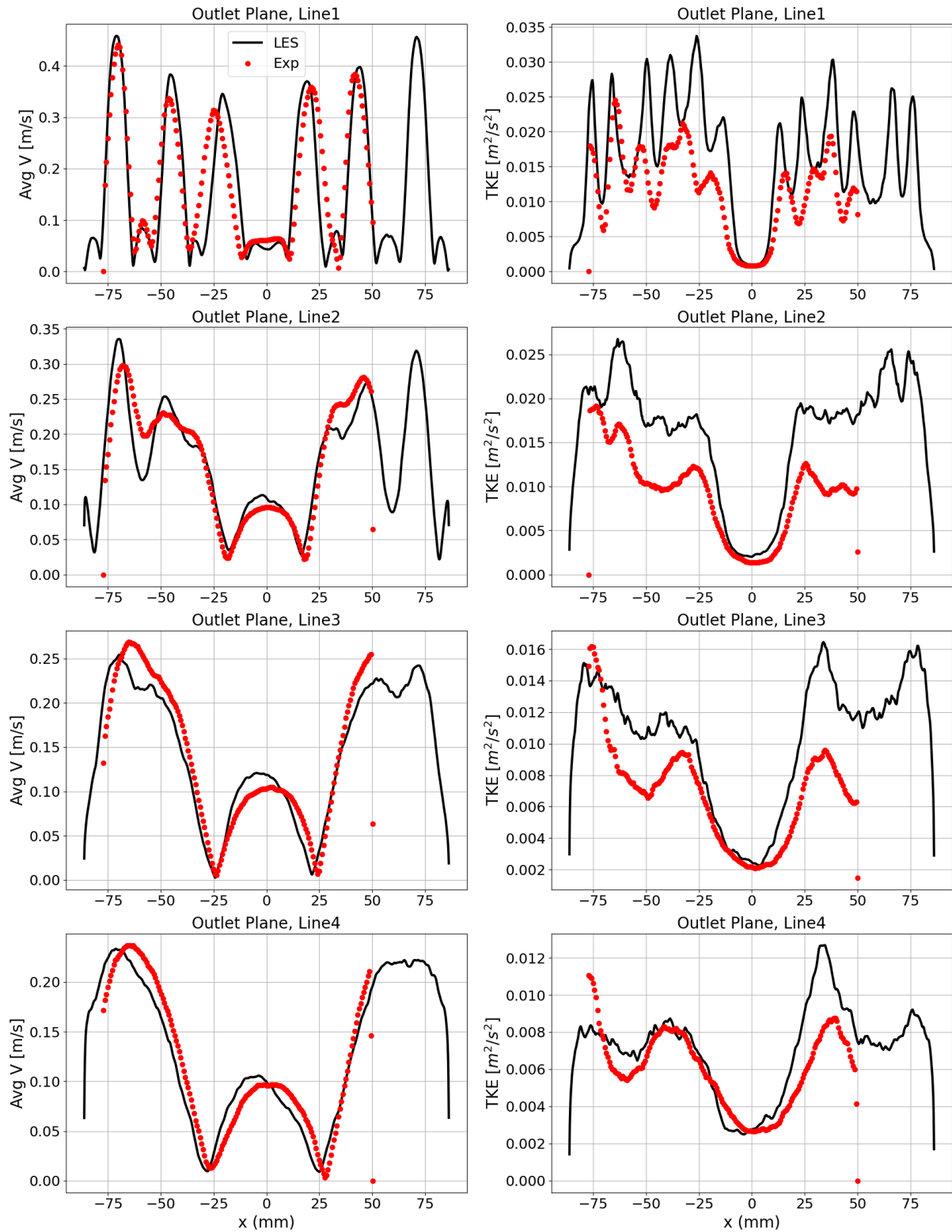


Figure 36: Average velocity and TKE for lines 8-11 (outlet) on the symmetry plane (z=0)

4.3 FSI simulation results for 0.125 m/s bulk velocity

The drag and lift force time histories corresponding to the fluid loading on the center tube within the bundle are shown in Figure 37. Both signals show an initial transient period, followed by a statistically stationary regime. The drag force rapidly increases from near zero and stabilizes around a mean value with relatively small fluctuations. In contrast, the lift force exhibits stronger oscillatory behavior with fluctuations about a smaller mean value, reflecting the influence of unsteady vortex shedding within the bundle. These force signals provide the input for the one-way coupled structural analysis.

Using the computed fluid forces, a one-way coupled mass–spring model was solved to estimate the structural response of the center tube. The displacement solution was initialized at 5 seconds, to avoid initial transient effects. The resulting displacement signals in both the streamwise (Y) and transverse (X) directions show small-amplitude oscillations about a mean position highlighted in Figure 38. The transverse displacement remains relatively low when compared to the displacement in the drag direction. Table 13 provides a statistical description of the displacement for both the experiment and the one-way coupled LES. There is reasonable agreement between the mean displacements from LES and the experiment; however, LES is underpredicting the RMS from the experiment.

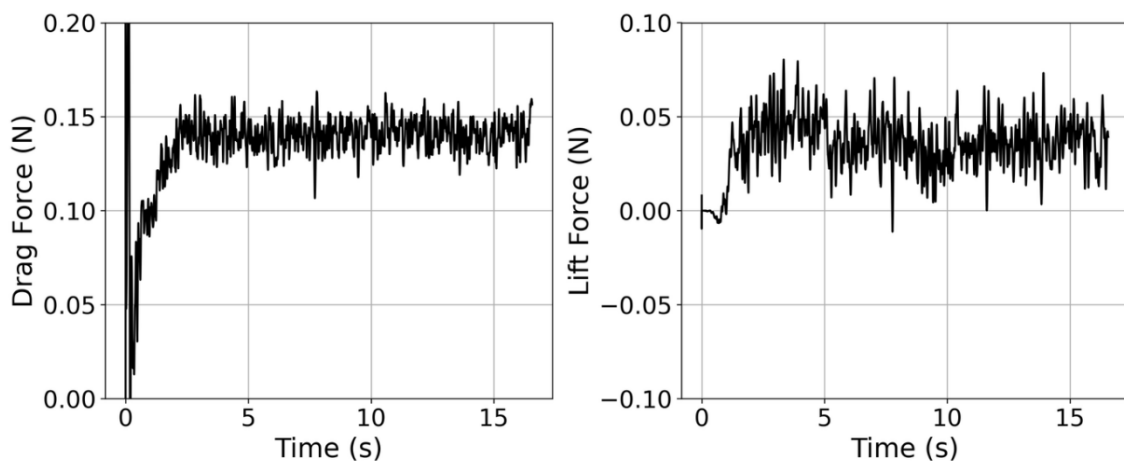


Figure 37: Drag (left) and Lift (right) force history from LES on the center tube of the bundle.

Table 13: Mean and RMS displacement of the center tube in the Lift (x) and Drag (y) direction between LES and the experiment.

	Mean X Displacement (mm)	Mean Y Displacement (mm)	RMS X (mm)	RMS Y (mm)
Experiment	0.006430	0.02566	0.008323	0.008503
LES	0.005064	0.02014	0.001929	0.001366

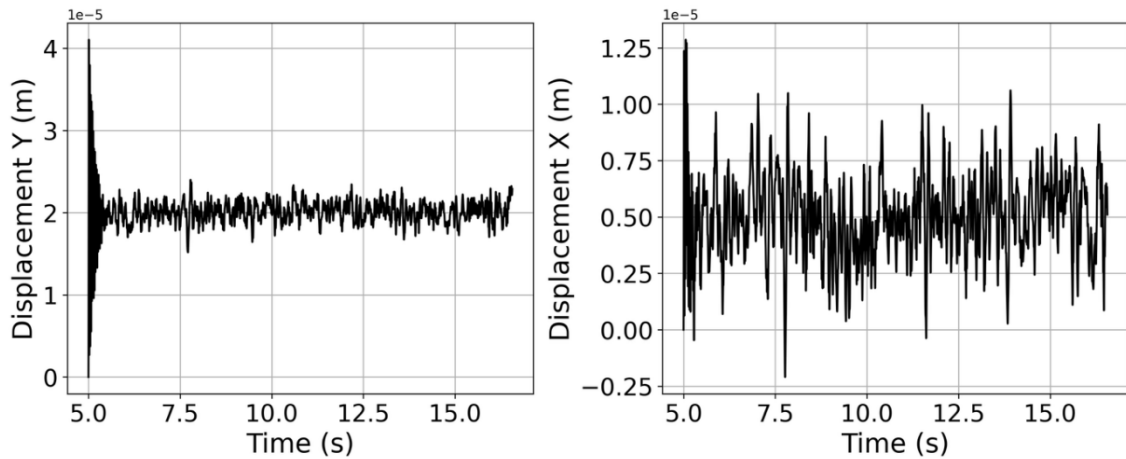


Figure 38: Displacement in the X and Y direction using a one-way coupled spring mass solve from the time series force data in the LES

4.4 Conclusions

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

5. Simulations performed by EDF

5.1 Method description

The open source CFD solver code_saturne v8.0 (www.code-saturne.org) developed by EDF has been utilized using massive parallel computing. Both supercomputers Cronos and Selena have been used. All the meshes are fully hexahedral and conformal and have been generated with Gmsh v4.7.1. Two turbulence models have been tested, the eddy viscosity model $k-\omega$ SST and the Reynolds stress model EB-RSM (Elliptic-Blending Reynolds Stress Model). All the walls are fully resolved and the inlet boundary conditions are fully developed, using an embedded 2D computation using the appropriate turbulence model. The same meshing strategy is used as in the AMOVI simulations performed by EDF, see Chapter 4 of GO-VIKING deliverable D3.2 (Filonov et al., 2026). The meshes adapt to the Reynolds number. The inlet and outlet are at a distance equal to 10D and 50D from the first and last rows of cylinders, respectively, with D the tube diameter.

The two RANS models are utilized in a URANS mode keeping the unsteady term in the transport equations. The time advancing scheme is an implicit Euler one. The convection scheme of the momentum equation is a SOLU (Second Order Linear Upwind) whereas a 1st order upwind is used for the turbulent quantities.

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

$$M\ddot{y}_i + C\dot{y}_i + Ky_i = f_y. \quad (12)$$

The structure coefficients are the ones provided by VKI and are set to $M = 0.1 \text{ kg}$, $C = 1.14 \text{ N.s.m}^{-1}$ and $K = 7000 \text{ N.m}^{-1}$.

Two main configurations are simulated, and of which the results are presented here, each at two bulk velocities:

- In-line tube bundle with rigid cylinders at the bulk velocities 0.5 m/s and 1 m/s.
- In-line tube bundle with a single moving cylinder at 0.125 m/s and 0.25 m/s.

An error has been introduced since the beginning of the simulations, and which was only discovered while post-processing the results. In the computations, the pitch $P = 0.0282$ and not 0.0288 as in the experiment. This results in $P/D = 1.41$ and not 1.44. This unfortunate error should not have dramatic effects, but has to be kept in mind. The profiles have been positioned according to this error.

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

5.2 Rigid tube bundle

5.2.1 Fluid flow prediction at a bulk velocity of 0.5 m/s

Two computations are presented in the present section on two different meshes, Mesh 0 and Mesh 1, containing 30 M and 87 M computational cells, respectively. Mesh 0 has 116 and 226 nodes in the azimuthal and z directions, respectively and the progression ratio for the refinement in the wall-normal direction is equal to 1.25. Mesh 1 has 140 and 400 nodes in the azimuthal and z directions, respectively and the progression ratio for the refinement in the wall-normal direction is equal to 1.2. Mesh 0 is fine enough for a URANS computation. Mesh 1 was initially introduced to perform a mesh refinement study, which unfortunately could not be concluded due to a lack of time. The $k-\omega$ SST and the EB-RSM are run using Mesh 1 and Mesh 0, respectively. The inlet boundary conditions are fully developed.

Figure 39 provides the imposed velocity and TKE profiles along a line compared to the experimental data. The experimental velocity profile is flatter than in the simulations in which a fully developed flow is imposed at the inlet. The numerically imposed turbulent kinetic energy is much higher in the simulations close to the walls, but the values in the centre are in reasonable agreement with the experiment. The time step is equal to 0.0001 s in both cases which leads to a time averaged maximum CFL number of 2.3 and 2.1. The time average of the maximum non-dimensional distance to the wall is equal to 1.1 and 1.7, respectively.

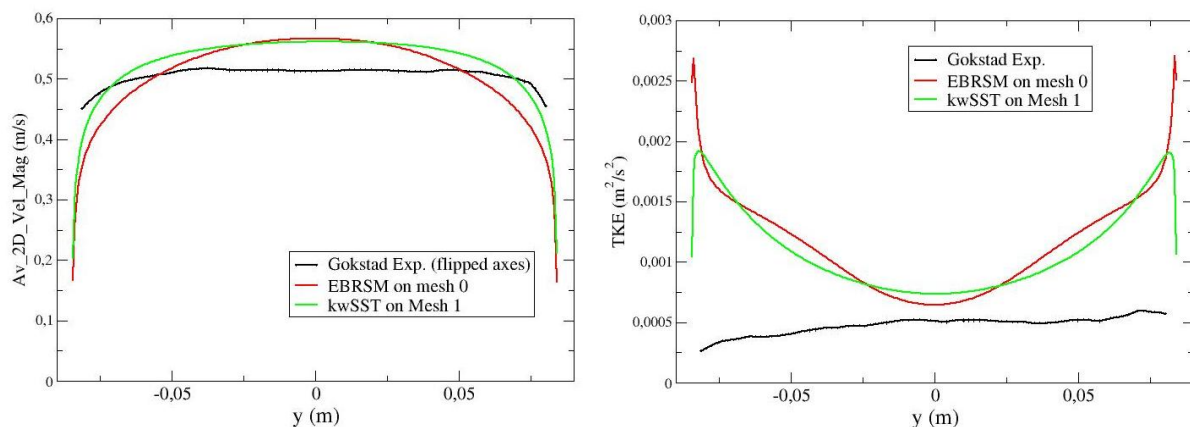


Figure 39: Mean stream-wise velocity and turbulent kinetic energy (TKE) at the inlet with the two turbulence models

Figure 40 shows an instantaneous snapshot of the non-dimensional distance to the wall for both computations. The maximum in these figures is lower than the one given above and corresponds to regions where the acceleration of the flow is high. Figure 41 shows an instantaneous snapshot of the CFL number in the centre plane. It doesn't exceed 1 in the present figure and the maximum values correspond again to the acceleration regions in between two tubes.

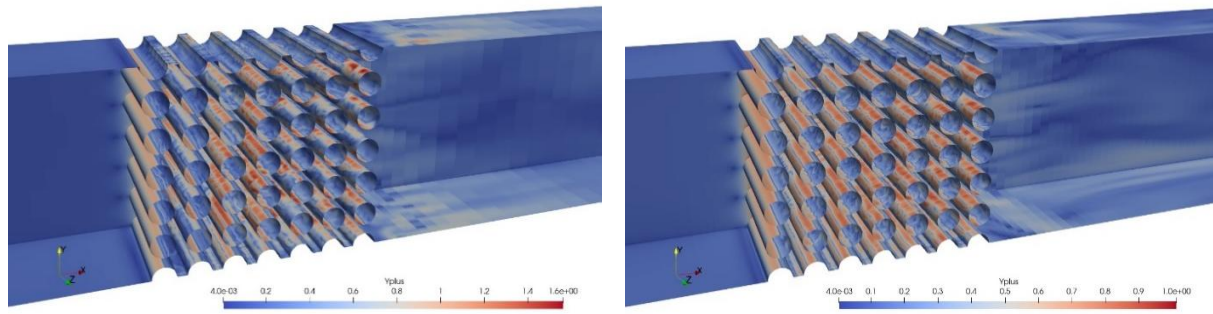


Figure 40: Instantaneous non-dimensional distance to the wall for a bulk velocity of 0.5 m/s – view of all the walls. Left: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.

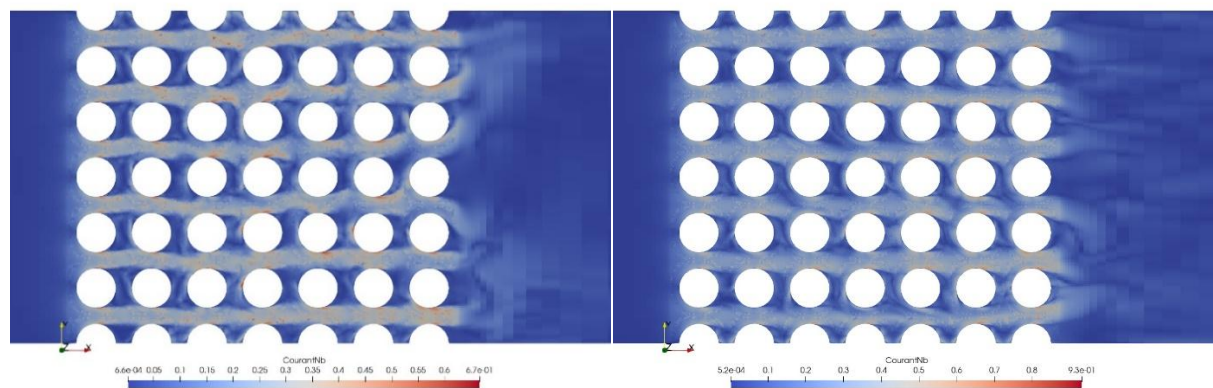


Figure 41: Instantaneous CFL number at a bulk velocity of 0.5 m/s on the center plane. Left: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.

Figure 42 compares the two-dimensional mean velocity magnitude and the streamlines in the centre plane (bundle region and outlet) obtained in the computations with the experimental data. The results obtained with the k- ω SST seem to be qualitatively closer to the experimental results. With EB-RSM, the flow at the outlet exhibits the large two recirculation zones as seen at the lower bulk velocity of 0.125 m/s.

Figure 43 shows the same fields as in the previous figure, but then focused on the bundle region. The EB-RSM predicts a more pronounced bifurcation than the k- ω SST with alternating impinging jets on the top/bottom of the tubes. The k- ω SST also shows these impinging jets, but instead it stays on same side of the cylinders, as is seen in the experiment. Figure 44 shows the predicted turbulent kinetic energy (TKE) in the bundle region as computed in the GOKSTAD experiment (Can Akkurt et al., 2026). The predicted levels of TKE by the EB-RSM are closer to the experimental ones, while the levels predicted by the k- ω SST are much lower. Note that the computed data are the sum of the mean modelled turbulent kinetic energy and the mean resolved one. As a conclusion, the k- ω SST exhibits a close behaviour to the experimental data with non-alternating jets in the bundle and an outlet which shows less pronounced recirculation zones.

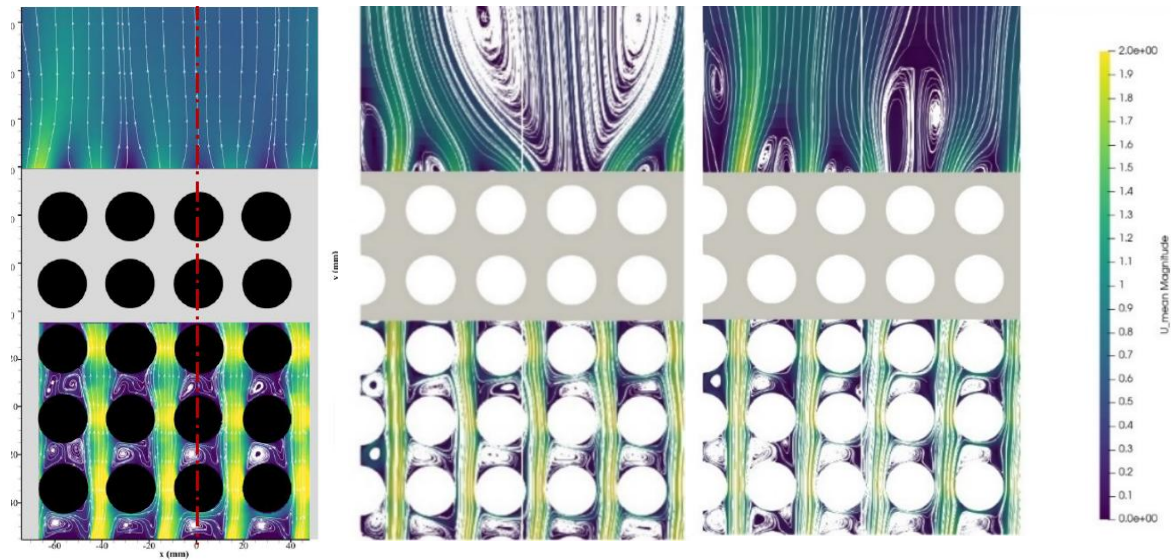


Figure 42: Velocity magnitude on the experimental plane (x,y) for a bulk velocity of 0.5 m/s – view of the bundle and the outlet. Left: experiment, middle: EB-RSM with Mesh 0, right: k-ω SST with Mesh 1.

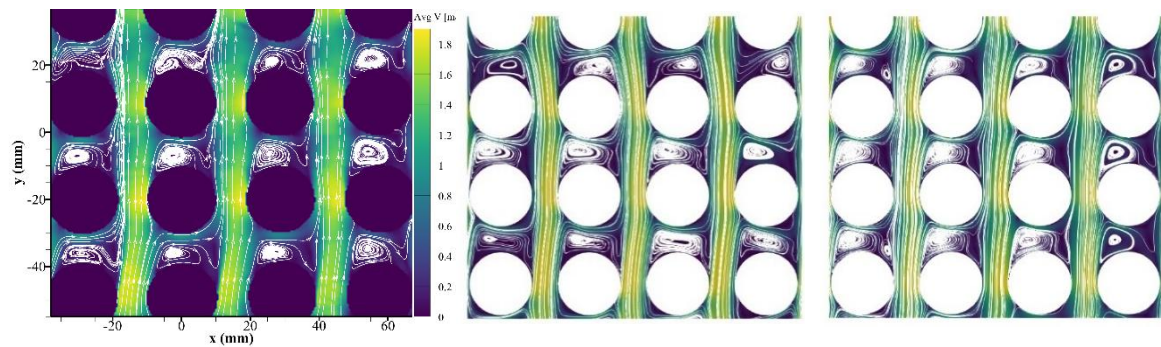


Figure 43: Velocity magnitude on the experimental plane (x,y) for a bulk velocity of 0.5 m/s – view of the bundle. Left: experiment, middle: EB-RSM with Mesh 0, right: k-ω SST with Mesh 1.

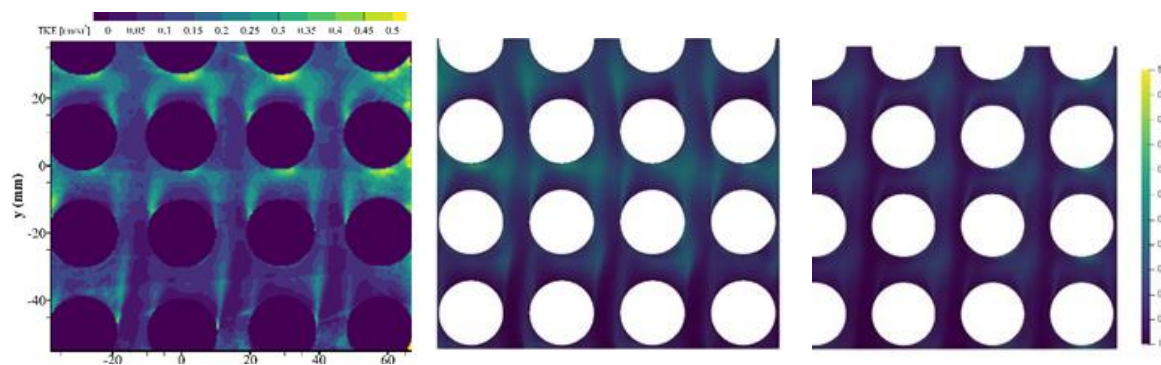


Figure 44: TKE on the experimental plane (x,y) for a bulk velocity of 0.5 m/s – view of the bundle. Left: experiment, middle: EB-RSM with Mesh 0, right: k-ω SST with Mesh 1.

The EB-RSM is less stable than the k-ω SST in the present configuration in the sense that there is at least one change in the regime of the flow, as will be shown in this section. Figure 45 provides the time evolution of the lift force applied to the central cylinder (4,3) and the

cylinder at the outlet (7,3), both of the middle column of cylinders, (7,3) for both computations. The lift force of the $k-\omega$ SST seems to be stable, whereas there is a change of regime of this force for the EB-RSM at around 285 s. This makes it difficult to compute the statistics and one might probably reconsider computing the statistics for the EB-RSM depending on the observed regime (conditional averaging). Note that the utilized statistics in the case of the EB-RSM used the last 20 seconds of the simulation (in hindsight, it would have been interesting to collect statistics between 250 s and 275 s to analyse the first regime).

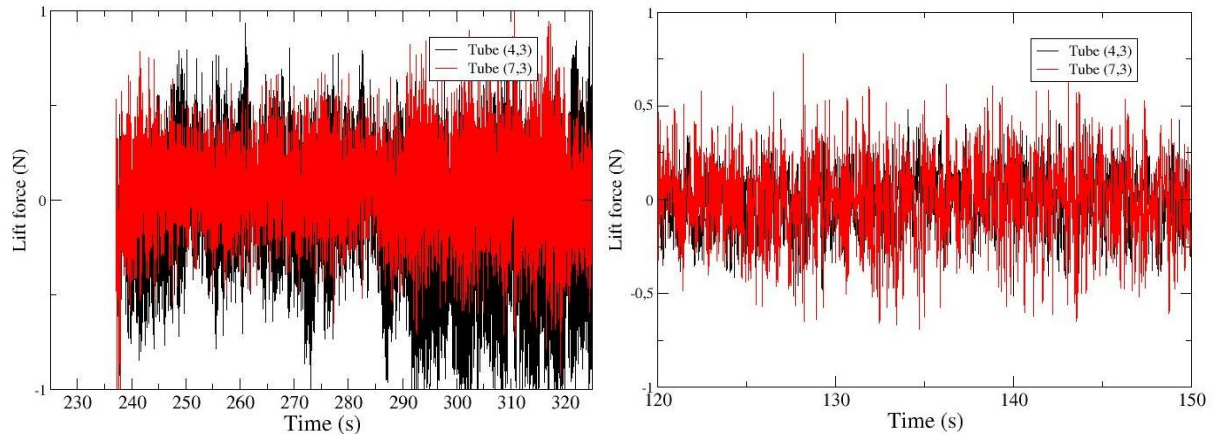


Figure 45: Time evolution of the lift force applied to cylinders (4,3) (the central one) and (7,3) (at the outlet) for a bulk velocity of 0.5 m/s. Left: EB-RSM with Mesh 0, right: $k-\omega$ SST with Mesh 1.

Figure 46 and Figure 47 provide comparisons to the experimental data for both turbulent models, showing the mean-averaged two-dimensional velocity magnitude and the turbulent kinetic energy as defined in GOKSTAD experiment, respectively. As it has been seen previously quantitatively, the $k-\omega$ SST is closer to the experimental results for the velocity magnitude along the lines P2 and P6 and the results of both models are comparable along the line P4. The deviations due to the jets are observable for both models on the third and fourth peaks of the velocity magnitude along P2 and P6 departing from the left. These deviations are more pronounced for the EB-RSM model. The results of TKE with EB-RSM are closer to the experimental data along P2 and P6 than with $k-\omega$ SST, as $k-\omega$ SST is unable to correctly predict some peaks that are clearly observed experimentally. The TKE is dramatically overpredicted with the EB-RSM along P4. This has to be further analysed to understand why.

It is interesting at this stage to understand the unsteadiness of the two employed URANS models. Figure 48 shows the resolved and the modelled parts of the TKE separately along line P6 for both turbulence models. For the EB-RSM, the modelled and resolved parts are of similar magnitude, however the shape of TKE is given by the resolved part (the two peaks on each “plateau”). While analysing the different resolved Reynolds stress components separately, the three diagonal components have the same weight in the resolved turbulent kinetic energy. This is not the case for the $k-\omega$ SST in which the R_{zz} component (along the tubes) is much smaller than the other two. Moreover, Figure 48 shows that the modelled part is much smaller than the resolved one. This tends to show that the $k-\omega$ SST simulation is a quasi 2D unsteady simulation. Note that the resolved part of the turbulent kinetic energy doesn’t predict the different peaks at the appropriate locations.

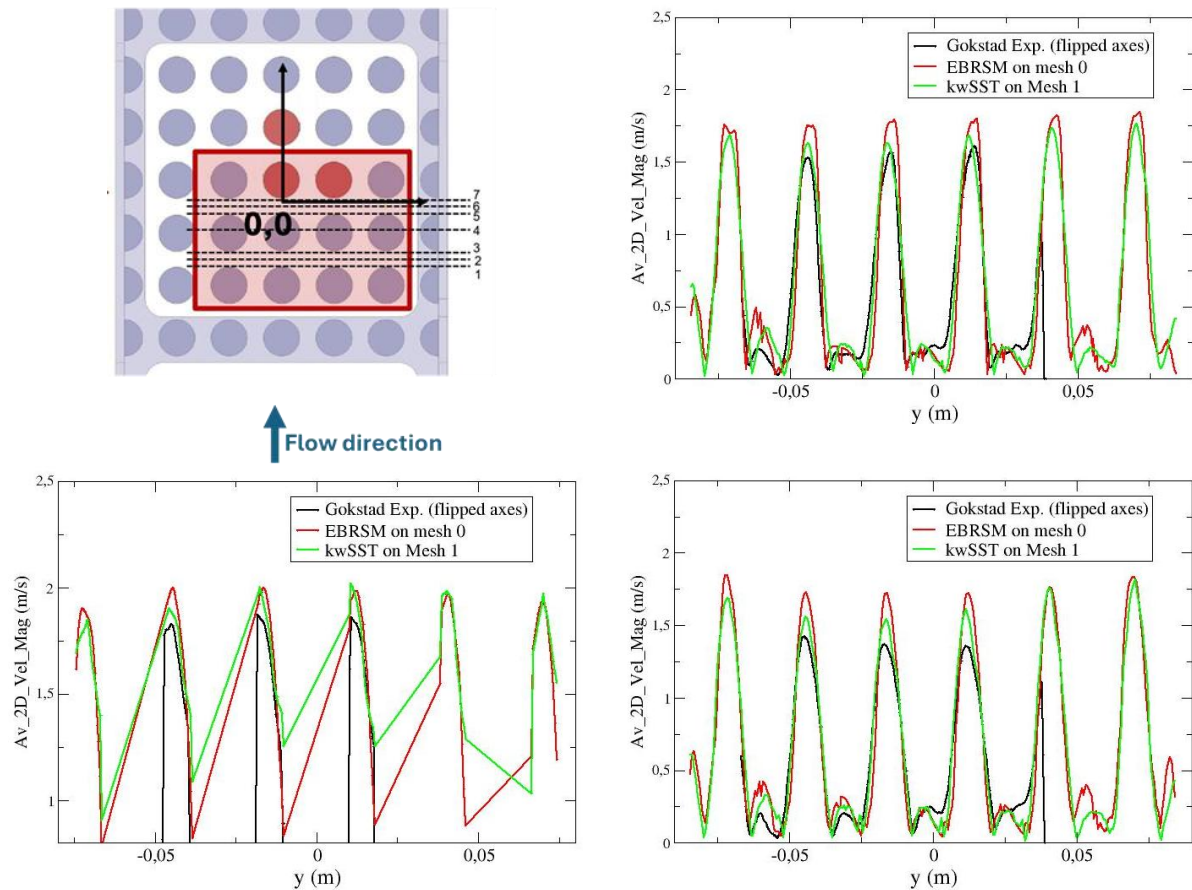


Figure 46: Velocity magnitude on the center plane for a bulk velocity of 0.5 m/s – comparisons of profiles along the lines P2 (top right), P4 (bottom left) and P6 (bottom right), see top left for profile locations.

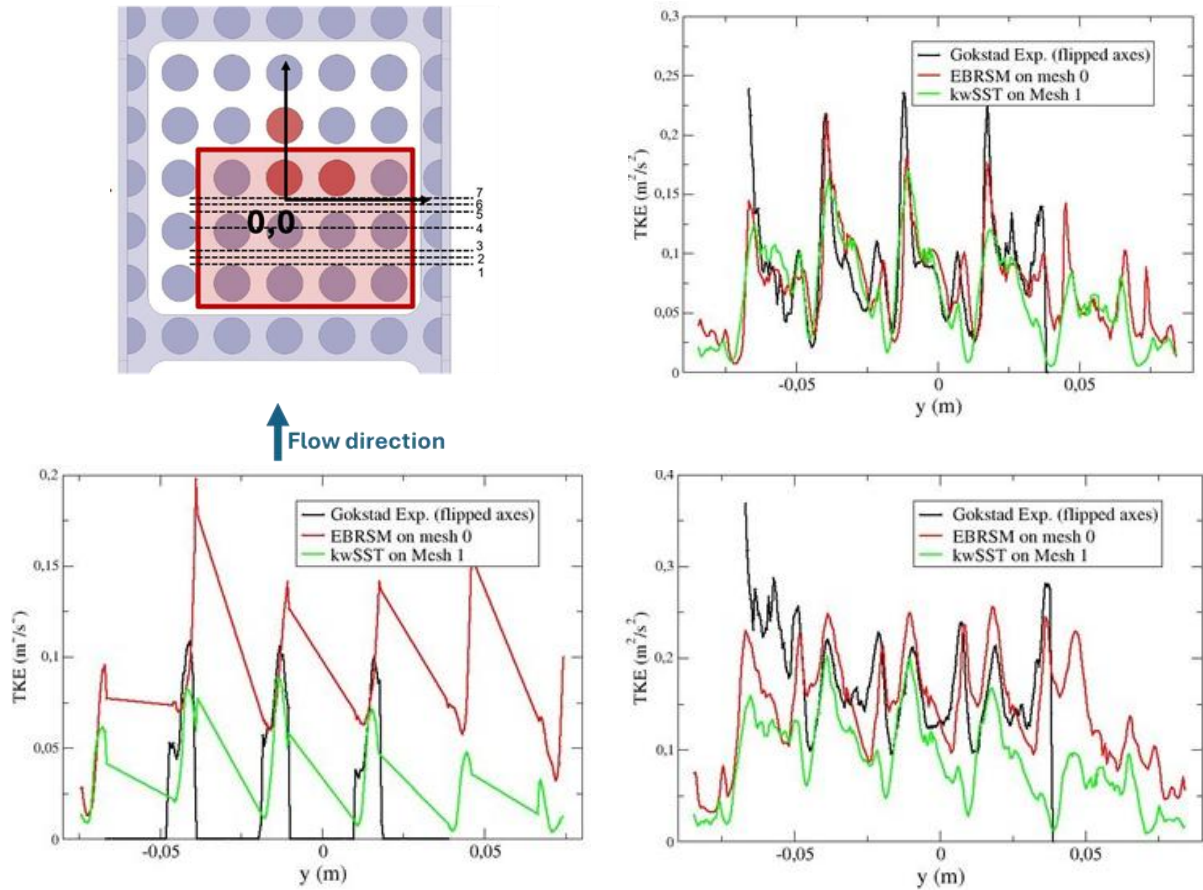


Figure 47: TKE on the center plane for a bulk velocity of 0.5 m/s – comparisons of profiles along the lines P2 (top right), P4 (bottom left) and P6 (bottom right), see top left for profile locations.

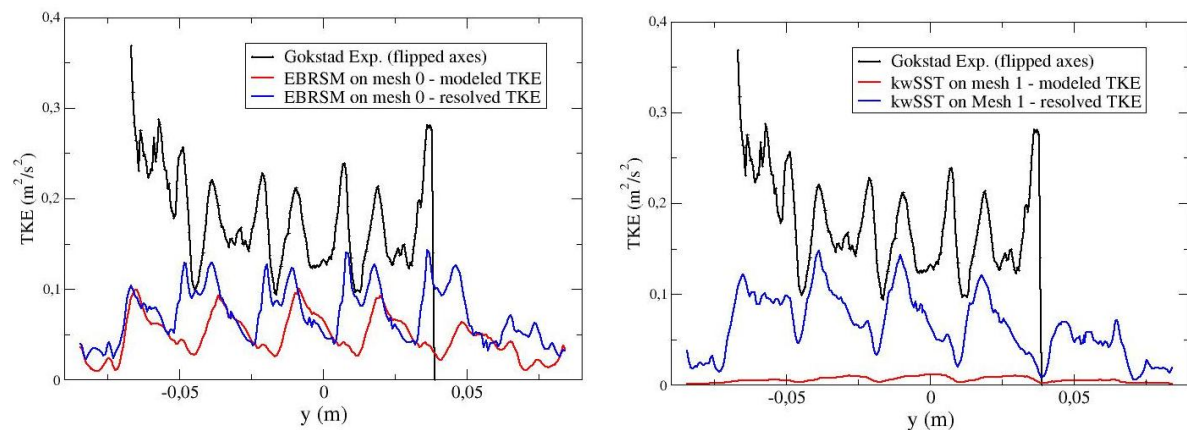


Figure 48: Modeled and resolved TKE in the center plane along the line p6 for a bulk velocity of 0.5 m/s. Left: EB-RSM with Mesh 0, right: k- ω SST with Mesh 1.

5.2.2 Fluid flow prediction at a bulk velocity of 1 m/s

The objective of the present section is to quantitatively check the ability of two turbulence models to predict the flow physics at 1 m/s, as the experiment showed totally different flow patterns at this velocity than at the lower bulk velocities. The utilized mesh is the equivalent

of Mesh 0 in the previous section; it contains 127 M cells (200 nodes in the azimuthal direction and 380 nodes along the tubes).

The EB-RSM and the $k-\omega$ SST exhibit bifurcation particularly visible on the drag force applied to the tube (7,3), see Figure 49. The change in the EB-RSM computation is abrupt, which may indicate that the flow regime in the outlet has totally changed. Flow statistics have been collected from 50 s to 73 s for the EB-RSM and during the last 5 s with the $k-\omega$ SST. The flow structure at the outlet is qualitatively the same as the one observed for the bulk velocity equal to 0.5 m/s, see Figure 50. However, it is clear that the inclination of the flow in the bundle is only correctly captured by the EB-RSM. This is confirmed by Figure 51, Figure 52 and Figure 53. Note that the levels of the TKE are globally in better agreement with the ones in the experiment than found at a bulk velocity of 0.5 m/s.

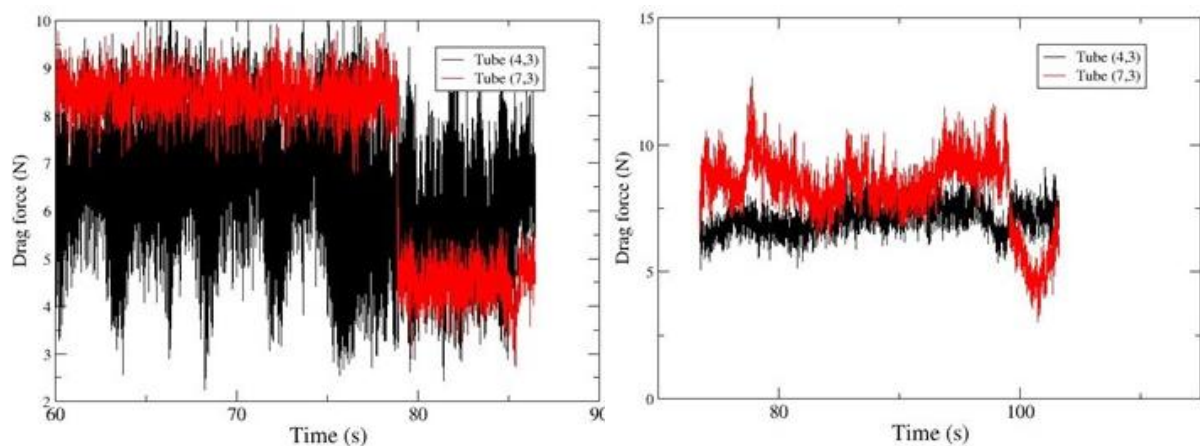


Figure 49: Time evolution of the lift force applied to cylinders (4,3) (the central one) and (7,3) (at the outlet) for a bulk velocity of 1.0 m/s. Left: EB-RSM, right: $k-\omega$ SST.

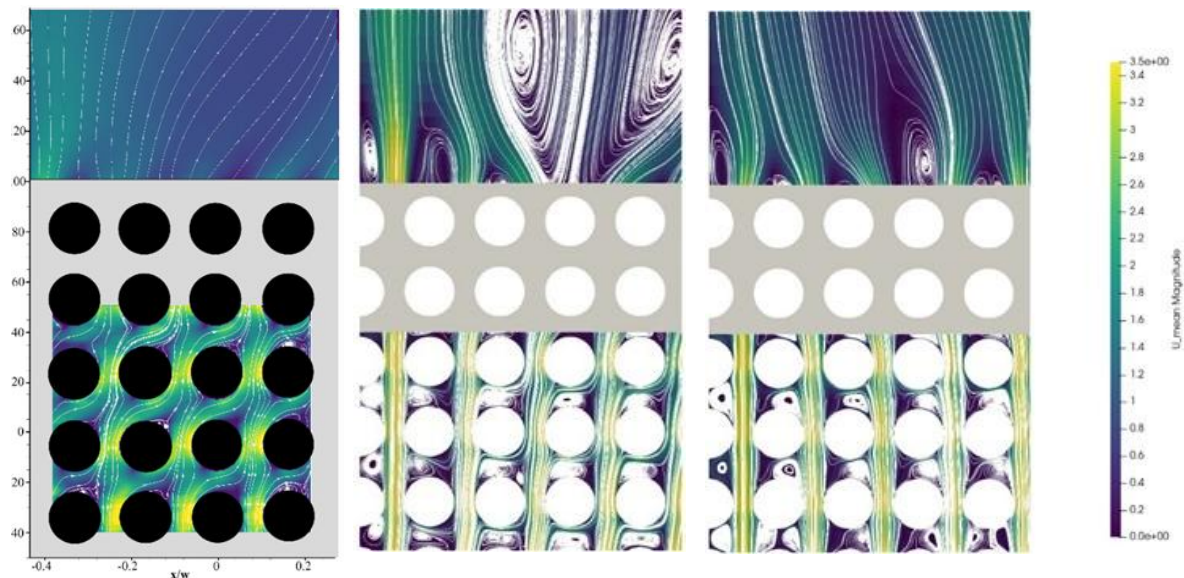


Figure 50: Velocity magnitude on the experimental plane (x,y) for an approximate bulk velocity of 1.0 m/s – view of the bundle and the outlet. Left: experiment, middle: EB-RSM, right: $k-\omega$ SST.

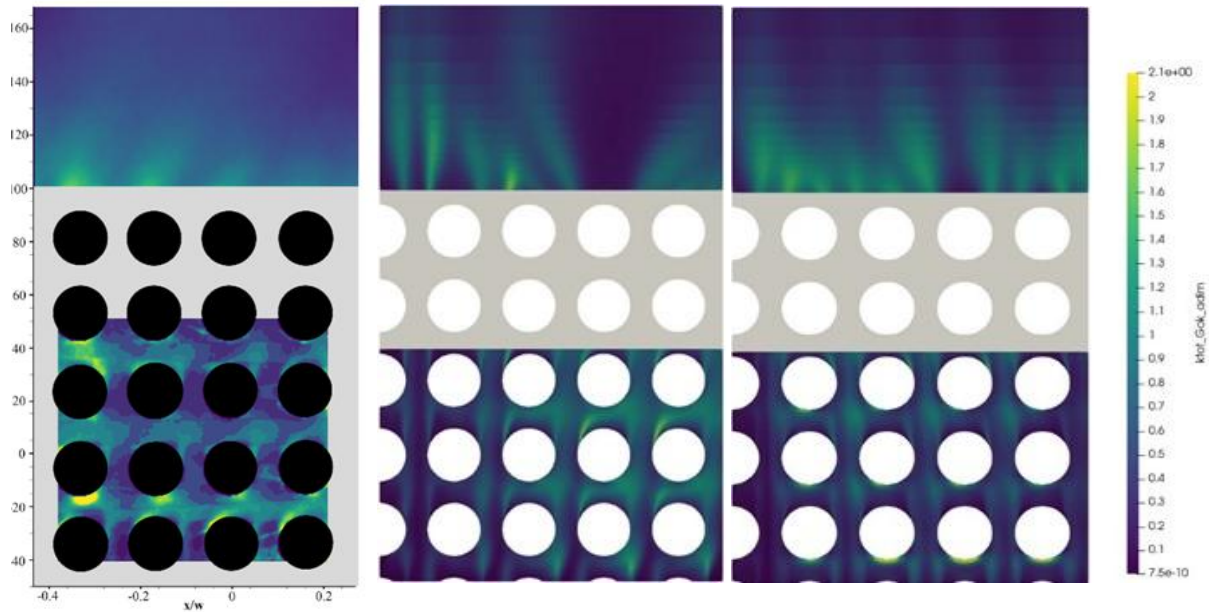


Figure 51: Velocity magnitude on the experimental plane (x,y) for an approximate bulk velocity of 1.0 m/s – view of the bundle and the outlet. Left: experiment, middle: EB-RSM, right: k- ω SST.

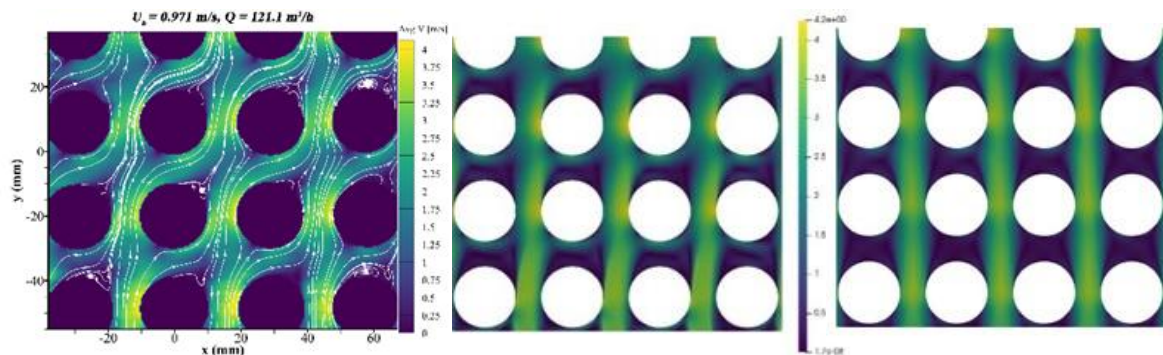


Figure 52: Velocity magnitude on the experimental plane (x,y) for a bulk velocity of ~1.0 m/s – view of the bundle. Left: experiment, middle: EB-RSM, right: k- ω SST.

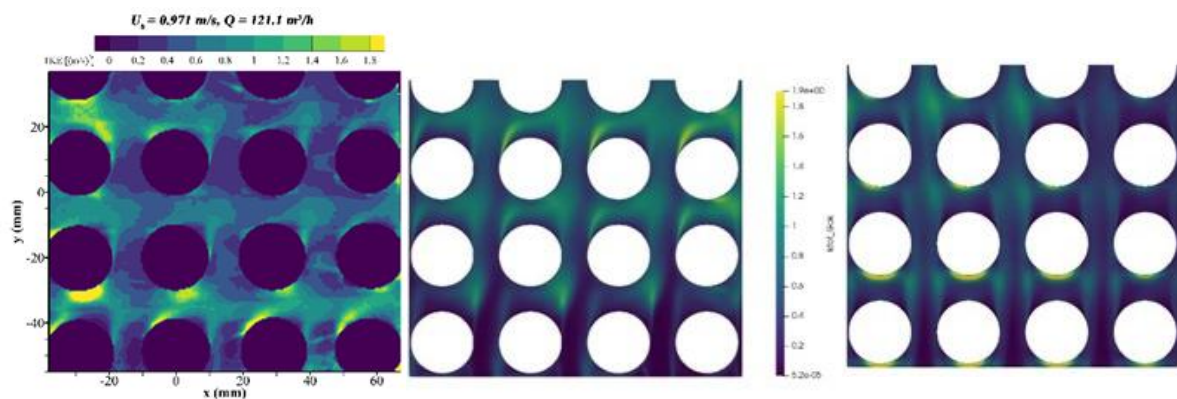


Figure 53: TKE on the experimental plane (x,y) for an approximate bulk velocity of 1.0 m/s – view of the bundle. Left: experiment, middle: EB-RSM, right: k- ω SST.

5.3 Simulations with one moving cylinder in a tube bundle

Successful simulations of a cylinder moving in two directions (lift and drag) at bulk velocities of 0.125 m/s and 0.25 m/s were performed. Only the EB-RSM has been used. Coarsening the meshes was needed for 0.25 m/s to allow the larger displacements observed at that velocity using ALE. The meshes are fine enough in the near-wall regions (which move like a rigid body) but coarsened in the bulk regions. Without this coarsening, the very small bulk cells would deform too much, resulting in negative cell volumes and unphysical results, if not crashing of the simulations. The results of the EB-RSM turbulence model at 0.125 m/s are part of the benchmark shown in Chapter 3. The mesh contains 10 M cells (120 nodes in the azimuthal direction and 90 along the tubes). For 0.25 m/s, the mesh contains 26 M cells (100 nodes in the azimuthal direction and 226 along the tubes).

The computation at 0.125 m/s was run for a very long time (more than 361 s). The obtained RMS displacements in the stream-wise and normal directions are respectively equal to 0.0023 mm and 0.0026 mm. The experimental data are 0.0085 and 0.0083. The under-estimation is of a factor of about 3. The computation at 0.25 m/s has been run for 14 s. The obtained RMS displacements in the stream-wise and normal directions are respectively equal to 0.025 mm and 0.014. The obtained experimental data are 0.011 mm and 0.013 mm. The agreement is much better, in particular for displacement in the lift direction. The time used for the simulation is probably too short. Moreover, another computation with a different mesh of the same refinement gave displacements equal to 0.068 mm and 0.022 mm, which is far from the results obtained with the first mesh. The present results have been obtained recently and need deeper analysis and additional computations. Concerning the frequencies, the displacement in the stream-wise direction exhibits three main energetic frequencies at 10.4 Hz, 20.7 Hz and 31 Hz. The main frequencies of the displacement in the lift direction are at 20.7 Hz and 31 Hz. The experimental results show one main frequency at 34.7 Hz and 32.7 Hz for the stream-wise and normal displacements, respectively. The spectra of the simulation at a bulk velocity of 0.25 m/s are shown in Figure 54.

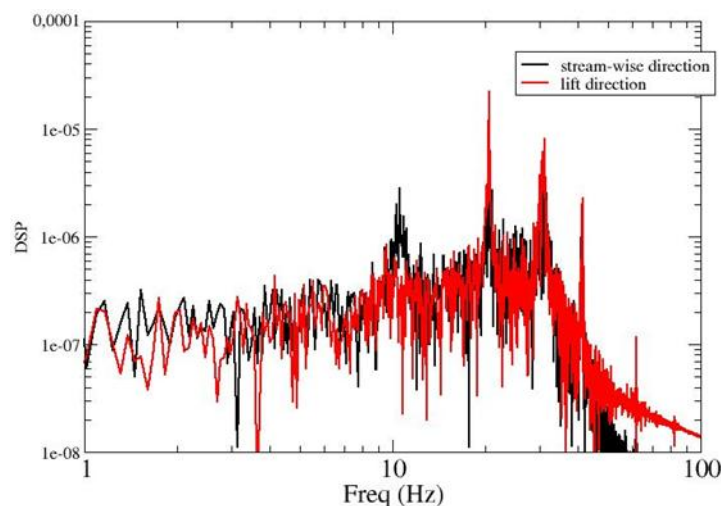


Figure 54: Displacement spectra for the single moving cylinder case at a bulk velocity of 0.25 m/s

5.4 Conclusions

The following can be concluded for the flow through a rigid tube bundle:

- For a bulk velocity of 0.5 m/s, the $k-\omega$ SST exhibits a closer behaviour to the experimental data with non-alternating jets in the bundle and an outlet which shows less pronounced recirculation zones than the EB-RSM. The simulation seems to be quasi-bidimensional as the variations are small in the z direction along the tubes. The EB-RSM shows an alternating jets regime with two large re-circulations downstream the bundle. An overall over-estimation of the velocity magnitude in the centre plane is observed for the two models, with EB-RSM showing the highest over-estimation. The turbulent kinetic energy is better predicted with the EB-RSM in the bulk region between the tubes. However, it seems to be largely overestimated in the gap regions.
- For the bulk velocity of 1 m/s, the $k-\omega$ SST is unable to predict the flow inclination observed experimentally. The EB-RSM, however, is able to do so, but more investigations on the reason for that and of the quantitative quality of the results is needed.
- The EB-RSM (and the $k-\omega$ SST for the bulk velocity 1 m/s) show a bifurcation phenomenon. This make it difficult to compute statistics as the stable position may change and provide an average which is not representative of the actually predicted flow physics. This is out of the scope of the present report and should be analysed in the future.

For the single moving cylinder configuration, the displacements are underestimated by a factor of 3 for the lowest bulk velocity (0.125 m/s). This case is further analysed as part of the benchmark exercise, where it is also compared to other computations.

One of the most recent computations gives encouraging results for the displacements at 0.25 m/s. However, it is today too early to draw conclusions and further investigations are needed.

6. Simulations performed by NRG PALLAS

6.1 Introduction

In Pressurized Water Reactors (PWRs), heat produced in the reactor core is transferred to the secondary side through steam generators containing large tube bundles. To enhance heat transfer, the secondary-side fluid flows perpendicular to these tubes, creating a cross-flow configuration. Under such conditions, flow-induced vibrations (FIV) may occur due to turbulence and vortex shedding in the wake of the tubes. These vibrations can lead to structural fatigue and damage of the steam generator tubes. At sufficiently high flow velocities, a more severe mechanism known as fluid-elastic instability (FEI) may develop, where the energy transferred from the flow to the structure exceeds the structural damping, resulting in rapidly growing tube oscillations. Understanding and predicting these phenomena is therefore essential for ensuring the safe and long-term operation of nuclear steam generators.

While traditional design approaches rely on empirical correlations derived from experiments, numerical methods based on coupled Computational Fluid Dynamics (CFD) and Computational Structural Mechanics (CSM) have increasingly been applied to study FIV. However, accurately resolving the complex three-dimensional flow structures within tube bundles remains challenging, especially when using computationally affordable turbulence models such as URANS or coarse LES. Within the VIKING collaboration, the VISCACHE single-moving-rod experiment was used as a benchmark to assess the capability of such medium-resolution fluid-structure interaction (FSI) approaches (Granger et al., 1993). These studies provided valuable insights into modelling requirements, mesh resolution, and the sensitivity of predicted vibration behaviour to turbulence modelling choices (Benhamadouche & Benguigui, 2023, Vivaldi et al., 1993, Zwijsen et al., 2024, Zwijsen et al., 2025). Building on these lessons, further numerical investigations are currently performed within Work Package 3 (WP3) of the GO-VIKING project using the newly developed GOKSTAD facility at the Von Karman Institute (Can Akkurt et al., 2026). This state-of-the-art experimental setup provides detailed flow and vibration measurements for a tube bundle subjected to cross-flow, offering new opportunities to validate and further improve simulation approaches for predicting flow-induced vibrations in nuclear steam generator tube bundles.

Within WP3, NRG PALLAS agreed to perform medium-resolution simulations of the GOKSTAD facility, thereby focusing on the use of Unsteady Reynolds Averaged Navier Stokes turbulence models, or models of similar computation costs. This chapter describes the work done by NRG PALLAS on the modelling of the GOKSTAD facility. First, in Section 6.2, the CAD and mesh used are described. Then, in Section 6.3, the setup of the performed simulations is presented. Subsequently, in Section 6.4, results obtained for the flow-only test cases are shown and, where possible, compared to the experiment. Finally, in Section 6.5, the focus is shifted to the FSI simulation results.

6.2 CAD & mesh

As presented in Chapter 2, the GOKSTAD experiment consists of a 7x7 tube bundle, with tube diameter $D = 20$ mm and length $L = 200$ mm, placed inside a channel with height $H = 172.8$

mm. These dimensions are respected in the geometry used in the simulations. To save on computational cost, however, the full length of the channel was reduced compared to the experiment. The experiment has an upstream square channel with a length of 2040 mm, a test section length of 520 mm and a downstream square channel with a length of another 520 mm. For the simulations, a square channel of 200 x 172.8 mm is used, similar to the experiment, though the total length is reduced to 2000 mm, with the tube bundle placed centrally in the channel. This means the full domain is 10 times the spanwise width of the channel, or equivalently equals 10 times the tube's length. The resulting CAD, which was created with STAR-CCM+ is shown in Figure 55.

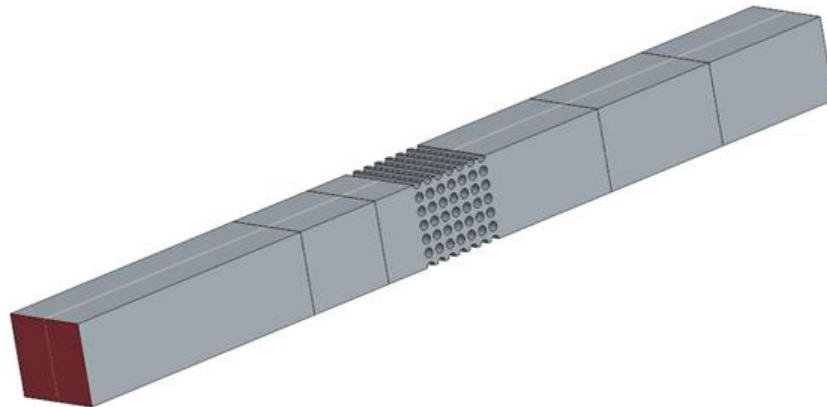


Figure 55: Geometry of GOKSTAD used by NRG PALLAS in its simulations.

The mesh was created in GAMBIT, making use of its Cooper meshing capabilities to create a fully hexahedral mesh. To save on the number of cells, it was decided to have local mesh refinements closer to the tube bundle, and have a relatively coarse mesh near the inlet and outlet. To be more precise, two different refinement zones are used. The first refinement runs from a distance of about $-1.5H$ before the bundle till $\sim 3.0H$ after the bundle and uses cells with a base size that is 50% in size of the cells in the coarse region. The second refinement runs from approximately $-0.5H$ upstream of the tube bundle till about $1.5H$ downstream of the tube bundle, with cells being 20% in size of those of the general coarse region.

From previous studies (Zwijssen et al., 2024), it was determined that in order to get good results, a $y^+ < 1$ mesh is needed. Hence boundary cell layers are used near all the walls, such that the first cell height satisfies this criterion for all the investigated flow velocities. A growth rate factor of 1.2 is used to ensure smooth transitions in cell height between the different layers. In total 25 cells are used in the boundary layer, which ensured a smooth transition between the last boundary layer and the bulk cells. No boundary layers are used in the spanwise bounding surfaces. This was done because tests are performed using both walls and periodic boundaries in that direction. Hence, to avoid influence of the spanwise mesh distribution, the mesh was kept the same for both settings.

To resolve the separation points of the flow around the cylinders with sufficient accuracy, for all full cylinders in the domain, a total of 64 cells is used in circumferential direction (and half that for the half tubes). Finally, a total of 50 layers were used in spanwise direction.

The only parameter left to set to create the mesh is the base size of the cells in the coarse region. To come to a manageable number of cells for the full domain in terms of computational

cost, this was set to 5.0 mm. This means cells have sizes of ~ 2.5 mm in the first refinement region, and 1.0 mm in the bundle region. This results in a total cell count of about 10.7M cells. Figure 56 shows several views and zooms of the final mesh.

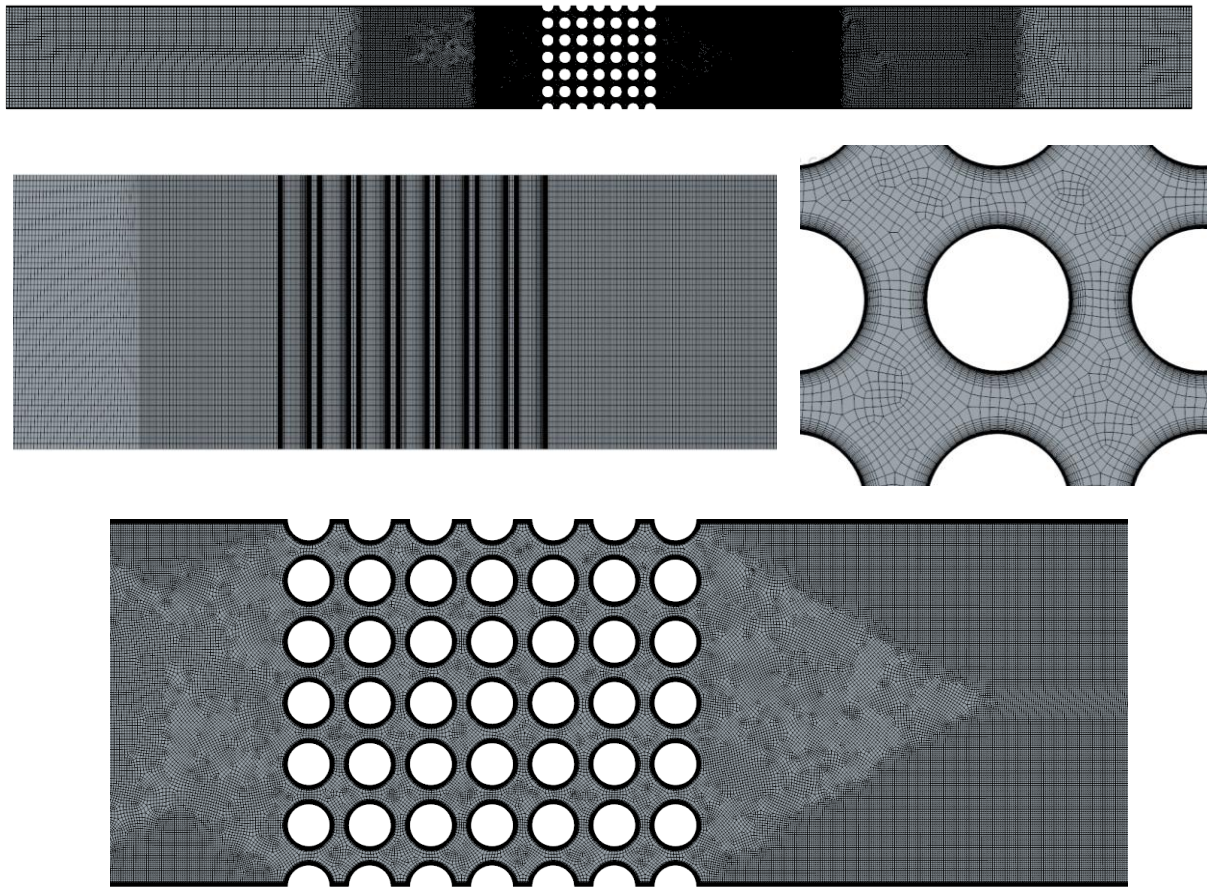


Figure 56: Mesh used by NRG PALLAS for its simulations with, top: front view of the full domain, middle left: top view of bundle area, middle right: mesh inside bundle around cylinders, and bottom: front view zoomed on the bundle region.

6.3 Numerical model

All simulations were performed using STAR-CCM+. For the flow-only simulations, STAR-CCM+ V2406 was used, while for the FSI simulations STAR-CCM+ V2510 was used. The goal of the flow-only simulations is to test various turbulence models, in particular more advanced turbulence models not part of the regular linear eddy-viscosity RANS turbulence models group (such as Improved Delayed Detached Eddy Simulation (IDDES) and Reynolds Stress Models (RSM)), as it was seen previously that these models may (severely) overpredict displacements of tubes inside tube bundles (Zwijzen et al., 2024). Therefore, for the flow-only simulations, the following turbulence models available in STAR-CCM+ V2406 were tested, within brackets the name used henceforth when referring to the individual turbulence models:

- URANS $k-\varepsilon$ with cubic constitutive relationship (URANS $k-\varepsilon$)
- $k-\omega$ SST-IDDES (IDDES)

- Elliptic Blending (EB) RSM (EB-RSM)
- Lagrangian Elliptic Blending (lagEB) RSM (LagEB)
- LES using the Wall-Adapting Local Eddy-viscosity (WALE) subgrid-scale model (LES)

All these models were run at an inlet velocity of 1.0 m/s, the maximum velocity at which experimental tests were done. All walls are no-slip walls, including the front and back walls. Additionally, for the URANS $k-\epsilon$ and LES, simulations were also performed with periodic front and back walls. A velocity inlet boundary condition is used for the inlet, set to 1.0 m/s. A turbulence intensity of 5% was set, which is similar to what has been observed in the experiments. For the outlet, a pressure outlet is used with 0 Pa gauge pressure.

A second order convection scheme and Second-Order Backward Differentiation Formula (BDF2) time integration scheme is selected. The time step is set equal to 2.0E-4s, which ensures Courant–Friedrichs–Lewy number $CFL < 1$ throughout most of the domain. A total of 25 iterations per time step are performed. The flow-only simulations were run for in total 10.0s of flow time, with statistics obtained after 2.0s, to give the flow some time to establish.

For the FSI-simulations, similar flow settings are used as for the flow-only simulations. The main difference is that the time-step size was increased depending on the inlet velocity. The resulting CFL remains below one throughout most of the domain. All FSI simulations use no-slip front and back walls.

To account for the tube motion in the FSI-simulations, since the tube can only translate in lift (horizontal, y-) and drag (vertical, x-) directions, STAR-CCM+'s Dynamic Fluid-Body Interaction (DFBI) solver is used for the FSI simulations. This is a rigid body motion solver, in which the Degrees-Of-Freedom (DOFs) of the structure of interest can be set. Hence for the current case, the only free DOFs are the x- and y-directions. The mass of the cylinder is set to 0.1 kg. Two spring-damper constraints are applied as body couplings: one for each DOF. Values for the spring stiffness and damping constant are set to $K = 7000 \text{ N}\cdot\text{m}^{-1}$ and $C = 1.14 \text{ N}\cdot\text{s}\cdot\text{m}^{-1}$, respectively, consistent with the values provided by the experimentalists, see Table 1.

Mesh deformation is handled using the Arbitrary Lagrangian–Eulerian (ALE) formulation, with the tube displacement and computational mesh updated at each time step. The FSI simulations were performed for an inlet velocity of 0.125 m/s and 0.5 m/s, and based on the outcome of the flow-only simulations, three different turbulence models were used, viz. cubic $k-\epsilon$, $k-\omega$ SST IDDES and WALE LES. The FSI simulations were run for 74.0s and 26.0s in total for an inlet velocity of 0.125 m/s and 0.5 m/s, respectively. These time periods were chosen based on the estimated flow-through times of the simulation, which is about 16.0 s for 0.125 m/s (2m domain, with flow velocity of 0.125 m/s) and 4.0s for an 0.5 m/s inlet velocity. Based on this, the simulation is first run for 1 Flow Pass (FP, time needed for the flow to pass through the full domain) for the flow to establish. During the subsequent 2FPs the flow statistics are obtained. Then the cylinder is released, with a ramp time of 2.0s. Then the cylinder is allowed to adjust to the flow for 0.5FP (1.0 FP for 0.5 m/s), and FSI statistics are finally calculated over the final 1.5FP (2.0 FP for 0.5 m/s).

6.4 Flow-only simulation results

In this section, the results obtained for the pure CFD simulations are presented. This is done for three different flow velocities. First, results for a flow velocity of 1.0 m/s are presented. These were performed using all five turbulence models listed in Section 6.3. The main goal of these simulations was to inter-compare the results obtained using different turbulence models, to shortlist those to be used for the eventual FSI simulations. Subsequently, results obtained during the flow-only parts of the FSI simulations with an inlet velocity of 0.125 m/s and 0.5 m/s are presented, respectively. These results will be compared with experimental results.

6.4.1 Inlet velocity of 1.0 m/s – inter-comparison of models

As mentioned before, the main goal of these flow-only simulations was to inter-compare the flow and force results obtained using different kinds of turbulence models, and based on that, make a selection of turbulence models to be used for the FSI simulations. As a first step, the suitability of the mesh is checked in terms of y^+ and CFL number. Figure 57 shows the instantaneous y^+ values on the walls of the tube bundle for the URANS k- ϵ and LES simulations at $t = 10.0$ s. As can be seen, y^+ is mostly below one everywhere, except for some isolated locations. For the FSI simulations, which are performed at lower velocities, $y^+ < 1$ will hold everywhere. Similar results were obtained with the other three turbulence models.

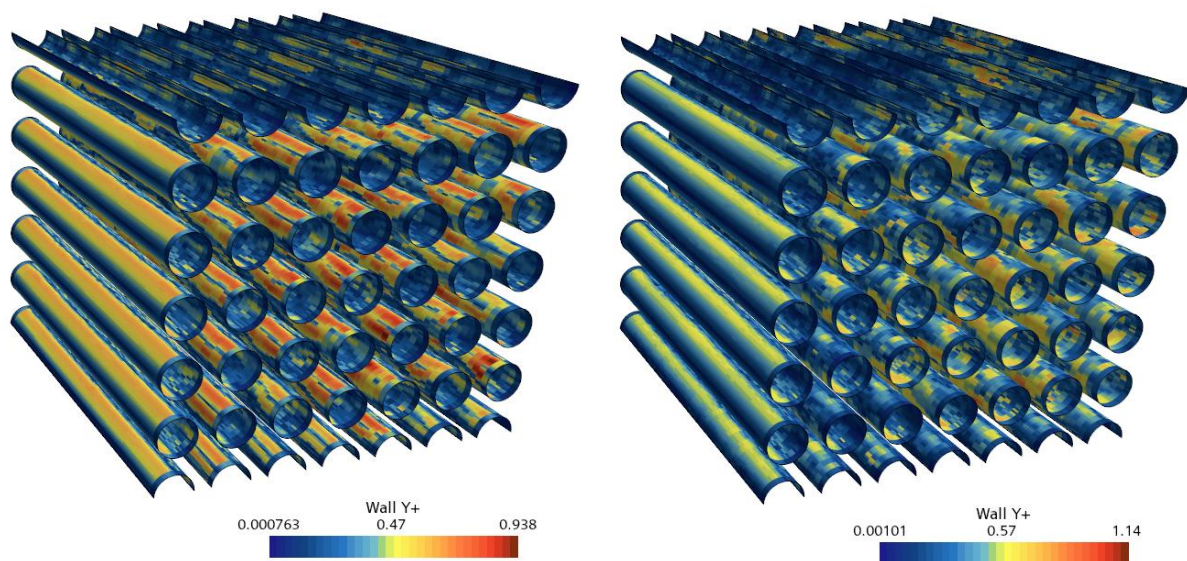


Figure 57: Wall y^+ as obtained with the URANS k- ϵ (left) and LES (right) turbulence models.

Results obtained for the CFL number with the URANS k- ϵ turbulence model at $t = 10.0$ s are shown in Figure 58 and Figure 59, which show respectively, the obtained CFL values on a central plane through the tube bundle, and a histogram of the CFL values of all cells in the domain. As can be seen, CFL values are mostly below one. Only in the Central Volume of the domain, in the cells near the tubes, are CFL values above one found. However, since an implicit time scheme is used (BDF2), CFL values above one does not pose any stability problems. Only in terms of accuracy should CFL not become too large. Hence, the found values are deemed acceptable. Similar results were obtained with the four other turbulence models.

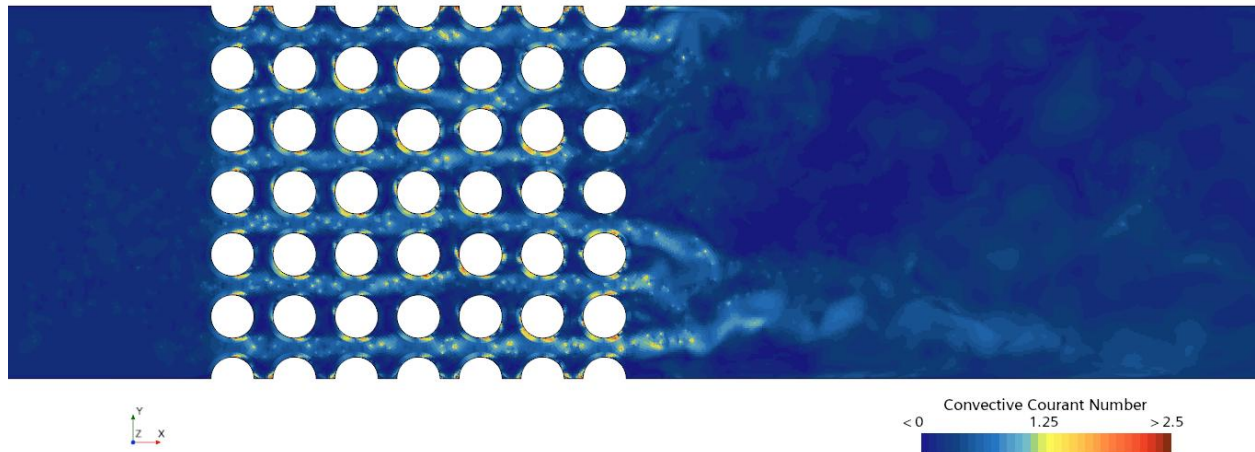


Figure 58: CFL values on the central plane, zoomed in around the tube bundle, obtained with URANS $k-\epsilon$.

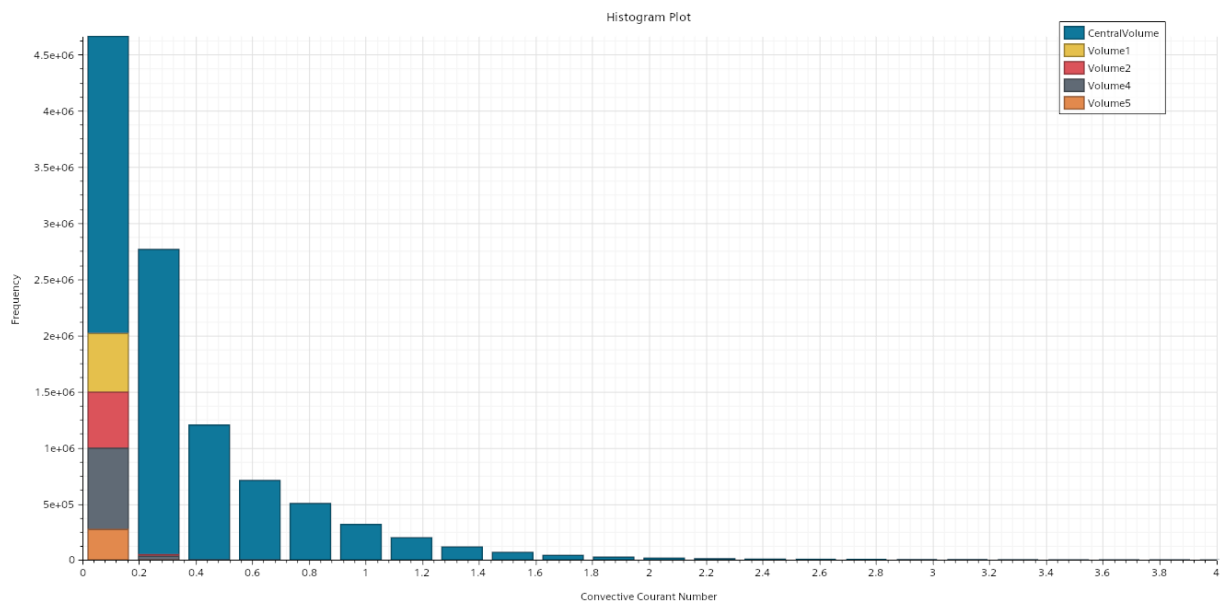


Figure 59: Histogram of the CFL values obtained with URANS $k-\epsilon$, split over the various mesh refinement zones. Mesh volumes are numbered from left to right per refinement zone, as displayed in Figure 56, with the Central Volume being the bundle region.

Based on the above, it can be concluded that the mesh is acceptable for performing the intended turbulence simulations. Hence, as a next step, the turbulence models are compared in terms of predictions of the flow quantities. For all presented time-averaged fields, the averaging was done over eight seconds, starting at $t = 2.0$ s and ending at $t = 10.0$ s, or equivalently for ~ 4 FPs. Figure 60 shows the time-averaged streamwise velocity fields on the central plane, zoomed in around the tube bundle, as obtained with the five different turbulence models, using wall boundary conditions in the spanwise directions. As can be seen, there is quite a large discrepancy between the different turbulence models, both inside the tube bundle as at the outlet section of the tube bundle.

Within the tube bundle, URANS k- ϵ seems to primarily predict strong jets moving through the tube bundle rows, without much cross-flow inside the bundle through the tube columns. IDDES also mostly recovers jets through the bundle rows, though these decrease in velocity deeper inside the bundle. The other three turbulence models also predict some cross-flow inside the bundle, with the most cross-flow found with the LES, where dominant cross-flow already forms after about three rows of tubes.

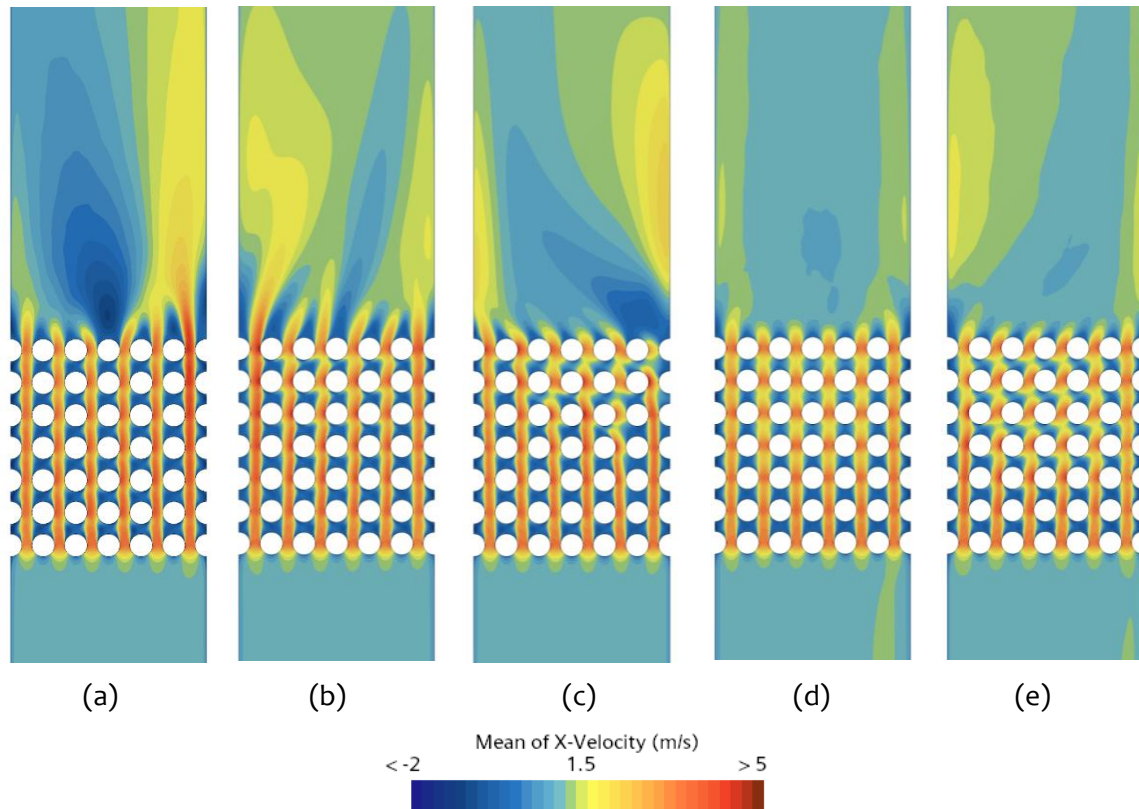


Figure 60: Time-averaged streamwise velocity, capped between -2.0 m/s and 5.0 m/s, for a) URANS k- ϵ , b) EB-RSM, c) Lag-EB, d) IDDES and e) LES.

The formation of jets and/or cross-flow inside the tube bundle greatly affects the behaviour of the water at the outlet of the bundle. For URANS k- ϵ , a large jet is formed at the right side of the test section, where several outlet jets merge into a large jet and move jointly towards the test section wall. A similar, though weaker, jet is formed on the left side. This also results in a sort of recirculation zone in the middle of the outlet section. Strong outlet jets are also seen with the EB-RSM, though they remain more isolated instead of merging into a stronger, larger jet. For IDDES, only the outer two gap jets seem to merge into jets, though these are significantly weaker in streamwise velocity than those found with URANS k- ϵ and EB-RSM. With Lag-EB and LES, significant cross-flow is observed at the outlet, with Lag-EB also showing a recirculation zone at the right wall.

The difference in amount of cross-flow through the tube columns predicted by the various turbulence models is further visualized in Figure 61, which shows the time-averaged horizontal velocity on the central plane for all the turbulence models used. Negative values indicate rightward flow, while positive values indicated flow to the left. As can be seen, there is very little cross-flow for both URANS k- ϵ and IDDES, consistent with what was observed in

Figure 60. There is quite significant cross-flow for the other three turbulence models, with EB-RSM and LES predicting rightward (negative) cross-flow, and Lag-EB leftward (positive) cross-flow. In particular LES predicts strong cross-flow within a large section of the outlet zone. More predominant 3D flow is expected at the relatively high Reynolds numbers under investigation here (Zwijzen, 2025).

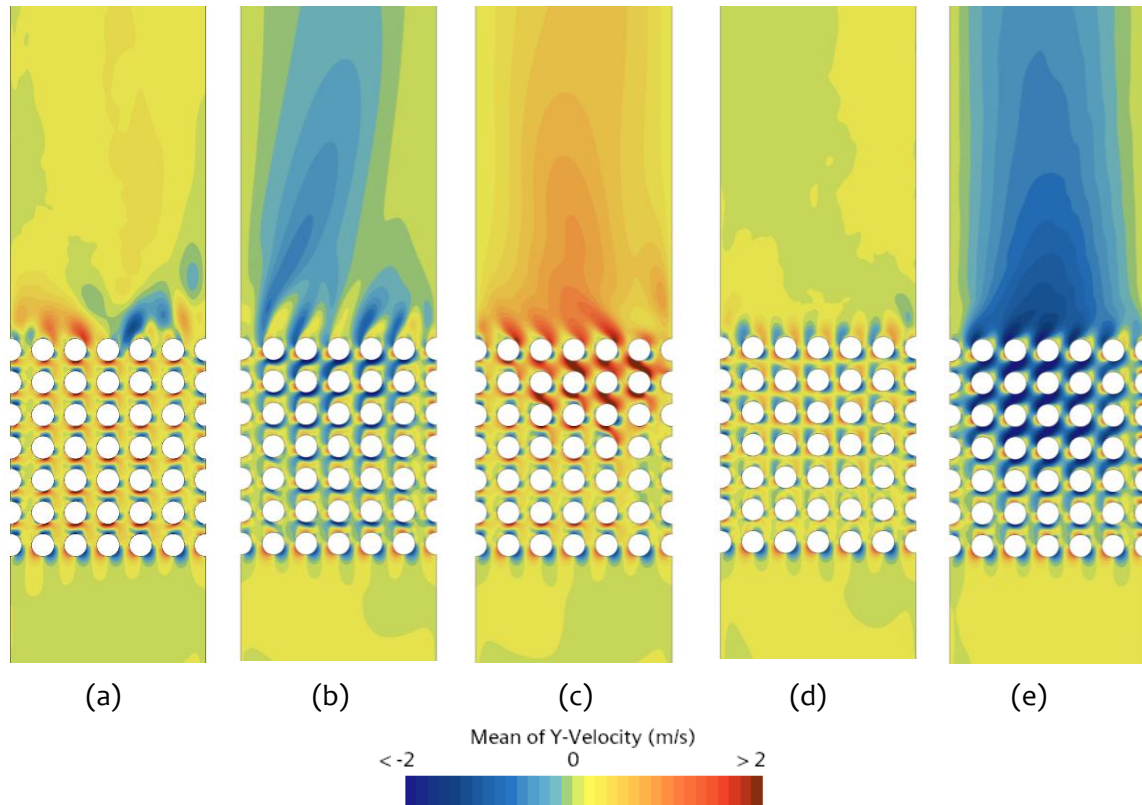


Figure 61: Time-averaged horizontal velocity, capped between -2.0 m/s and 2.0 m/s, for a) URANS k- ϵ , b) EB-RSM, c) Lag-EB, d) IDDES and e) LES

The third time-averaged velocity component, in the spanwise direction, is shown in Figure 62, with positive values being upward (out-of-plane) flow and negative values downward (into-plane) flow. It is interesting to see that both URANS k- ϵ and IDDES also predict the least spanwise motion, with especially URANS k- ϵ predicting very little spanwise flow. This indicates that these models mainly predict straight flow through the tube bundle, without much three-dimensional mixing taking place, in line with the strong inner-column jets observed in Figure 60. Very strong spanwise flow is observed with Lag-EB, which even gave values close to 1.0 m/s in magnitude, for both upward and downward flow. The spanwise velocities predicted by EB-RSM and LES are lower than those found with Lag-EB, but still significant. In particular, LES gives upward flow in the right half of the domain, and downward flow in the left half. Together with the other velocity components, this seems to indicate that LES predicts rightward and upward flow in the right half of the domain, and left- and downward flow in the left half of it.

Based on the above, it can be concluded that three predominantly anisotropic turbulence models, being LES, EB-RSM and Lag-EB, predict as expected more three-dimensional anisotropic flow than URANS k- ϵ and IDDES, at least in a time-averaged sense. To investigate the sensitivity of the solution to the length of the time-averaging, an additional simulation using LES was done, for which the simulation time was increased to 20.0 s. Time-averaging

was done over the final 18.0 s, meaning for a total of 9 FPs. Not much difference was observed between the obtained time-averaged fields. It should be noted here that, in order to have full confidence in the length of time-averaging period used to reach statistical stationarity, a much longer simulation should be performed, as it is known that flow through a tube bundle experiences bifurcation with a very low frequency (Benhamadouche & Benguigui, 2023, Zwijsen et al., 2025). Unfortunately, due to time and computational limitations, longer time averaging was not possible.

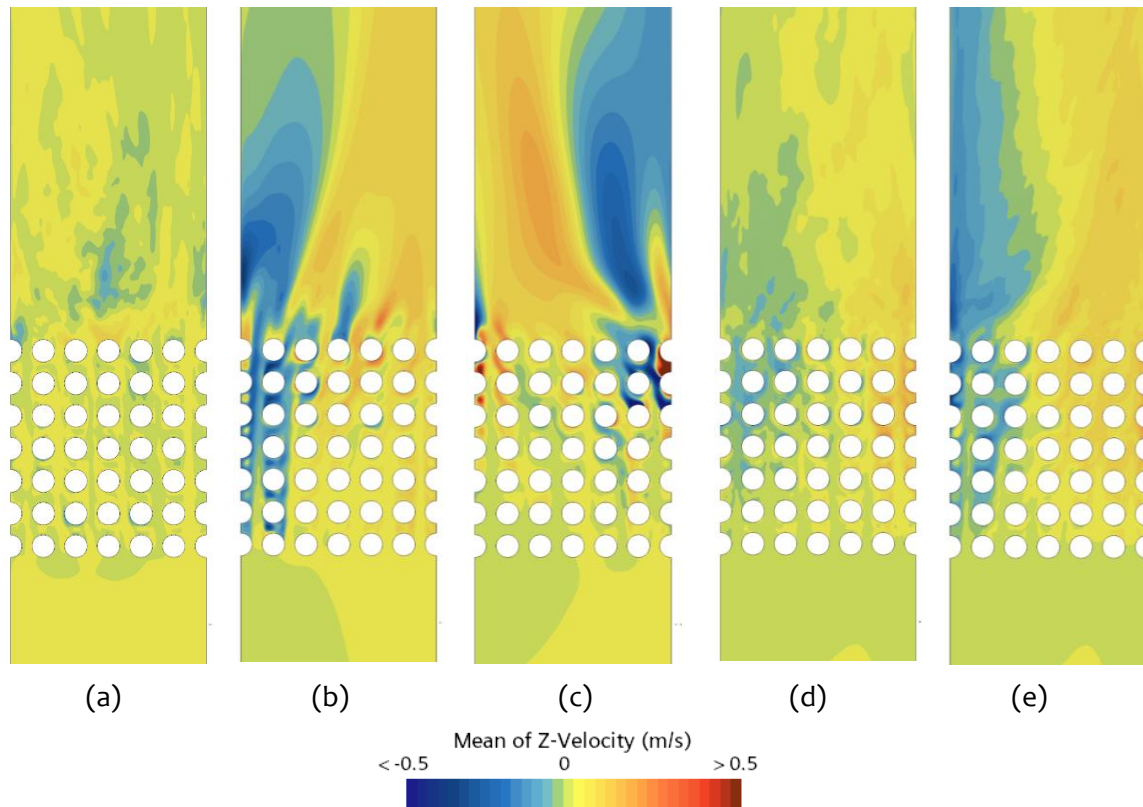


Figure 62: Time-averaged spanwise velocity, capped between -0.5 m/s and 0.5 m/s, for a) URANS k-ε, b) EB-RSM, c) Lag-EB, d) IDDES and e) LES

To get an indication of the instantaneous solution, Figure 63 shows the instantaneous vorticity magnitude on the same central plane through the tube bundle as used for the time-averaged velocity components. The vorticity magnitude has been capped at 1.0×10^3 /s for all turbulence models, to make the comparison clearer. There are once more significant differences between the different turbulence models. Lag-EB and EB-RSM predict very low amounts of vorticity, indicating that a limited range of scales of turbulence are resolved. More small scale structures are resolved with the other three turbulence models. URANS k-ε shows quite high levels of vorticity, though the structures are relative large and coherent. The other two show more smaller turbulent scales, with in particular LES resolving small scales, starting already very early inside the tube bundle.

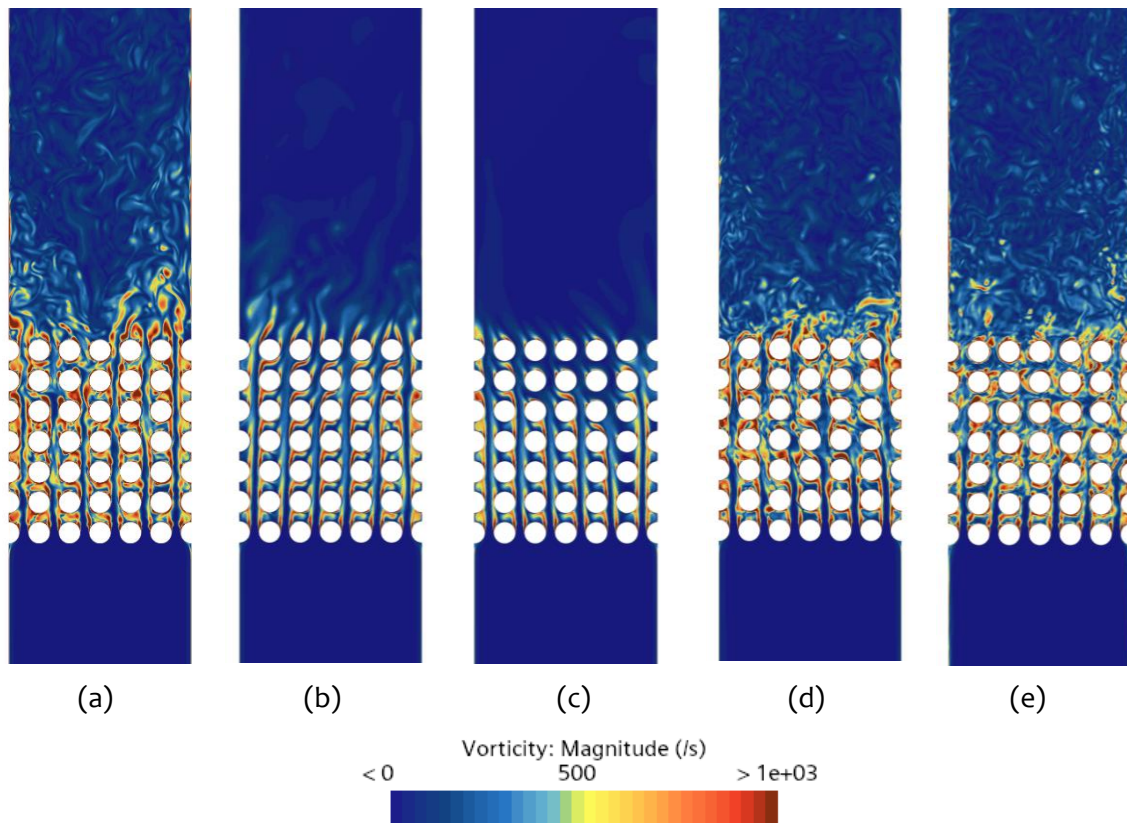


Figure 63: Instantaneous vorticity magnitude, capped at $1.0e+3$ /s, for a) URANS $k-\epsilon$, b) EB-RSM, c) Lag-EB, d) IDDES and e) LES

To understand the impact of the different observed flow fields on the forces, Figure 64 shows the time-averaged Lift Force Coefficient (LFC) spectra on the first four cylinders of the middle column as predicted by the different turbulence models. Here, the first row is the most upstream row of five full cylinders plus two half cylinders, and the fourth row is the central row. The bottom right figure thus shows the spectra of the lift force coefficient on the central tube inside the tube bundle, as indicated by Rod 1 in Figure 5. There is a clear distinction in magnitude of the LFC on all cylinders coming from the Lag-EB (orange) and EB-RSM (grey) models, and those from URANS $k-\epsilon$ (yellow), IDDES (blue) and LES (red), with the former models giving significantly lower LFCs than the others. This is in line with what was seen in Figure 63, as lower levels of turbulence (i.e., of vorticity) generally result in lower pressure fluctuations and thus lower forces on the tubes.

URANS $k-\epsilon$, IDDES and LES predict similar values of the LFCs for the lower frequency range, up to about 100 Hz. Above that frequency, the LFC values of LES are slightly higher than those of the other two. This indicates that LES resolves smaller turbulence scales, which was already observed in Figure 63.

Finally, it is interesting to see the various peaks in the LFC spectra predicted by the different turbulence models. These peaks generally indicate coherent structures, such as those resulting from vortex shedding. In particular Lag-EB predicts strong peaks, though it is unclear where these come from. The peaks predicted by the other models seem to be more consistent with vortex shedding, with peaks at higher frequencies being harmonics of the lowest frequency peak. It can also be seen that the magnitude of the peaks decreases going

downstream the tube bundle, with the highest peaks observed for the most upstream cylinder, while most peaks have disappeared for the central cylinder, except for Lag-EB, and EB-RSM to a lesser degree. In particular, LES shows very little to no peaks already at the third row. This indicates that any coherent structure, such as vortices being shed, break apart very quickly with the flow moving through the bundle, suggesting very little VIV of rods further down the tube bundle, with motion more driven by local turbulence.

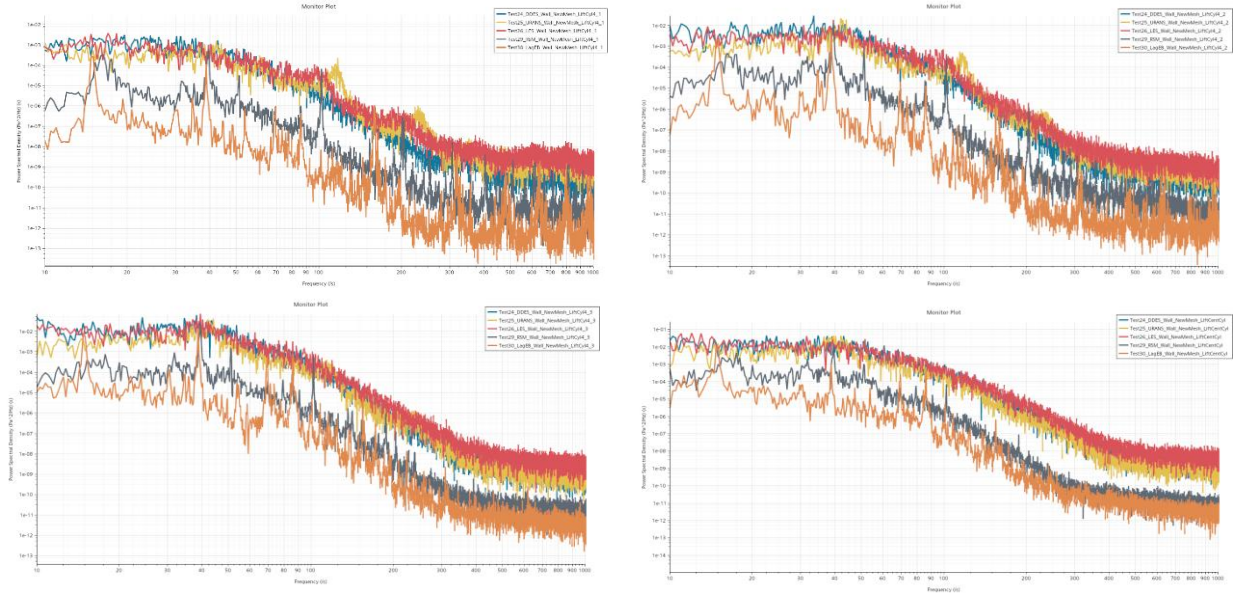


Figure 64: Time-averaged lift force coefficient on the central cylinder of row 1 (upper left), row 2 (upper right), row 3 (lower left) and row 4 (lower right) for URANS $k-\epsilon$ (yellow), EB-RSM (grey), Lag-EB (orange), IDDES (blue) and LES (red).

To understand the (primary) contributions to the force coefficients, Figure 65 shows the spectra of the lift and drag force coefficients based on the pressure stresses only, shear stresses only and total (pressure + shear) stresses on the central cylinder and using URANS $k-\epsilon$. As can be clearly seen, force coefficient values based on shear stresses alone are several orders of magnitude lower than those based on the pressure stresses. Hence, the pressure force is the primary contributing force that drives that motion of the cylinders.

The above spectra consider the forces over the full length of the cylinder. The forces are generally not equal over the cylinder's full length, but instead vary, due to the 3D nature of the turbulent flow. To investigate this, the forces have not only been tracked as function of time of the full cylinder, but also per axial layer of cells, i.e., over all 50 layers in spanwise direction. Using the Pearson correlation coefficient $\rho_{X,Y}$, which is defined as:

$$\rho_{X,Y} = \frac{\text{cov}(X,Y)}{\sigma_X \sigma_Y}, \quad (13)$$

with $\text{cov}(X,Y)$ being the covariance between X and Y , and σ_X and σ_Y the standard deviations of X and Y , respectively, it is possible to calculate the correlation between two sets of data, and hence also between the lift forces at two different locations along the length of a tube. Figure 66 shows the resulting Pearson correlation coefficient obtained with URANS $k-\epsilon$ for the lift force along the central tube's length. The three different curves use three different

locations as reference, i.e., three different locations for the variable X . For each curve, all other 49 locations are used for the variable Y . The curve is centered on the reference location, to demonstrate similarity in the decorrelation. The result of this is that some curves extend further to the left or right, though all have a total range of 0.2 m, being the tube length.

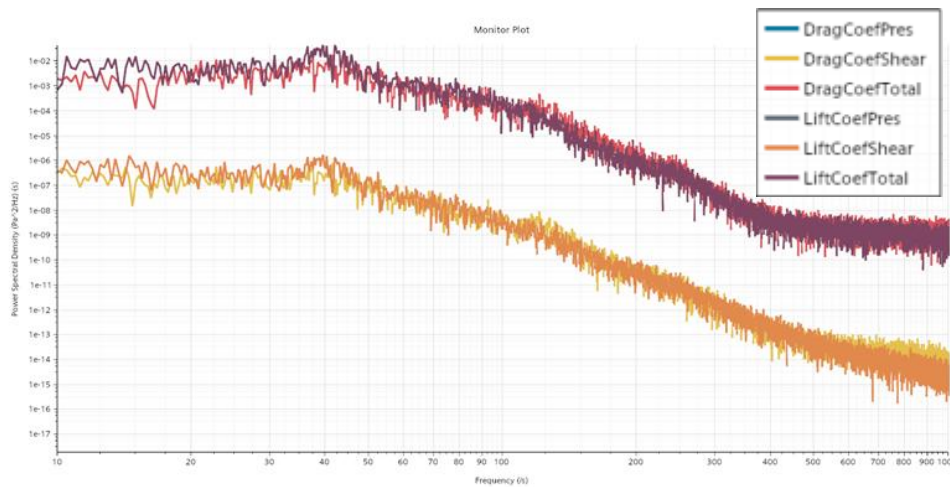


Figure 65: Time-averaged total drag and lift force coefficients, and the resulting contributions from the pressure (normal) and shear (tangential) stresses, on the central cylinder in the tube bundle as predicted by URANS $k-\epsilon$.

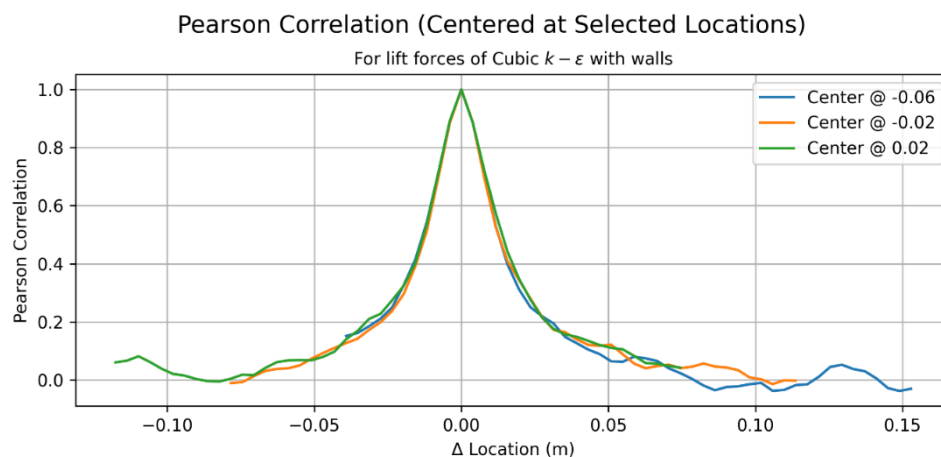


Figure 66: Pearson correlation coefficient between the lift forces along the central tube obtained with URANS $k-\epsilon$. Different lines indicate different locations used as reference location.

As can be clearly seen in the figure, all three curves overlap very well, showing near decorrelation ($\rho_{X,Y} \approx 0$) after about 0.06 m, which equals three times the tube's diameter. A similar decorrelation length is also found for the drag forces. Similar decorrelation lengths are found for IDDES and LES, as can be seen in Figure 67, which shows the Pearson correlation coefficient obtained for all turbulence models, using all 50 locations as reference location. IDDES shows more predominant regions of blue, indicating negative correlation, though these are very low values in magnitude, corresponding to almost no correlation. The correlation patterns obtained with EB-RSM and Lag-EB show very strange behaviour, suggesting unphysical fluctuating correlation along the tube's length. In particular, Lag-EB shows very odd

patterns, with at some locations very quick decorrelation with neighbouring layers, and larger correlations further down the tube's axis.

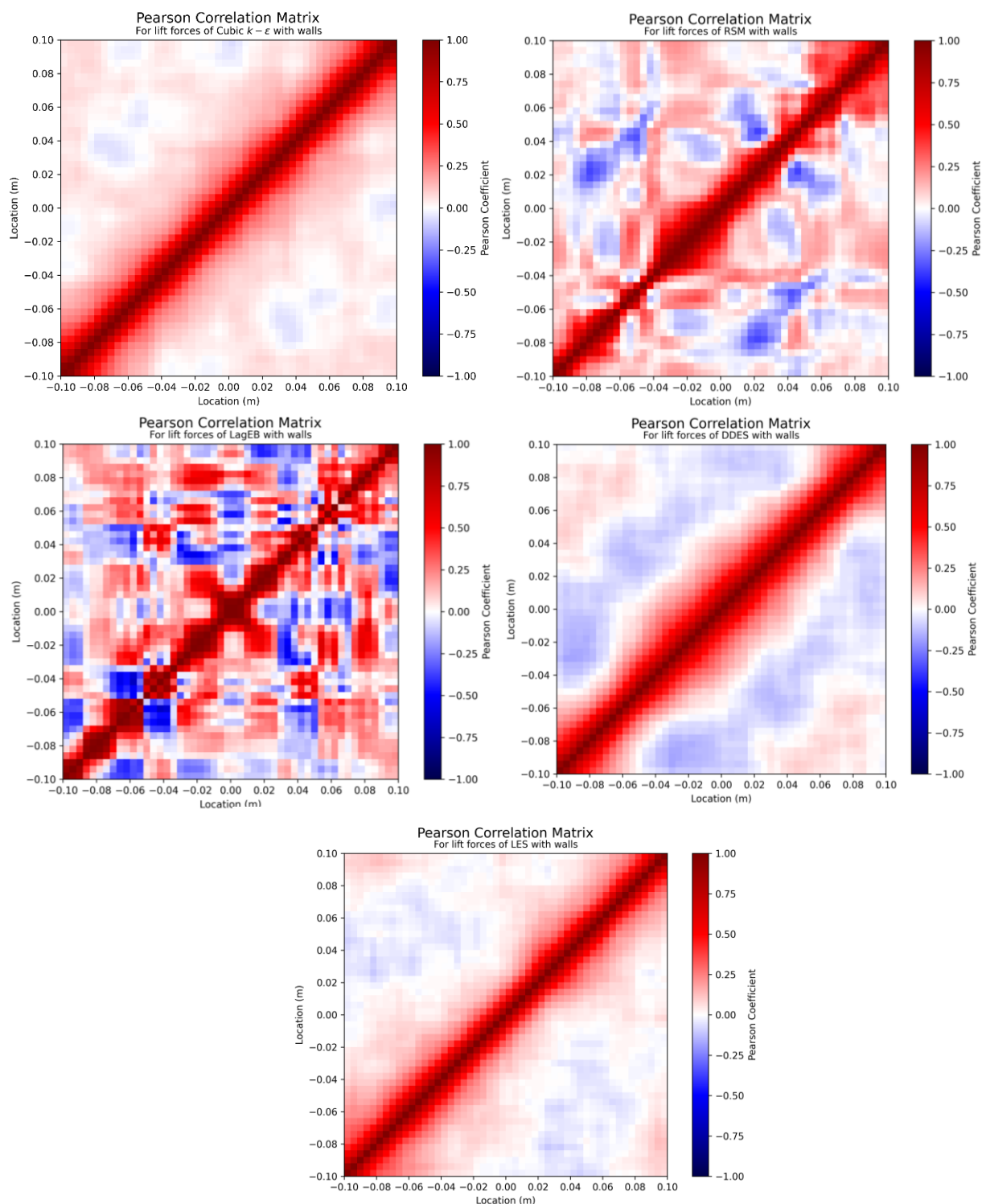


Figure 67: Pearson correlation coefficient between the lift forces along the central tube, for URANS $k-\epsilon$ (upper left), EB-RSM (upper right), Lag-EB (middle left), IDDES (middle right) and LES (bottom)

Besides spatial correlation, the forces are also correlated in time. Figure 68 shows the autocorrelation (correlation in time) of the lift forces coming from the different turbulence

models, plotted for three different locations along the tube. These plots have been time-averaged, using the time lag as reference quantity. URANS k- ϵ , IDDES and LES again give very similar autocorrelation plots, for all the three locations shown. And once more EB-RSM and Lag-EB give very strange correlation behaviour, showing fluctuation correlation in time, reaching values of upwards of 0.5 again later on.

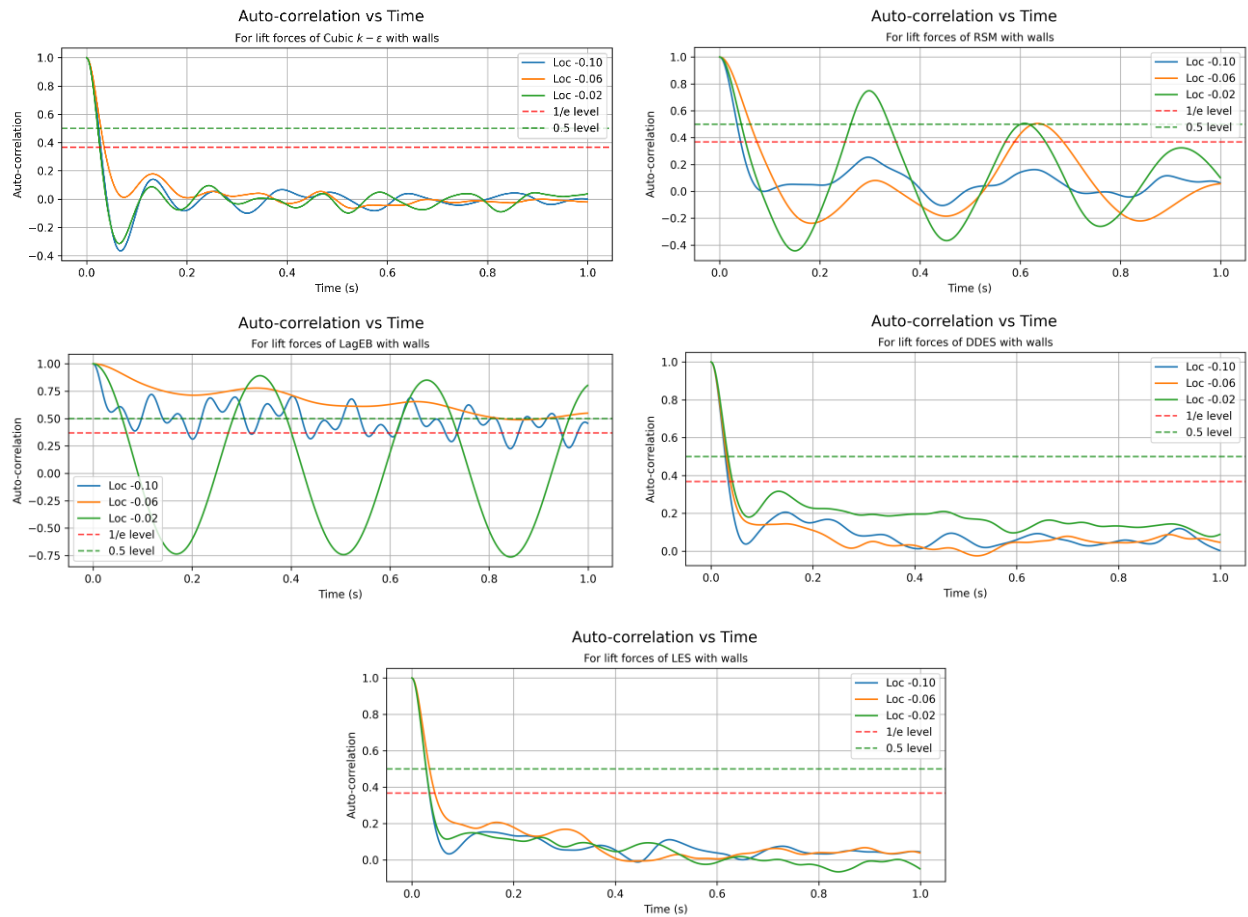


Figure 68: Autocorrelation of the lift forces at three different locations along the central tube, for URANS k- ϵ (upper left), EB-RSM (upper right), Lag-EB (middle left), IDDES (middle right) and LES (bottom).

To assess decorrelation time, the fall in autocorrelation below a specific value, such as below 0.5 or below $1/e$, is tracked. To better see this in the autocorrelation plots, horizontal lines were added for both quantities in Figure 68. URANS k- ϵ , IDDES and LES give very similar values for the decorrelation times, with values found of about 0.2s for the 0.5 level, and around 0.3 for the $1/e$ level.

The decorrelation times for each location along the tube are extracted, resulting in the plots as shown in Figure 69. URANS k- ϵ and LES look very similar, with increased decorrelation times at a distance of the tube's quarter length from the walls. It is not fully clear why this is, though it could very well have to do with the side walls, and the boundary layers formed there. These will suppress the formation of turbulent structures there, and will result in increased flow in the bulk area of the domain. The decorrelation times found with LES are slightly higher though than those predicted by URANS k- ϵ . A similar pattern is found by the EB-RSM. The values are however much larger. Lag-EB again gives strange behaviour, while IDDES gives mostly constant

decorrelation times along the full length of the tubes, with only lower values found near the walls.

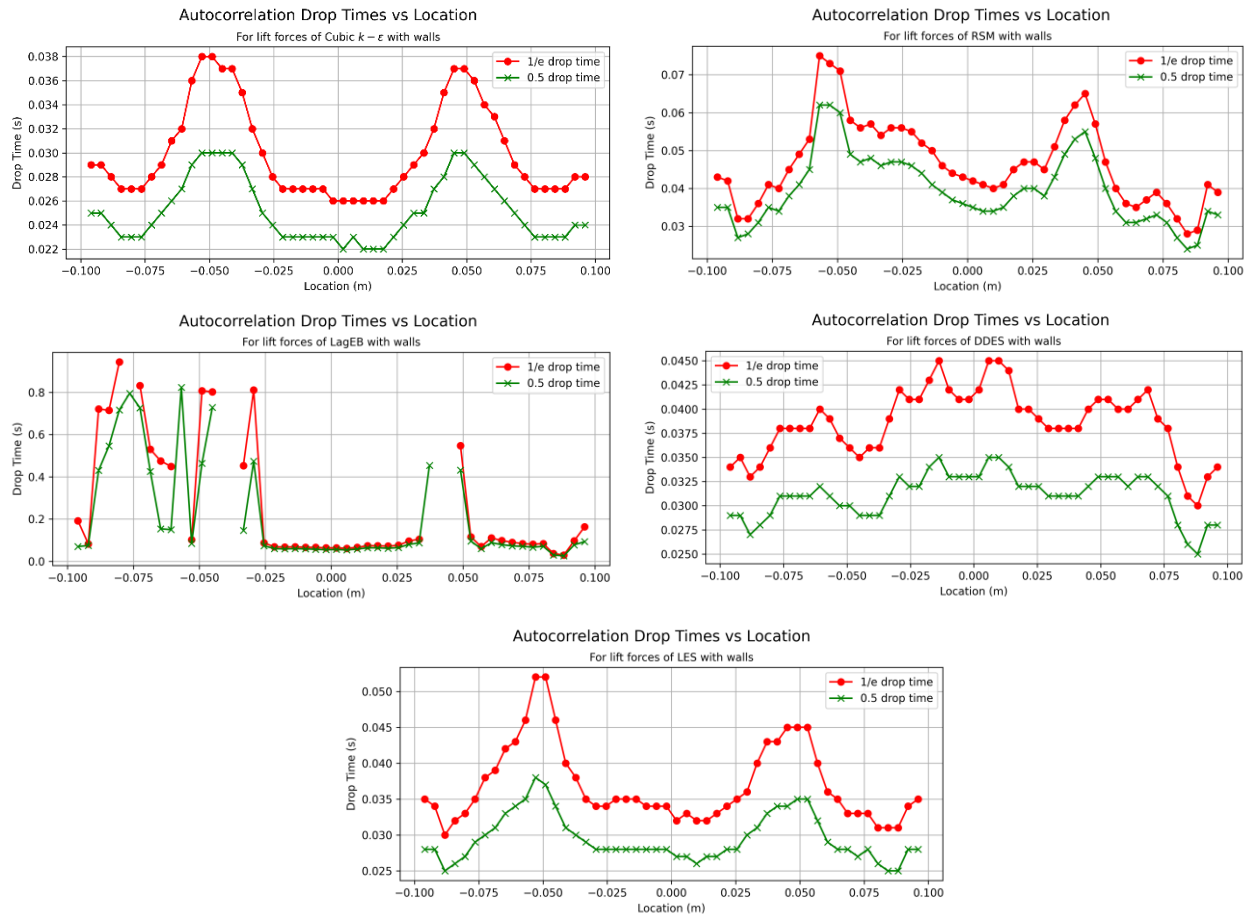


Figure 69: Autocorrelation drop times of the lift forces as function of the location along the central tube, for URANS $k-\epsilon$ (upper left), EB-RSM (upper right), Lag-EB (middle left), IDDES (middle right) and LES (bottom).

As a final comparison between the different turbulence models, Figure 70 shows the pressure coefficient on all seven tubes in the central row of the tube bundle as simulated using each of the five turbulence models. A general pattern between all of the models is that the pressure coefficient decreases downstream the tube bundle. This is a result of frictional pressure losses. What is more interesting to see, however, is how the pressure coefficient changes along the tube's length with each increasing tube column. On the first tube, the pressure coefficient is relatively uniform along its full length. As the water moves down the tube bundle, regions of high and low pressure coefficient values are found over the length of the tube. Higher values are found at the upstream side of the cylinder. This is consistent with the formation of a stagnation point near the side of the cylinder facing the flow. On top, lower values are found, as is generally seen for flow around a cylinder. These zones of high and low values of the pressure coefficient are not consistent across the full length of the cylinder however, and in particular URANS $k-\epsilon$, IDDES and LES show variation of the pressure coefficient along the tube's length, with lower values of the pressure coefficient found at the wall than in the central part of the tube.

Based on the above, it can be concluded that the EB-RSM and Lag-EB simulations give quite unreliable results. Hence only the URANS $k-\epsilon$, IDDES and LES models will be used for the FSI simulations.

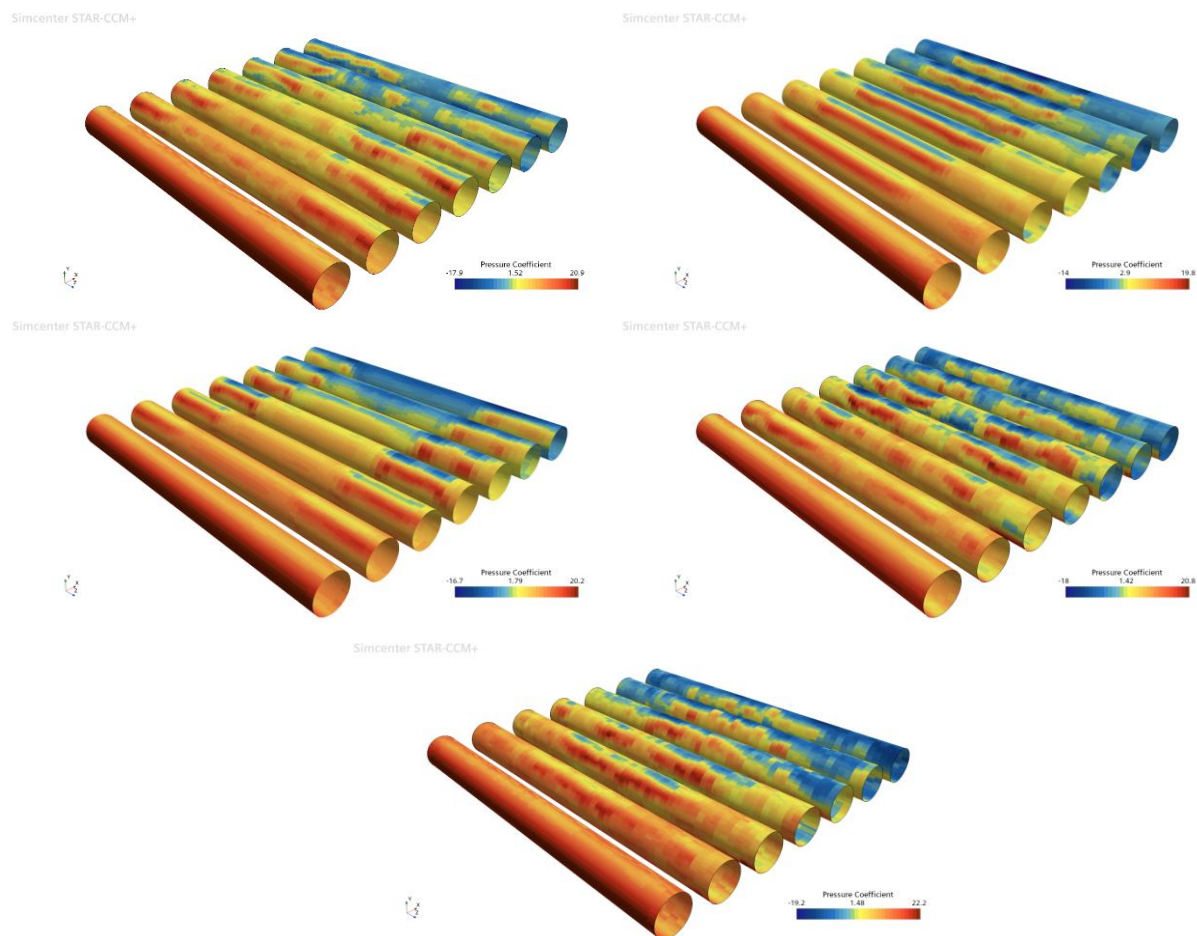


Figure 70: Time-averaged pressure coefficient on the central row of tubes for URANS $k-\epsilon$ (upper left), EB-RSM (upper right), Lag-EB (middle left), IDDES (middle right) and LES (bottom).

One open question remaining is whether to include the full tube's length in the simulation, together with the side walls, or if reduced lengths of the tubes can be used along with periodic side conditions. In particular, Figure 67 suggests the forces are no longer correlated after a distance of about 2-3D, suggesting that using a tube length of 5D would be sufficient. To study the effect of periodic boundary conditions, the simulations with URANS $k-\epsilon$ and LES were repeated, using periodic boundary conditions in the spanwise direction instead of walls. The fully length of the tubes, being 10D, was kept.

Figure 71 shows a comparison for the time-averaged streamwise velocity obtained with URANS $k-\epsilon$ (left two images) and LES (right two images), using walls in spanwise direction (first and third image) and periodic boundary conditions (second and fourth plot). Using periodic boundary conditions seems to have a pronounced effect on the flow field, resulting in a much more symmetric flow across the height of the tube bundle than what is obtained with actual side walls. There is also much less cross-flow between the tube rows. This seems to suggest

the flow becomes much more unidirectional, in the streamwise direction. This is also apparent when looking at the other velocity components (not shown here for brevity).

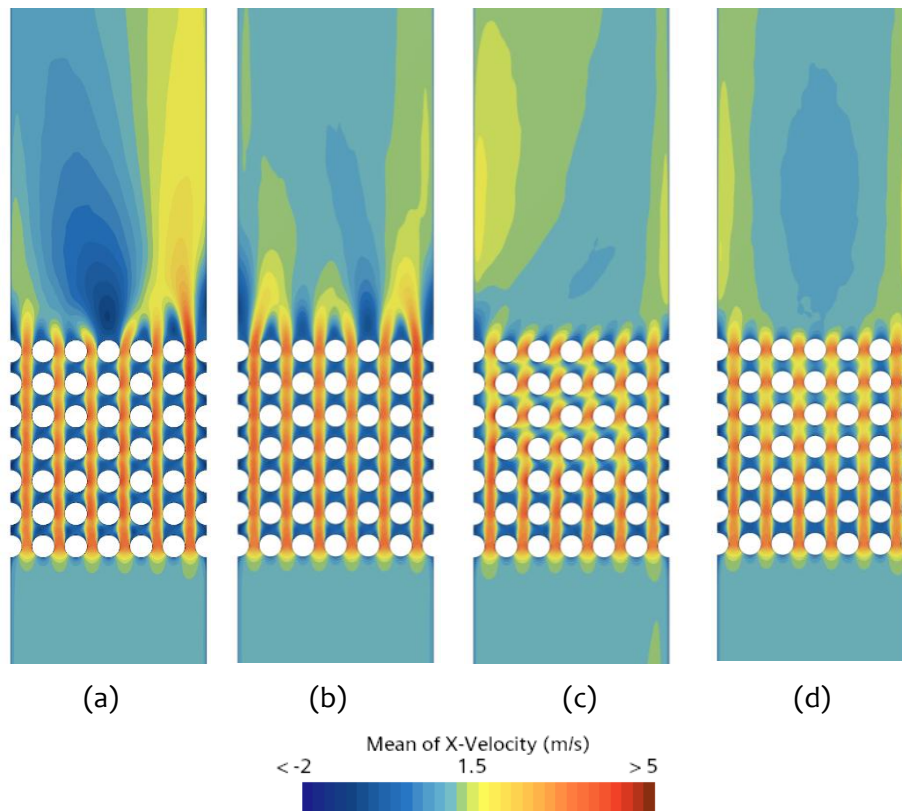


Figure 71: Time-averaged streamwise velocity, capped between -2.0 m/s and 5.0 m/s, for a) URANS $k-\epsilon$ + walls, b) URANS $k-\epsilon$ + periodic, c) LES + walls, d) LES + periodic.

While the influence of the spanwise boundary conditions is minimal on the spatial correlation of the forces, there seems to be a clear influence on the autocorrelation, as shown in Figure 72. URANS $k-\epsilon$ with periodic spanwise boundaries predicts a more or less uniform decorrelation time along the length of the tubes. LES, on the other hand, now peaks in the middle, with relatively consistent values towards the walls. The overall decorrelation time values have also increased compared with walls as boundaries.

As a final comparison between the two different options for boundary conditions in the spanwise direction, Figure 73 compares the pressure coefficient on the central row of tubes for URANS $k-\epsilon$ and LES with walls on the side (left) and periodic boundaries (right). These plots seem to further illustrate that using periodic boundary conditions results in a more uniform flow through the bundle, with less variation in spanwise direction, as the simulations with periodic boundary conditions show less variation along the lengths of the tubes than no-slip walls do.

Based on this, for the FSI simulations, the no-slip condition will be used on the walls in spanwise direction. To match the experiment, the cylinder can slide along these walls. This better mimics the actual experiment, in which the full cylinder displaces due to the springs attached to its ends.

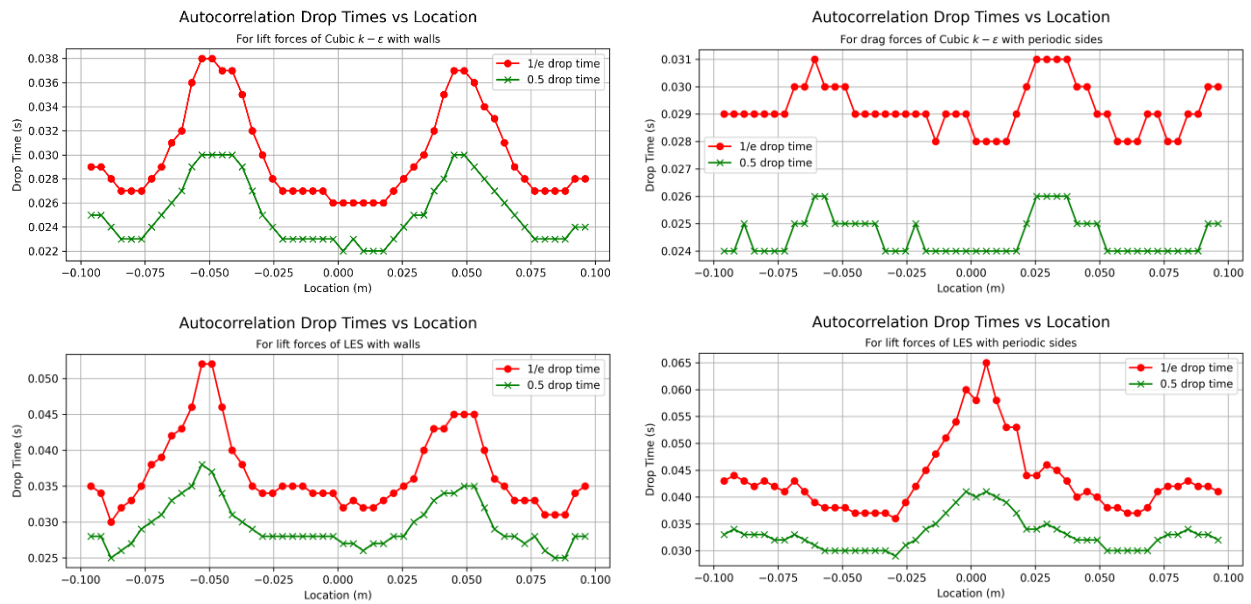


Figure 72: Autocorrelation drop times of the lift forces as function of the location along the central tube, for URANS $k-\epsilon$ (up), and LES (bottom), with side walls (left) and periodic sides (right).

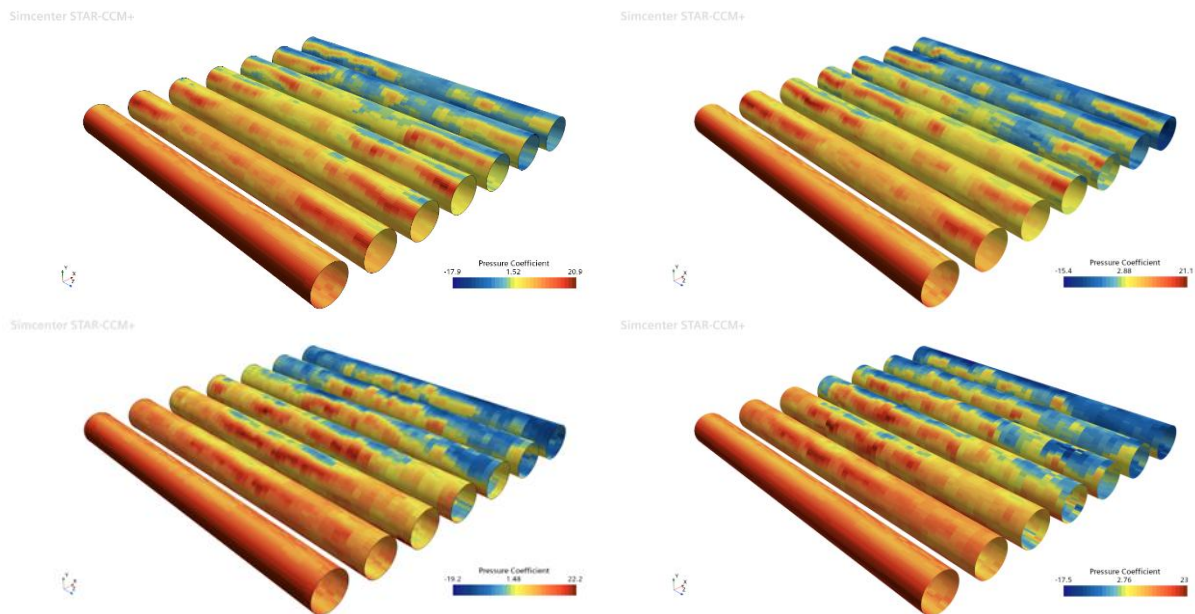


Figure 73: Time-averaged pressure coefficient on the central row of tubes for URANS $k-\epsilon$ (up), and LES (bottom), with side walls (left) and periodic sides (right).

6.4.2 Inlet velocity of 0.125 m/s – comparison with experiment

In this section, simulation results are compared with the experimental data for an inlet velocity of 0.125 m/s. Based on the results presented in the previous section, this will be done for the following three turbulence models:

- URANS k- ϵ with cubic constitutive relationship (URANS k- ϵ)
- k- ω SST-IDDES (IDDES)
- LES using the Wall-Adapting Local Eddy-viscosity (WALE) subgrid-scale model (LES)

As mentioned in Section 6.3, the simulations were run for a total of 74.0s. This involves the following phases:

- T = 0.0 – 16.0 s: Flow development
- T = 16.0 – 40.0s: Obtain flow statistics
- T = 40.0s: Release time of cylinder
- T = 40.0 – 42.0s: Ramp time of cylinder (slow increase of fluid forces)
- T = 42.0 – 50.0s: FSI adjustment
- T = 50.0 – 74.0s: Obtain FSI statistics

Hence, results presented in this section are for $t = 16.0 - 40.0$ s. As a first comparison, Figure 74 compares the average velocity between the simulations and experiment as function of the x-coordinate on the $z = 0.0$ m plane inside the Inlet FOV plane (see Figure 2 and Table 2). Several things can be noticed. First of all, there is a good match between simulations and experiment for the velocity in the bulk region, even though the simulations slightly under-predict the velocity there. This is probably related to the second point, and that is that the simulation profiles look more developed than those in the experiment, as evidenced by the flatter and broader simulation profiles, with stronger drops nears the walls.

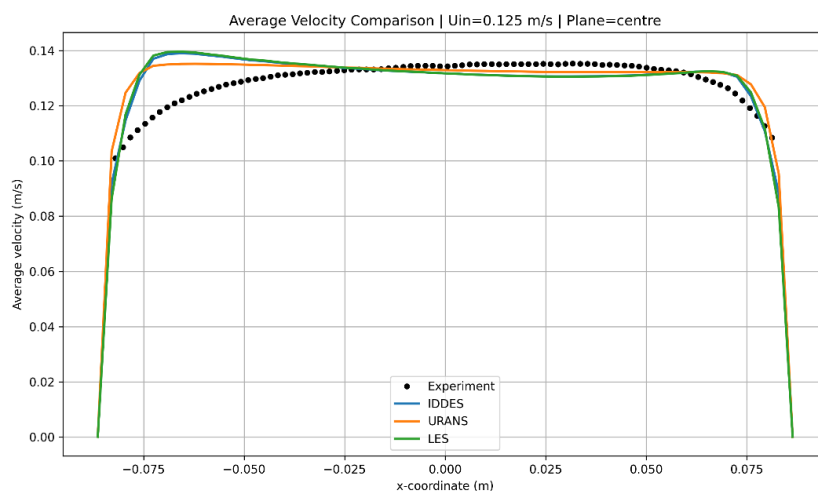


Figure 74: Comparison between experiment and simulations of the average velocity as a function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region, for $u = 0.125$ m/s.

Thirdly, it is interesting to see that the profiles are not symmetric for neither the experiment, nor the simulation. From the experiment, this probably is a result of the bend upstream the bundle. From the simulation, the cause is more unclear. Potential causes are insufficient time-averaging or a not fully symmetric mesh in x-direction, i.e., not fully symmetric on a yz-plane. Do note that the geometry of the simulation is fully symmetric, hence negative or positive x-values in the simulation hold little meaning, and the fact that the profiles of the simulation peak towards the left is just a plotting choice.

And lastly, there is little difference between the profiles predicted by the three different turbulence models, and in particular between IDDES and LES. Similar conclusions can be drawn for the velocity on the other two planes, as can be seen in Figure 75 for the near-wall ($z = 0.065\text{m}$) plane.

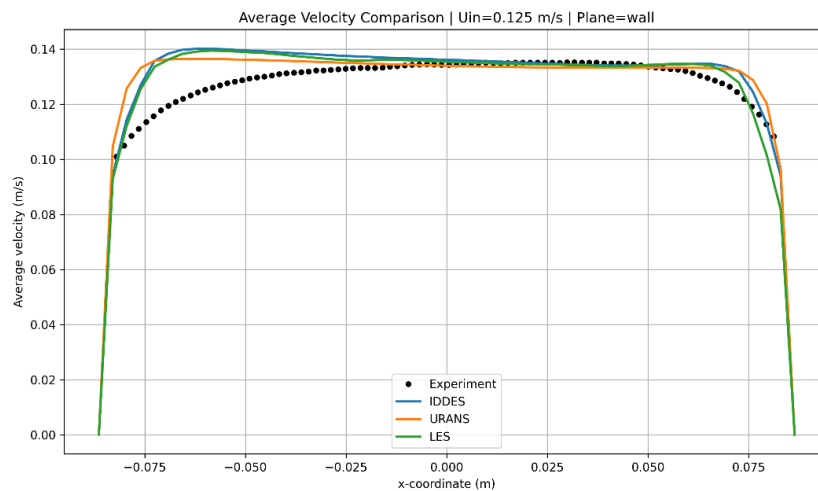


Figure 75: Comparison between experiment and simulations of the average velocity as a function of the x-coordinate on the near wall $z = 0.065\text{ m}$ plane in the inlet region, for $u = 0.125\text{ m/s}$.

As a next comparison, Figure 76 shows the normalized turbulent kinetic energy for the experiment and simulations as function of the x-coordinate inside the inlet region on the central plane. Normalization was done by dividing the calculated TKE by the square of the set inlet velocity, which is 0.125 m/s in this case. For all three simulations, the TKE includes both the resolved TKE, coming from the three different velocity fluctuation components, and the modelled TKE. For the experiment, since only two velocity components were measured, the assumption was made that the third component contains a similar turbulent contribution as the other two components. It should be noted that, due to spatial and temporal filtering inherent to PIV measurements, the experimentally measured TKE levels may underestimate the actual values.

As can be seen in the figure, the level of TKE in the bulk region is quite well recovered by all models, with especially URANS k- ϵ being close to the experimental value. The simulations predict much higher peaks though in the near-wall region than the experiment does, where higher turbulence levels are expected. These peaks are absent in the experimental results probably due to the frame of the test facility blocking access of the PIV lasers to the zones

close to the walls. A peak in the experiment is observed however for the right wall. Asymmetry is also again observed, in particular for the experiment and for IDDES and LES.

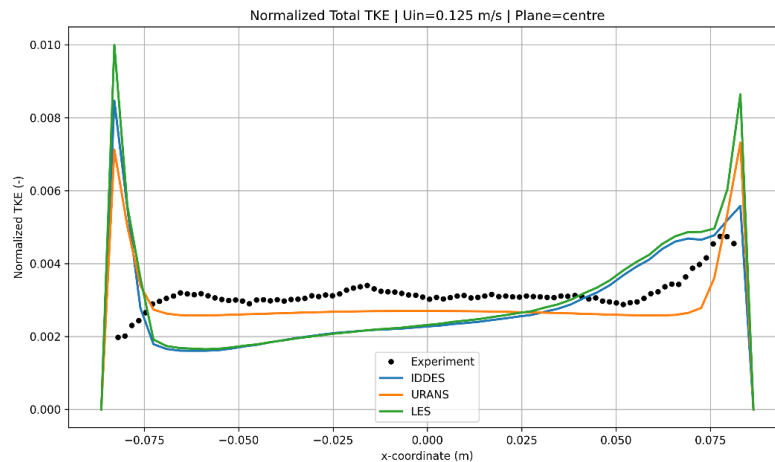


Figure 76: Comparison between experiment and simulations of the normalized TKE as a function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region, for $u = 0.125$ m/s.

To get an impression of the level of turbulence that is resolved versus modelled by each turbulence model, Figure 77 shows the modelled, resolved and total TKE profiles on the same line of Figure 76 for each of the turbulence models. There are some clear differences between the three turbulences models in terms of the amount that is modelled versus resolved. For URANS $k-\epsilon$, practically everything is modelled, with the resolved part being (almost) zero. This was to be expected, as the flow inside the inlet channel is undisturbed, hence no flow structures are present in the resulting predicted flow field.

For LES, all the turbulence seems to be resolved, with little to no contribution from the modelling part. This is a bit surprising, as it would indicate that the mesh is fine enough to resolve all turbulent scales. This is very unlikely to be true. It may be that the incorrect modelled TKE was stored. This needs further investigation though. As for IDDES, both the modelled as the resolved TKE contribute to the total TKE. The former contributes predominantly near the walls, where the URANS $k-\epsilon$ model is active, and is zero in the bulk region. In that bulk region, the contribution is (almost) fully from the resolved part.

To compare the simulations results of the flow field inside the tube bundle and outlet, velocity contours similar to those generated by the PIV are created in the different simulations. The results hereof are shown in Figure 78, which shows the time-averaged flow field inside the field of view indicated in the left of the figure for the experiment (far left), URANS $k-\epsilon$ (middle left), IDDES (middle right) and LES (far right), all scaled to the same range. The flow lines have been further visualized using streamlines, thereby using STAR-CCM+ Line Integral Convolution plotting option.

Comparing the simulations with the experiment, it can be seen that the simulations reproduce the main features of the flow quite well. All of them show strong flow inside the bundle lanes, with the jets meandering through the bundle, recirculation zones between the successive cylinders columns, and several jets merging at the outlet of the bundle, thereby leading to the

formation of large recirculation zones in the middle of the bundle. There are also some differences between the simulations. URANS $k-\varepsilon$ predicts, similar to the experiment, for the outlet recirculation zone to be above the central column of cylinders. IDDES and LES have this zone above one column further towards the wall. This recirculation zone is less wide in URANS $k-\varepsilon$ though than in the experiment. IDDES and LES predict wider zones, with especially IDDES finding a similar width as in the experiment.

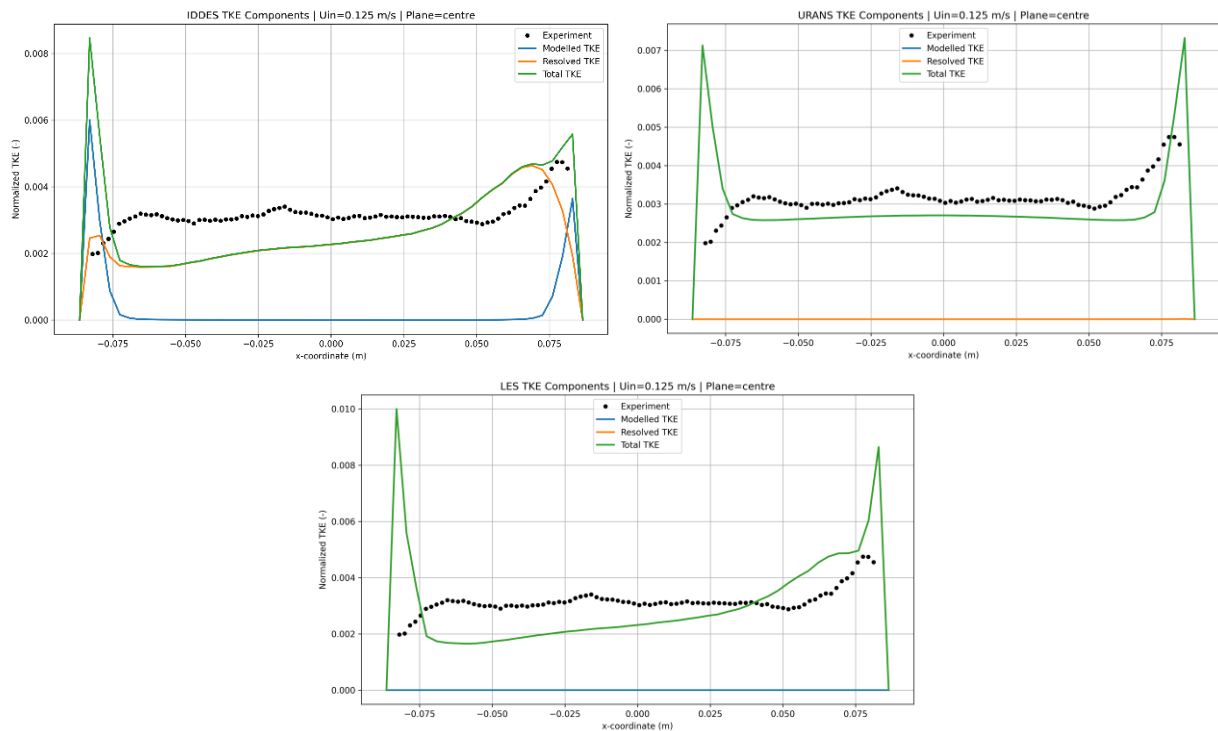


Figure 77: Modelled (blue), resolved (yellow) and total (green) TKE profiles as function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region for IDDES (upper left), URANS $k-\varepsilon$ (upper right) and LES (bottom), for $u = 0.125$ m/s.

Looking at the flow inside the tube bundle, the experiment predicts predominantly a single recirculation zone downstream tubes, while the simulations at times predict two vortices to co-exist in a space between two adjacent cylinders. The meandering of the jets inside the bundle is also not the same for all simulations. For URANS $k-\varepsilon$, it seems to go from left to right between each successive cylinder column. For the other two turbulence models, it can stay attached to a particular row of cylinders for more than a single column. This pattern is also seen in the experiment.

The normalized TKE is plotted on the same plane, and again compared to the experiment, in Figure 79. As to be expected, the largest amount of turbulent kinetic energy is found in the highly turbulent zones at the outlet, corresponding to the location where the jets are merged. The found levels of TKE in the simulation are quite consistent with those in the experiment, though they seem to remain somewhat higher downstream the bundle in the simulations than in the experiment.

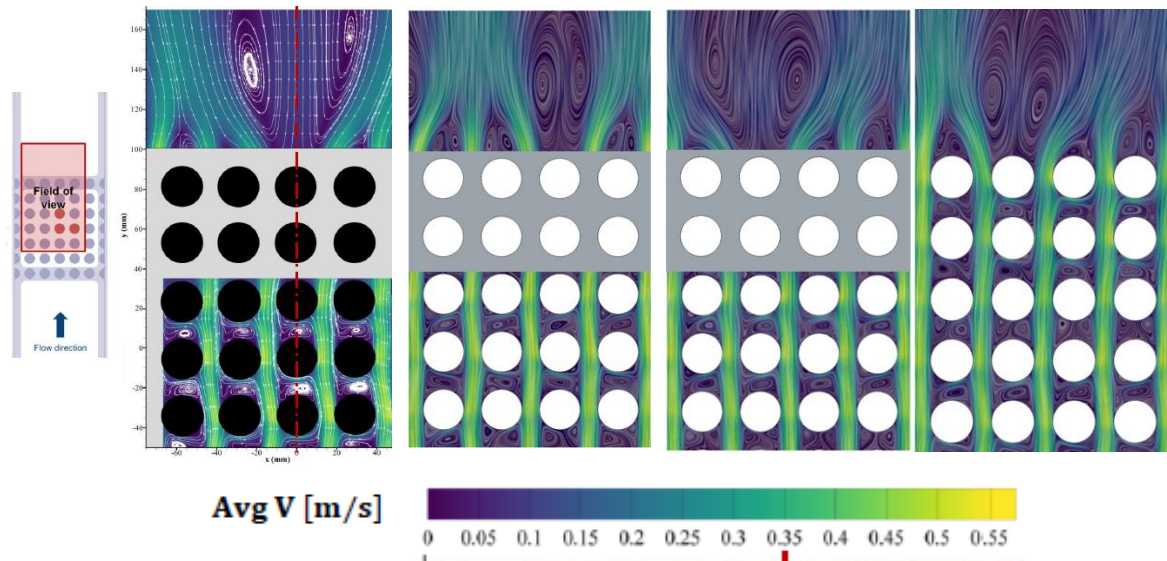


Figure 78: Comparison of the time-averaged velocity on a streamwise plane inside the test section between experiment (left) and simulation, for URANS k- ϵ (middle-left), IDDES (middle-right) and LES (right), for $u = 0.125$ m/s.

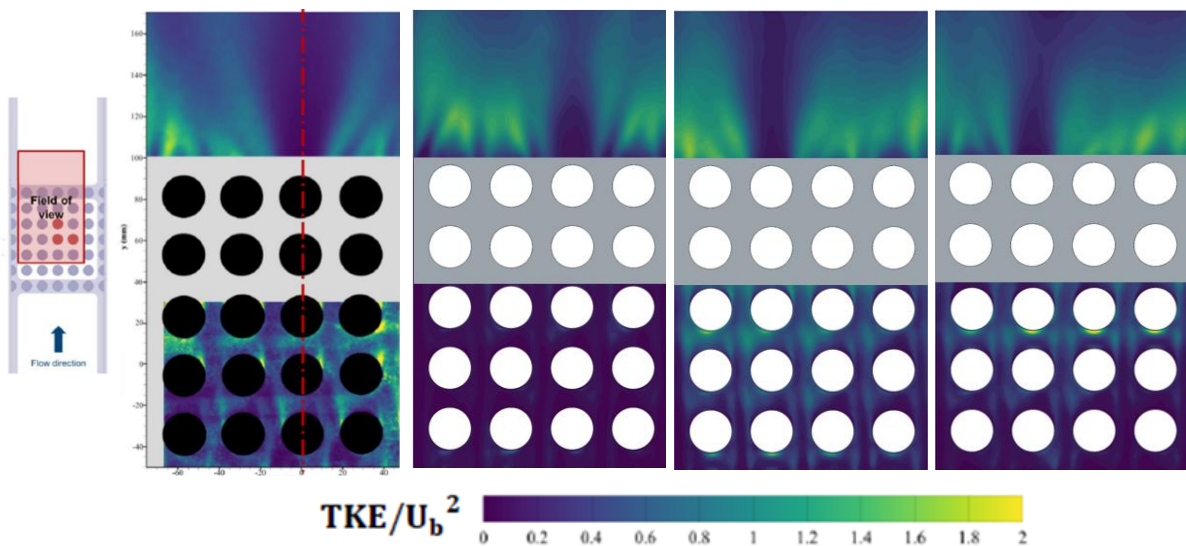


Figure 79: Comparison of the time-averaged normalized TKE on a streamwise plane inside the test section between experiment (left) and simulation, for URANS k- ϵ (middle-left), IDDES (middle-right) and LES (right), for $u = 0.125$ m/s.

Larger differences are observed within the tube bundle. URANS k- ϵ hardly predicts any turbulence in the bundle, with only minor levels of turbulence found in the later visualized rows. Higher levels of TKE are recovered by IDDES and LES, with especially the latter finding similar amounts of TKE in the bundle as the experiment, in particular at the stagnation zones of the last row of cylinders for which PIV data is available. Overall though, both for the velocity and turbulent kinetic energy, the simulations match the experiment reasonably well.

Line profiles inside the bundle and outlet region along the transverse direction of, amongst others, the velocity and turbulent kinetic energy are also available from the experiment at

various streamwise locations. Figure 80 shows a comparison between the experimental results and those obtained with URANS $k-\varepsilon$ (blue), IDDES (orange) and LES (green) for lines P4 till P7 (see Figure 4 for exact location of the profiles). There is quite a good match between the experiment and the simulations, for all profiles, both in the flow channels as in the spaces between successive cylinder rows. The flow magnitudes match the experiment quite well, with the simulations also predicting the secondary peaks between the flow channels at P7.

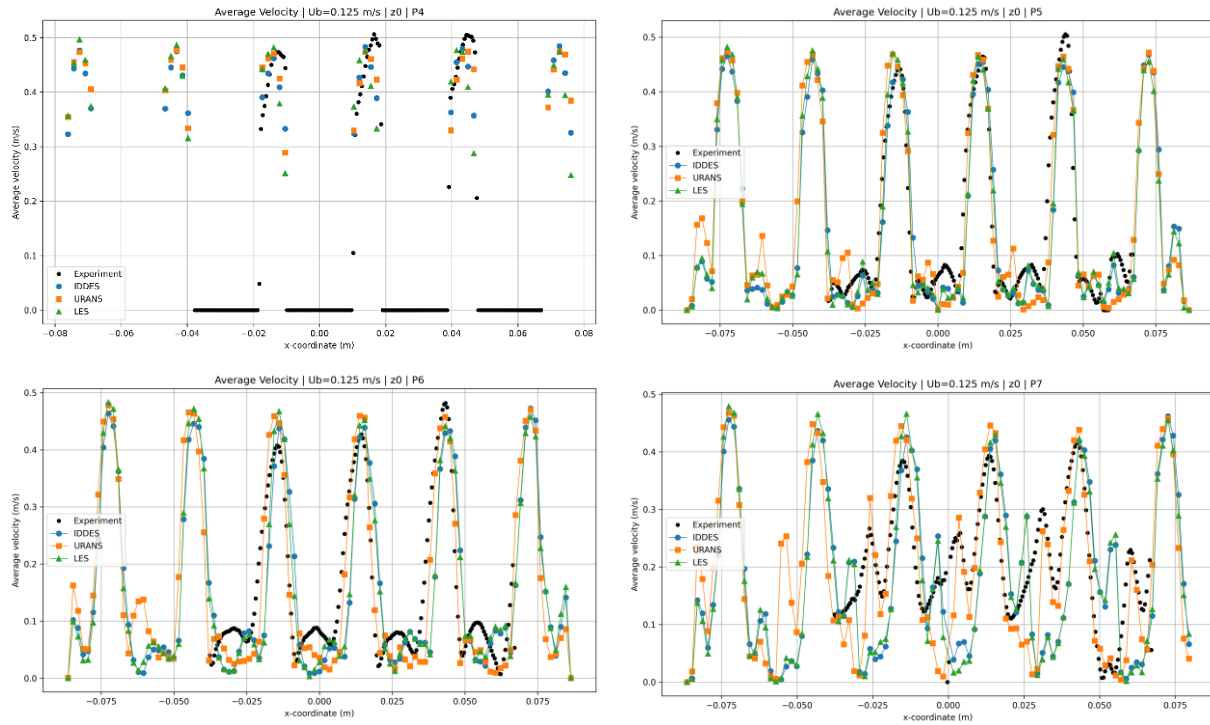


Figure 80: Comparison between experiment and simulations of the velocity inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right), for $u = 0.125$ m/s.

Looking at the TKE on the same lines, which are compared to the experiment in Figure 81, the simulations to capture the general trend of TKE, with larger values of the TKE found in the flow channels, as to be expected. There are differences between the three different turbulent models in terms of the level of TKE predicted, with IDDES finding levels closest to those measured in the experiment, with LES a close second. URANS $k-\varepsilon$ under-predicts the TKE quite significantly, which is consistent with the contour plots shown in Figure 79.

Figure 82 and 35 compares respectively, average velocities and TKE from the experiment with those from the simulations on the four different lines available in the outlet region. These plots largely confirm what was already observed from the contour plots of Figure 78 and Figure 79. For the velocity, the magnitudes are quite well predicted by the simulations, especially by IDDES and LES. URANS $k-\varepsilon$ generally overpredicts the found values in the experiment, probably due to there being more 3D flow in the experiment, and in the other two simulations, than in URANS $k-\varepsilon$, viz., more flow in transverse direction. On the other hand, the velocity profile as found by URANS $k-\varepsilon$ matches the experiment better, which is a direct result of IDDES and LES predicting the recirculation zone not to be located directly behind the central row, but one row shifted towards the wall. The width of the recirculation zone is better

recovered by IDDES and LES however, which also recover the bump in velocity in the recirculation zone better than URANS k- ϵ , in particular IDDES.

The shape of the velocity profiles in the outlet zone have obviously a direct impact on the shape of the corresponding TKE profiles. Profiles from URANS k- ϵ match those of the experiment best, but also over-predict experimental values the most. Profiles do flatten out further downstream the outlet zone, with lower levels of TKE found, both experimentally as numerically.

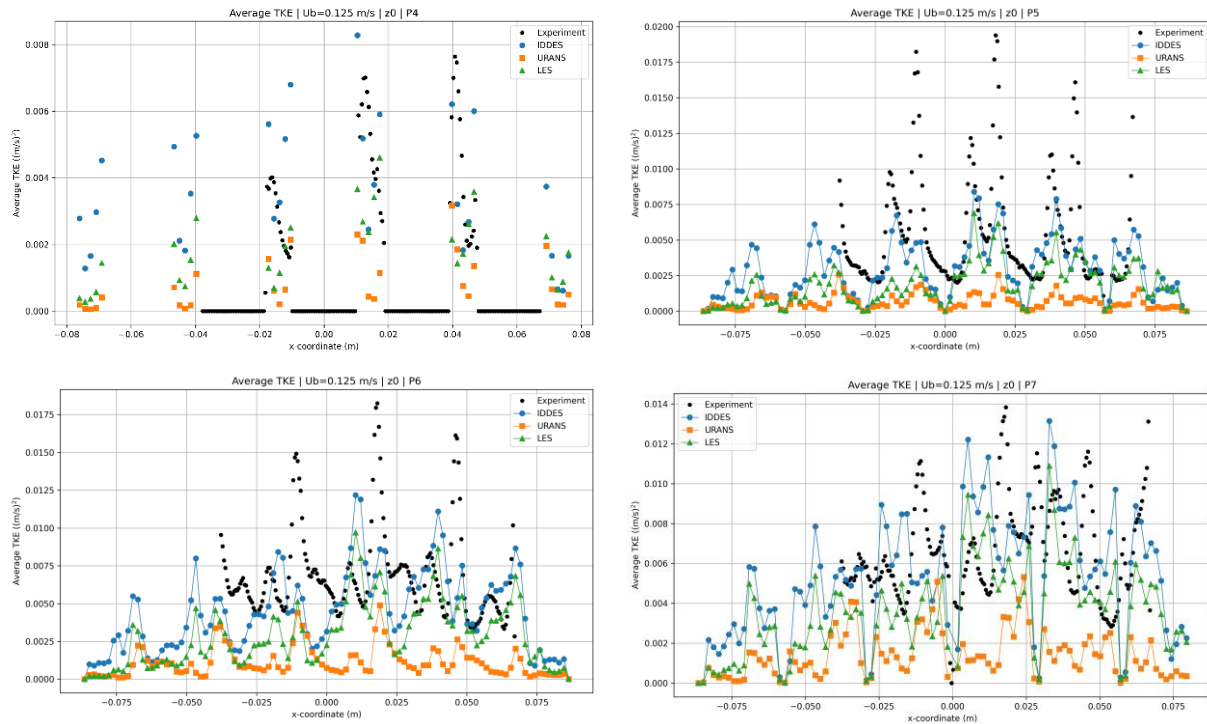


Figure 81: Comparison between experiment and simulations of the TKE inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right), for $u = 0.125$ m/s.

As final results obtained from the simulations, Figure 84 and Figure 85 show the Power Spectral Density (PSD) of the lift and drag force coefficients, respectively, obtained from the flow-only section of the FSI simulation on three cylinders of the central column of cylinders, viz., cylinder 4.1, which is the most upstream cylinder in this column, cylinder 4.4, which is the central cylinder in the bundle which later on will be allowed to vibrate, and cylinder 4.7, the most downstream cylinder. When looking at the PSD of the LFC, it is obvious that URANS k- ϵ predicts lower lift forces than the other two models, in particular at higher frequencies. This was also observed in Section 6.4.1. IDDES and LES predict similar levels of lift forces.

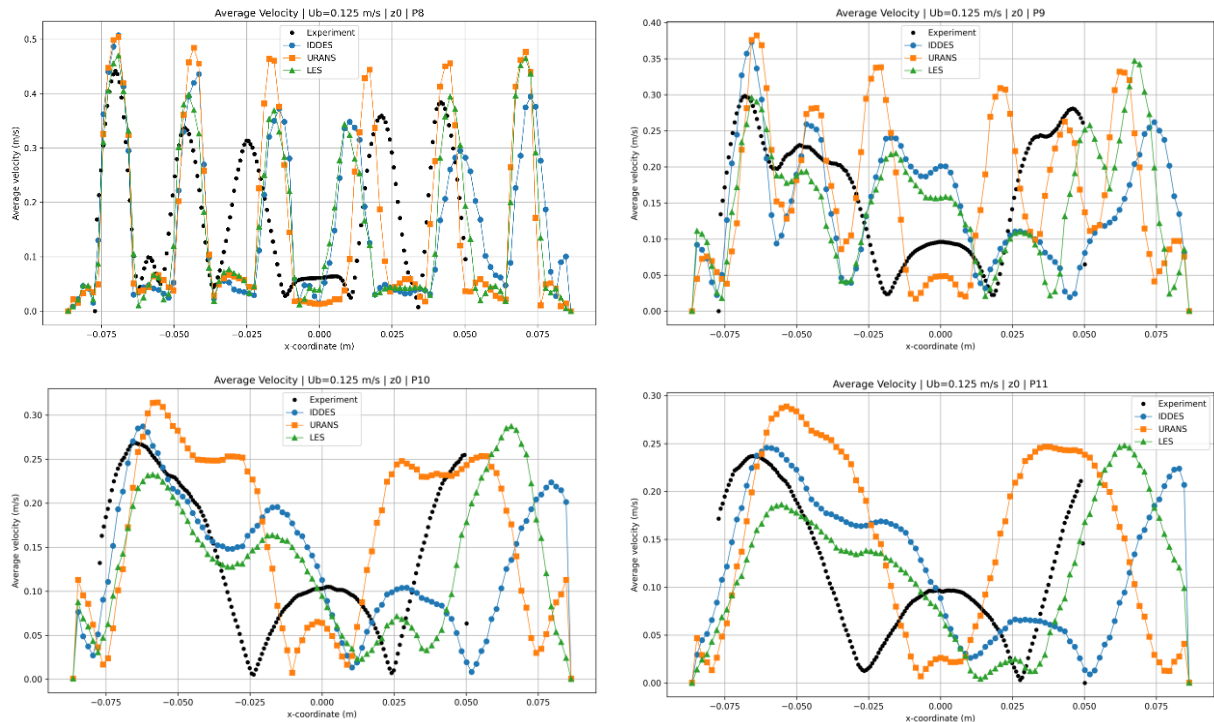


Figure 82: Comparison between experiment and simulations of the velocity in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right), for $u = 0.125$ m/s.

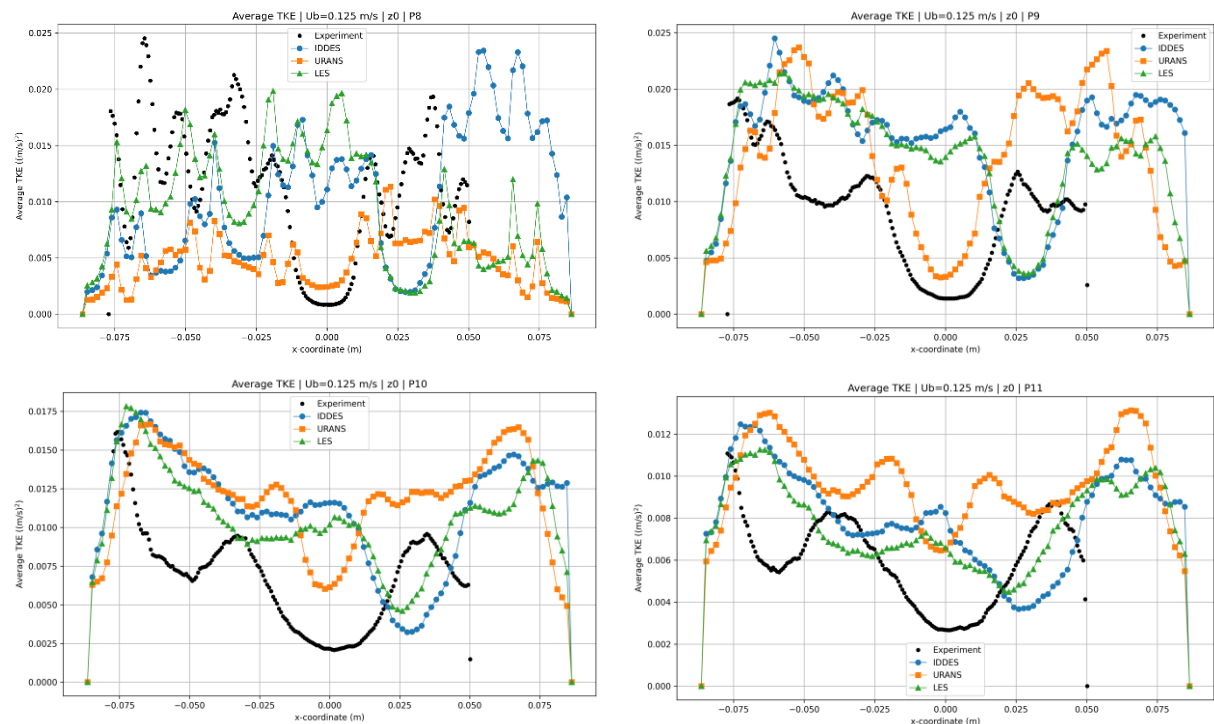


Figure 83: Comparison between experiment and simulations of the TKE in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right), for $u = 0.125$ m/s.

What is more interesting to see is that for the lift forces, the largest lift forces are found on cylinder 4.7, followed by cylinder 4.1, with cylinder 4.4 having the lowest fluid lift forces. This is probably caused by the fact that cylinder 4.1 is primarily subjected to vortex shedding, which generally results in significant levels of lift forces. These vortices do not form though on rows further downstream the bundle, due to the small pitch between cylinders. Instead, the initially formed vortices are broken apart, and turbulent structures are formed. The amount of turbulent intensity increases downstream the cylinder, resulting in larger turbulent forces on cylinders further downstream the bundle, such as cylinder 4.7 compared to cylinder 4.4.

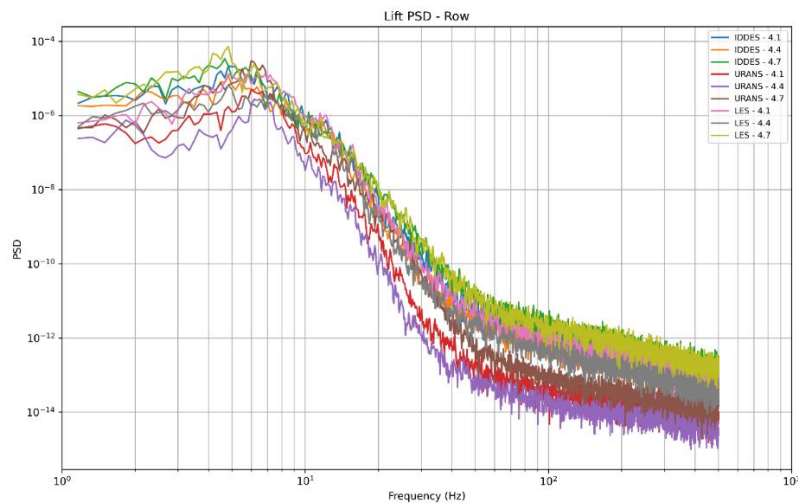


Figure 84: PSD of the lift force coefficient on the most upstream cylinder (4.1), central cylinder (4.4) and most downstream cylinder (4.7) of the central column of cylinders obtained by the three different turbulent models, for $u = 0.125$ m/s.

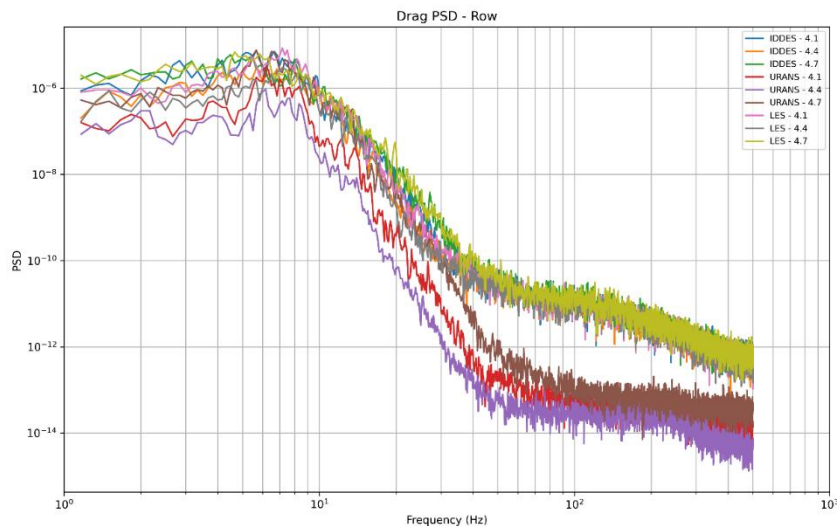


Figure 85: PSD of the drag force coefficient on the most upstream cylinder (4.1), central cylinder (4.4) and most downstream cylinder (4.7) of the central column of cylinders obtained by the three different turbulent models, for $u = 0.125$ m/s.

While the above is true for the lift forces, the drag forces remain more or less constant throughout the tube bundle, as a result of the more or less stable jet that passes through the tube bundle, which results in consistent levels of drag forces on all rows of cylinders.

6.4.3 Inlet velocity of 0.5 m/s – comparison with experiment

FSI simulations with an inlet velocity of 0.5 m/s were also performed. In this section, simulation results are compared with the experimental data for the flow only part of this FSI case, using the same three turbulence models as done for $U_b = 0.125 \text{ m/s}$. Due to the four times higher flow rate, the time for a single flow passage is reduced by about a factor of four. Therefore, the timing of the various phases of the simulation was also adjusted, hence:

- $T = 0.0 - 4.0 \text{ s}$: Flow development
- $T = 4.0 - 12.0 \text{ s}$: Obtain flow statistics
- $T = 12.0 \text{ s}$: Release time of cylinder
- $T = 12.0 - 14.0 \text{ s}$: Ramp time of cylinder (slow increase of fluid forces)
- $T = 14.0 - 18.0 \text{ s}$: FSI adjustment
- $T = 18.0 - 26.0 \text{ s}$: Obtain FSI statistics

Furthermore, to ensure CFL-values are still mostly below 1.0, the time-step size was reduced by a factor of four, i.e., to $\Delta t = 2.5 \cdot 10^{-4} \text{ s}$. A similar comparison will be done with the experiment for an inlet velocity of 0.5 m/s as was done for 0.125 m/s, focusing on line profiles in the inlet region, contour plots in the joint bundle-outlet region with also line profiles for each separate region, and finally some PSD plots of the forces on selected cylinders.

For starters, Figure 86 and Figure 87 shown again the time-averaged velocity and normalized TKE, respectively, coming from the simulations and compared with the experiment. Similar conclusions can be drawn as was done for an inlet velocity of 0.125 m/s, with the simulations' velocity profiles looking more developed (flatter bulk profiles) and their TKE profiles showing the expected peaks near the walls, which are absent in the experiment due to restricted access to the near-wall region.

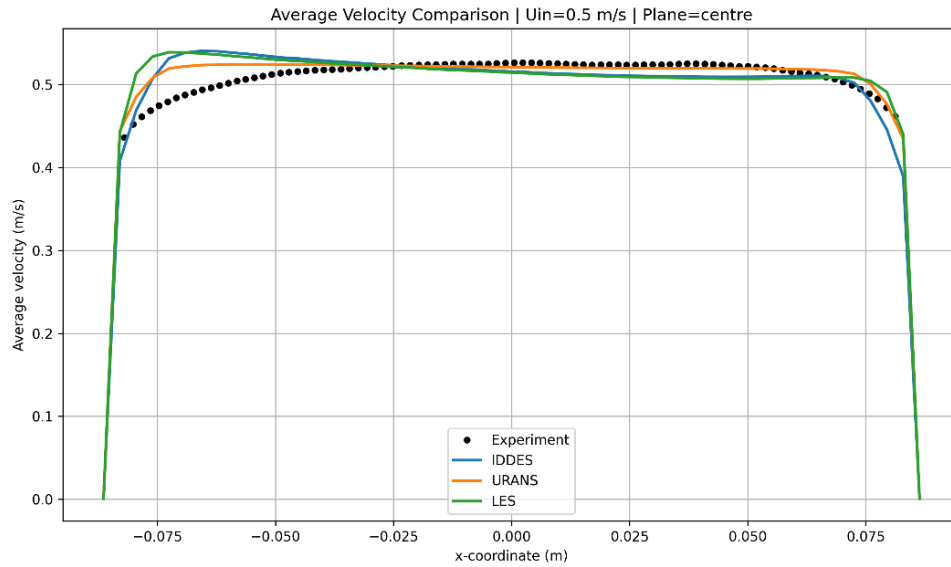


Figure 86: Comparison between experiment and simulations of the average velocity as a function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region, for $u = 0.5$ m/s.

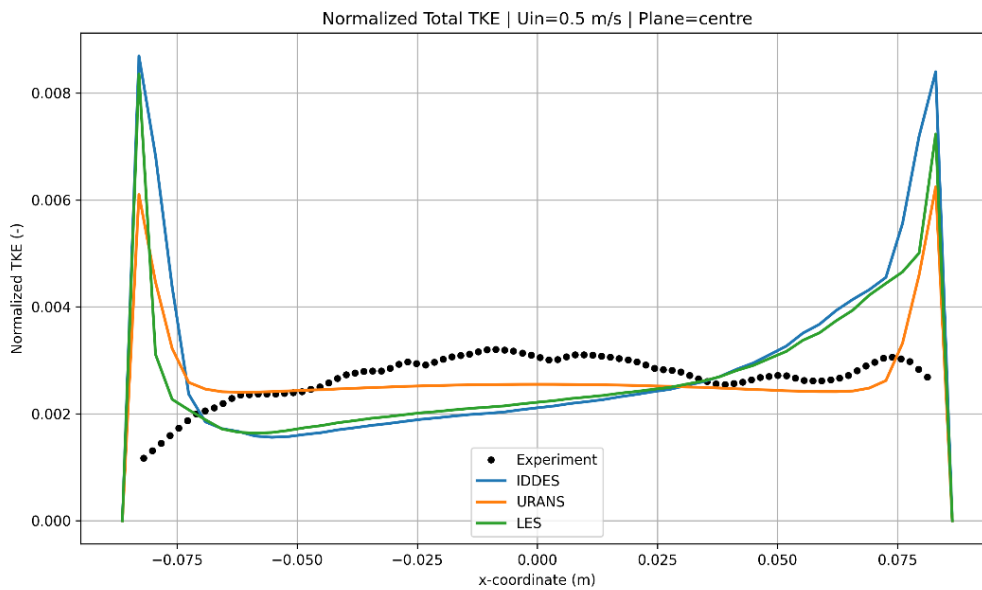


Figure 87: Comparison between experiment and simulations of the normalized TKE as a function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region, for $u = 0.5$ m/s.

Figure 88 shows the contributions of the modelled and resolved parts of the TKE of the simulations to the total TKE, matching again what was seen for $U_b = 0.125$ m/s, i.e., total TKE levels matching the experiment quite well, with URANS $k-\varepsilon$ having most TKE modelled while LES has most of it resolved. IDDES has contributions of both near the wall, and mostly resolved TKE in the bulk region.

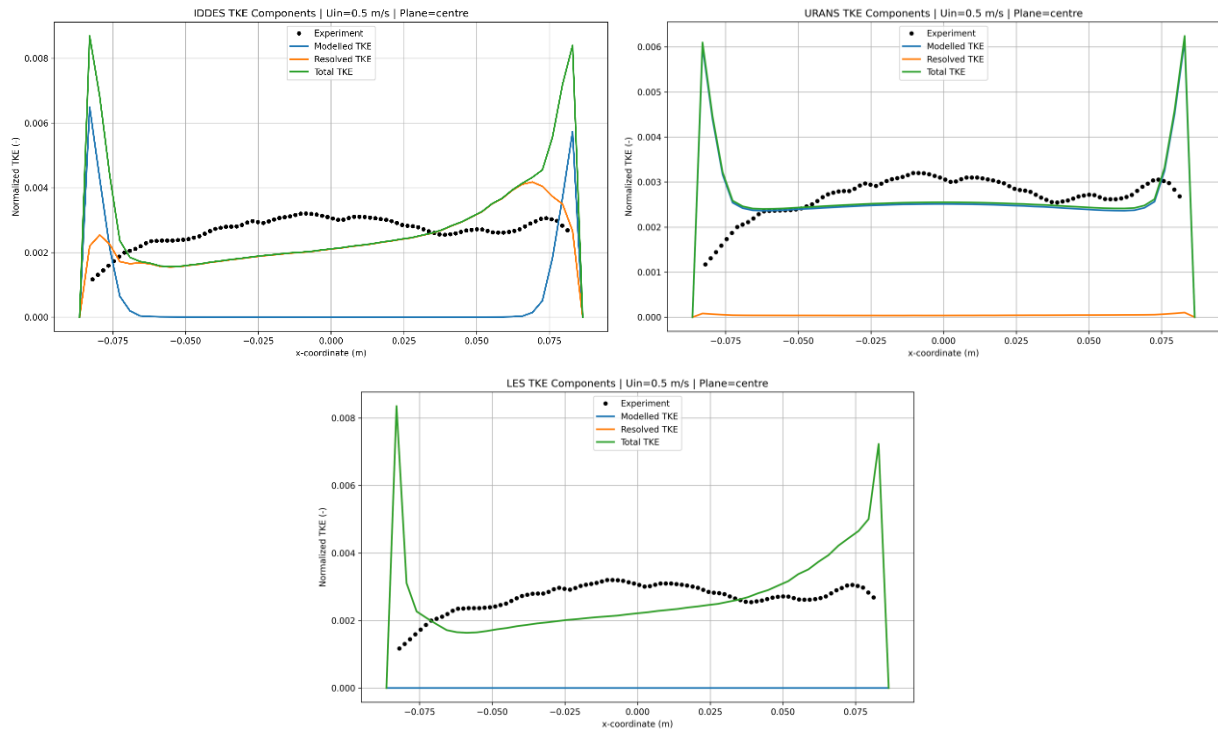


Figure 88: Modelled (blue), resolved (yellow) and total (green) TKE profiles as function of the x-coordinate on the central $z = 0.0$ m plane in the inlet region for IDDES (upper left), URANS $k-\varepsilon$ (upper right) and LES (bottom), for $u = 0.5$ m/s.

Next, Figure 89 and Figure 90 compare the time-averaged velocity and TKE, respectively, inside a selected 2D-window in the joint bundle-outlet region with the experiment. Some interesting observations can be made, especially also in comparison to the lower inlet velocity case. For $U_b = 0.5$ m/s the experiment no longer shows the formation of two merged jets at the outlet separated by a wide recirculation zone. Instead, the individual jets exiting the bundle remain separated. This changed outlet profile is captured by the LES, and also to a slightly lesser extent by IDDES. URANS $k-\varepsilon$, on the other hand, still predicts a similar outlet pattern as it did for the lower inlet velocity. There are also differences inside the bundle itself, with URANS $k-\varepsilon$ predicting a quite narrow jet inside the flow channels, while those in IDDES and LES are wider, and hence less high velocities are seen in the bundle.

The outlet flow profiles obviously have a significant effect on the outlet TKE profiles, visualized in Figure 90. URANS $k-\varepsilon$ finds high zones of TKE towards the sides of the test section in the outlet region, corresponding to areas of high velocity. These high zones of TKE continue relatively far downstream. This is not the case of IDDES and LES, where TKE levels drop off quicker in the outlet zone. Additionally, the TKE is more distributed along the test section's width, matching as expected the velocity profiles. Within the bundle, much higher levels of TKE are found by LES and IDDES than by URANS $k-\varepsilon$, with the locations of high TKE matching those of the experiment quite well.

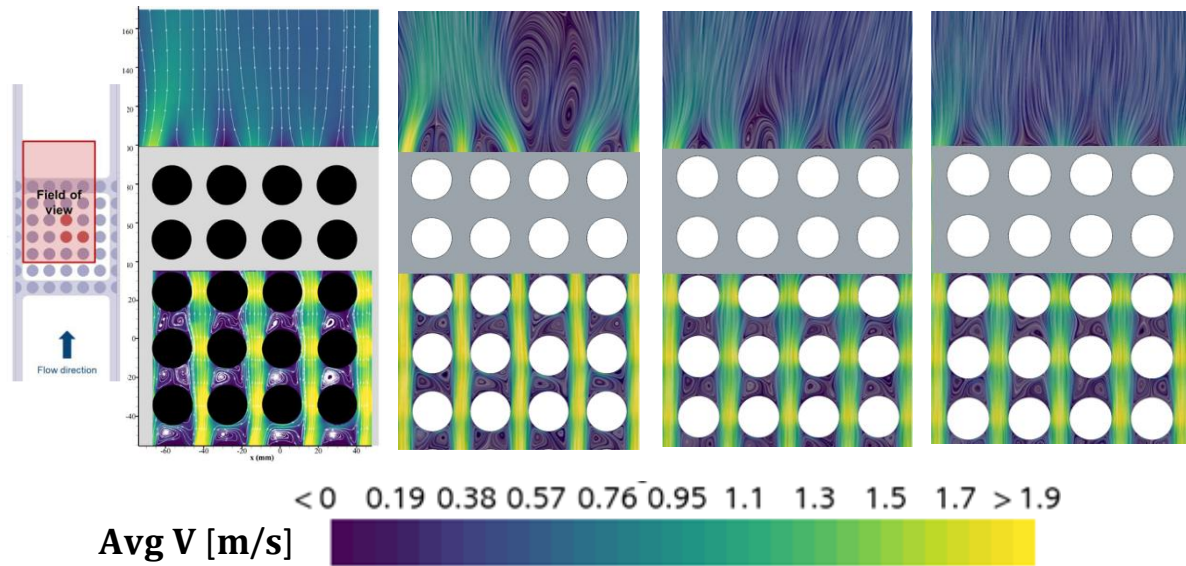


Figure 89: Comparison of the time-averaged velocity on a streamwise plane inside the test section between experiment (left) and simulation, for URANS $k-\epsilon$ (middle-left), IDDES (middle-right) and LES (right), for $u = 0.5$ m/s.

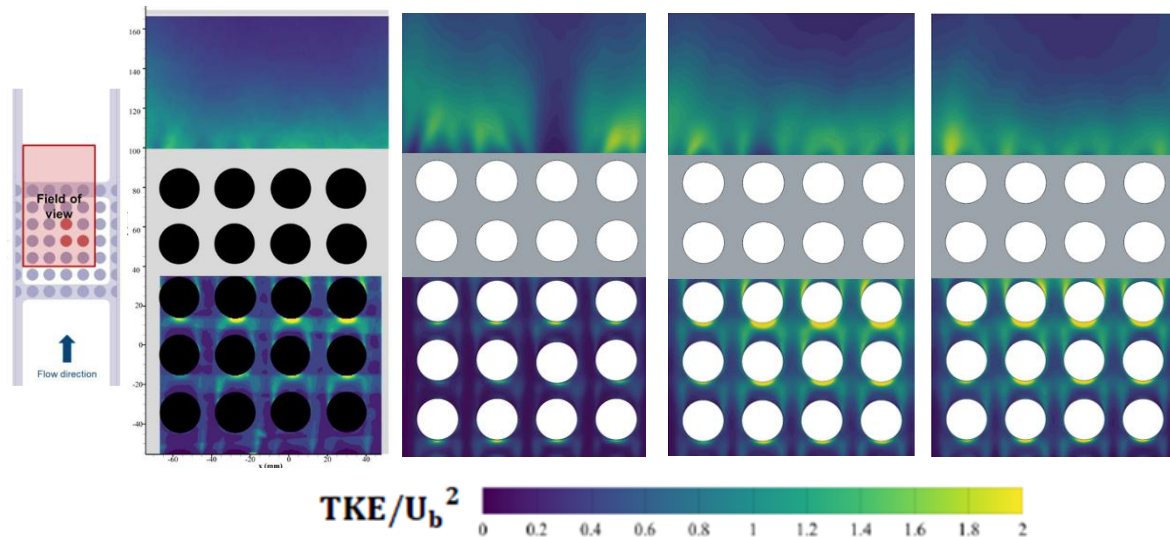


Figure 90: Comparison of the time-averaged normalized TKE on a streamwise plane inside the test section between experiment (left) and simulation, for URANS $k-\epsilon$ (middle-left), IDDES (middle-right) and LES (right), for $u = 0.5$ m/s.

To further illustrate the differences in the flow and TKE patterns in the different simulations, Figure 91 and Figure 92 compare the velocity and TKE profiles, respectively, along selected transverse lines in the bundle region. The location of the high peaks match well between the experiment and simulations, as was to be expected as these correspond to the flow channels. These profile plots also confirm the higher velocities found by URANS $k-\epsilon$ than by IDDES and LES, and also the experiment, as the maximum velocity is highest with URANS $k-\epsilon$, and clearly overpredicts those of the experiment. IDDES and LES match the experiment very well, both in terms of velocity magnitude as width of the peaks. All three turbulence models also well recover the velocities in between the two rows of cylinders. The location of the secondary

peaks do not always match the experiment, however, this is partly due to the choice of positive transverse direction; the simulations are perfectly symmetric around the central YZ-plane, and hence the plots could easily be flipped, resulting in a much better match in terms of location.

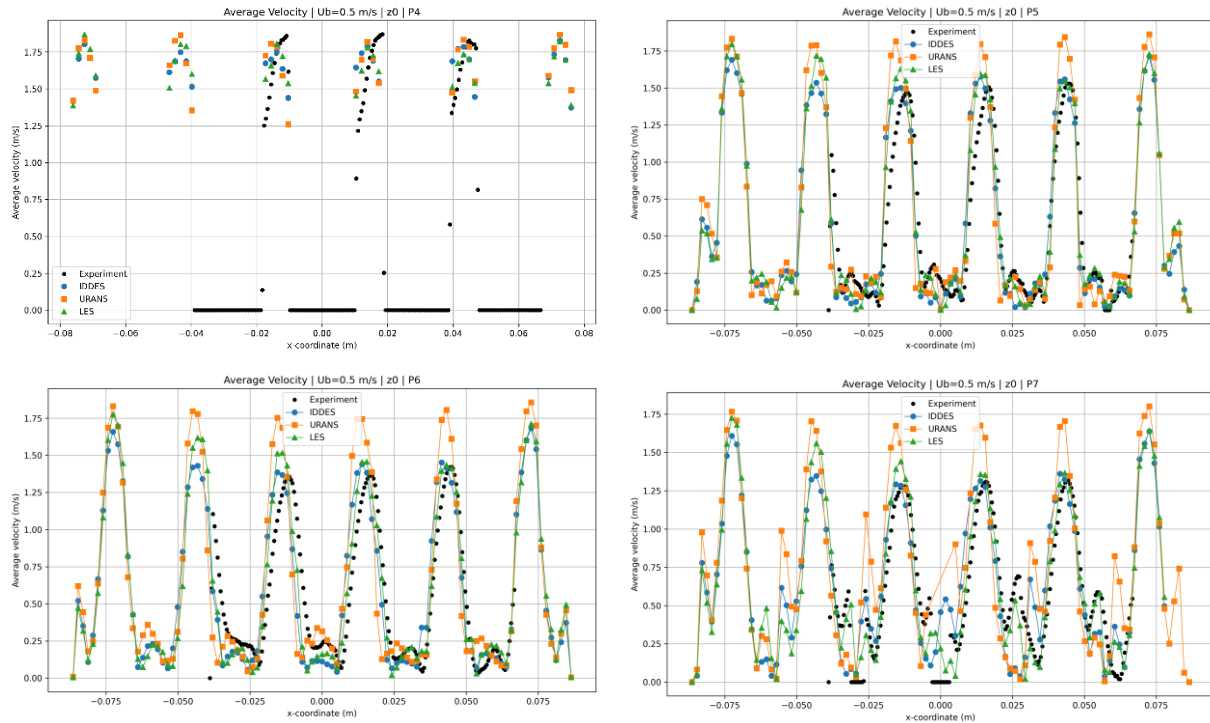


Figure 91: Comparison between experiment and simulations of the velocity inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right), for $u = 0.5$ m/s.

The corresponding TKE profiles are plotted in Figure 92. Similar as for $U_b = 0.125$ m/s, URANS $k-\epsilon$ under-predicts the levels of TKE, especially for P6 and P7, while IDDES and LES match the levels found in the experiment quite well. The patterns also generally match decently well.

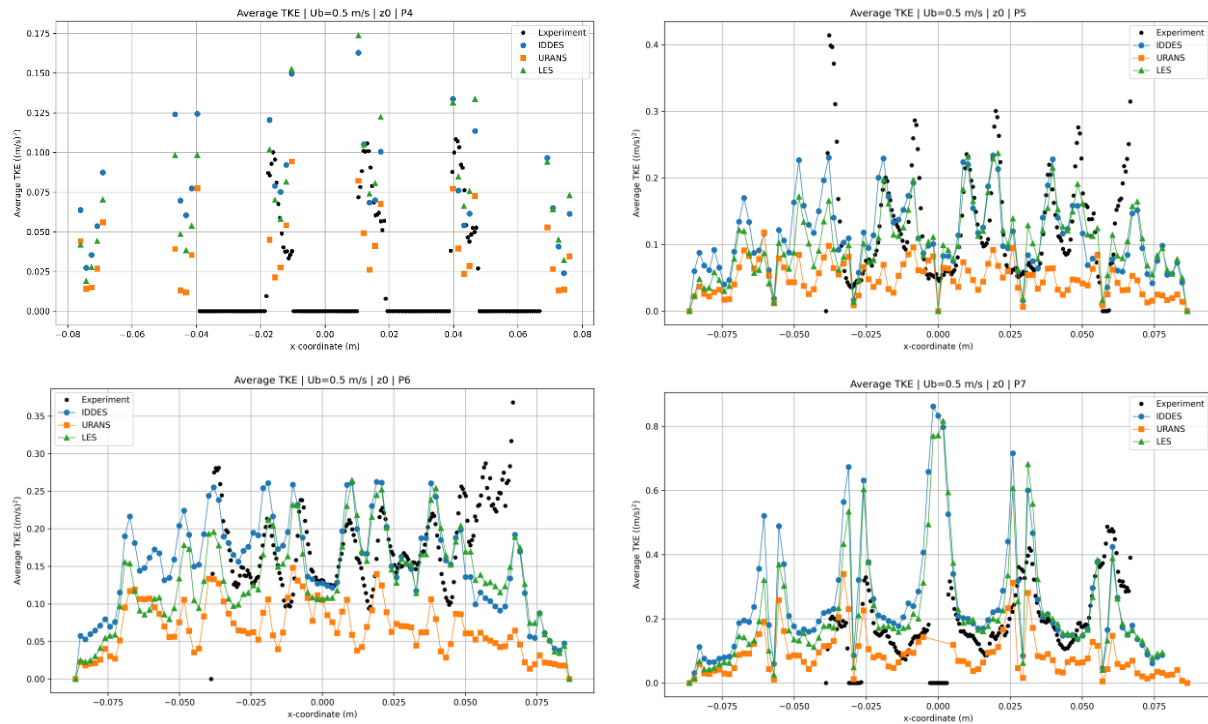


Figure 92: Comparison between experiment and simulations of the TKE inside the bundle on lines P4 (upper left), P5 (upper right), P6, (lower left) and P7 (lower right), for $u = 0.5$ m/s.

Figure 93 and Figure 94 do a similar comparison as the previous two figures, but then for lines in the outlet region. Similar to the earlier analysed case of an inlet velocity of 0.125 m/s, these profiles confirm observations made when studying the contour plots in Figure 89 and Figure 90, as to be expected. Velocity profiles at P8 from all three simulations match the experiment reasonably well, with all simulations predicting the isolated jets seen in the experiment. The simulations do over-predict the velocities, especially URANS $k-\epsilon$. At the other profiles, URANS $k-\epsilon$ predicts the merging of the isolated jets into two larger jets towards the wall, as was seen for the lower inlet velocity case. This is not in line with the experiment and results in large differences, both in terms of profile and local velocities. IDDES and LES, on the other hand, match the experiment remarkably well, especially at P10 and P11, with both simulation profiles overlapping the experimental one largely.

When looking at the TKE levels, fluctuations in the simulated profiles are seen at P8, which can also be observed in the experiment. LES predicts similar levels of TKE as the experiment, with IDDES finding slightly lower levels, followed by URANS $k-\epsilon$. At the other locations, the simulations predict higher levels of TKE, with LES again being closest, followed by IDDES. URANS $k-\epsilon$ gives the largest over-predictions. It again should be noted here that for LES, the modelled TKE was not tracked correctly, hence this contribution is missing in the plots. Since the flow is quite turbulent at the outlet though, it is not expected this contribution to be very significant. Additionally, it is known that PIV does not measure the full range of turbulent scales, hence the experimental results may also not fully reflect the actual levels of turbulence occurring in the test case.

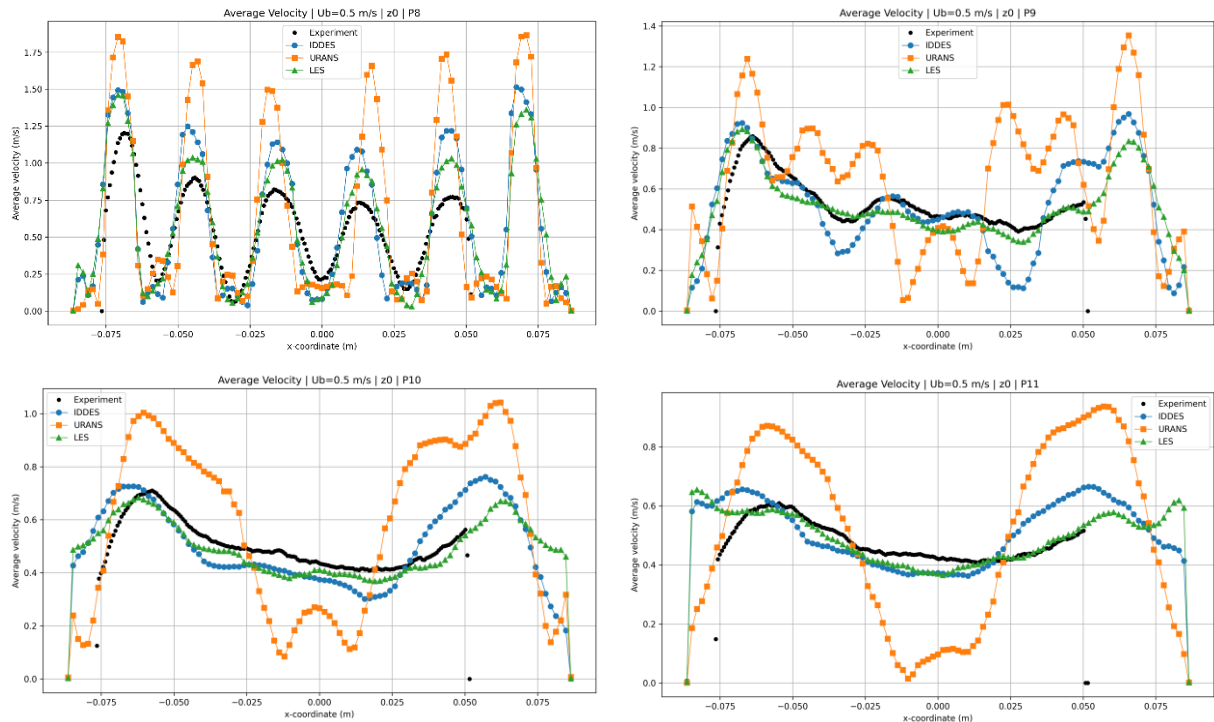


Figure 93: Comparison between experiment and simulations of the velocity in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right), for $u = 0.5$ m/s.

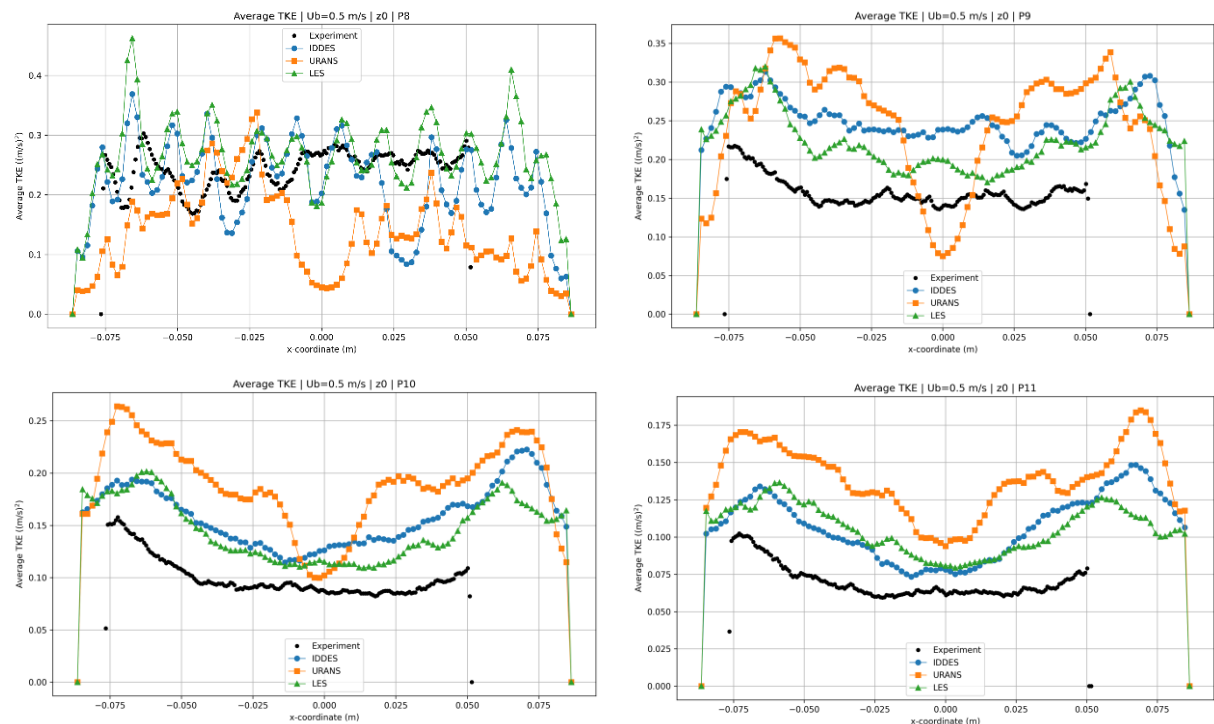


Figure 94: Comparison between experiment and simulations of the TKE in the outlet region on lines P8 (upper left), P9 (upper right), P10, (lower left) and P11 (lower right), for $u = 0.5$ m/s.

Lastly, Figure 95 and Figure 96 show the PSD of the lift and drag force coefficient, respectively, on cylinders 4.1, 4.4 and 4.7, similar as was done for the lower inlet velocity. Once again, URANS predicts lower values of the LFC and DFC than the other two models, especially at higher frequencies. This is in line with the lower levels of turbulence that it resolved by URANS $k-\varepsilon$. This time, however, there does not seem to be a lot of difference between the values of the coefficients on the three different cylinders, showing similar values throughout the bundle. The values are significantly higher for $U_b = 0.5 \text{ m/s}$ than what was seen for $U_b = 0.125 \text{ m/s}$.

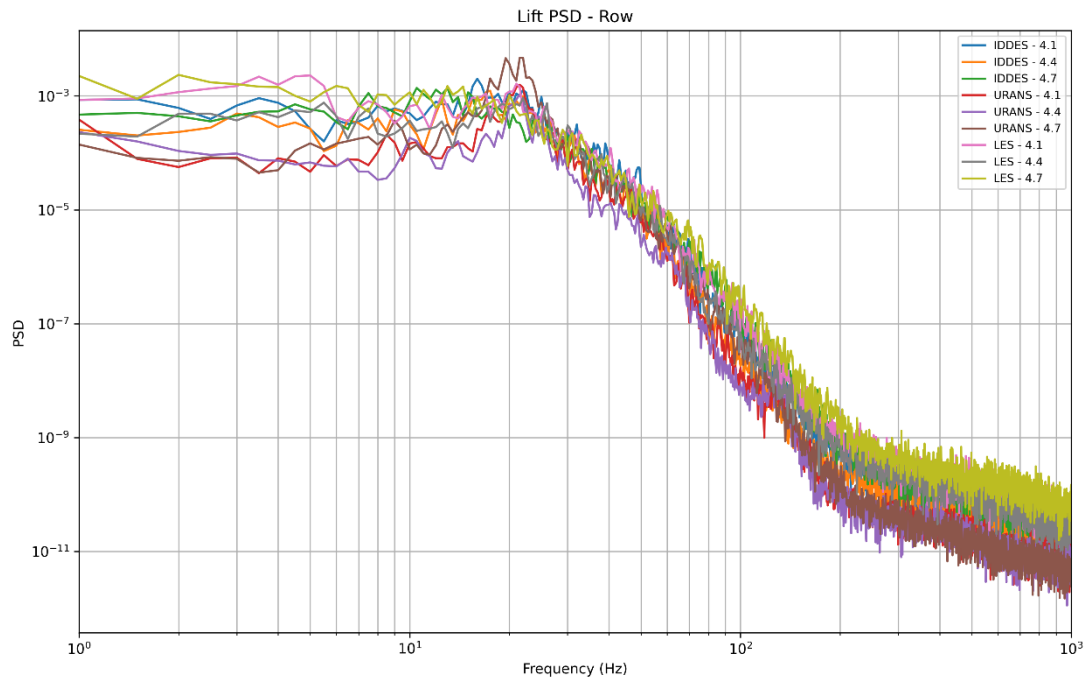


Figure 95: PSD of the lift force coefficient on the most upstream cylinder (4.1), central cylinder (4.4) and most downstream cylinder (4.7) of the central column of cylinders obtained by the three different turbulent models, for $u = 0.5 \text{ m/s}$.

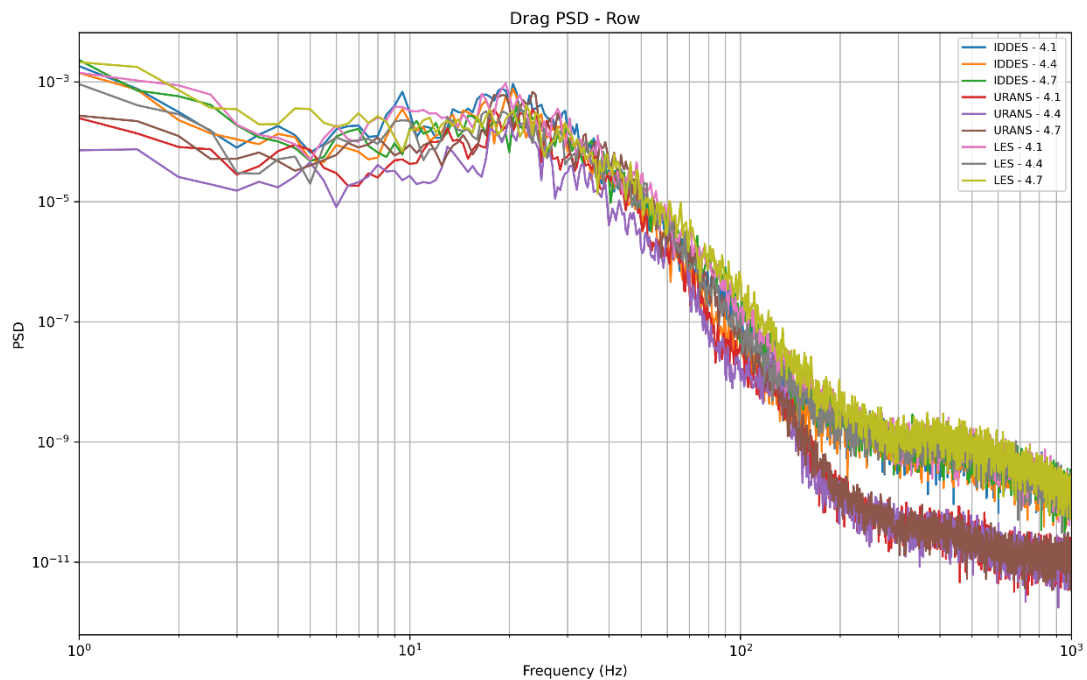


Figure 96: PSD of the drag force coefficient on the most upstream cylinder (4.1), central cylinder (4.4) and most downstream cylinder (4.7) of the central column of cylinders obtained by the three different turbulent models, for $u = 0.5$ m/s.

6.5 FSI simulation results

Following the comparison done between simulation results and experimental data for the flow-only phase of the FSI simulations, now it is time to compare FSI related parameters, thereby focusing mainly on the forces on and displacement of the moving, central, cylinder. This will be done again first for an inlet velocity of 0.125 m/s, followed by an inlet velocity of 0.5 m/s.

6.5.1 Inlet velocity of 0.125 m/s

As mentioned in the introductory part of this section, FSI results for an inlet velocity of 0.125 m/s will be compared with those obtained with the three turbulence models and those measured during the experimental campaigns. Figure 97 shows the time series of the displacements (left) and forces (right) for the lift (transverse, top) and drag (streamwise, bottom) directions for IDDES (blue), URANS $k-\varepsilon$ (orange) and LES (green). Unfortunately, no experimental time series are available, hence only the simulation data can be inter-compared. Looking at the lift forces, and resulting displacements, it seems that IDDES and LES predict larger peaks than URANS $k-\varepsilon$, which seems to fluctuate around a more narrow band. In terms of frequency, as far as one can tell, all three turbulence models show similar rapid fluctuations.

When looking at the drag direction, what is in particular interesting is that all three models find a negative mean displacement, corresponding to the gravity direction. Hence this suggests the fluid drag is not strong enough to counter the tube's weight. Furthermore, URANS $k-\varepsilon$ has a larger mean displacement than IDDES and LES, and also shows again less strong peaks in both the forces and displacements.

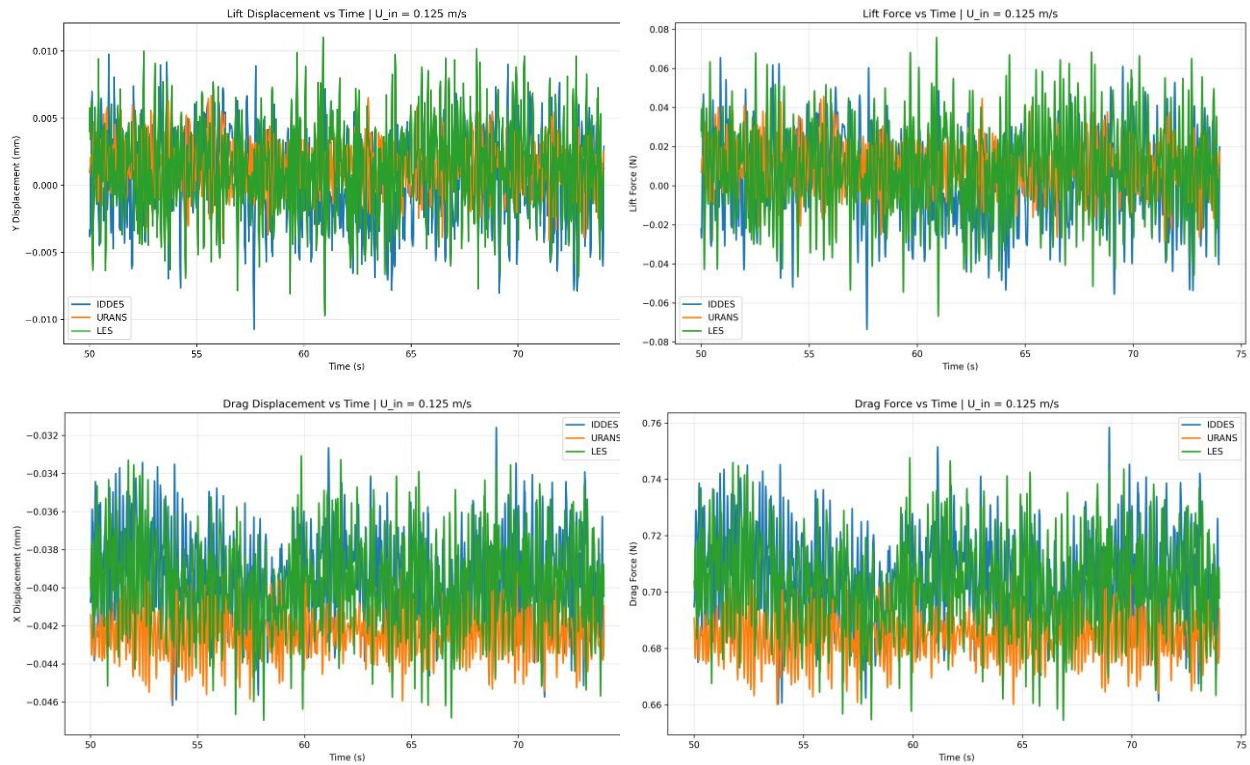


Figure 97: Lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for IDDES (blue), URANS k- ϵ (orange) and LES (green), for $u = 0.125$ m/s.

Figure 98 shows the Power Spectral Density of the previously shown time series. These largely confirm the earlier made observations, being that URANS k- ϵ predicts lower fluctuations in terms of amplitude in force and displacements, and that the predicted frequencies from all simulations are nearly equal. It is interesting to compare the PSD of the forces during the FSI phases of the simulations, shown in Figure 98, with those during the flow-only phases of the simulations, shown in Figure 84 and Figure 85 for the lift and drag direction, respectively. There are quite some significant differences. First of all, there is a clear peak now in the spectra around 31 Hz, corresponding to the main vibration frequency. Additionally, there is a local dip in the spectra at 42 Hz, which is the natural frequency of the cylinder in air, see Table 1.

The time series and PSD were further post-processed to obtain the vibration frequency in both directions, as well as the mean and RMS displacements. These are listed in Table 14, along with the experimental ones for the mean and RMS displacements. Due to the low amplitudes of displacement, it was not possible to discern a clear vibration frequency, relative to the noise, from the experimental spectral plot. All simulations predict the same main vibration frequency of 31 Hz, which is found in both directions. Looking at the mean displacements, the values confirm the negative values found numerically in the drag directions, with a nearly zero mean displacement in lift direction. Oddly, in the experiment, there is a negative mean displacement in lift direction, and a near zero mean, albeit positive, displacement in drag direction. Hence in terms of mean displacement there is quite some discrepancy between the experiment and simulations. Additionally, perhaps the values could change if longer time averaging was performed.

As for the RMS displacements, the simulations under-predict the experimental values in both directions. The largest under-prediction is found with URANS k- ϵ . IDDES and LES find values

closer to the experiment, under-predicting the lift RMS displacement by about a factor of 2.8 and 2.5, respectively, and the drag one by about 3.6-3.7. Such under-predictions are frequently found for relatively low inlet velocities (Zwijsen et al., 2025)

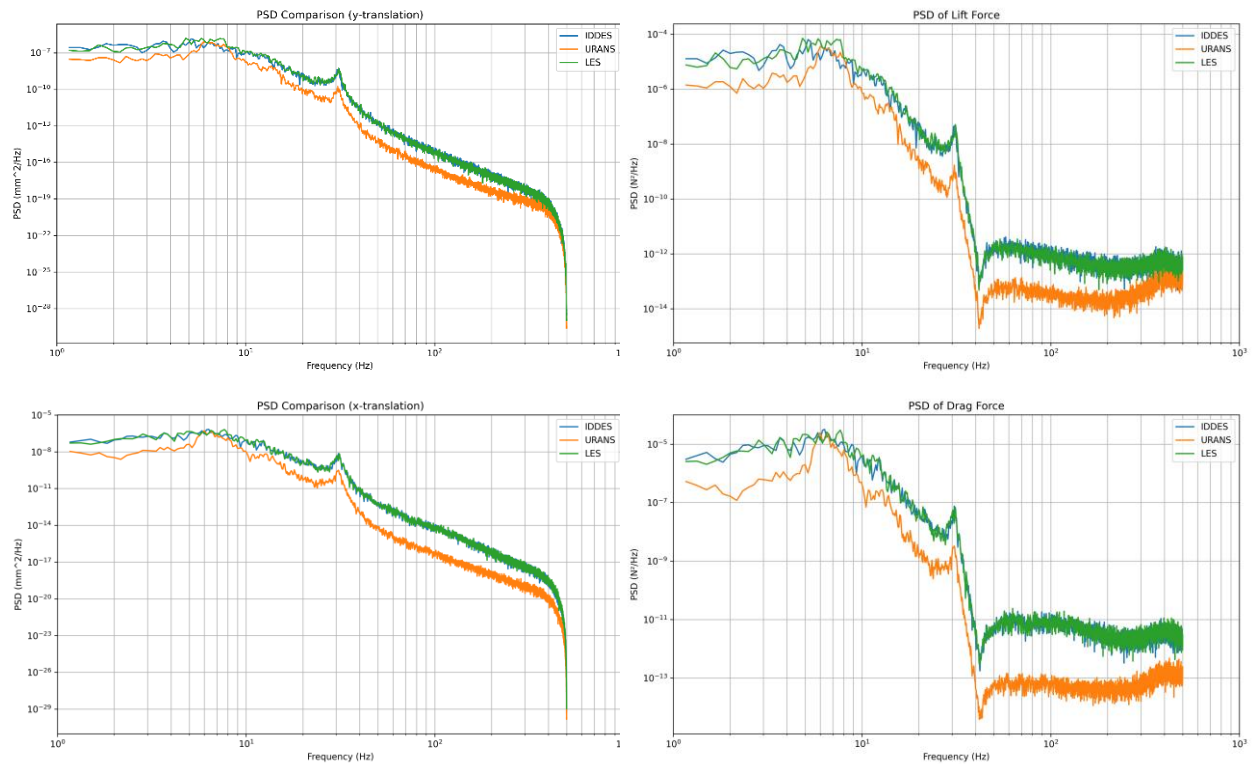


Figure 98: PSD of the lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for IDDES (blue), URANS $k-\epsilon$ (orange) and LES (green), for $u = 0.125$ m/s.

Table 14: Comparison between experiment and simulations of relevant FSI statistics for $u = 0.125$ m/s

U = 0.125 m/s	Frequency [Hz]		Mean displ. [mm]		RMS displ. [mm]	
	Lift	Drag	Lift	Drag	Lift	Drag
Experiment	N/A	N/A	-2.57E-2	6.43E-3	8.50E-3	8.32E-3
IDDES	31	31	4.81E-4	-3.94E-2	3.00E-3	2.23E-3
URANS	31	31	1.28E-3	-4.22E-2	1.78E-3	1.24E-3
LES	31	31	1.25E-3	-3.98E-2	3.33E-3	2.30E-3

6.5.2 Inlet velocity of 0.5 m/s

In this sub-section, results of the FSI-phase of the simulations will be presented, and, where possible, compared to the experiment. Time series of the displacements and forces are presented in Figure 99, with resulting PSDs shown in **Figure 100**. The results are quite different from the lower inlet velocity case. For the lift direction, the time series and PSDs do not show

much difference between the three turbulence models, showing very similar levels of forces and displacements. The PSD of the lift forces also shows a stronger recovery after the dip, going back up to higher values, in particular for URANS k- ϵ .

Large differences do appear however for the drag direction, with URANS k- ϵ predicting much higher forces, and resulting displacements, than the other two turbulence models. This results in a much stronger peak in the PSDs at the main vibration frequency, and even harmonics of this frequency at higher frequencies. These larger forces and displacements probably are a result of the higher velocities seen inside the tube bundle with URANS k- ϵ , see Figure 89 and Figure 91.

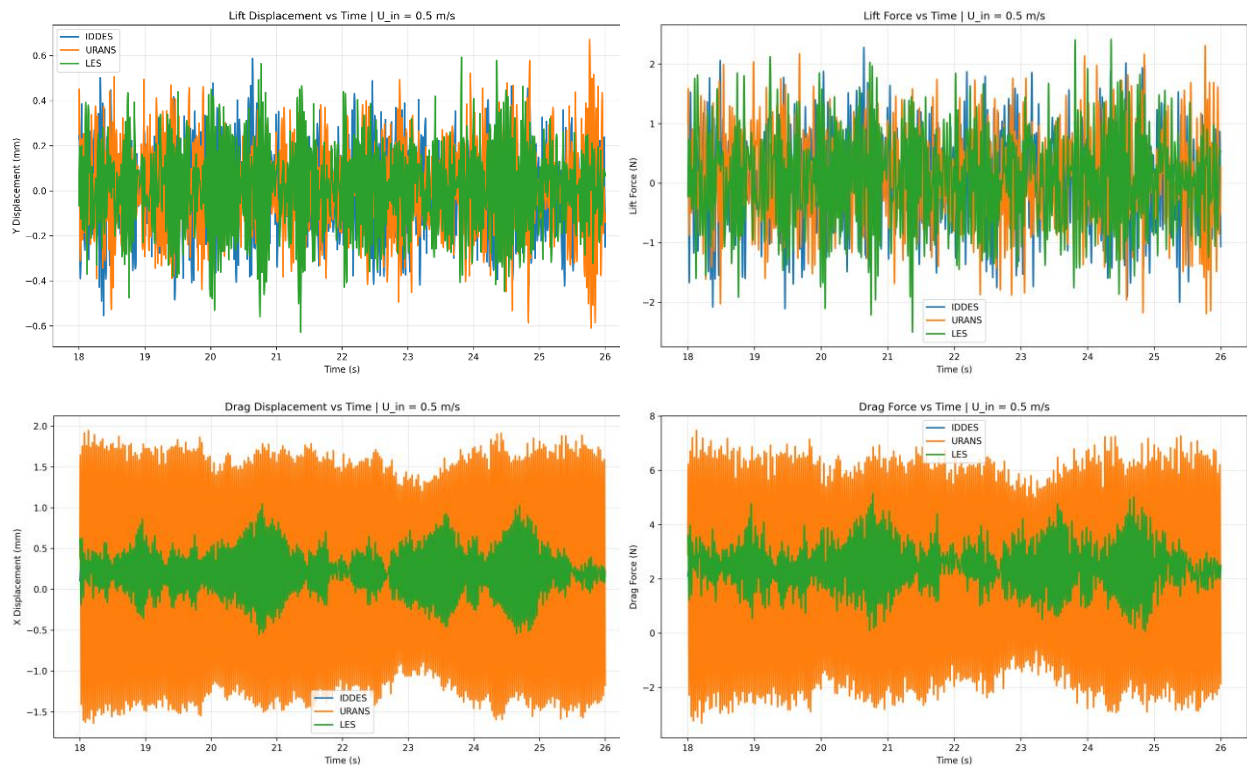


Figure 99: Lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for IDDES (blue), URANS k- ϵ (orange) and LES (green), for $u = 0.5$ m/s.

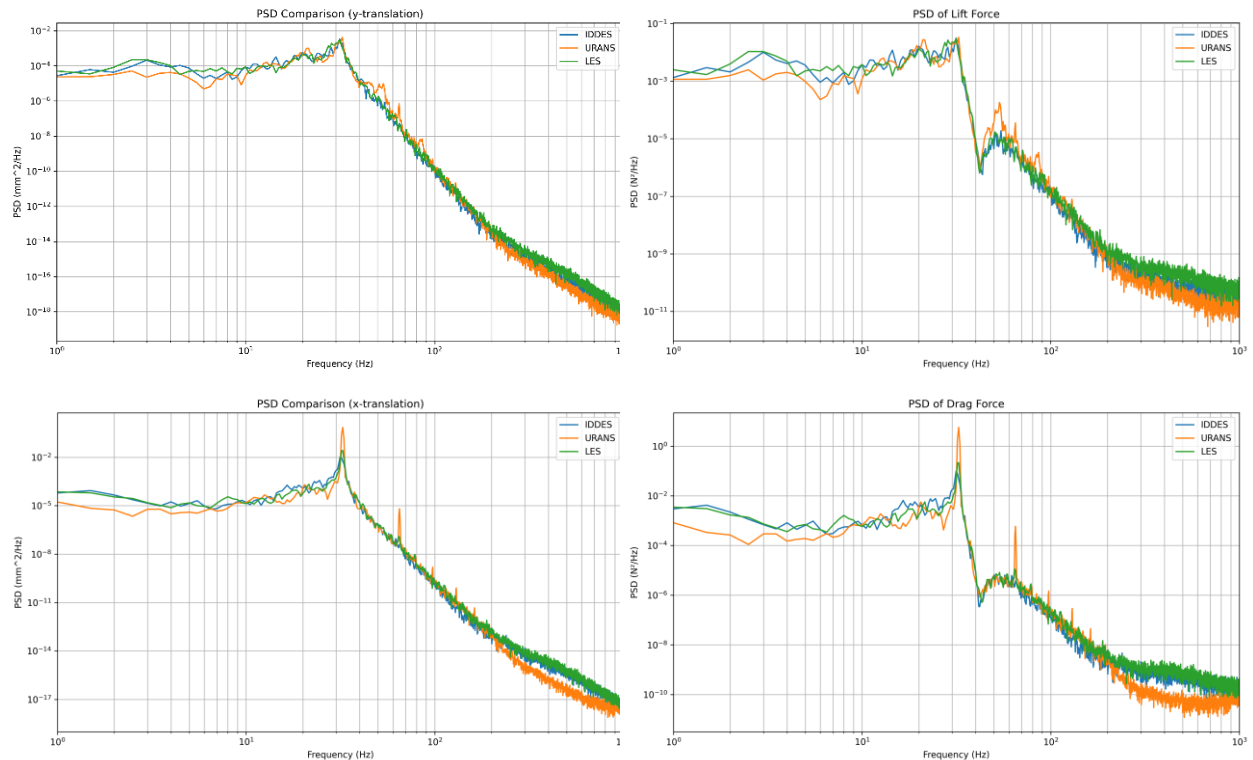


Figure 100: PSD of the lift (up) and drag (bottom) forces (left) and displacement (right) as function of time for IDDES (blue), URANS k- ϵ (orange) and LES (green), for $u = 0.5$ m/s.

To quantify the observed patterns, Table 15 lists the main vibration frequencies and mean and RMS displacements for the two directions obtained from the simulations, alongside those of the experiment. Looking at the frequency, the simulated ones match the experimental frequency in drag direction very well. The lift frequency is under-predicted by a few Hertz. It is rather surprising though that the experimental frequencies in lift and drag direction differ.

Comparing mean displacements, experimentally a negative value is found for the lift direction, while numerically positive values are predicted. These are all very small though and are relatively close to zero. In drag direction, all mean values are positive, with the simulations under-predicting the experimentally obtained value. But at least they are all positive now, opposed to the lower velocity case, where all models predicted negative values.

Table 15: Comparison between experiment and simulations of relevant FSI statistics for $u = 0.5$ m/s

U = 0.5 m/s	Frequency [Hz]		Mean displ. [mm]		RMS displ. [mm]	
	Lift	Drag	Lift	Drag	Lift	Drag
Experiment	35.2	32.2	-4.67E-2	2.97E-1	6.50E-2	5.82E-2
IDDES	32	32	4.88E-3	2.15E-1	1.71E-1	2.20E-1
URANS	32-33	32-33	1.17E-2	1.51E-1	1.82E-1	1.05E+0
LES	32	32	9.15E-3	2.08E-1	1.76E-1	2.60E-1

Finally, the RMS displacements are over-predicted by all models, for both the lift direction and drag directions. For lift, the over-prediction is by a factor of 2.6 (IDDES) to 2.8 (URANS $k-\epsilon$), while for drag IDDES and LES over-predict it by 3.8x and 4.4x, respectively. URANS $k-\epsilon$, as seen already in the displacement time series, predicts a even much larger RMS displacement in drag direction, over-predicting the experimental ones by nearly a factor of 20. Hence, the simulations went from under-predicting RMS displacements at low velocities to over-predicting at higher velocities. This is in line with earlier observations made in literature (Zwijssen et al., 2025).

7. Overall conclusions

This report describes the collaborative effort performed by VCU, EDF and NRG PALLAS in the GO-VIKING project on the modelling of the GOKSTAD experimental facility. GOKSTAD is a water loop facility with a test section containing a 7 x 7 (the side columns are half tubes) tube bundle subjected to cross-flow. A rich experimental campaign was performed in which high-resolution and CFD-grade data was obtained for thorough validation of CFD and FSI approaches. The current report, which constitutes deliverable D3.4 of the aforementioned GO-VIKING project, describes results obtained by the three partners for the benchmark case of 0.125 m/s and any additional cases each partner may have simulated.

In addition to the specific benchmark conclusions given in Section 3.4, the following overall conclusions can be drawn:

- High-resolution reference data: for proper validation of CFD and FSI simulations, it is really crucial to have high-resolution reference data, be it either from experiments or simulations. These data should not only cover first-order statistics, such as mean velocities, but also second-order statistics such as turbulent kinetic energy and Reynolds stresses. And such statistics should be available at all regions of interest, such as the inlet, areas surrounding moving tubes, and outlet regions. A large goal of Task 3.2 of GO-VIKING, of which this deliverable is part, was to ensure such data is available.
- Turbulence modelling: scale-resolving methods give better predictions for the flow fields than medium-resolution models, such as those based on Reynolds-averaging or filtering of (large) scales. This is in particular true for higher-order flow statistics. This does come at an increased computational costs, however. The aforementioned medium-resolution models can still give reasonably accurate predictions, given they resolve sufficient scales and resolve the boundary layer sufficiently. In that regard, it is beneficial to use a Reynolds Stress Model or under-resolved LES on a URANS quality mesh.
- Structural modelling: cylinders subjected to cross-flow can be reasonably accurately modelled as mass-spring-damper systems with a single or two degrees-of-freedom, given the modelled cylinder does not show any rocking motion or bending due to the flow field. It is however important to include sufficient length of the moving cylinder in the model. In particular for wall-bounded moving cylinders, including the full length may be needed due to pressure distribution along the tube's length not being uniform.
- FSI modelling: For small displacement, using a one-way approach gives just as good results as using a two-way approach.

Bibliography

Benhamadouche, S., and Benguigui, W. (2023). Wall-resolved LES and Low-Reynolds Number URANS combined to an Arbitrary Lagrangian Eulerian approach for Predicting Water Cross-Flow Induced vibrations of a Single Flexible Tube in a Normal Square Tube Array. Proc. of the NURETH-20 conference, Washington, USA, August 20 – August 25.

Can Akkurt, M., Delsipee, M., Koloszar, L, & Planquart, P. (2026). D3.3 - Single-phase GOKSTAD experiments and benchmark setup. GO-VIKING deliverable D3.3.

Dong, S., Karniadakis, G., and Chrysosostomidis, C. (2014). A robust and accurate outflow boundary condition for incompressible flow simulations on severely-truncated unbounded domains. Journal of Computational Physics, 261, pp. 83–105.
<https://doi.org/https://doi.org/10.1016/j.jcp.2013.12.042>

Filonov, V., Filonova, Y., Dubyk, Y., Mistry, H., Papukchiev, A, and Benhamadouche, S. (2026). GO-VIKING deliverable D3.2: AMOVI model preparation and simulation results.

Geuzaine, C., and 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.

Granger, S., Campistron, R., and Lebre, J. (1993). Motion-dependent excitation mechanisms in a square in-line tube bundle subject to water cross-flow: an experimental modal analysis,” Journal of Fluids and Structures, 7, pp. 521–550.

Nek5000. (2017). Nek5000 17.0 Documentation.

Novak, A.J., Bourdot Dutra, C., Shaver, D., and Merzari, E. (2025). CFD simulation of interassembly bypass flow in Sodium Fast Reactors, Nuclear Engineering and Design, Volume 439, 2025, 114044, ISSN 0029-5493.
<https://doi.org/10.1016/j.nucengdes.2025.114044>.

Tomboulides, A., Saini, N., Shaver, D.R., Obabko, A.V., Yuan, H., Merzari, E., and Fischer, P.F. (2024). A robust spectral element implementation of the $k-\tau$ RANS model in Nek5000/NekRS, International Journal of Heat and Fluid Flow, Volume 112, 2025, 109679, ISSN 0142-727X. <https://doi.org/10.1016/j.ijheatfluidflow.2024.109679>.

Vivaldi, D., and Baccou, J. (2023). URANS And Hybrid Urans/Les Coupled Fluid-Structure Simulations Of Fluid-Induced Vibrations In A Square-Pitch Tube Bundle Subjected To Water Cross-Flows. Proc. of the NURETH-20 conference, Washington, USA, August 20 – August 25.

Zwijssen, K., Hussain, M.M.M.D., & Roelofs, F. (2024). URANS Simulations Of Vortex-Induced Vibrations Of Cylinders In Cross-Flow. FIV2024, Iguaçu Falls, Brazil.

Zwijssen, K., Tajfirooz, S., Roelofs, F., Papukchiev, A., Vivaldi, D., Hadžić, H., Benhamadouche, S., Benguigui, W., Norddine, T., Iacovides, H., Cioncolini, A., Nabawy, M.R.A., Angele, K., Lillberg, E., Chazot, B., and Iyamabo, E. (2025). Outcome of the VIKING project: status and

perspectives of numerical modeling of flow-induced vibrations of nuclear power plant components. Nuclear Engineering and Design, 440, pp. 114131.

Zwijssen, K. (2025). On Flow-Induced Vibrations of Single and Multiple Cylinders in Cross-Flow and using Medium-Resolution Methods for Numerical Predictions. Proc. of the NURETH-21 conference, Busan, South Korea, 31 August – 5 September.