



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



Simulation of ALAIN 5x5 PWR rod bundle benchmark

Authors : Hidajet HADZIC (Framatome GmbH)

GO-VIKING - Contract Number: 101059603

Project officer: Angelgiorgio IORIZZO

Document title	Simulation of ALAIN 5x5 PWR rod bundle benchmark
Author(s)	Hidajet HADZIC
Number of pages	167
Document type	Deliverable
Work Package	WP2
Document number	D2.4
Issued by	Framatome GmbH
Date of completion	2026-06-03 00:28:52
Dissemination level	Public

Summary

Description and discussion of the final results from the numerical simulations.

Approval

Date	By
2026-06-03 00:45:25	Hidajet HADZIC (Framatome GmbH)
2026-06-03 09:05:50	Papukchiev ANGEL (GRS)



GO-VIKING deliverable D2.4: Simulation of ALAIN 5x5 PWR rod bundle benchmark

Month 39

Version 1.0

May / 2026

Hidajet Hadžić¹, Antoine Michel², Joachim Herb³, Christoph Bläsius³, Vladislav Filonov⁴,
Yaroslav Dubyk⁴, Yuliia Filonova⁴

¹Framatome GmbH (FRA-G), Paul-Gossen-Strasse 100, 91052 Erlangen, Germany

²Commissariat à l'énergie atomique et aux énergies alternatives (CEA), Rue Leblanc 25, 75015 Paris, France

³Gesellschaft für Anlagen- und Reaktorsicherheit gGmbH (GRS), Schwertnergasse 1, 50667 Köln, Germany

⁴IPP Centre LLC (IPP), Budindustrii Street 5B, 01013 Kyiv, Ukraine



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.

Document information

Grant Agreement / Proposal ID	101060826
Project Title	Gathering expertise On Vibration ImpaKt In Nuclear power Generation
Project Acronym	GO-VIKING
Scientific Coordinator	Angel Papukchiev, mailto:Angel.Papukchiev@grs.de , GRS
Project starting date (duration)	1 st June 2022 – 31 st May 2026 (48 Months)
Related Work Package	WP2
Related Task(s)	Task 2.2 ALAIN 5x5 PWR rod bundle
Lead Organisation	Framatome GmbH (FRA-G)
Contributing Partner(s)	FRA-G, CEA, GRS, IPP
Due Date	31.08.2025
Submission Date	31.05.2026
Dissemination level	PU

History

Date	Version	Submitted by	Reviewed by	Comments
31.05.2026	1.0	H. Hadžić	A. Papukchiev, K. Zwijsen	First submitted version

Authors matrix

Name	Organization	Contribution
Hidajet Hadžić	Framatome GmbH (FRA-G), Paul-Gossen-Strasse 100, 91052 Erlangen, Germany	Chapters: 1, 2, 3, 4, 8 and 9
Antoine Michel	Commissariat à l'énergie atomique et aux energies alternatives (CEA), Rue Leblanc 25, 75015 Paris, France	Chapter 5
Joachim Herb Christoph Bläsius	Gesellschaft für Anlagen- und Reaktorsicherheit gGmbH (GRS), Schwertnergasse 1, 50667 Köln, Germany	Chapter 6
Vladislav Filonov Yaroslav Dubyk Yuliia Filonova	IPP Centre LLC (IPP), Bolsunovska Street 8, 01014 Kyiv, Ukraine	Chapter 7

Table of contents

1	Purpose	20
2	Introduction	21
3	Problem description.....	23
3.1	Experimental setup	23
3.2	Test fuel assembly	24
3.3	Natural frequencies.....	26
3.4	Flow velocity profiles setup.....	26
3.5	Benchmark setup and objectives	26
4	Simulation performed by Framatome	29
4.1	Introduction.....	29
4.2	CFD simulations	30
4.2.1	Turbulence modeling approach	30
4.2.2	Overview of CFD models	32
4.2.3	Numerical mesh	32
4.2.4	Computational settings and simulation workflow	36
4.2.5	Flow statistics monitors	37
4.2.6	Monitoring of fluid forces	38
4.2.7	Simulation results.....	38
4.2.7.1	Single span configuration	39
4.2.7.2	Full-scale configuration	48
4.2.7.3	Fluid forces along the fuel assembly.....	53
4.3	Mechanical model	56
4.3.1	Description of the Mechanical model	56
4.3.2	Effects of fluid on dynamic behavior.....	58
4.4	FIV simulations	61
4.5	Summary and conclusions.....	63
5	Simulation performed by CEA.....	65
5.1	Modeling approach	65
5.1.1	Governing equations	65
5.1.1.1	RANS equations.....	65
5.1.1.2	Large eddy simulation	66
5.1.2	The arbitrary Lagrange- Euler formalism	67

5.1.3	Boundary conditions	68
5.1.4	Numerical simulation approach	69
5.1.4.1	CFD code used	69
5.1.4.2	Description of the numerical mesh	70
5.1.4.3	Post processing setup	72
5.1.5	Calculated configuration	73
5.2	Simulations results	73
5.2.1	Fluid flow profiles	73
5.2.1.1	Influence of computational domain on RANS velocity profiles	73
5.2.1.2	Influence of the turbulence modeling approach for the single span configuration	80
5.2.2	Conclusion on the flow simulations in the ALAIN experiment	90
5.3	Fluid forces and assembly vibrations	91
5.3.1	Modeling of the assembly vibration and modal coefficients	91
5.3.1.1	Mode shape: beam theory and finite elements model	91
5.3.2	Simulations with imposed vibration	93
5.3.2.1	Force acting on the bundle	95
5.4	Conclusions	99
6	Simulations performed by GRS	103
6.1	Setup of CFD simulations	103
6.2	CFD simulation results	106
6.2.1	Mesh sensitivity study	106
6.2.2	Discretization sensitivity study	108
6.2.3	Time step size sensitivity study	109
6.2.4	Turbulence model sensitivity study	110
6.2.5	Turbulent quantities/velocity fluctuations	111
6.2.6	Model selection for benchmark/input for mechanical simulations	115
6.3	Mechanical simulation	115
6.3.1	Model description	115
6.3.2	Load application	118
6.3.3	Results	119
7	Simulations performed by IPP	121
7.1	Introduction	121
7.2	CFD model of the ALAIN experimental assembly	122

7.2.1	Geometry of the flow section of the experimental assembly	122
7.2.2	Computational meshes of the ALAIN FA CFD model	122
7.2.3	CFD model and applied boundary conditions	123
7.3	Analysis of the computational mesh and initial state	124
7.4	Three-point mechanical model of fuel assembly displacement	128
7.4.1	Model definition	129
7.4.2	Restoring system characteristics from experimental data	130
7.4.3	Fluid effects on the vibrational model	133
7.5	Results of the coupled FIV analysis	135
8	Benchmark	145
8.1	Introduction	145
8.2	Flow field	147
8.2.1	Axial mean flow profiles	148
8.2.2	Cross mean flow profiles	148
8.2.3	Axial flow fluctuations profiles	148
8.2.4	Cross flow fluctuations profiles	149
8.3	FIV fuel assembly response	156
8.4	Benchmark summary and conclusions	159
9	Overall summary and conclusions	161
10	Bibliography	163

List of figures

Figure 2-1: PWR fuel leak causes worldwide in 2006–2010 (left) and 2011–2015 (right).	21
Figure 3-1: Principal sketch of the “ALAIN” test loop (left) and picture of the test section of the test loop with measurement system (right).	23
Figure 3-2: Overview of 5x5 fuel assembly.	24
Figure 3-3: Overview of the fluid flow profiles and FIV measurements setup.	28
Figure 4-1: Overview of the CFD models investigated.	33
Figure 4-2: Numerical mesh details – around spacer grid and in rod bundle area.	34
Figure 4-3: Distribution of the viscosity ratio and sub-grid activity over the computational domain for single-span configuration (left) and full-scale configuration (right).	35
Figure 4-4: Courant number distribution in single-span model (left) and full-scale model (right).	36
Figure 4-5: Execution of the CFD simulation.	37
Figure 4-6: Mapping of fluid forces to the mechanical model.	38
Figure 4-7: Single span flow profiles - average deviation axial flow.	41
Figure 4-8: Single span flow profiles - average deviation cross flow.	41
Figure 4-9: Single span axial flow mean velocity profiles along traverse T1 left and T2 right.	42
Figure 4-10: Single span cross flow mean velocity profiles along traverse T1 left and T2 right.	43
Figure 4-11: Single span axial flow velocity RMS profiles along traverse T1 left and T2 right.	44
Figure 4-12: Single span cross flow velocity RMS profiles along traverse T1 left and T2 right.	45
Figure 4-13: Single span mean flow profiles - quantitative comparison.	46
Figure 4-14: Single span RMS flow profiles - quantitative comparison.	47
Figure 4-15: Flow profiles average deviation axial flow: full-scale vs. span model.	48
Figure 4-16: Flow profiles average deviation cross flow: full-scale vs. span model.	48
Figure 4-17: Full-scale model axial flow mean velocity profiles - traverse T1 and T2.	49
Figure 4-18: Full-scale model crossflow mean velocity profiles - traverse T1 and T2.	50
Figure 4-19: Full-scale model axial flow RMS velocity profiles - traverse T1 and T2.	51
Figure 4-20: Full-scale model crossflow RMS velocity profiles - traverse T1 and T2.	52
Figure 4-21: Time histories of the fluid forces acting on spacer grids.	53
Figure 4-22: Fluid force profile along the fuel assembly – average (static components).	54
Figure 4-23: Fluid force profile along the fuel assembly – RMS (dynamic components).	55
Figure 4-24: PSD of fluid forces in lateral direction x acting on fuel assembly at SG positions.	55

Figure 4-25: PSD of fluid forces in lateral direction y acting on fuel assembly at SG positions.	56
Figure 4-26: Mechanical model overview.	57
Figure 4-27: Simplified single-span FSI model for evaluation of fluid dynamic parameters. ...	59
Figure 4-28: Added mass of the single span in stagnation water as function of frequency....	59
Figure 4-29: Fluid damping as function of frequency and axial velocity (right).	60
Figure 4-30: Rayleigh damping parameters implemented in mechanical model based on the fluid damping evaluation using FSI simulation for single span.....	60
Figure 4-31: Fuel assembly vibration response PSD for lateral displacements ux and uy at spacer grids 2, 3 and 4 compared with experimental results.....	62
Figure 5-1: Sketch of a single span showing the recirculation box employed to determine the inlet velocity condition (in blue), the meshed grid region (in yellow) and the bundle region downstream of the grid obtained by extrusion (in green).	69
Figure 5-2: Original geometry of the grid region (left panel) and simplifications made to improve the mesh quality (right panel).	71
Figure 5-3: Average value of the Taylor microscale (λ , left panel) and Kolmogorov's lengthscale (ηk , right panel), estimated from a RANS simulation of a recirculation box with a mesh cell size $\Delta = 0.6$ mm.	72
Figure 5-4: Average axial velocity predicted by a RANS model 10 mm upstream of the mixing grid for the single span (left panel), the first grid of the whole assembly (center panel) and the second grid of the whole assembly (right panel). The discrepancies in the inlet velocity field influence the flow downstream of the grids.	74
Figure 5-5: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 10 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	75
Figure 5-6: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 20 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	76
Figure 5-7: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 50 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	76
Figure 5-8: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 100 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	76

- Figure 5-9: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 250 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation. 77
- Figure 5-10: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 10 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation. 78
- Figure 5-11: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 20 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation. 78
- Figure 5-12: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 50 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation. 79
- Figure 5-13: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 100 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation. 79
- Figure 5-14: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 250 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation. 79
- Figure 5-15: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 10 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation. 81
- Figure 5-16: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 20 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation. 81
- Figure 5-17: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 50 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation. 81
- Figure 5-18: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 100 mm downstream of the mixing grid for the single span domain. Black squares

denote the experimental measurements and error bars materialize $\pm$ one standard deviation.	82
Figure 5-19: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 250 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	82
Figure 5-20: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 10 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	83
Figure 5-21: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 20 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	83
Figure 5-22: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 50 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	84
Figure 5-23: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 100 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	84
Figure 5-24: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 250 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	84
Figure 5-25: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 10 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	86
Figure 5-26: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 20 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	86
Figure 5-27: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 50 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	86
Figure 5-28: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 100 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	87
Figure 5-29: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 250 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	87

Figure 5-30: RMS of the transverse velocity along the traverse T1 (left panel) and T2 (right panel) 10 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	88
Figure 5-31: RMS of the transverse velocity along the traverse T1 (left panel) and T2 (right panel) 20 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	88
Figure 5-32: RMS of the transverse velocity along the traverse T1 (left pannel) and T2 (right pannel) 50 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements.	88
Figure 5-33: RMS of the transverse velocity along the traverse T1 (left pannel) and T2 (right pannel) 100 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements.	89
Figure 5-34: RMS of the transverse velocity along the traverse T1 (left pannel) and T2 (right pannel) 250 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements.	89
Figure 5-35: First and second modes for the whole assembly computed by Cast3M.....	92
Figure 5-36: First and second mode of the bundle vibration. FEM: cast3m model. EB: Euler-Bernoulli beam theory. modif. EB: Euler Bernoulli beam theory modified to rigidify the bundle at the grid location. The grids positions are represented by thin vertical red lines.	93
Figure 5-37: Average of the axial velocity along the traverse T1 (left pannel) and T2 (right pannel) 10 mm downstream of the second grid of the full domain and for the single span computed with the RANS model. The results are shown for the static structure, and with imposed vibration. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	94
Figure 5-38: Average of the axial velocity along the traverse T1 (left pannel) and T2 (right pannel) 50 mm downstream of the second grid of the full domain and for the single span computed with the RANS model. The results are shown for the static structure, and with imposed vibration. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	94
Figure 5-39: Average of the axial velocity along the traverse T1 (left pannel) and T2 (right pannel) 250 mm downstream of the second grid of the full domain and for the single span computed with the RANS model. The results are shown for the static structure, and with imposed vibration. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.	94
Figure 5-40: Normalized norm of the average force acting on the center (left panel) and an adjacent (right panel) fuel rod when the flow is computed by LES. Several points were added upstream of the grid for informative purpose. Red squares represent the pressure force and blue squares the total force (sum of pressure and viscous forces). Error bars materialize $\pm$ one standard deviation around the average.	95

Figure 5-41: Normalized norm of the average force acting on the center (left panel) and an adjacent (right panel) fuel rod when the flow is computed by RANS. Red squares represent the pressure force and blue squares the total force (sum of pressure and viscous forces). Error bars materialize $\pm$ one standard deviation around the average....	96
Figure 5-42: Spectral density of the axial (left panel) and transverse (right panel) component of the force acting on the central rod of the single span assembly in quiet fluid, 250 mm downstream of the grid for the single span domain. RANS and LES refer to the mesh employed for the computation. A vertical black line materializes the 30 Hz frequency of the imposed vibrations.	97
Figure 5-43: Spectral density of the axial (left panel) and transverse (right panel) component of the force acting on the central rod of the assembly in quiet fluid, 10 mm downstream of the second grid for the whole domain. RANS and LES refer to the mesh employed for the computation. Vertical black lines materialize the 6 and 12.5 Hz frequencies of the imposed vibrations.....	97
Figure 5-44: Spectral density of the axial (left panel) and transverse (right panel) component of the force acting on the central rod of the assembly in quiet fluid, 250 mm downstream of the second grid for the whole domain. RANS and LES refer to the mesh employed for the computation. Vertical black lines materialize the 6 and 12.5 Hz frequencies of the imposed vibrations.....	98
Figure 6-1: Sketch of a single span used in the GRS CFD simulations.....	103
Figure 6-2: Mean velocities on trace 1 at 10 mm downstream of spacer grid using different meshes.	107
Figure 6-3: Mean velocities on trace 2 at 10 mm downstream of spacer grid using different meshes.	108
Figure 6-4: Mean velocities on trace 1 at 10 mm downstream of spacer grid using different discretization.....	109
Figure 6-5: Mean velocities on trace 1 at 10 mm downstream of spacer grid using different turbulence models	110
Figure 6-6: Mean velocities on trace 1 at 10 mm downstream of spacer grid using different turbulence models.	111
Figure 6-7: RMS values of velocities on trace 1 at 10 mm downstream of spacer grid using different turbulence models.	112
Figure 6-8: Calculated RMS values of velocities on trace 1 at 10 mm downstream of spacer grid using different URANS models compared with the experiment.	113
Figure 6-9: Calculated RMS values of velocities on trace 1 at 250 mm downstream of spacer grid using different URANS models compared with the experiment.	114
Figure 6-10: Schematic representation of the mechanical model.....	116
Figure 6-11: Recalculation of the lateral deflection of SG2 in the impact test.....	117
Figure 6-12: Values for fluid damping measured on shaker table tests for real fuel assemblies under different axial velocities in [58], [59], [60], [64], [66] and [67].	118
Figure 6-13: Visualization of the data extension process	118

Figure 6-14: APSD of lateral displacement at spacer grid 2 for damping $D = 0.5$	120
Figure 6-15: APSD of lateral displacement at spacer grid 2 for damping $D = 0.8$	120
Figure 7-1: General view of the modeled flow part of the ALAIN test assembly	122
Figure 7-2: General view of the prepared computational meshes for the ALAIN FA CFD model	123
Figure 7-3: Boundary conditions specified to the ALAIN FA CFD model.....	124
Figure 7-4: Directions for the LDA velocity measurements in the ALAIN experiment	125
Figure 7-5: Axial velocity profiles in the Transverse 1 (a) and Transverse 2 (b) directions. ..	126
Figure 7-6: Transverse velocity profiles in the Transverse 1 (a) and Transverse 2 (b) directions.	127
Figure 7-7: Possible representation of the FA model.	129
Figure 7-8: Restored mode shapes.....	131
Figure 7-9: Comparison of restored versus experimental frequencies	132
Figure 7-10: Modal Assurance Criterion	133
Figure 7-11: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SST, Force XY: PF, solver: NW)	136
Figure 7-12: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SAS - SST, Force XY: PF, solver: DH)	136
Figure 7-13: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SAS - SST, Force XY: SF, solver: DH)	136
Figure 7-14: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SAS - SST, Force XY: PF, solver: NW)	137
Figure 7-15: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SAS - SST, Force XY: SF, solver: NW)	137
Figure 7-16: Oscillations of the assembly at first seconds of transient	138
Figure 7-17: Distribution by height of the time-averaged axial flow velocity in characteristic cross-sections in two directions.....	139
Figure 7-18: Distribution by height of the time-averaged transverse flow velocity in characteristic cross-sections in two directions.....	140
Figure 7-19: Vibration spectrum in X-direction for SG2.....	142
Figure 7-20: Vibration spectrum in Y-direction for SG2.....	142
Figure 7-21: Vibration spectrum in X-direction for SG3.....	142
Figure 7-22: Vibration spectrum in Y-direction for SG3.....	143
Figure 7-23: Vibration spectrum in X-direction for SG4.....	143
Figure 7-24: Vibration spectrum in Y-direction for SG4.....	143
Figure 8-1: Strategies for mechanical modelling of fuel assembly structure.	146

Figure 8-2: Benchmark - axial flow mean velocity profiles - traverses T1 and T2.	150
Figure 8-3: Benchmark - cross flow mean velocity profiles - traverses T1 and T2.	151
Figure 8-4: Benchmark - axial flow RMS velocity profiles - traverses T1 and T2.	152
Figure 8-5: Benchmark - cross flow RMS velocity profiles - traverses T1 and T2.	153
Figure 8-6: Benchmark: Axial flow field mean velocity profiles - deviation form experimental results.....	154
Figure 8-7: Benchmark: Cross flow field mean velocity profiles - deviation form experimental results.....	154
Figure 8-8: Benchmark: Axial flow field RMS velocity profiles - deviation form experimental results.....	155
Figure 8-9: Benchmark: Flow field RMS velocity profiles - deviation form experimental results.	155
Figure 8-10: Benchmark: RMS of lateral displacements u_x and u_y – deviation from experimental results.	157
Figure 8-11: Benchmark: Comparison of time histories of the lateral displacements u_x and u_y	158
Figure 8-12: Benchmark: Comparison of the PSD of lateral displacements u_x and u_y	159

List of tables

Table 3-1: Flow channel and test fuel assembly bundle main data.....	25
Table 3-2: Structural mass of single FA parts relevant for mechanical modelling.....	25
Table 3-3: Material properties.	26
Table 3-4: Natural frequencies for fuel assembly and skeleton deduced from impact tests..	26
Table 4-1: Overview of CFD calculation cases.....	39
Table 4-2: Fuel assembly vibration response – RMS vibration amplitude.....	61
Table 5-1: Summary of the numerical methods.	70
Table 5-2: summary of the mesh characteristics. The target mesh cell size given as well as the number of control volumes are indicators of the computational cost. The average value of the wall mesh cell size y^+ is also provided for reference.....	72
Table 5-3: Average of the absolute error (in SI units) on the axial velocity profiles obtained from RANS simulations at several locations downstream of the grid. Results are given for the traverses T1 and T2, for the single span domain, and for the whole domain behind the first and second grid (Grid 1 and Grid 2, respectively).	77
Table 5-4: Table 4.2.1.4: average of the absolute error (in SI units) on the transverse velocity profiles obtained from RANS simulations at several locations downstream of the grid.	

Results are given for the traverses T1 and T2, for the single span domain, and for the whole domain behind the first and second grid (Grid 1 and Grid 2, respectively).	80
Table 5-5: Average of the absolute error (in SI units) on the average axial velocity profiles at several locations downstream of the grid. Results are shown for the RANS and LES models (at different refinement) and for the two traverses.....	82
Table 5-6: Average of the absolute error (in SI units) on the average transverse velocity profiles at several locations downstream of the grid. Results are shown for the RANS and LES models (at different refinement) and for the two traverses.....	85
Table 5-7: Average of the absolute error (in SI units) on the axial velocity rms profiles at several locations downstream of the grid. Results are shown for the RANS and LES models (at different refinement) and for the two traverses.....	87
Table 5-8: Average of the absolute error (in SI units) on the transverse velocity rms profiles at several locations downstream of the grid. Results are shown for the RANS and LES models (at different refinement) and for the two traverses.....	89
Table 5-9: Material properties employed for the EB beam and shell elements used in the finite element model of the bundle	91
Table 5-10: Natural frequency (in Hz) of the first modes of the skeleton (guide-tubes and grids) and the bundle measured in ALAIN experiment impact test and predicted by the Cast3M finite element model.	92
Table 5-11: Value of the first added modal coefficient computed for the single span domain. The results are given for the two different meshes.	99
Table 6-1: summary of the mesh characteristics as output by the checkMesh tool	104
Table 6-2: boundary conditions of the CFD simulations	104
Table 6-3: numerical schemes.....	105
Table 6-4: Natural frequencies and mode shapes after fitting.....	116
Table 7-1: The main characteristics of the prepared computational meshes	123
Table 7-2: Characteristic values of velocity RMS	141
Table 8-1: Strategies for CFD simulations followed by different benchmark participants....	146
Table 8-2: CFD simulations used for the flow field benchmark.	147
Table 8-3: Benchmark: Comparison of the RMS of lateral displacements u_x and u_y	157

List of annexes

Appendix 5-1: Computation of the added modal coefficients.....	100
--	-----

Abbreviations and acronyms

Acronym	description
BOL	Begin Of Life
CAD	Computer Aided Design
CFD	Computational Fluid Dynamics
CT	Cladding Tube
DES	Detached-Eddy Simulation
DNS	Direct Numerical Simulation
FA	Fuel Assembly
FIV	Flow-Induced Vibration
FR	Fuel Rod
FSI	Fluid-Structure Interaction
GO-VIKING	Gathering experience On Vibration ImpaKt In Nuclear power Generation
GT	Guide Tube
GTRF	Grid-To-Rod Fretting
IAEA	International Atomic Energy Agency
ITTL	Implicit Three-Time Levels
LDA	Laser Doppler Anemometer
LES	Large-Eddy Simulation
NPP	Nuclear Power Plant
PSD	Power Spectral Density
PWR	Pressurized Water Reactor
RANS	Reynolds-averaged Navier-Stokes
RMS	Root Mean Square
SG	Spacer Grid
SGS	Sub-Grid Scale
SRS	Scale Resolving Simulation
WALE	Wall Adapting Local Eddy-viscosity

Summary

The GO-VIKING deliverable D2.4 is concerned with providing the results of the numerical simulations for the Benchmark 2 of Work Package 2, referred to as the ALAIN Benchmark. Numerical simulations related to this benchmark are based on the experimental results of the flow-induced vibration of a reduced 5x5 pressurized water reactor fuel assembly subjected to axial water flow in the ALAIN test loop provided by Framatome in the deliverable D2.3 [1]. The area of interest for the numerical simulations is the test section that includes the test channel, and the test fuel assembly fixed inside the channel at its bottom and top side and subjected to axial flow.

According to the available experimental results, the focus of the numerical simulations is on the prediction of the fluid flow distribution within the test fuel assembly as well as on the vibration response of the fuel assembly for given flow rate corresponding to average axial velocity of 5.2 m/s i.e. Reynolds number of 88500 based on hydraulic diameter of the cross-section in the rod bundle area. The results of numerical simulations in terms of the flow profiles along two profile lines at five axial positions downstream the second spacer grid build the first part of the benchmark. The second part of the benchmark is related to vibration response of the fuel assembly in terms of the Root Mean Square (RMS) and Power Spectral Density (PSD) of the lateral displacement at three middle spacer grids.

Four contributing partners: Framatome GmbH (FRA-G), Commissariat à l'énergie atomique et aux énergies alternatives (CEA), Gesellschaft für Anlagen- und Reaktorsicherheit gGmbH (GRS) and IPP Centre LLC (IPP) were involved in the benchmark. They performed numerical simulations using different simulation approaches and different tools.

The main conclusions and take aways from the performed simulations are:

The turbulence modeling approach used in the CFD simulation plays a significant role for the predictability of the vibration response. The scale-resolving simulation approaches turned out as the right choice for the purpose of simulation of FIV. Wall modeling is the only option that can be realized for given configuration in combination with scale-resolving simulations. Wall-modeled LES was found to perform best in prediction of unsteady turbulent flow and to provide the turbulence-induced fluid forces. On the other hand, the methods based on the use of RANS turbulence models showed poor performance in prediction of turbulent flow fluctuations relevant for FIV and SAS-SST model showed significant damping resulting in a significant underestimation of vibration amplitude.

Computational effort is a significant limitation in the analysis of the ALAIN benchmark. Especially the requirement for more advanced turbulence modeling approaches such as LES makes it even more difficult to manage a CFD simulation. This is also a significant limitation in terms of performing coupled FSI simulations, which proved to be unfeasible for most participants.

Because of the long slender structure of a fuel assembly, the mechanical model can be represented very well by beam elements. Thereby, the whole fuel assembly may be represented by just one or by multiple beams. Additional elements can be included to capture some special features, such as the clamping of the fuel rods in the spacer grids, to simulate the effects of these features on the overall structural dynamic properties of the fuel assembly. This typical

approach for structural modeling of the fuel assemblies facilitates significantly the handling of the structural model and increases the computational efficiency.

Small vibration amplitude of just a few μm RMS measured in the FIV tests justify the use of the decoupled analysis for evaluation of the FIV response. In this case the mechanical model must be extended to include the effects of fluid on the dynamic behavior of the fuel assembly. Here, added mass and fluid damping play the most important role. One must be very careful in evaluation of these parameters since they may have a significant effect on the vibration response.

Key results from the benchmark show that vibration response of the reduced-scale fuel assembly can be predicted qualitatively both in terms of vibration frequency and vibration amplitude. Qualitatively, an accuracy of $\pm 85\%$ of the vibration amplitude measured in the FIV tests could be achieved. Considering a very small vibration amplitude measured in the tests, of just a few μm RMS, and the overall uncertainties of the measurements, which could not be quantified exactly, the achieved results are quite reasonable.

Keywords

ALAIN, Flow-Induced Vibration, Fluid Structure Interaction, FIV, FSI, Turbulence Modeling, Numerical Simulation

1 Purpose

Vibration induced by coolant flow represent an important challenge for the integrity and operability of different components of the Nuclear Power Plants (NPP). The Flow Induced Vibration (FIV) leads to increased wear and tear and/or material fatigue, so that it has been and remains important contributor to failures of key NPP components. Such failures can degrade NPP safety features and fail confinement barriers. Some (typical) examples of the NPP components affected by FIV are steam generators and fuel assemblies.

The GO-VIKING (**G**athering experience **O**n **V**ibration **I**mpaKt **I**n **N**uclear power **G**eneration) project is aimed to contribute to an improvement of the current state-of-the-art knowledge on FIV and its analysis in support of a safe and reliable operation of NPP. The overall objective of GO-VIKING project is the increase of the expertise and improvement of the tools and skills of the European nuclear stakeholders in the analysis of complex FIV phenomena, which, in turn, will help maintain and further improve the operation and safety of NPP. This will be accomplished by:

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

The contribution of the GO-VIKING project to the improvement of the safety of contemporary reactors and the design evaluation of new concepts is in making available new experimental results and in the improvement of numerical approaches for the evaluation of FIV.

The GO-VIKING deliverable D2.4 is concerned with providing the results from the numerical simulations of the FIV of a reduced 5x5 Pressurized Water Reactor (PWR) fuel assembly subjected to axial single-phase flow based on the experimental data provided in deliverable D2.3. The main contribution of the deliverable D2.4 to the overall project objectives is through development of numerical simulation approaches for FIV analysis of fuel assemblies under axial single-phase flow. Because of its specific configuration based on use of real components and incorporating all relevant features of a real PWR fuel assembly, this benchmark is especially attractive for numerical simulation since it can greatly contribute to improvement of the understanding of complex FIV phenomena in fuel assemblies. On the other hand, although significantly smaller compared to the fuel assemblies in the real scale, the configuration investigated in this benchmark is still very challenging for numerical simulation. Besides complex physical phenomena that require the use of high-level multi-physics modelling approaches, a big challenge represents also the computational effort required for its simulation. Therefore, the essential contributions of this benchmark are in understanding of the physics of the fluid flow inside fuel assembly and associated FIV mechanism as well as strategies for efficient simulation.

2 Introduction

During operation nuclear fuel assemblies are subjected to FIV. The main concern for fuel assemblies with respect to FIV was for long time a damage of the fuel rod cladding due to fretting wear that may occur at the points of contact with the Spacer Grids (SG). According to International Atomic Energy Agency (IAEA), Grid-To-Rod Fretting (GTRF) has been the main identified fuel failure mechanism in PWR from the mid-1990s to 2015. Figure 2-1 shows the distribution of the main fuel failure causes for PWR fuel for periods 2006-2010 and 2011-2015 from the Review of Fuel Failures in Water Cooled Reactors (2006–2015) IAEA report [2].

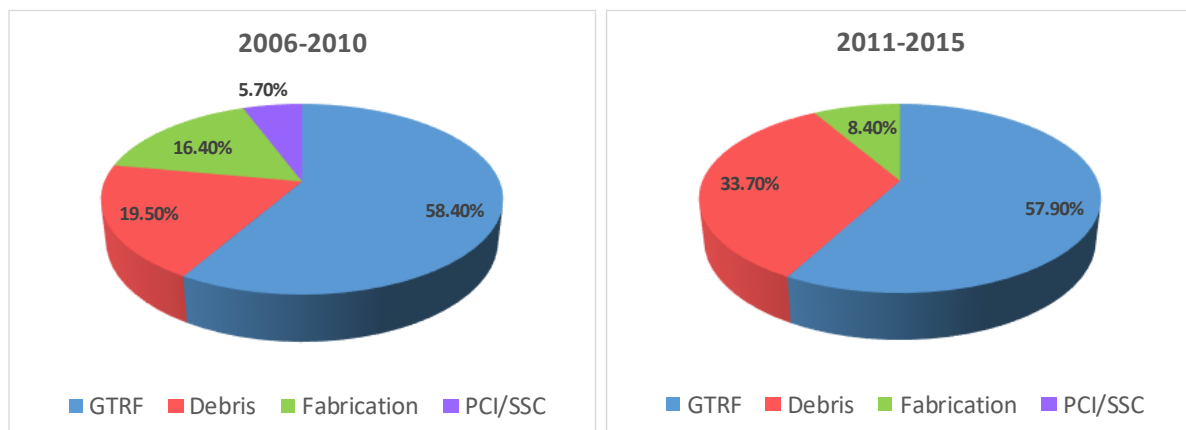


Figure 2-1: PWR fuel leak causes worldwide in 2006–2010 (left) and 2011–2015 (right).

Any defects of fuel rods or fuel assemblies in general may cause unplanned shutdowns of the nuclear power plants, which are typically associated with considerable operational costs. There is a strong interest in a reliable free-of-failure operation of a fuel assemblies during their in-reactor operation time. Robust operational behavior with respect to FIV is an important milestone for the achievement of this ultimate design objective.

The analysis of FIV in fuel assemblies is a quite complex task both, from the technical point of view and from the standpoint of the physical phenomena to be investigated. This high complexity is the main reason why experimental investigations have been primarily utilized for investigation of FIV ([3], [4], [5]). Analytical and numerical studies are applied only to a limited extent and usually to support experimental investigations.

The analysis of the FIV by numerical simulation has experienced a significant increase in recent years. With permanently growing computer power and advancements in the analysis methods, the numerical simulation approach is becoming increasingly an important tool in the FIV analysis of nuclear fuel assemblies. In particular, the use of Computational Fluid Dynamics (CFD) for the FIV analysis is increasing considerably. Many CFD studies have been performed with the aim to investigate different aspects of FIV of fuel assemblies. Depending on the objectives of a particular study and the mechanisms driving the FIV, the analyses vary from steady-state fluid flow simulations to fully coupled unsteady Fluid-Structure Interaction (FSI) simulations.

Steady-state fluid flow simulations are focused mainly on evaluation of global (e.g. within nuclear reactors [6]) or local (e.g. within a fuel assembly) flow conditions. Global and local flow distributions provide important qualitative input for the FIV design assessment. Unsteady CFD

simulations are used either as stand-alone or as a part of coupled FSI simulations in combination with structural models. Stand-alone unsteady CFD simulations are typically used to provide further refined information about the flow field and in most cases to provide the unsteady (dynamic) fluid forces acting on the fuel assembly and fuel rods [7]. Fluid forces calculated by unsteady CFD simulations can be used in subsequent structural dynamics simulations to evaluate structural response in the frame of a decoupled FIV analysis [8], [9]. In the case of coupled FSI simulations the vibration response is calculated directly as the main result of the simulation [10], [11].

Most of the numerical simulations available in the literature focus on the FIV analysis for fuel rods in connection with turbulence-induced vibration and GTRF. On the other hand, studies analyzing FIV of the whole fuel assemblies or rod bundles equipped with spacer grids subjected to axial flow are rare. The main reason for this is the difficulty to deal with the complete (full-scale) fuel assemblies in numerical simulations. As addressed in [3] and [4] the FIV behavior can play a significant role in the operational reliability of PWR fuel assemblies and is therefore highly rated in the design verification process emphasizing the importance of the of FIV analysis and assessment.

The ALAIN Benchmark analyzed in this report represent a step forward towards numerical simulation of FIV of nuclear fuel assemblies. This report presents the results of the numerical simulations of FIV for a reduced 5x5 PWR fuel assembly performed by four partners within the ALAIN Benchmark of the Work Package (WP) 2 of the GO-VIKING project. Different numerical simulation approaches, tools and strategies are used by different partners providing a wide range of possibilities to analyze given problem. The numerical simulation approaches utilize unsteady CFD simulations to analyze complex turbulent fluid flow within the fuel assembly generated by spacer grids. The results of CFD simulations in terms of mean flow field and flow fluctuations field are compared with the experimental results. Dynamic fluid forces calculated by CFD simulations are used in combination with structural dynamic models of the fuel assembly to calculate the vibration response in the frame of decoupled or coupled FSI simulations.

The report is organized as follows: In chapter 3 a brief description of the experimental setup is provided as a shorten extract from the D2.3 deliverable report [1]. Chapters 4 to 7 provide the separate contributions from the partners involved in the benchmark, namely, Chapter 4 provides the results from FRA-G, Chapter 5 contains the contribution from CEA, Chapter 6 provides the contribution from GRS and Chapter 7 contains the contribution from IPP. In Chapter 8 the benchmark case is presented along with comparison of the selected simulation results from different partners with the experimental results. The overall summary and conclusions are provided in Chapter 9.

3 Problem description

The basis for the simulations is the experimental setup of the ALAIN Benchmark which is described in detail in the deliverable D2.3 document [1]. Some details related to the experimental setup relevant for this report are recalled and briefly summarized in this chapter.

3.1 Experimental setup

The experiments were carried out in the “ALAIN” test loop that was operated by the technical center of Framatome in Erlangen, Germany. The principal sketch of the test loop is given in Figure 3-1 left, and Figure 3-1 right shows the test section of the ALAIN loop with systems for measurements of vibration and flow field during tests. The experimental bench consists of a Plexiglas channel (1) with a 2.3 m long test fuel assembly (2) placed inside the test channel (1). The test channel has a square cross-section with inner dimensions 69x69 mm, and length of 2.7 m. Usage of the Plexiglas walls has allowed for the optical access to the test bundle and application of non-invasive and non-disturbing optical measurements methods. The flow of the operating medium (water) in the loop is forced by the pump (3). The flow rate through the flow channel is generally controlled by means of the two throttling valves placed upstream (4) and downstream (5) of the test section. In addition, the by-pass section (6) can also be employed for the flow control. The volume flow rate of the operating medium in the loop is measured by means of the magneto-inductive flow meter (7). The flow channel is generally controlled by means of the two throttling valves placed upstream (4) and downstream (5) of the test section. In addition, the by-pass section (6) can also be employed for the flow control. The volume flow rate of the operating medium in the loop is measured by means of the magneto-inductive flow meter (7).

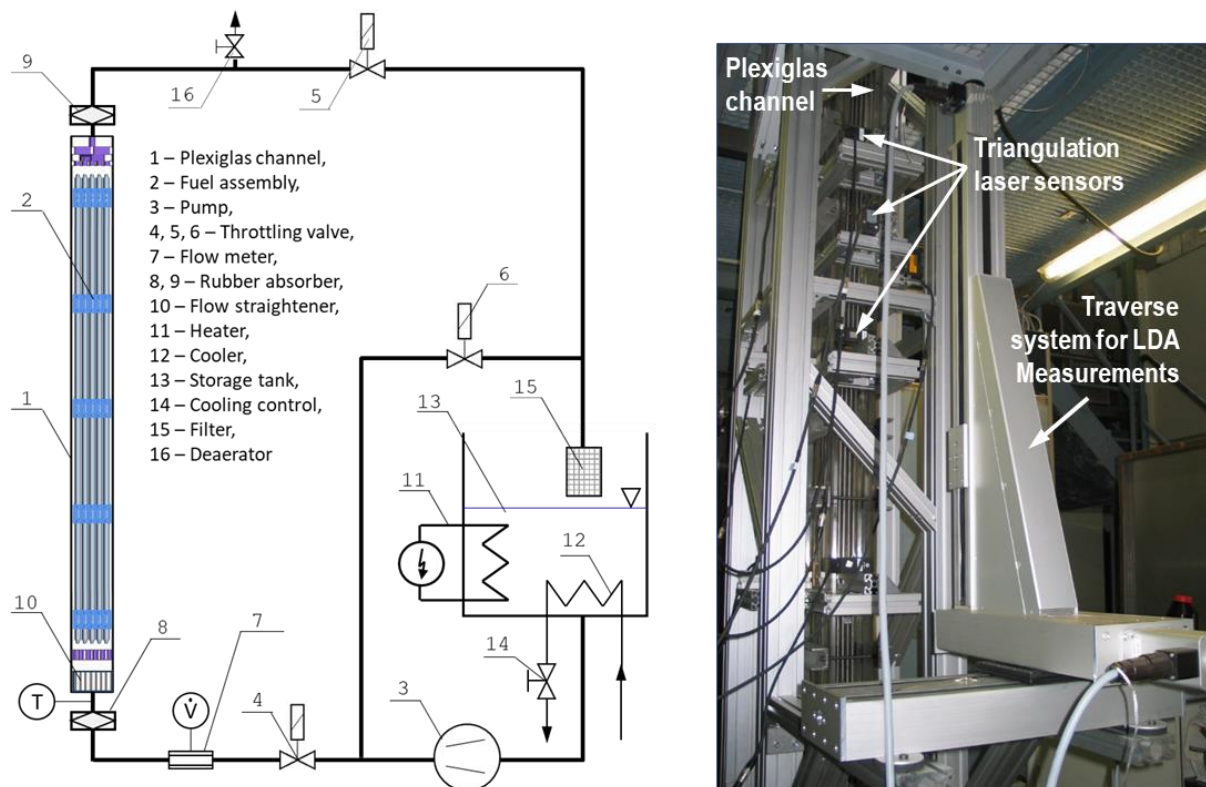


Figure 3-1: Principal sketch of the “ALAIN” test loop (left) and picture of the test section of the test loop with measurement system (right).

Rubber absorbers are implemented at inlet (8) and outlet (9) of the test section to prevent transmission of mechanical vibrations generated through the pump to the test section. To

achieve a smooth flow distribution, a flow straightener (10) is implemented at the entrance in the tests section. The temperature of the operating fluid in the test loop is adjusted by means of the heater (11) and/or cooler (12) placed in the storage tank (13). The throttling valve (14) is responsible for control of the cooling capacity. Process medium is filtered by the filter (15). The manual deaerator (16) is placed at the top of the test loop.

3.2 Test fuel assembly

An overview of the test fuel assembly is given in Figure 3-2. It consists of a 5x5 square rod bundle built of 23 Fuel Rods (FR) and 2 Guide Tubes (GT). Fuel rods have outer diameter d_{FR} of 9.5 mm and the pitch between rods is p is 12.32 mm. The 5x5 spacer grid sections have a width of 66 mm resulting in a nominal side gap between bundle and test channel of 1.5 mm.

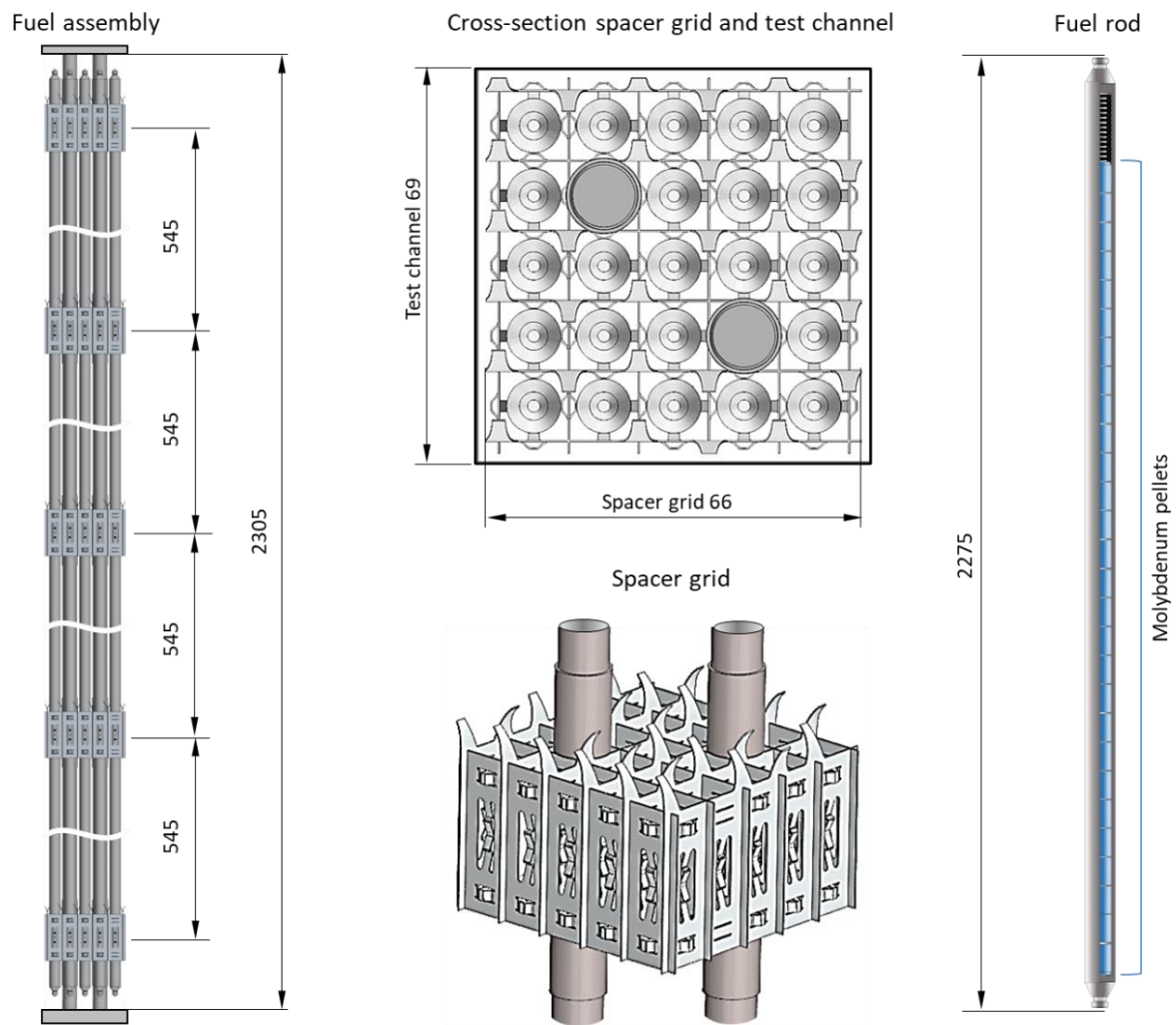


Figure 3-2: Overview of 5x5 fuel assembly.

Two guide tubes together with five spacer grids, a top and a bottom plate form the skeleton of the fuel assembly. The 5x5 spacer grid sections are obtained by cutting out from the original 16x16 spacer grids used in real fuel assemblies. Spacer grids are connected rigidly to the guide tubes and distributed equidistantly along the bundle providing a span length of 545 mm. The bottom plate, which is cut out from original bottom nozzle filter is fixed to the base plate of the test loop. At the top side an upper plate is implemented with a hold down system to fix

the bundle in axial position inside the test channel. The total length of the test fuel assembly corresponding to the distance between lower and upper tie plate is 2305 mm.

The test bundle is equipped with test rods of the length of 2275 mm. The test rod consists of a cladding tube (CT) with the outer diameter of $d_{CT,out}=9.5$ mm and the inner diameter of $d_{CT,in}=8.22$ mm made of Zircaloy. The test rods are filled with Molybdenum pellets to simulate the mass of Uranium oxide pellets in real fuel assemblies. At the bottom side there is a supporting tube and at the top side is a plenum spring which keeps the Molybdenum pellets in position. The rods are clamped in the spacer grid cells in accordance with Begin Of Life (BOL) rod support conditions corresponding to a nominal clamping spring force of 20 N. Main parameters of the test fuel assembly and material properties relevant for the mechanical modelling are summarized in Table 3-1, Table 3-2 and Table 3-3 below.

Table 3-1: Flow channel and test fuel assembly bundle main data.

Parameter	Symbol	Value	Unit
Flow channel width	B_{fch}		
Rod arrangement		5x5	
Number of fuel rods		23	
Number of guide tubes		2	
Fuel assembly length	L_{FA}	2305	mm
Fuel rod length	L_{FR}	2275	mm
Span length	L_{SP}		
Fuel rod outer diameter	d_{FR}	9.5	mm
Fuel rod (cladding tube) inner diameter	$d_{FR,i}$	8.22	mm
Guide tube outer diameter	$d_{GT,o}$	11.96	mm
Guide tube inner diameter	$d_{GT,i}$	11.05	mm
Rod filling – Mo pellets column ($D \times H$)*		8.05 x 2160	mm
Spacer grid dimensions ($B \times H$)*		66 x 38	mm
*) D stands for diameter, B stands for width, H stands for height			

Table 3-2: Structural mass of single FA parts relevant for mechanical modelling.

	Mass [kg]	
	single parts	total
Fuel Rod	1.4011	32.224
Guide Tube	0.3366	0.673
Spacer Grid	9.183E-02	0.4592
Total Fuel Assembly*		
*) Only components which are relevant for the fuel assembly mechanical model. Other components are fixed and hence their mass does not affect the dynamic behavior of the fuel assembly		

Table 3-3: Material properties.

Material	Density [kg/m ³]	E-modulus [GPa]		Parts
		20°C	40°C	
Zircaloy	6600	98.1	96.79	CT, GT
Alloy 718	8200			SG
Molybdenum	10200			pellets

3.3 Natural frequencies

Natural frequencies relevant for the mechanical model, which are deduced from the lateral displacements recorded during impact tests are summarized in Table 3-4. Natural frequencies in different lateral directions are very close to each other, especially for fuel assembly, indicating homogeneous/symmetric mechanical properties in both lateral directions.

Table 3-4: Natural frequencies for fuel assembly and skeleton deduced from impact tests.

	Natural frequency mode →	1	2	3	4	5	6
Fuel assembly	<i>x</i> direction (0°-180°)	6.0	14.4	23.9	---	---	---
	<i>y</i> direction (90°-270°)	6.0	14.5	23.9	---	---	---
Skeleton	<i>x</i> direction (0°-180°)	9.1	16	23.9	34.4	44.3	51.15
	<i>y</i> direction (90°-270°)	8.8	15.8	23.55	34.1	43.9	50.8

3.4 Flow velocity profiles setup

During the test campaign LDA (Laser Doppler Anemometer) measurements were carried out to evaluate flow field behind the second spacer grid. The LDA measurements were performed at five different elevations downstream the second spacer grid. At each elevation the flow field profiles were measured along two lines (traverses) located in the middle of the flow sub-channels next to the central rod row. A detailed presentation of the test setup for the flow velocity measurements is provided in Figure 3-3. Note that for the profiles at axial position 10 mm behind the 2nd spacer grid a correction in the horizontal positioning of the profile lines is made because of the presence of the guide tube sleeves at this position which have a diameter larger than the fuel rod pitch.

3.5 Benchmark setup and objectives

The experimental setup described above serves as the basis for the setup of the numerical simulations of Benchmark 2 (“ALAIN Benchmark”) of the WP2. The numerical simulations include fluid flow and structural mechanics analyses.

For the fluid flow simulations, the numerical setup needs to reflect the following global flow boundary conditions used in the tests:

- Fluid flow is considered as incompressible,
- Operating conditions in the test:

- Temperature: $T = 40^{\circ}\text{C}$,
- System pressure: $p_{\text{abs}} = 4 \text{ bar}$,
- Average axial flow velocity of $\langle w \rangle = 5.2 \text{ m/s}$ in the test channel cross-section in the pure bundle area¹.

According to H₂O IAPWS-IF97 standard [12], the corresponding fluid properties of water are obtained for the operating conditions given above:

- Density: $\rho = 992.36 \text{ kg/m}^3$,
- Dynamic viscosity: $\mu = 6.53 \cdot 10^{-4} \text{ Pa s}$.

Mass flow rate corresponding to average axial velocity $\langle w \rangle = 5.2 \text{ m/s}$ in the test channel cross-section in the pure bundle area is:

- Mass flow rate: $\dot{m} = 15 \text{ kg/s}$.

The geometry of the fluid flow domain is defined by the CAD (Computer Aided Design) models of the fuel assembly structure and of the test channel which are delivered in [1].

For the structural model following global input parameters are to be used:

Mass of different components given in Table 3-2 and properties of corresponding materials given in Table 3-3.

Table 3-3 In addition, depending on the modelling strategy for structural model, the following boundary conditions are recommended to be used:

- fully clamped boundary conditions for guide tubes or whole bundle at bottom and top end (zero displacements and zero rotations),
- clamping force of 20 N for the fuel rods inside spacer grids.

Independently of the modelling strategy, final structural models should match as good as possible first three natural frequencies of the fuel assembly that were deduced from the mechanical impact tests, which are provided in Table 3-4. For the case that skeleton is also modelled explicitly (e.g. in the frame of a multi-beam model) the natural frequencies of the skeleton deduced from impact tests can also be used as additional reference for the fine tuning of the structural dynamic model used in the simulations.

¹ Test channel cross-section area reduced by the cross-section area of all rods and guide tubes

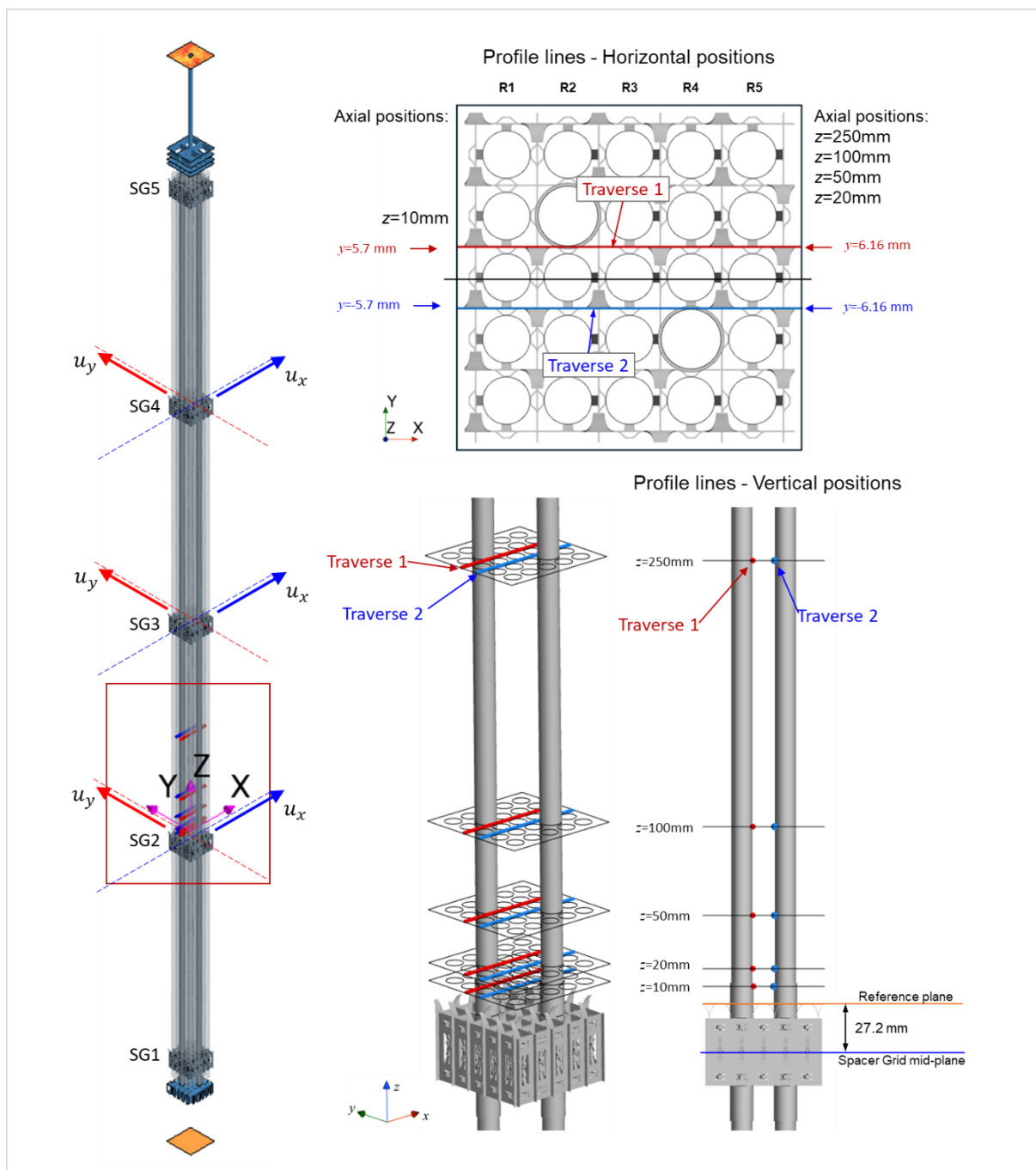


Figure 3-3: Overview of the fluid flow profiles and FIV measurements setup.

4 Simulation performed by Framatome

In this chapter the numerical simulations of the ALAIN benchmark performed by Framatome (FRA-G), and results of these simulations are described and presented in detail.

4.1 Introduction

The configuration that needs to be analyzed includes a small-scale 5x5 fuel assembly equipped with spacer grids resulting in a quite complex geometry. Although the considered fuel assembly is quite small compared to original fuel assembly size, from the standpoint of a CFD simulations this configuration is still very large. This makes it quite challenging for numerical simulation requiring a large computational effort even for a moderate resolution. Before making the decision on the simulation strategy, it is worthwhile to look at the experimental results because this can in some way influence the selection of a suitable simulation strategy for this complex configuration.

The vibration response of reduced fuel assembly measured in the FIV tests is characterized by a very low vibration amplitude of just a few $\mu\text{m RMS}^2$, which is typical for turbulence-induced vibration. In that case the vibration is driven by dynamic fluid forces, which are transmitted to the structure by fluctuating turbulent pressure and shear stress [13]. Turbulent fluctuations result in a kind of random excitation force composed of many periodic modes continuously distributed over a wide range of frequencies with amplitudes corresponding to turbulence spectrum. The main assumption in the analysis of turbulence-induced vibration, among the methods evolved over time (see e.g. [14]), is that there is no strong interaction between fluid flow and structural motion. In other words, it is assumed that the structural motion does not affect the characteristic of the fluid forces exciting the vibration of the structure. Such assumption is typically applicable to vibration analysis of structures exhibiting stable forced vibration with small amplitude.

All this raises the question of what kind of analysis is suitable or sufficient for the configuration analyzed here? A coupled FSI simulation is clearly the most straight forward analysis approach, but also the most complex and for the given case extremely challenging one. Compared to stand-alone CFD simulations, FSI simulations involve mesh movement which increases significantly complexity of the whole simulation, not only from the standpoint of computational effort but also from the standpoint of the numerical stability and convergence. For the configuration to be analyzed here, both effects play a role. In addition, a special strategy for FSI coupling based on the approach that involves non-trivial implementation of user codes embedded in the commercial CFD code is required. On the other hand, there are some arguments speaking for decoupled simulation approach as a reasonable and suitable first step for the analysis of this challenging configuration. The main supporting argument speaking for a decoupled simulation approach is very small vibration amplitude of just a few $\mu\text{m RMS}$ measured in the FIV tests. For this level of vibration one can assume that the vibration is turbulence-induced, and that the assumptions associated with turbulence-induced vibration are applicable to present case.

² Under RMS amplitude only the dynamic component (static component removed) of displacements or forces is considered throughout this report.

Based on the discussion presented above, the strategy followed in present studies is based on decoupled analysis. Turbulence-induced fluid forces are calculated by means of the unsteady CFD simulation on non-moving geometry for the configuration used in the FIV tests described in Chapter 3, using the Simcenter™ STAR-CCM+™ software. The time histories of the (turbulence-induced) fluid forces calculated by CFD simulation are used as forcing functions in subsequent mechanical simulations to evaluate the vibration response using a structural dynamic model of the fuel assembly. In the following sections the CFD and structural dynamic simulations applied in the present studies are described in detail including the results produced by different simulations and models. At the end of the chapter a summary and conclusions drawn from the results of different simulations are provided.

4.2 CFD simulations

4.2.1 Turbulence modeling approach

The purpose of the present CFD simulations is to analyze the fluid flow inside the fuel assembly and to provide the fluid forces acting on the fuel assembly structure. Based on the discussion in previous section, the main driver of vibration is presumably turbulence. Therefore, to correctly capture the dynamic (turbulence-induced) fluid forces acting on the fuel assembly structure exciting its vibration, it is essential that turbulent fluctuations are captured properly by CFD simulation.

The methods capable to resolve turbulent flow structures, usually called in the literature as Scale Resolving Simulation (SRS) [15], seem to be appropriate candidates for this purpose. Given the fact that Direct Numerical Simulation (DNS), which stands for direct discretization and solution of the Navier-Stokes equations, is out of reach for most of the industrial applications, and so also for the present case, the approaches that involve certain level of turbulence modeling must be used.

The most prominent representative of this class of methods is the Large-Eddy Simulation (LES), which is second ranked behind DNS. LES provides for a numerical simulation in which only the energy-containing large-scale turbulent motions (large eddies) are represented directly while the small-scale motions are modeled. The process of removal of the small scales from the solution is called “filtering” and the resulting equations which are solved by LES are called filtered Navier-Stokes equations [16]. Since the small scales are not resolved, the computational effort of the LES is significantly smaller compared to DNS. Due to the universal character of the small scales, it is expected that they can be modeled by relatively simple models. However, despite significant reduction of the computational effort, the application of LES to industrial configurations is still strongly limited. Especially the requirement for the fine mesh resolution close to the solid walls makes the application of classical (wall-resolved) LES to industrial configurations prohibitively expensive.

On the other hand, the approaches based on the solution of Reynolds-averaged Navier-Stokes (RANS) equations (see Wilcox [17] and Celik [18]), which is the method of choice for most industrial applications, are considered less suitable for the present case. Contrary to LES where a part of turbulent flow structures is resolved, in the RANS approach all the turbulence is modeled. The turbulent flow fluctuations are removed and the effect of the whole turbulence on the mean flow, which is the result of the solution of the RANS equations, is represented by an additional diffusion term given by the Reynolds stress tensor. The effect of turbulent diffusion

can be expected to be significantly higher in the case of RANS equations compared to the filtered Navier-Stokes equations solved in LES. Hence, the standard RANS models inherently have stronger tendency to damp the flow fluctuations, which limits their capability to predict correctly turbulence-induced fluid forces acting on the structure. As pointed out by Menter [19] and Acton *et al.* [20], special modifications of standard RANS models are needed in order to capture the unsteady flow features representing the turbulence specific structures. Some developments in this direction can be found in Menter *et al.* [21] and [22], Baglietto *et al.* [23] and Jakirlić *et al.* [24].

One way to overcome the limitations of LES and transient RANS approaches is to use hybrid methods whose main idea is to combine the good properties of LES and RANS approaches. A representative of hybrid approaches is the Detached-Eddy Simulation (DES) first proposed by Spalart *et al.* [25]. DES combines LES and RANS methods in a single model by applying each method in the areas where it works well, i.e., the RANS mode in the wall boundary layer and the LES Sub-Grid Scale (SGS) mode in the area farther from the wall. The main idea of detached eddy simulation is to limit the eddy viscosity to allow for development of the large-scale turbulent eddies. In its original formulation [25]) this is achieved by replacing the distance to the wall, d , which is used as the length scale in the Spalart-Almaras model, by

$$\tilde{d} = \min(d, C_{DES}\Delta) \quad (4.1)$$

where C_{DES} is a constant and Δ is the representative local cell size. Thus, close to the wall, where distance to the wall d is smaller than $C_{DES}\Delta$, the model operates in RANS mode. Farther from the wall, where d is larger than $C_{DES}\Delta$, the model operates in LES mode.

The main advantage of a hybrid approach such as DES described above is in application of computationally less demanding RANS models in the region close to the walls while the LES is applied in the area farer from the walls where the requirements for the fine mesh resolution are significantly smaller. Thereby, the main attractiveness of the use of RANS models in wall boundary layer is in combination with wall models.

The use of wall models is in the meantime also available in combination with LES, the so-called Wall-Modeled (WM) LES. This is an additional great possibility to reduce further the computational costs of LES making WM LES competitive to DES. A detailed overview of the possibilities for wall models in conjunction with LES is provided by Piomelli *et al.* [26]. Such capabilities are meanwhile available also in the commercial CFD codes and can be used easily.

Another possible alternative to calculate turbulence-induced forces worth mentioning, that utilize the advantages of the RANS models regarding computational effort, is based on the solution of pressure fluctuation equation. This approach was used by some authors for simulation of FIV (see e.g. Senthoooran [27] or Kottapali *et al.* [28]), based on self-implementation in combination with different CFD codes, e.g. OpenFOAM in later case.

Because of restriction on the available commercial CFD code STAR-CCM+, this option is not eligible for present simulations. Based on the discussion on turbulence modeling presented above LES and DES approaches based on different SGS i.e. turbulence models and in combination with wall modeling, which are available in STAR-CCM+ code, were used in the present simulations. Turbulence modeling approaches are first assessed on a small-scale single-span model and based on this assessment a favorite approach is selected for the final simulation on the full-scale model representing the entire setup of the ALAIN benchmark.

In the following sections the corresponding CFD simulations performed on single-span and full-scale setup configurations are described and the results from these CFD simulations are presented in detail.

4.2.2 Overview of CFD models

CFD simulations were performed for two geometrical configurations:

- a) reduced-scale configuration in which only one span of the fuel assembly was modeled,
- b) full-scale configuration in which complete test setup was investigated.

The two configurations are shown in Figure 4-1. The single span configuration includes a spacer grid and a portion of the rod bundle and guide tubes of a length corresponding to the span length of 545 mm. In this case the periodic boundary conditions are used to couple inlet and outlet in terms of a fully developed flow with specified flow rate corresponding to the conditions realized in the FIV tests. The domain is extended upstream of the spacer grid for 1/5 of the span length measured from the mid-plane of the spacer grid, while the part downstream the spacer grid corresponds to 4/5 of span length.

The geometry of the computational domain considered in the full-scale CFD model consists of the whole test fuel assembly (includes guide tubes, fuel rods and spacer grids), the lower tie plate, upper tie plate and the hold-down system. All geometrical parts are represented in detail in the CFD model according to their CAD representation. The flow channel surrounding the test fuel assembly is extended appropriately upstream the lower tie plate and downstream the upper tie plate to adapt the inflow and outflow boundaries. The flow straightener that was installed in the tests before the entrance in the test section to ensure a regular flow distribution before entering the test section is not considered in the model. The effect of the flow straightener is modeled equivalently by the extension of the fluid domain upstream the entrance in the test section and by specifying a uniform flow profile at the inlet. Since the flow at the entrance in the fuel assembly is strongly formed by the flow filter of the lower tie plate, the need for the explicit modeling of the flow straightener in the simulation is further reduced. At the inlet the mass flow rate is specified as boundary conditions while constant pressure is applied at the outlet. All solid surfaces in both models are considered as walls with no slip conditions.

4.2.3 Numerical mesh

The CFD mesh is generated using the automated grid generation approach based on the trim mesh technology of STAR-CCM+ code in combination with part-based meshing method. Trim mesh provides the Cartesian hexahedral mesh in the core of the computational domain. At solid walls, prism cell layers are applied, while the space between core mesh and the prism cell layers is filled by trimmed cells. Local grid refinement is applied appropriately to resolve fine geometrical details, such as the contact between spacer grids and fuel rods. The number of the wall cell layers, and the cell layer thickness are selected carefully to ensure an optimal topology of the trim mesh and the desired y^+ values along the surface of the rods and on spacer grids.

Principally the same mesh settings were used to generate both meshes for single and full-scale models. The single span mesh was generated first and the full-scale mesh was generated afterwards. Some slight optimizations in the meshing setup were implemented for the full-scale mesh after certain experience was gathered on the single span model. The resulting mesh contains about 45 million of cells for the single span configuration and about 245 million cells in the full-scale model. Some details of the CFD mesh around the spacer grids and in the rod bundle area are shown in Figure 4-2.

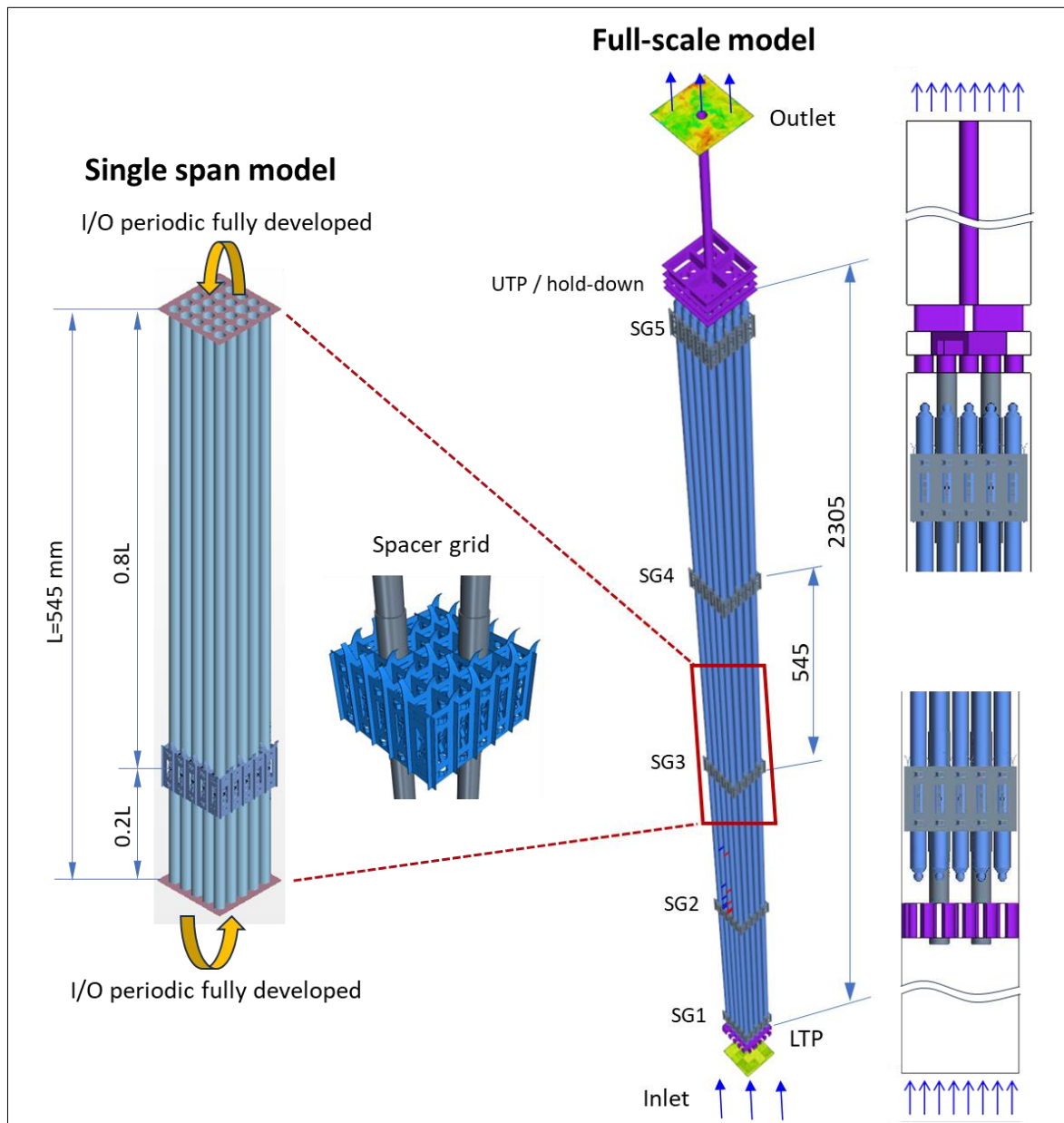


Figure 4-1: Overview of the CFD models investigated.

The numerical mesh is the central part of a CFD simulation. The overall quality of the simulation depends strongly on the quality and resolution of the numerical mesh. Mesh quality and resolution play an essential role for LES / DES simulation. The cell quality is a crucial precondition for a successful simulation, because of the non-bounding numerical schemes used in combination with LES / DES, which are much more sensitive to mesh quality compared to bounding

schemes, which are typically used in combination with RANS methods. Further, the mesh resolution is another important parameter, especially for LES, since the cell size of the mesh implicitly defines the filter size and therewith the range of turbulent scales (eddies) that are resolved by the simulation.

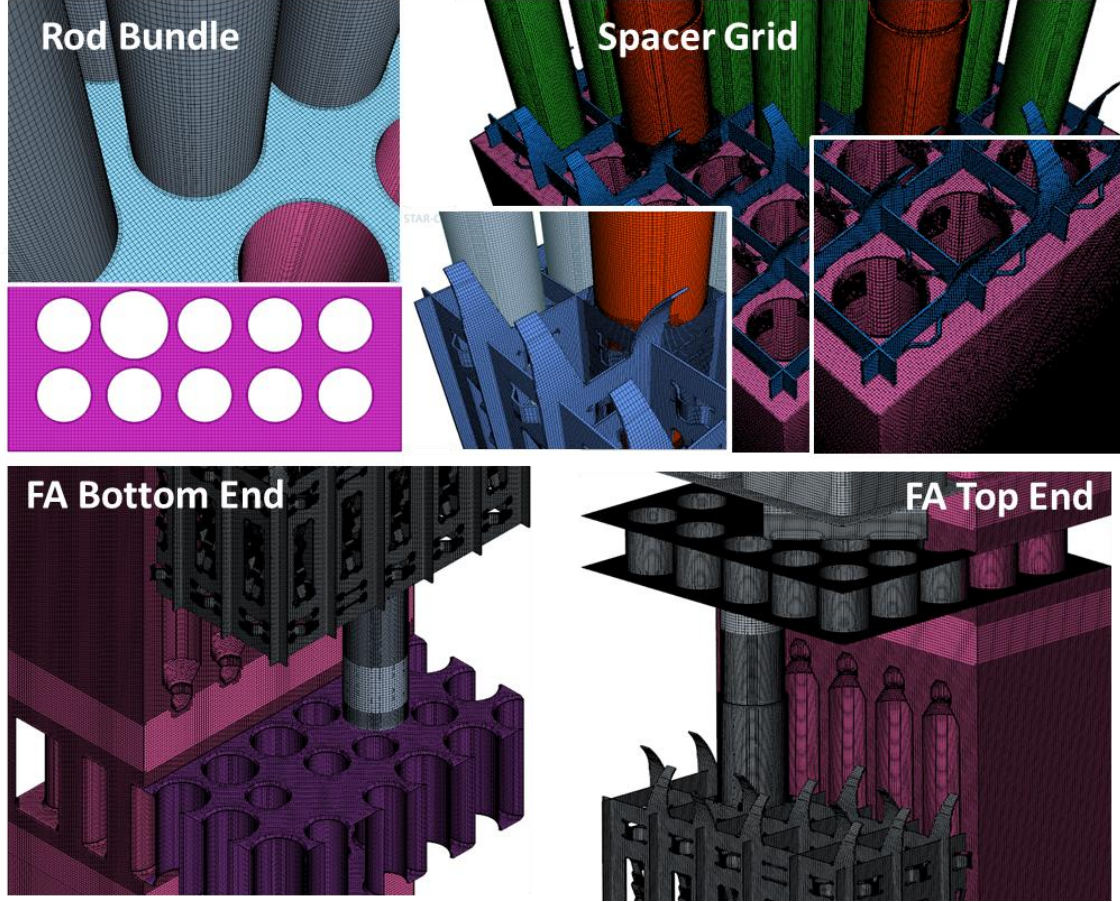


Figure 4-2: Numerical mesh details – around spacer grid and in rod bundle area.

Therefore, a special attention is paid to both aspects of the numerical mesh in the present simulations. The mesh quality was ensured by small geometrical modifications to smooth the geometry (typically at the contacts between rods and spacer grids) at critical areas that may result in bad cells.

The mesh size is estimated based on the criteria related to integral turbulent length scale L_t and Kolmogorov length scale η , which are defined as:

$$L_t = \frac{k^{3/2}}{\varepsilon} \quad \eta = \left(\frac{\nu^3}{\varepsilon}\right)^{1/4} \quad (4.2)$$

where k is turbulent kinetic energy, ε turbulent kinetic energy dissipation and ν is kinematic viscosity. The typical mesh size criteria related to these two length scales are [16], [29]:

$$\Delta x_{\text{base}} \leq \frac{1}{6} L_t \text{ and } \Delta x_{\text{base}} \leq 25\eta \quad (4.3)$$

The turbulent kinetic energy k and turbulent kinetic energy dissipation rate ε are obtained from a steady RANS simulation that was performed prior to the unsteady simulations with preferred scale-resolving models.

The selected base size satisfies criteria (4.3) in most of the bulk flow area of the fully developed axial flow within the rod bundle. Closer to solid walls criteria are not fulfilled due to the use of the wall-modeling approach.

The criteria described above based on the turbulence length scales allow for the a priori estimate of the mesh cell size prior to performing the unsteady (LES) simulation. Additional criteria can be used for a posteriori assessment of the numerical mesh. These are based on the viscosity ratio R_μ , defined as the ratio between sub-grid viscosity³ μ_{SGS} and molecular (dynamic) viscosity μ of the fluid, and the sub-grid activity parameter s (see [15]) defined as:

$$R_\mu = \frac{\mu_{SGS}}{\mu} \quad s = \frac{\mu_{SGS}}{\mu_{SGS} + \mu} \quad (4.4)$$

For viscosity ratio $R_\mu < 10$ the numerical mesh is considered sufficiently fine, and for the viscosity ratio $R_\mu > 100$ it is considered as too coarse.

Figure 4-3 shows the distribution of the viscosity ratio R_μ and sub-grid activity s for both configurations. It is evident that for a large portion of the domain in both configurations the viscosity ratio is below 10. Sub-grid activity parameter shows principally the same distribution but somehow rearranged because of its mathematical formulation.

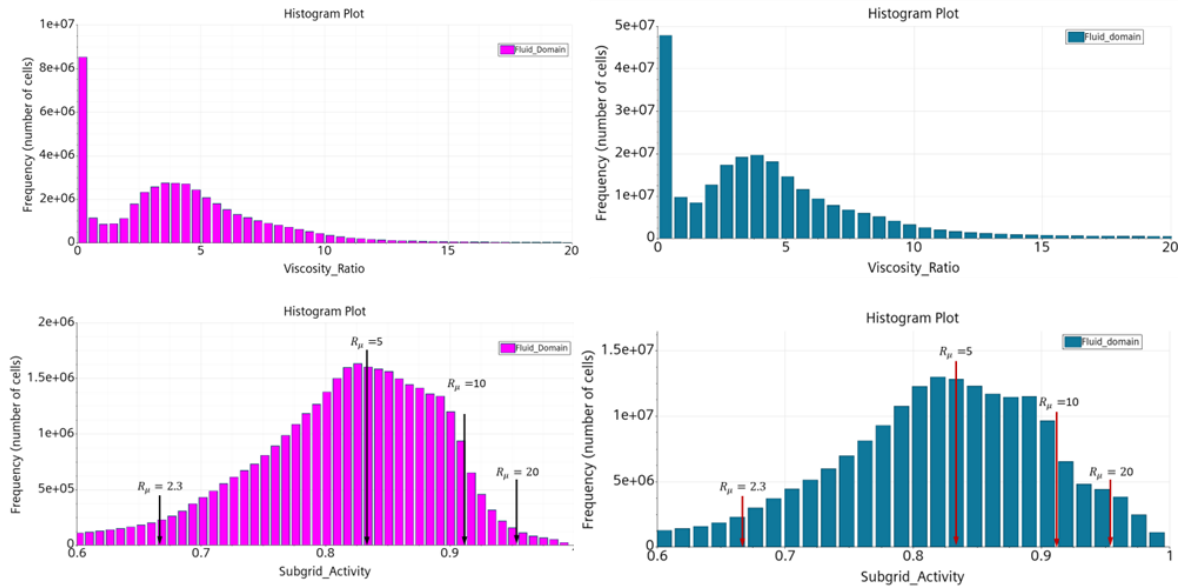


Figure 4-3: Distribution of the viscosity ratio and sub-grid activity over the computational domain for single-span configuration (left) and full-scale configuration (right).

The block-by-block meshing strategy is used, in which different mesh blocks are generated separately and the final mesh is obtained by assembling all mesh blocks together. Thereby, the repetition of different geometrical features like spacer grids as well as the long regular structure of the rod bundle is utilized. This allows for multiple use of different mesh blocks and mesh extrusions in the final volume mesh assembly. In this way the mesh generation is facilitated, which is especially advantageous for the mesh optimization which can be performed much easier for small mesh blocks due to shorter turn-around time of the mesh generation process and easier handling of smaller mesh blocks. This procedure is very beneficial

³ This definition is strictly related to terminology used for LES. For DES the turbulence model used is acting as a sub-grid scale model, and accordingly the turbulent viscosity represents also the sub-grid scale viscosity.

for the mesh generation of the full-scale model, in which case the small mesh blocks generated separately with an optimized mesh are used several times.

During mesh generation, care is taken to ensure conformal meshes at the interfaces between the mesh blocks. Thus, the final mesh could be assembled by fusing the conformal vertices on the interfaces providing a smooth continuous mesh over the whole computational domain, which is favorable for the numerical simulation.

4.2.4 Computational settings and simulation workflow

All CFD simulations are performed as unsteady incompressible single-phase flow. In all cases the corresponding models for treatment of turbulence are used in combination with the so called “all y^+ ” wall model of STAR-CCM+.

The governing equations are solved using the segregated solution procedure in combination with implicit unsteady solver based on SIMPLE algorithm [30].

For spatial discretization of the convective fluxes the non-diffusive bounded central differencing scheme is used with a small amount of 15% of upwind scheme⁴. The use of non-diffusive schemes in conjunction with scale-resolving methods (LES and DES) is mandatory to capture the sharp gradients resulting from turbulent fluctuations.

For integration in time, second order Implicit Three-Time Levels (ITTL) scheme [30] is used. The time step is determined based on the accuracy requirements given by Courant number (Co) criterion. Based on the target $Co \leq 1$ and considering the selected cell size and average flow velocity, a time step of $\Delta t = 5.0E - 5$ s (0.05 ms) results. This time step is used in all CFD simulations, in combination with all models. The distribution of Courant number realized in the simulations with single-span and full-scale models is shown in Figure 4-4, which shows similar distribution in both cases and that for the largest part of the domain the Courant number is below 1 and except for a negligibly small portion of the domain it has values higher than 1.5. These high values are typically associated with very small cells.

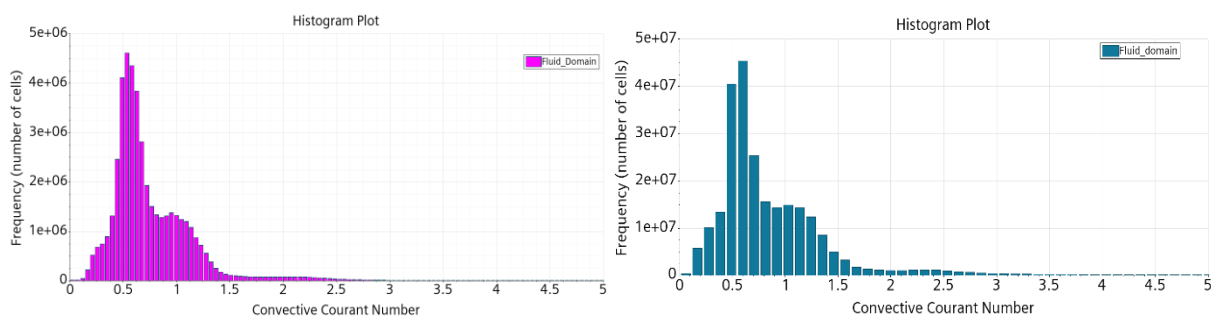


Figure 4-4: Courant number distribution in single-span model (left) and full-scale model (right).

According to the test conditions, the working fluid in the CFD simulations is water at a system pressure of 4 bar and temperature of 40°C. Corresponding values for fluid density and dynamic viscosity are calculated according to thermodynamic properties of water and steam standard H2O IAPWS-IF97 [12]. These are:

- Density (ρ): 992.35 kg/m³

⁴ Standard STAR-CCM+ setting.

- Dynamic viscosity (μ): 6.53E-04 Pa.s

The flow rate applied in the CFD simulation corresponds to an averaged axial velocity within the rod bundle of 5.2 m/s. According to fluid properties (density ρ), average axial velocity and the surface of the cross-section in the rod bundle area, the resulting mass flow rate for these conditions is $\dot{m} = 15$ kg/s.

The Reynolds number calculated based on the given flow conditions (axial flow velocity and fluid properties) and the hydraulic diameter of $D_h = 11.2$ mm derived from the flow cross-section area and vetted perimeter in the rod bundle area of the test section is $Re = 83,500$.

The CFD simulation is performed in two steps (see Figure 4-5). In the first step (phase 1) a pre-run is performed for a duration of several flow-through cycles (typically 2 -5 cycles), with a flow-through cycle being defined as the time that a fluid particle needs to travel through the computational domain. The purpose of this pre-run is to obtain fully developed unsteady turbulent flow conditions. Final runs were carried out for several further cycles (typically 5-10). During this final simulation phase, the relevant data is collected. In the present simulations the data related to the flow statistics (mean flow and flow fluctuations) and fluid forces acting on the fuel assembly structures was collected. The flow statistics is used to compare with the experimental LDA data, while the fluid forces are used in combination with the mechanical model to calculate the vibration response.

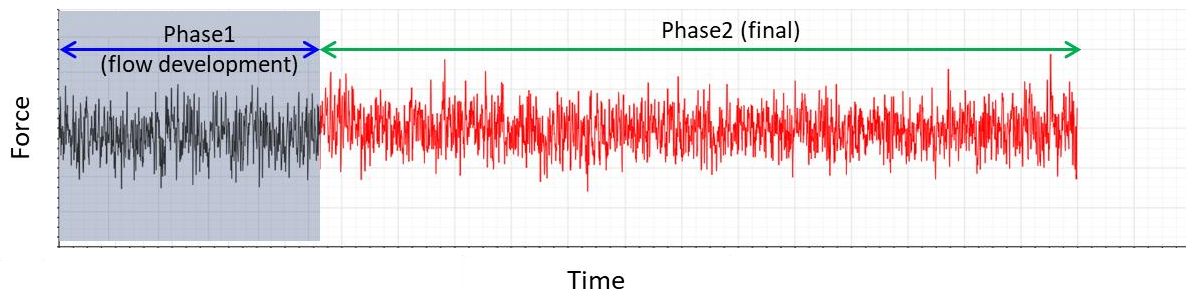


Figure 4-5: Execution of the CFD simulation.

4.2.5 Flow statistics monitors

To get insight into the flow statistics, the flow data are collected over the time during the final simulation step (phase 2 in Figure 4-5). This is achieved by activating the mean and variance field monitors in STAR-CCM+ for the flow velocity components, which collect the data over time and calculated corresponding fields based on the collected data.

The mean field for a variable ϕ is calculated as:

$$\Phi_{\text{mean}} = \frac{1}{N} \sum_{i=1}^N \phi_i \quad (4.5)$$

where ϕ_i is the value of variable ϕ at time step i and N is the number of time steps for which the data has been collected. Equation (4.5) provides the ensemble average of the variable ϕ which in this case corresponds to time average since the sample collection in equation (4.5) is made over the time.

The variance field for a variable ϕ is calculated as:

$$\Phi_{\text{variance}} = \frac{1}{N-1} \left[\sum_{i=1}^N \phi_i^2 - \frac{1}{N} \left(\sum_{i=1}^N \phi_i \right)^2 \right] \quad (4.6)$$

The variance fields calculated per equation (4.6) for velocity components provide indication of the level of flow fluctuations related to turbulence. The quantity calculated by (4.6) corresponds rather to turbulent kinetic energy (unit $\left(\frac{m}{s}\right)^2$). To have a better feeling about the level of velocity fluctuations and to compare the CFD with the experimental results, the root square of the variance field is calculated providing the values in basic units, i.e. in m/s for velocity fluctuations.

4.2.6 Monitoring of fluid forces

As described later in section 4.3, the mechanical model used for representation of the fuel assembly structure is based on beam elements, which is the typical way for modeling the slender long structure of fuel assemblies. Thus, the whole structure of fuel assembly is discretized by several nodes which are connected to each other by beam elements. To calculate vibration response of the fuel assembly induced by the flow, fluid forces need to be transferred to the nodes of the mechanical model. This is done by integration of pressure and shear stress over the corresponding surface of the fuel assembly that can be associated to corresponding calculation nodes in the mechanical model as illustrated in Figure 4-6.

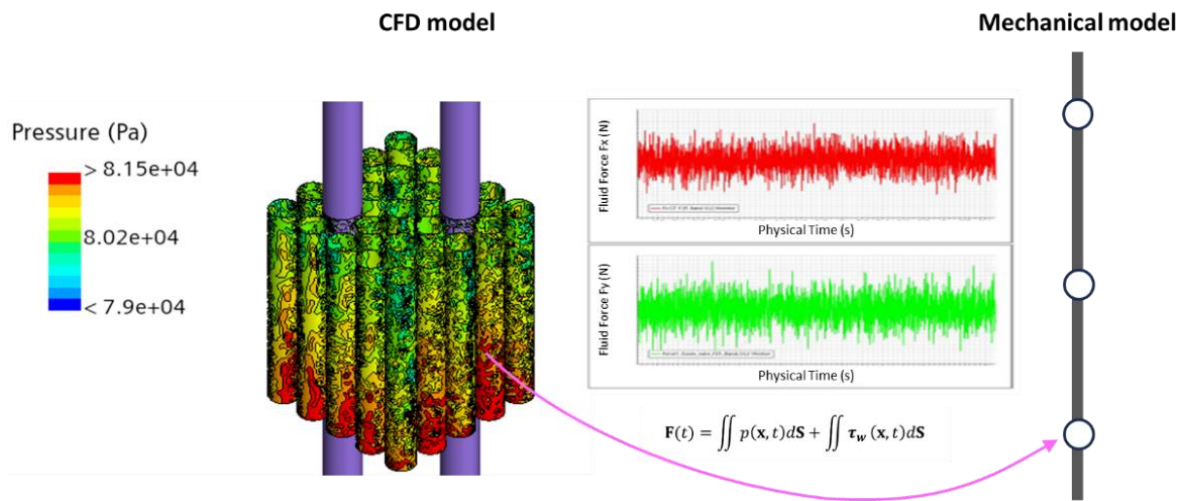


Figure 4-6: Mapping of fluid forces to the mechanical model.

4.2.7 Simulation results

An overview of all CFD simulations performed including geometrical configuration, modelling approach, turbulence model, running time in the final phase and number of samples for the flow statistics associated with each simulation are summarized in Table 4-1.

As explained above, first simulations were performed with reduced single span configuration with the objective to assess different simulation approaches and associated turbulence models in their capability to simulate properly the presents case. The modeling approach and the associated turbulence model for the CFD simulation of the full-scale configuration was made based on the assessment of the results obtained for single span configuration.

Table 4-1: Overview of CFD calculation cases.

Case Nr.	Geometrical Configuration	Modeling approach	Turbulence / SGS Model	Running time final phase	No. of samples for statistics
1	Single span	LES	WALE	2 s	40000
2			Smagorinsky		
3		DES	Spalart-Almaras		
4			Eliptic Blending		
5			SST		
6	Full-scale	LES	WALE	4.3	86000

Therefore, the results obtained for the single-span configuration are presented first. The results obtained with different approaches and models are assessed against experimental data for flow field profiles.

Based on some key findings from this assessment the decision was made regarding the modeling approach and the turbulence model to be used in the CFD simulation on the full-scale configuration. The results from this simulation on the full-scale model are presented afterwards.

4.2.7.1 Single span configuration

4.2.7.1.1 Flow field profiles

The flow velocity profiles from single span CFD simulations are compared directly with the experimental profiles in Figure 4-9 to Figure 4-12.

The comparison of the axial flow mean velocity profiles is shown in Figure 4-9, for traverse T1 on the left side and traverse T2 on the right side. In principle, a good qualitative agreement between CFD and experimental results can be observed. The shape of the velocity profiles is generally well reproduced at all positions, with smaller or larger local deviations. These deviations are particularly visible at positions close to guide tubes. The CFD results show a tendency to overestimate the experimental results at axial positions farther from the spacer grid. From the direct comparison of all profiles, it is hard to conclude which approach is closer to the experimental results. It is, however, evident that there is a clear difference between LES and DES results. The two LES results show similar tendency, which is generally different from the tendency that show all three DES results. One characteristic of the DES results is that these profiles show a wavier shape compared to both LES and experimental results.

The experimental profiles show more pronounced peaks to the bottom (reduced velocity) at positions close to the guide tubes. One would intuitively expect that these peaks appear exactly in the middle of the guide tube where the profiles are closest to the guide tube wall. This is, however, not always the case, and at some positions, like e.g. traverse T1 at $z = 50$ mm and $z = 250$ mm the peaks are clearly moved slightly to the left respectively to the right from the middle of the guide tube. The first assumption is that there was perhaps an uncertainty in the position of the measuring points during experiments. However, the DES results with Spalart-Almaras and SST models for traverse T1 at $z = 50$ mm predict correctly the position of this peak. On the other hand, the two LES results predict the peak location close to the middle of the guide tube with an asymmetric shape with a sharp slope on the right side. The

Elliptic Blending DES show a double peak at this location one corresponding well to the experimental and the other two DES results and the other one corresponding well to the two LES results. At the position $z = 250$ mm on traverse T1, all CFD results show a peak close to the middle of the guide tube, whereas the experimental result shows a slight displacement to the right. This demonstrates clearly the whole complexity of the flow field behind the spacer grid and the difficulty to predict it correctly. Some deviations from the experimental results are for sure due to uncertainties in the experimental results itself with respect to position of the experimental measurement points or/and of the measured velocity values, which cannot be exactly quantified for present case. Considering all these effects, a good agreement between experimental and simulation results can be concluded.

The comparison of the cross flow mean velocity profiles is shown in Figure 4-10, for traverse T1 on the left side and traverse T2 on the right side. Like for the axial flow mean velocity profiles, generally a reasonable qualitative agreement can be observed also in this case with local deviations especially at locations closer to the spacer grid. Again, a clear difference can be observed between LES and DES results. The tendency of DES to produce more wavy profiles is evident, the profile peaks are at many locations significantly overpredicted compared to LES results and partially also to experimental results. Poor predictions of experimental profiles can be observed at location $z = 10$ mm for both traverses, although the LES and DES show very similar results. On the other hand, the LES shows a very good predictions at location $z = 50$ mm.

Figure 4-11 shows the comparison of the axial flow RMS velocity profiles. In this case a relatively good agreement of LES with experimental results. On the other hand, DES results show systematically an underestimation of the experimental results especially at two locations immediately behind the spacer grid. One special observation can be made for profiles at $z = 50$ mm. At this location, the LES with WALE model shows significant peaks at the location of guide tubes for both T1 and T2. The experimental results, however, show a significant peak only at T2. It is difficult to judge whether this can be associated to experimental uncertainties, although this is the most likely the explanation. LES with Smagorinsky model show only a moderate peak at this location on both sides. Similar observations can be made also for the cross-flow RMS velocity profiles shown in Figure 4-12, where a relatively good agreement (within the range of accuracy) of LES with experimental results can be observed. DES results, on the other hand, show again underestimation of the experimental results.

Because of difficulties in the assessment of the results by direct comparison of the profiles an additional more quantitative comparison is performed by calculating directly the differences between simulation and experimental results as:

$$\Delta = \left(\sum_i^N |\phi_{i,\text{sim}} - \phi_{i,\text{exp}}| \right) / \left(\sum_i^N |\phi_{i,\text{exp}}| \right) \quad (4.7)$$

Where $\phi_{i,\text{sim}}$ and $\phi_{i,\text{exp}}$ represent the values of corresponding variables from simulations and experiments respectively at profile point i . The results are presented in Figure 4-11 and Figure 4-12 for mean and RMS velocity profiles respectively. For mean axial flow velocity similar results can be observed for LES and DES, while in the case of mean crossflow velocity tendentially larger deviations can be observed for DES results. In the case of RMS profiles, it is evident that the deviation is systematically larger for DES.

Based on all these results and analyses it can be concluded that LES predicts better the flow field compared to DES, especially in terms of flow fluctuations, in particular close to spacer grid. Regarding LES slight advantages can be derived for the WALE compared to Smagorinsky SGS model. This can be concluded directly from Figure 4-7 and Figure 4-8 where the deviations calculated by equation (4.7) for each profile are averaged further for all profiles (all traverses and axial positions) for axial and cross-flow.

Finally, the LES approach with WALE SGS model is selected for the full-scale model simulation.

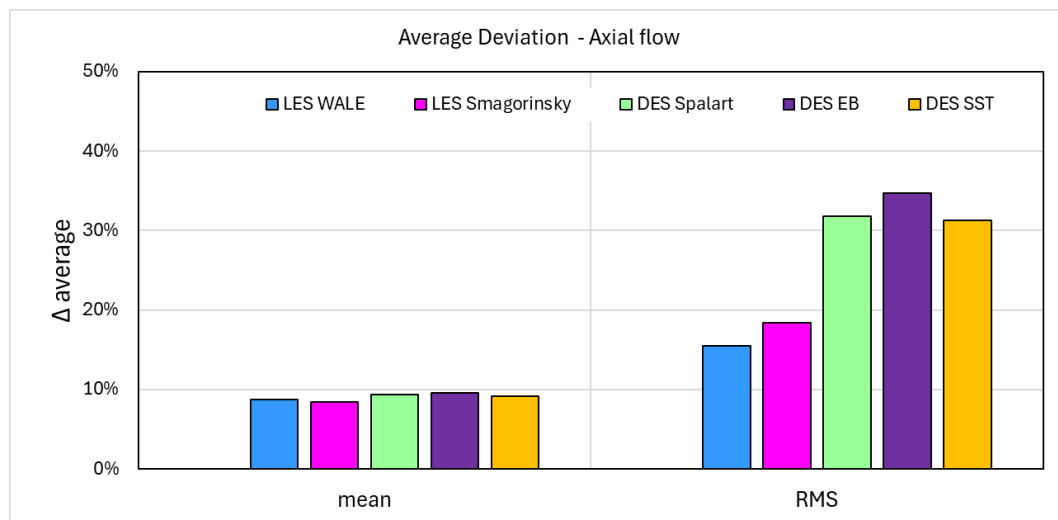


Figure 4-7: Single span flow profiles - average deviation axial flow.

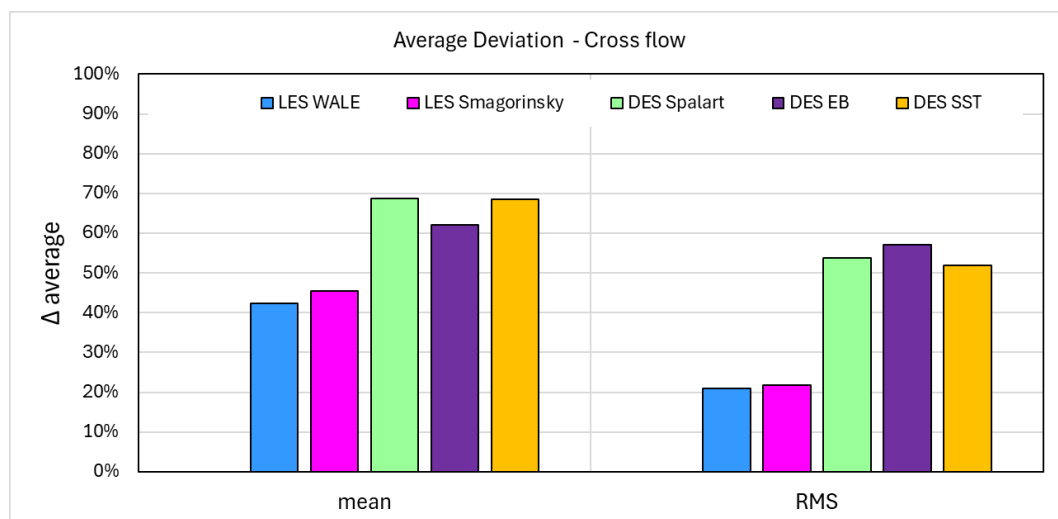


Figure 4-8: Single span flow profiles - average deviation cross flow.

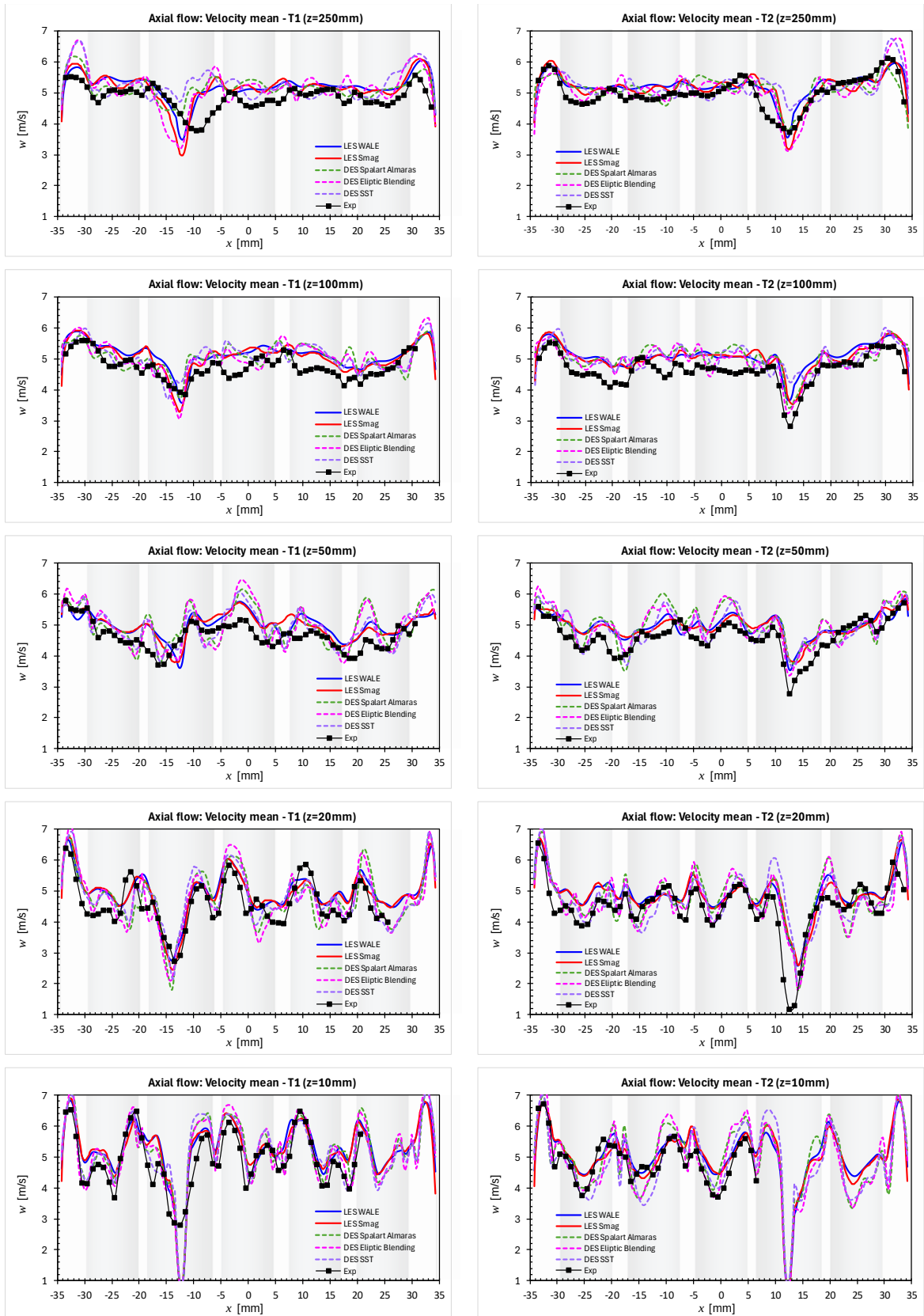


Figure 4-9: Single span axial flow mean velocity profiles along traverse T1 left and T2 right.

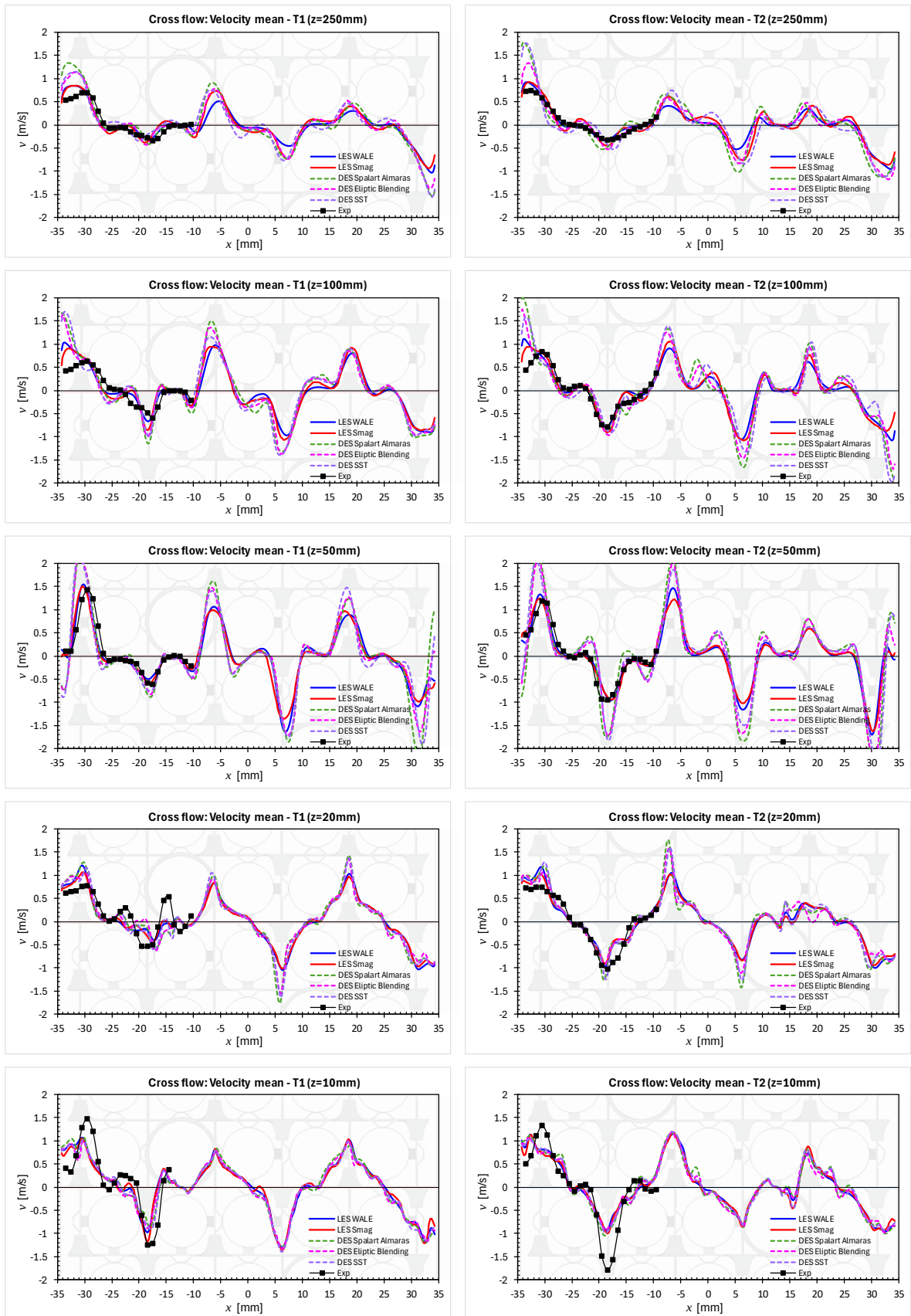


Figure 4-10: Single span cross flow mean velocity profiles along traverse T1 left and T2 right.

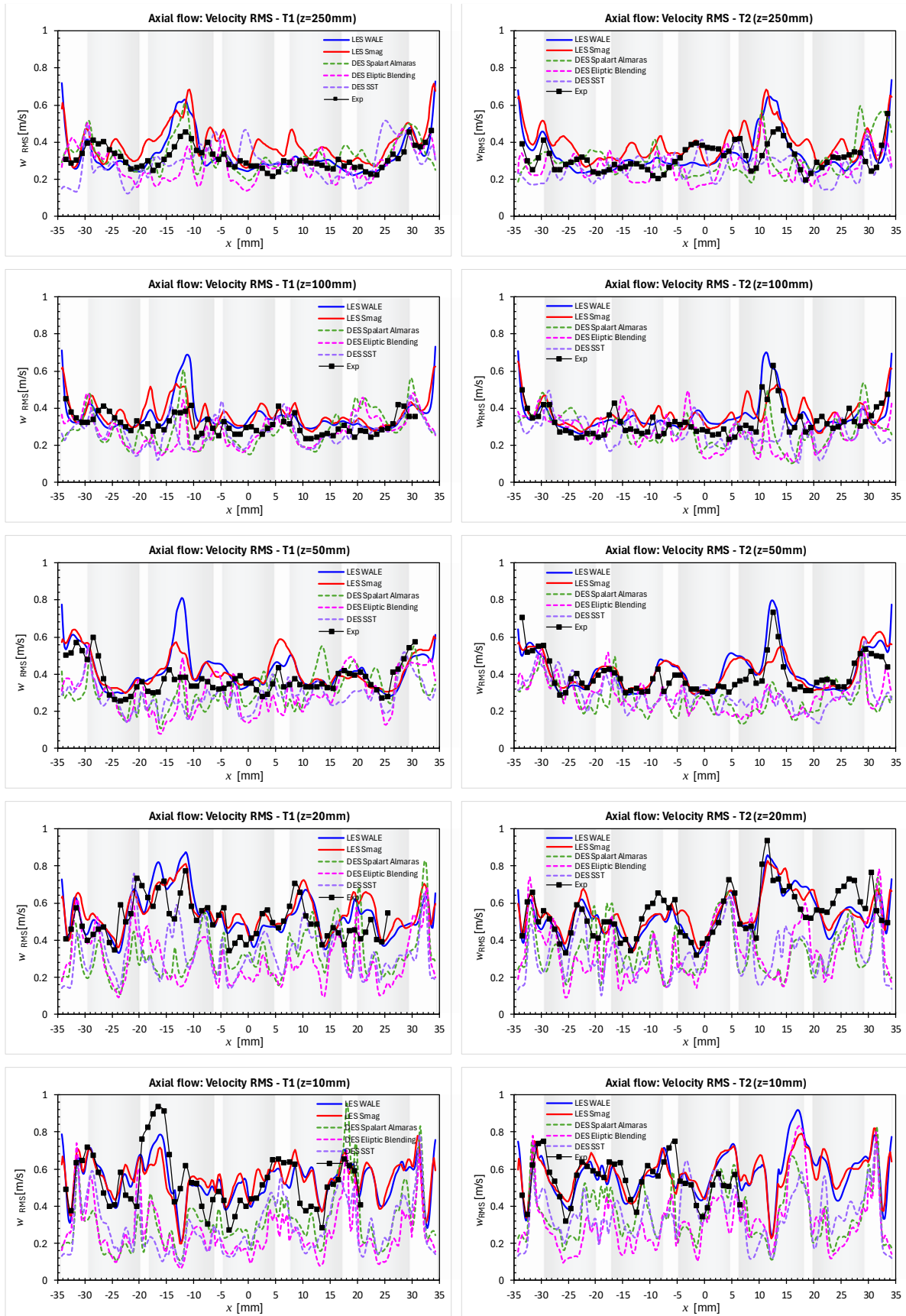


Figure 4-11: Single span axial flow velocity RMS profiles along traverse T1 left and T2 right.

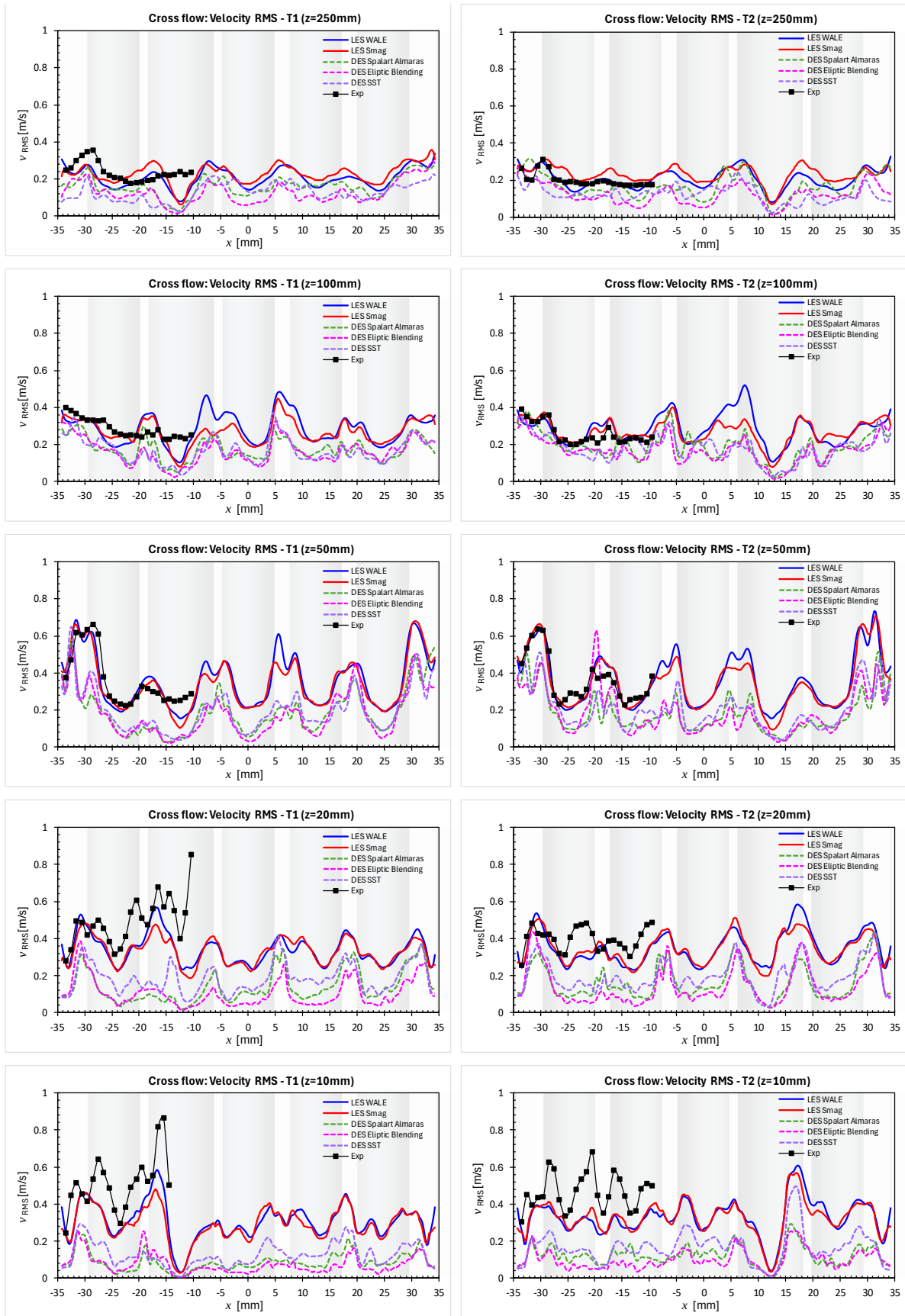


Figure 4-12: Single span cross flow velocity RMS profiles along traverse T1 left and T2 right.

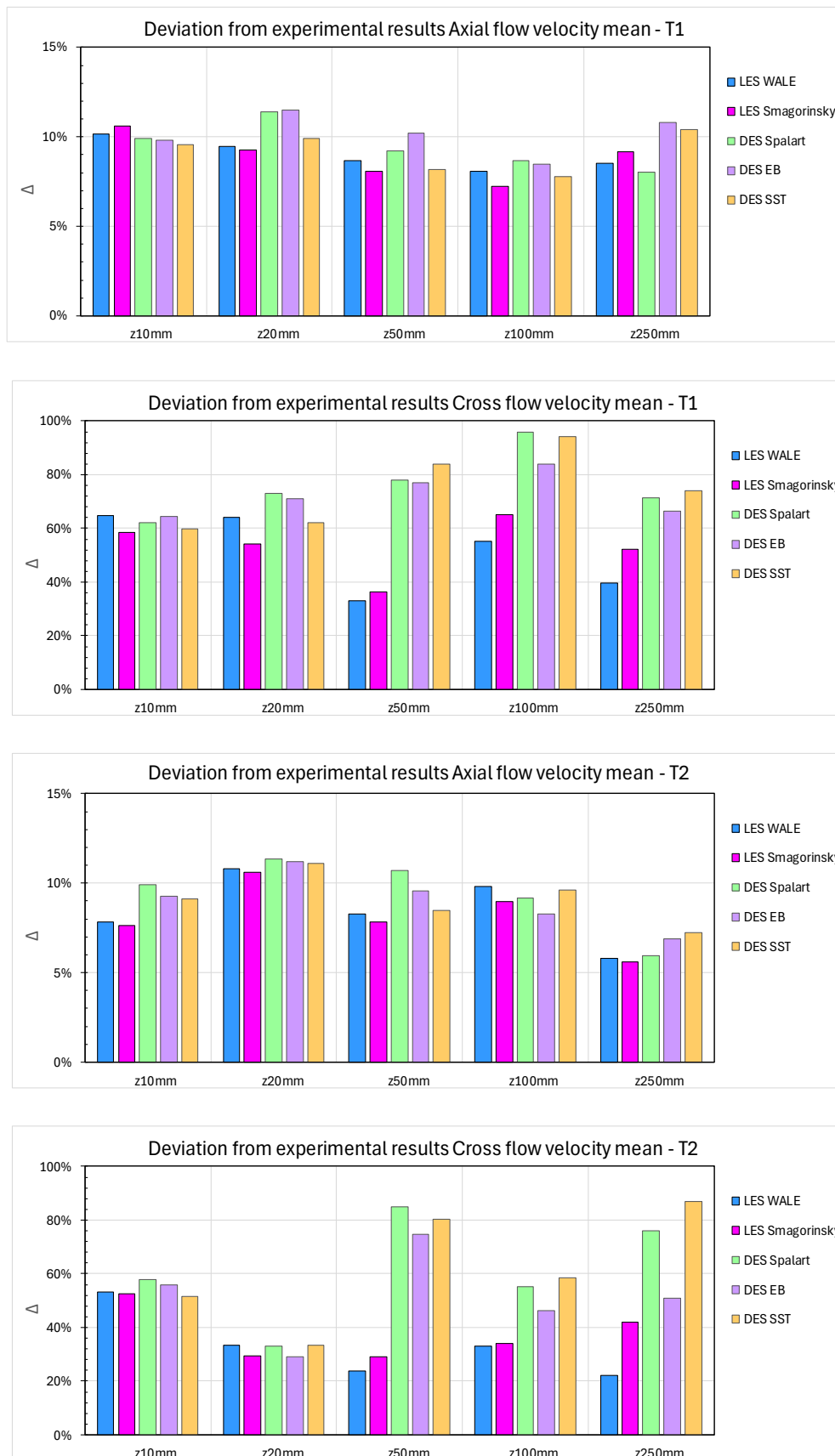


Figure 4-13: Single span mean flow profiles - quantitative comparison.



Figure 4-14: Single span RMS flow profiles - quantitative comparison.

4.2.7.2 Full-scale configuration

4.2.7.2.1 Flow field profiles

The direct comparison of the flow velocity profiles from full-scale model CFD simulations, with LES WALE model, against the experimental results is provided in Figure 4-17 to Figure 4-20. For comparison reasons the results from single-span CFD simulation with LES WALE model is also provided.

Generally, the behavior that was observed for the single-span CFD simulation results for LES WALE model can also be observed here. The performance of the LES approach with WALE model is approximately at the same level as already assessed from the single-span simulation. Nevertheless, some slight differences between LES results with WALE model obtained from single-span and full-scale configurations can be observed in the flow profiles.

Figure 4-15 and Figure 4-16 show an average deviation from the experimental results calculated for all profiles confirm this. LES results from the full-scale model show slightly lower deviation compared to all results from single span simulations.

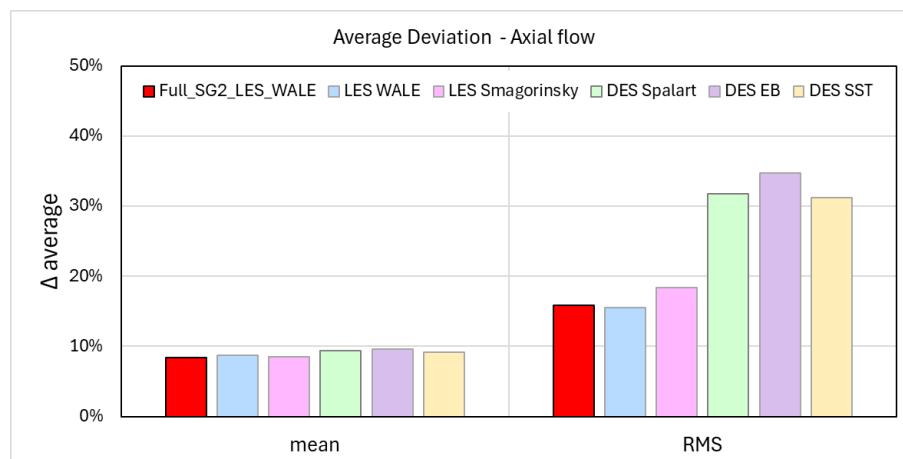


Figure 4-15: Flow profiles average deviation axial flow: full-scale vs. span model.

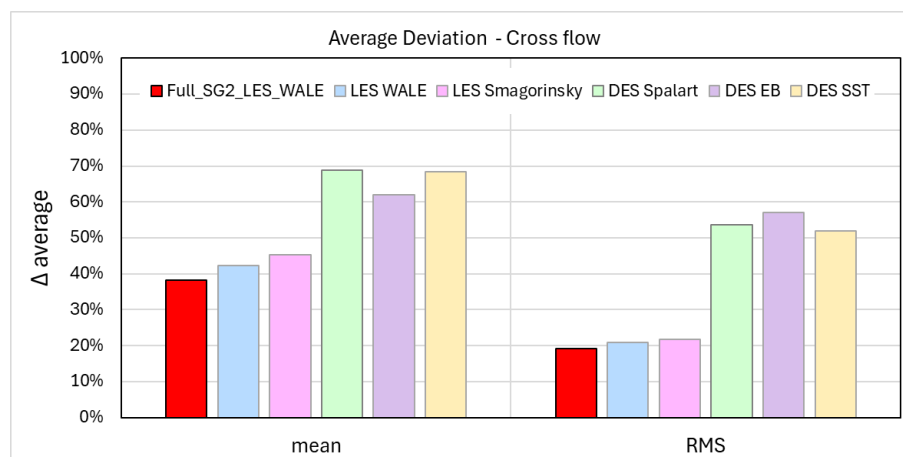


Figure 4-16: Flow profiles average deviation cross flow: full-scale vs. span model.

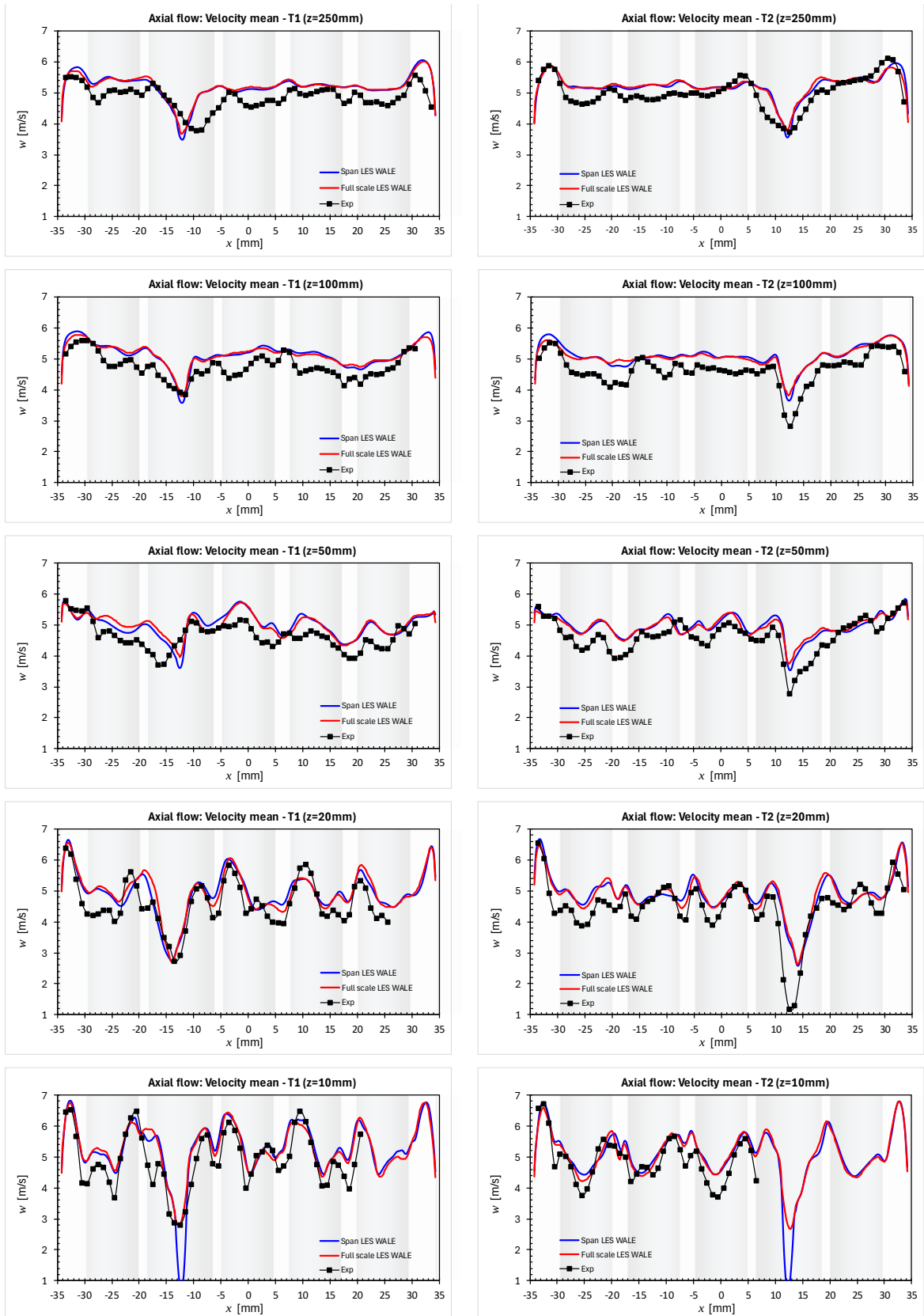


Figure 4-17: Full-scale model axial flow mean velocity profiles - traverse T1 and T2.

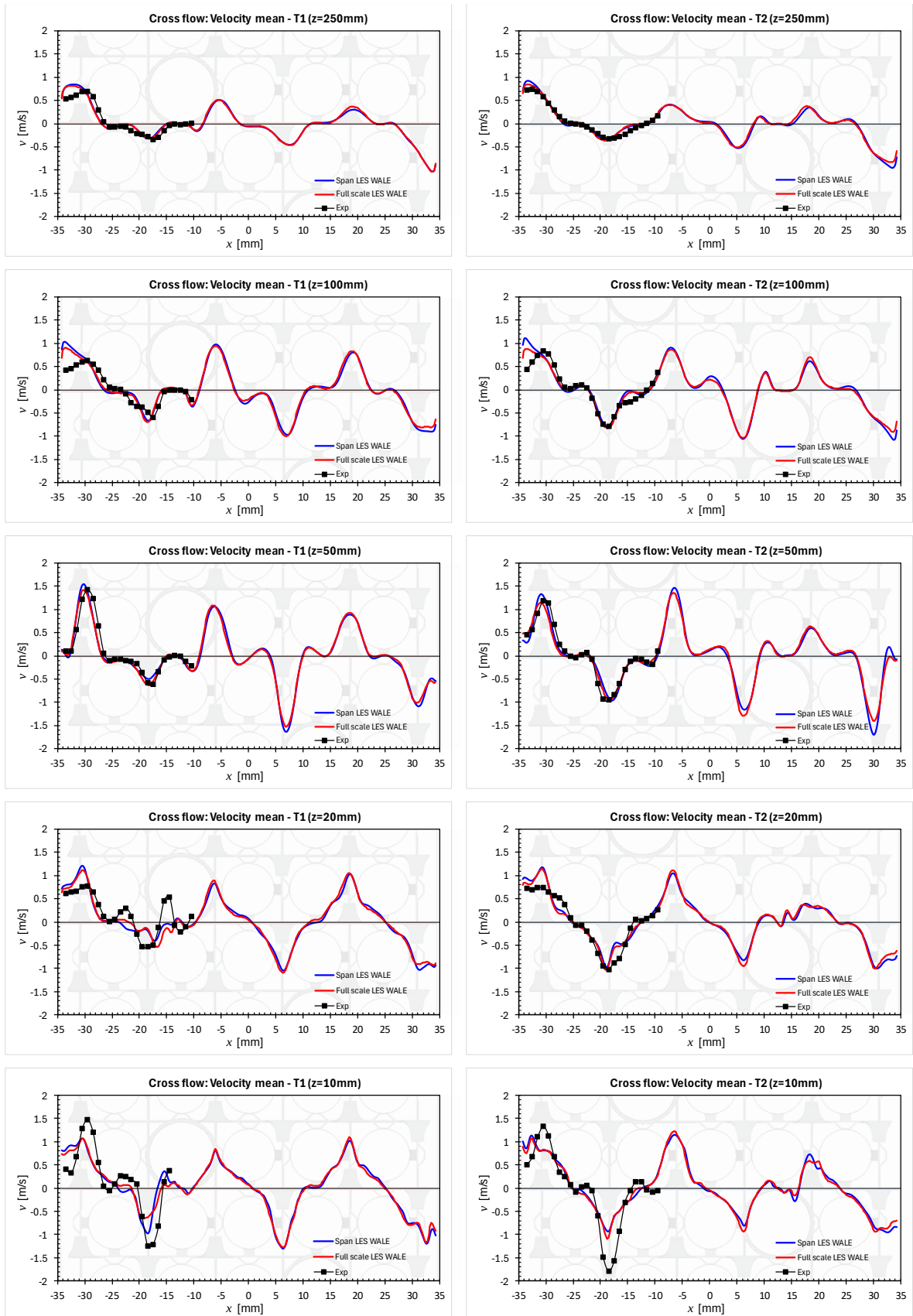


Figure 4-18: Full-scale model crossflow mean velocity profiles - traverse T1 and T2.

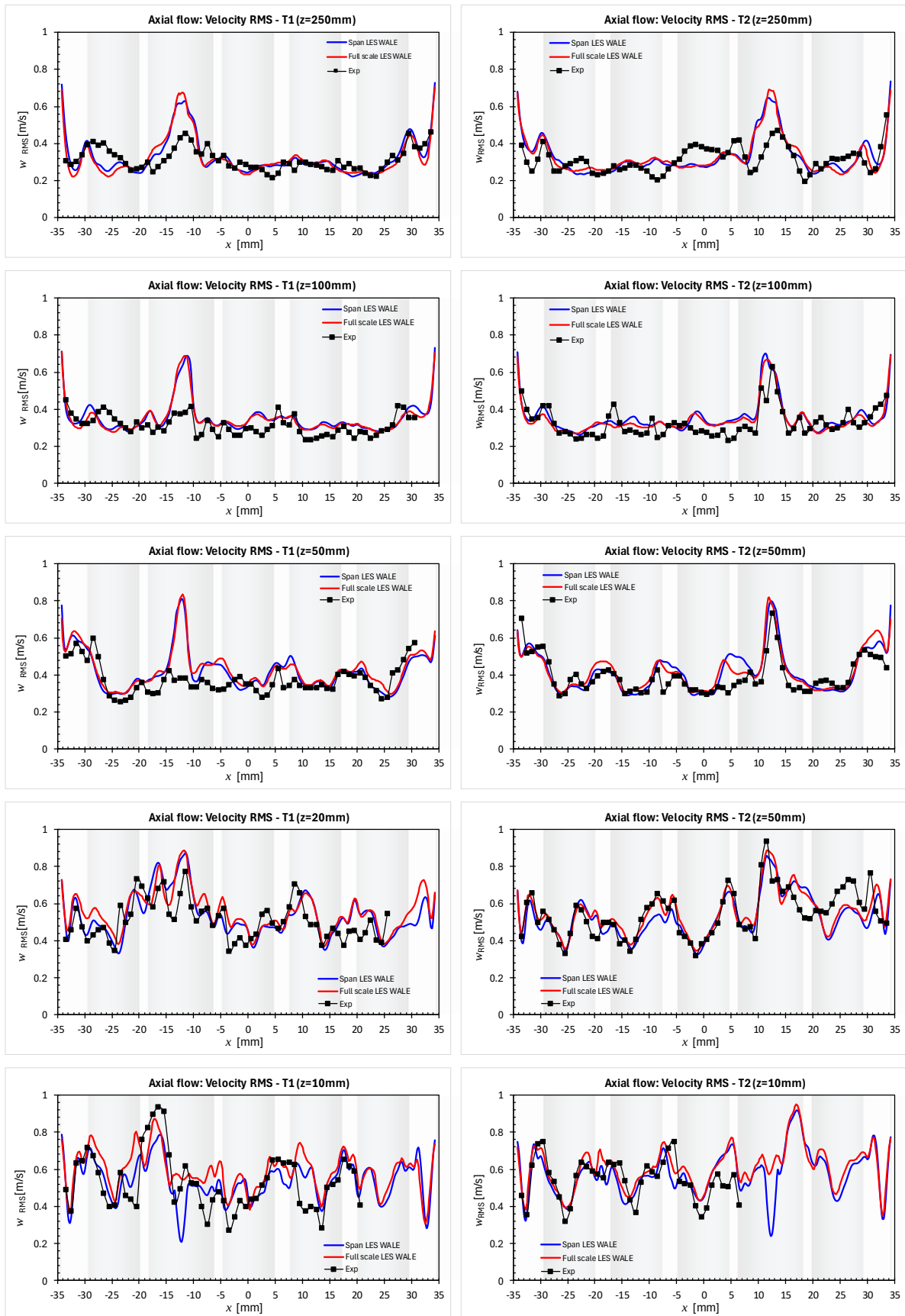


Figure 4-19: Full-scale model axial flow RMS velocity profiles - traverse T1 and T2.

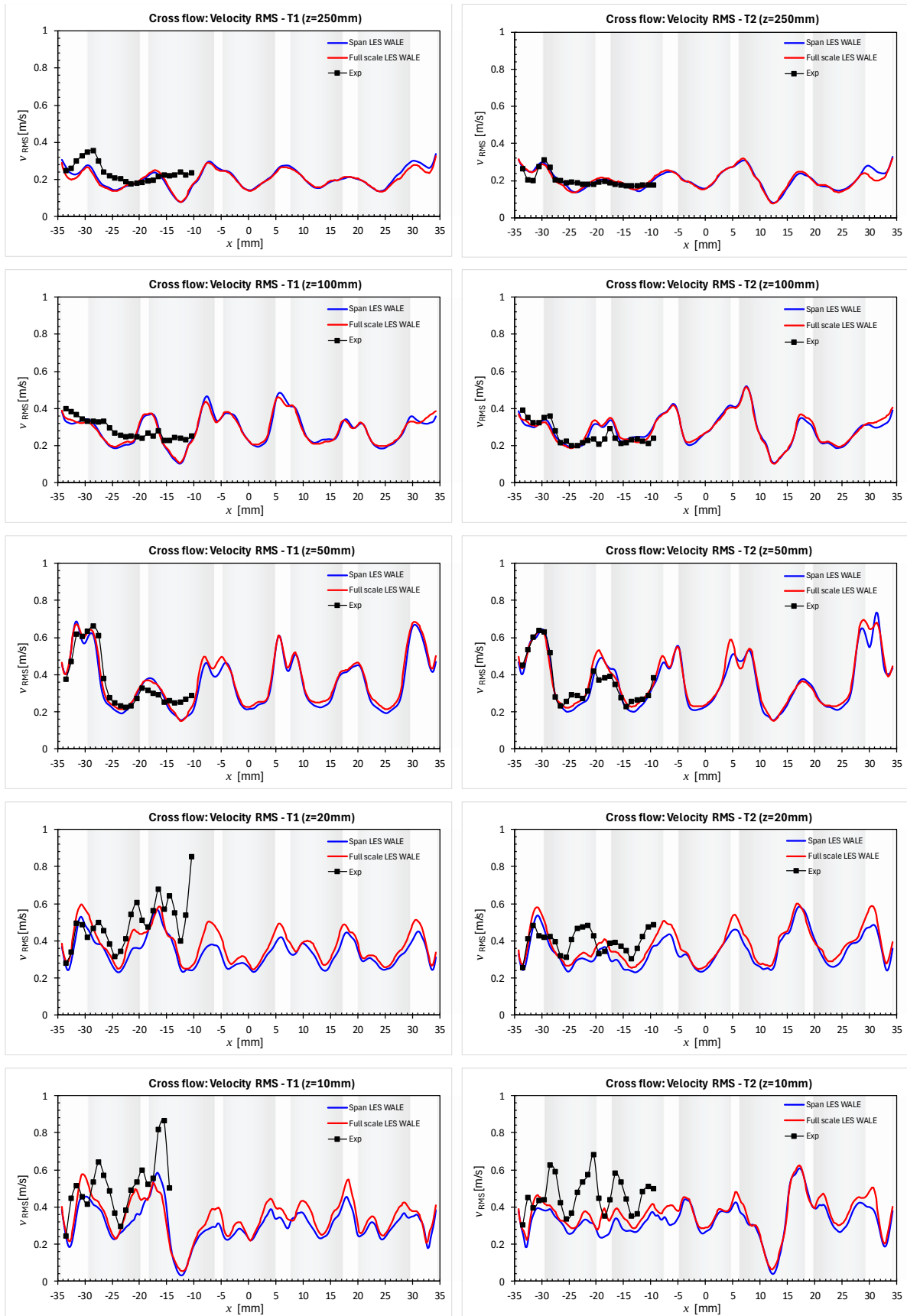


Figure 4-20: Full-scale model crossflow RMS velocity profiles - traverse T1 and T2.

4.2.7.3 Fluid forces along the fuel assembly

The fluid forces calculated by CFD are stored in corresponding force monitors during the simulation. The forces are integrated over selected surfaces distributed axially along the fuel assembly, which correspond to the calculation nodes on the mechanical model, according to procedure described in section 4.2.6. The forces are calculated for each axial segment containing for two lateral directions x and y in each time step and the data is stored over the time. The recorded time histories of fluid forces (F_x and F_y) are used in mechanical simulation. Examples of time histories for fluid forces acting on the fuel assembly at SG locations are shown in Figure 4-21. Axial division of the fuel assembly surface for force integration is indicated in the fuel assembly figure in the middle.

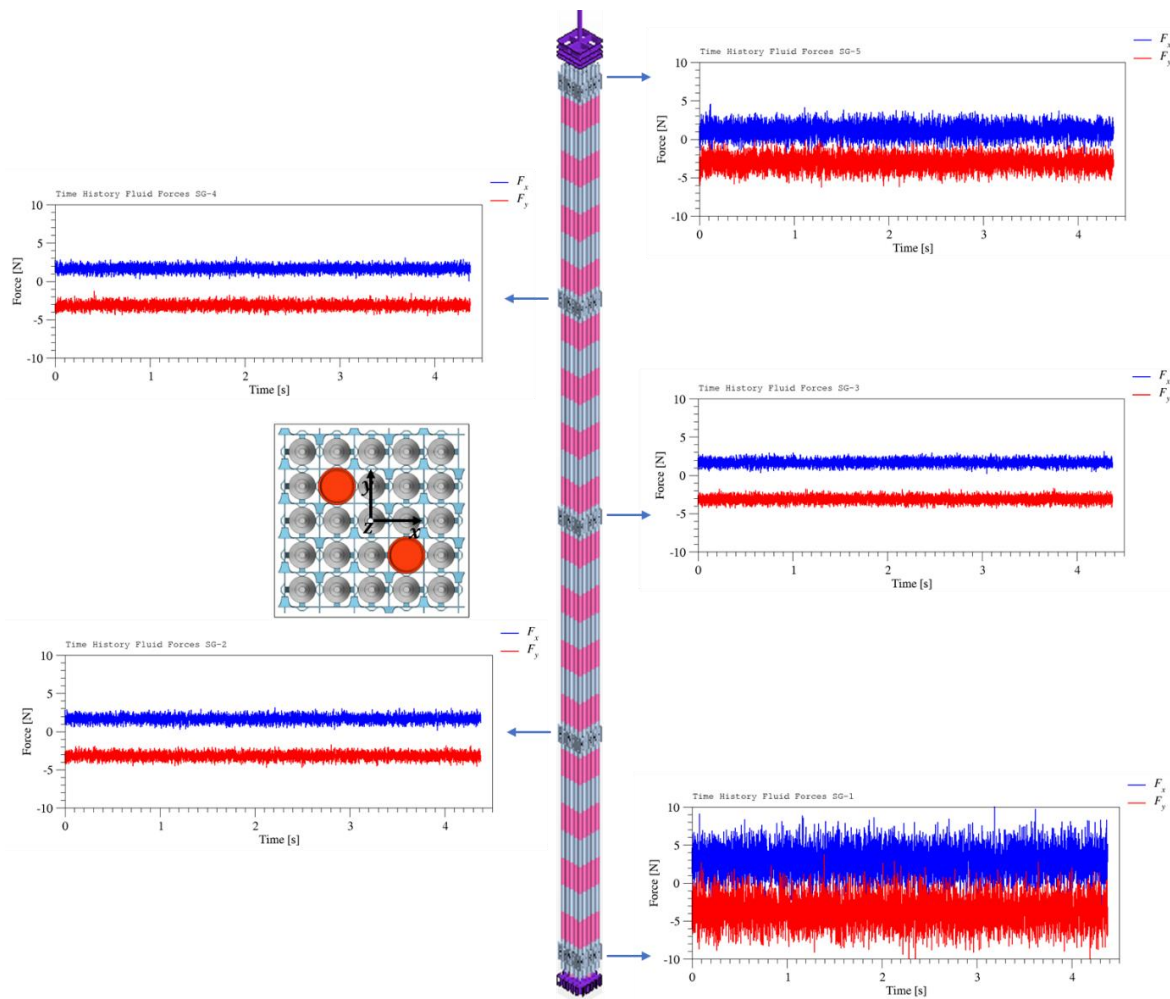


Figure 4-21: Time histories of the fluid forces acting on spacer grids.

To get a better insight in their characteristic several properties of the fluid forces were evaluated. From Figure 4-21 it is evident that the unsteady forces are oscillating around certain average value which is not zero. This average value represents the static fluid force component which is obtained from the corresponding time history as

$$\bar{F} = \frac{1}{T} \int_{t_0}^{t_0+T} F(t) dt = \frac{1}{N} \sum_{i=1}^N F_i \quad (4.8)$$

F_i is the fluid force at time step i , Δt is the time step, t_0 is the starting time point, T is the time interval over which the average value is calculated and N is the number of time step over the integration interval. The integration is performed within the time interval of the final simulation phase according to Figure 4-5.

The distribution of the average values (static component) of the lateral fluid forces along the fuel assembly is shown in Figure 4-22. It is evident that there are static force components acting on spacer grids in both directions. Static forces acting on the fuel assembly in the pure rod bundle area are very small in comparison to forces acting on spacer grids sections. Furthermore, the static force component in y direction, which is parallel to the shape of mixing vanes, is by factor 1.3 at bottom to 2.75 at top side larger than the force component in x direction, which is perpendicular to the shape of mixing vanes.

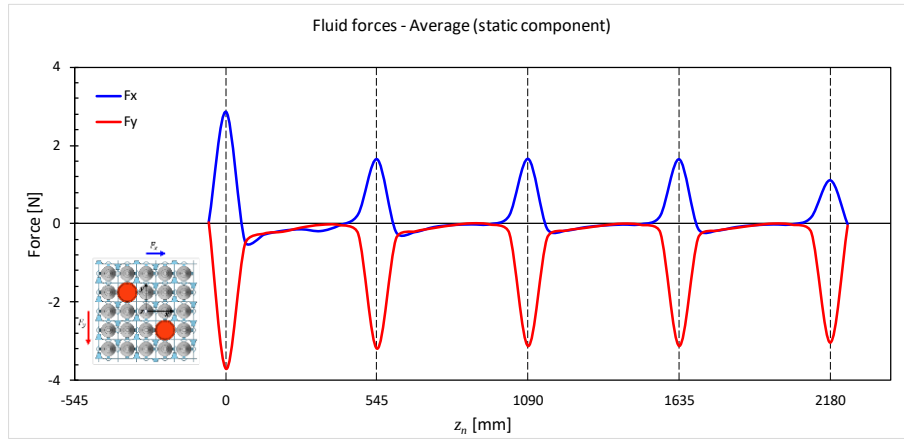


Figure 4-22: Fluid force profile along the fuel assembly – average (static components).

Another property of the fluid forces, which is particularly important for FIV is the dynamic force component (static component subtracted) which represents the fluctuation of the fluid force and is responsible for the excitation of the vibration. The level of fluctuations of the fluid forces is described by the RMS value of the dynamic fluid force component which is calculated from time histories as

$$F_{RMS} = \sqrt{\frac{1}{T} \int_{t_0}^{t_0+T} (F(t) - \bar{F})^2 dt} = \sqrt{\frac{1}{N} \sum_{i=1}^N (F_i - \bar{F})^2} \quad (4.9)$$

The distribution of the RMS values of the dynamic component of the lateral fluid forces along the fuel assembly is shown in Figure 4-23.

From the distribution of RMS values, it is evident that the dynamic component of the fluid forces is highest at the spacer grid locations with a generally decreasing trend within a span in downstream direction towards next spacer grid. After significant drop immediately behind the spacer grid, the further decay of the RMS values along a span is much slower. Furthermore, the dynamic fluid force is highest at the first spacer grid, while they are at comparable level

for the spacer grids 2 to 4 and again at a higher level at spacer grid position 5. This is associated with the increased incoming turbulence level and stronger flow disturbances before the first spacer grid generated by the lower tie plate. Similarly, the slightly higher force level at and behind the 5th spacer grid is also due to flow disturbances downstream the last space grid, i.e. in the space between rod bundle and the upper tie plate. Unlike the static force component for which significant differences in two lateral directions exist at all spacer grid positions, the deviations in the amplitude of dynamic force components are negligible.

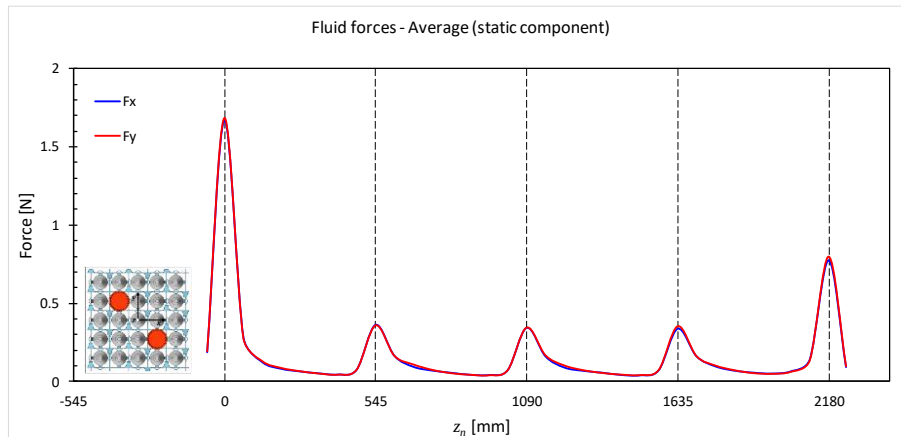


Figure 4-23: Fluid force profile along the fuel assembly – RMS (dynamic components).

The next property that describes the character of fluid forces is the power spectral density (PSD) that shows the intensity of the dynamic fluid forces as a function of frequency. The PSD provides important information about the spectral characteristic of the flow-induced excitation. Examples of the PSD for the fluid forces acting on the spacer grids are provided in Figure 4-24. The plots are shown in the log-log scale. The PSD plots show a rather flat behavior in the lower frequency range up to 100 Hz. With increasing frequency (higher than 100 Hz) the PSD curves show a slope which is nearly parallel to the slope $-5/3$ (dashed black line) which is characteristic for the slope of turbulence spectrum. At the frequency around 1000 Hz there is a slight peak in the PSD. It is, however, not clear if this peak can be associated with any kind of periodic force appearing at this frequency. At frequencies higher than 1000 Hz a rapid decay can be observed which is an indication of the limit of the turbulent flow structures resolved by the simulation given by the cell size of the CFD mesh.

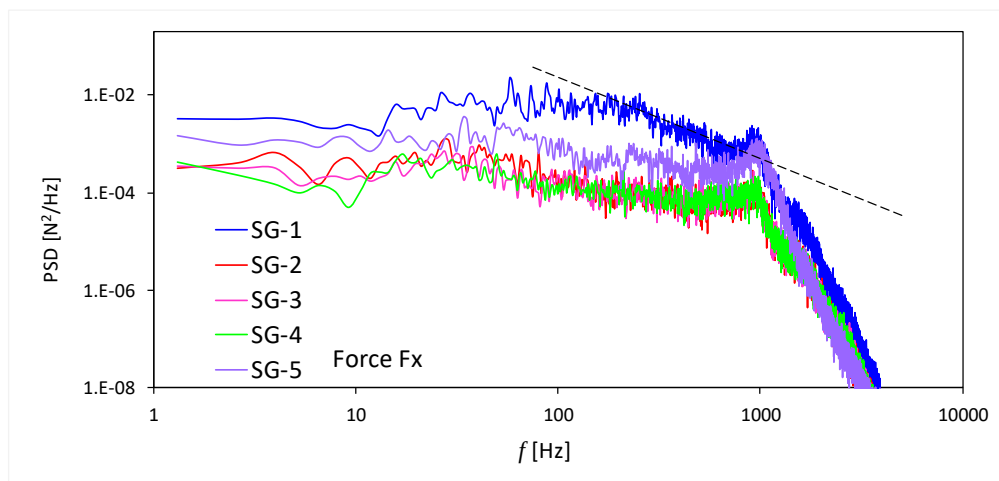


Figure 4-24: PSD of fluid forces in lateral direction x acting on fuel assembly at SG positions.

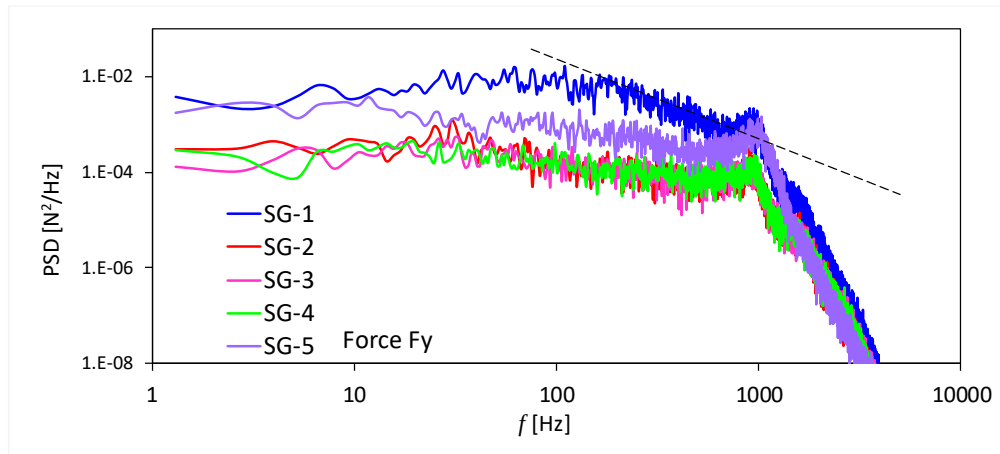


Figure 4-25: PSD of fluid forces in lateral direction y acting on fuel assembly at SG positions.

4.3 Mechanical model

In this section the mechanical model of the test fuel assembly and associated structural dynamic simulations to evaluate FIV response are described.

4.3.1 Description of the Mechanical model

Due to their specific slender structure, fuel assemblies are typically represented by beam elements in structural models. This kind of representation is usually sufficient to describe properly the dynamic behavior of fuel assemblies, which is, in the present case, the lateral vibration. In this way, the computational effort required for structural dynamic analysis can be significantly reduced and available computational resources can be efficiently used for fluid flow calculation.

In the present studies the structure of the fuel assembly is represented by the finite element mechanical model illustrated in Figure 4-26 a). The model consists of 35 nodes which are connected by linear Euler beam elements. At the spacer grid locations linear rotational springs are added to consider the rotational stiffness resulting from the connection of the spacer grids with the guide tubes and fuel rods. The fuel assembly is fixed at the bottom and at the top side in the lower and upper tie plate, respectively, which corresponds to the nodes 1 and 35 in the model. The clamping boundary conditions for the mechanical model are applied assuming zero displacements and rotations for these end nodes. The mass of the fuel rods is assumed to be homogeneously distributed over the whole fuel assembly length. The mass of spacer grids is added to the corresponding nodes at SG locations (nodes 2, 10, 18, 26 and 34). The mass and inertia are distributed over computational nodes according to lumped mass approach resulting in a diagonal mass matrix.

The main input data for the fuel assembly mechanical model are provided in deliverable D2.3 document [1] i.e. in chapter 3 of this report. The masses of the structural parts are calculated based on their volume and the density of corresponding materials based on data provided in Table 3-1 and Table 3-2.

The bending stiffness (EI) and the stiffness of the linear rotational spring elements (c_r) at spacer grid locations are adjusted to match the natural frequencies of the first three vibration modes listed in Table 3-4, which are derived from the dynamic response measured during the

mechanical impact tests described in [1]. This is illustrated in Figure 4-26 b) where the vibration response of the fuel assembly simulating the impact tests and the resulting frequency spectra are shown.

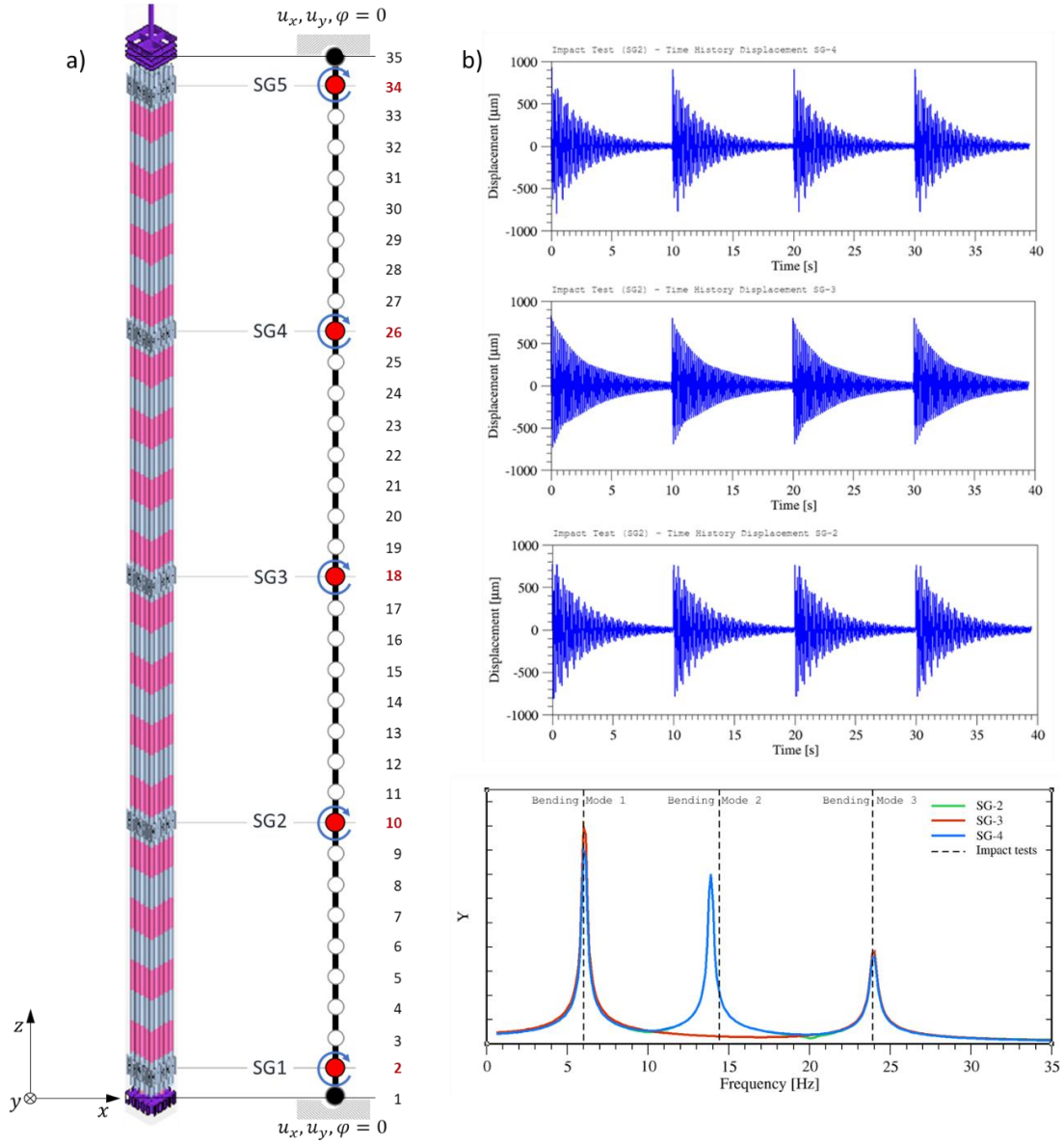


Figure 4-26: Mechanical model overview.

After applying the finite element discretization, the following equation of motion for the beam displacement and rotation, vector $\mathbf{u}_i$ is obtained

$$\mathbf{M} \cdot \ddot{\mathbf{u}}_i + \mathbf{C} \cdot \dot{\mathbf{u}}_i + \mathbf{K} \cdot \mathbf{u}_i = \mathbf{f}_{\text{ex}} \quad , \quad (4.10)$$

where index i represents lateral directions x and y according to coordinate system used in Figure 4-26 a). $\mathbf{M}$, $\mathbf{C}$ and $\mathbf{K}$ denote the mass, damping and stiffness matrices of the finite element model. The parameter $\mathbf{f}_{\text{ex}}$ denotes the vector of external forces and moments. In the case of FIV it represents the turbulence-induced fluid forces $\mathbf{f}_{\text{fl}}$ acting on the fuel assembly. The damping matrix $\mathbf{C}$ is represented as a linear combination of mass and stiffness matrices according to Rayleigh damping concept:

$$\mathbf{C} = \alpha_D \mathbf{K} + 2\beta_D \mathbf{M} \quad (4.11)$$

where α_D and β_D are the Rayleigh damping coefficients. Damping parameters α_D and β_D are related to the damping factor ζ_k of the vibration mode k with frequency ω_k by following equation:

$$\zeta_k = \frac{\beta_D}{\omega_k} + \frac{\alpha_D \omega_k}{2} \quad (4.12)$$

The vibrational response of the fuel assembly is obtained by integrating the equation of motion (4.10) in time using implicit Newmark- β method for time integration [31].

The basis of the Newmark- β method is the assumption that the system displacement vector $\mathbf{u}_{i,n+1}$ and its derivative $\dot{\mathbf{u}}_{i,n+1}$ at the new time step $n + 1$ can be written in the form:

$$\begin{aligned} \dot{\mathbf{u}}_{i,n+1} &= \dot{\mathbf{u}}_{i,n} + \left[\frac{(1-\gamma)\ddot{\mathbf{u}}_{i,n} + \gamma\ddot{\mathbf{u}}_{i,n+1}}{\text{weighted average}} \right] \Delta t \\ \mathbf{u}_{i,n+1} &= \mathbf{u}_{i,n} + \dot{\mathbf{u}}_{i,n} \cdot \Delta t + \left[\frac{(1-2\beta)\ddot{\mathbf{u}}_{i,n} + 2\beta\ddot{\mathbf{u}}_{i,n+1}}{\text{weighted average}} \right] \frac{\Delta t^2}{2} \end{aligned} \quad (4.13)$$

Where γ and β are weighting constants with the values $\gamma = 0.5$ and $\beta = 0.25$.

4.3.2 Effects of fluid on dynamic behavior

The surrounding fluid may have a significant effect on the dynamic behavior of the structures immersed in the fluid. This effect must be considered properly to evaluate correctly the dynamic response of the structure immersed in fluid [32]. The two main parameters which need to be considered are added (hydrodynamic) mass and fluid damping. The effect of added mass is characterized by a reduction of the frequency. The fluid damping increases the overall damping and thus influences the vibration amplitude under given external load. In coupled FSI simulations these effects are included per definition. In decoupled simulations, the fluid effects must be added explicitly in the mechanical model.

To estimate the added mass and fluid damping for given configuration consisting of a 5x5 fuel assembly placed in a channel and subjected to axial flow, coupled FSI simulations have been performed with a small model representing one span of the fuel assembly placed in the channel shown in Figure 4-27 and subjected to axial flow. In this case, the realizable $k - \varepsilon$ RANS turbulence model is used. RANS models were proven to be sufficient for accurate evaluation of the added mass and fluid damping effects (see [33]). The rod bundle is considered as a rigid body attached on a spring allowing its vibration in one lateral direction (x). The mass m of the structure considered in this small model is calculated according to the mass of the fuel assembly span, while the stiffness c of the spring is adjusted according to frequency f_0 which needs to be matched.

Geometrical symmetry of the configuration is utilized so that only a half of the fuel assembly cross-section is considered in this simplified CFD-FSI model. In axial direction periodic boundary conditions are applied with prescribed mass flow rate corresponding to the considered average axial velocity. At the beginning of the simulation, the span structure is displaced from the central position for a small displacement Δx and then released to exhibit free damped

vibration. The corresponding dynamic properties are derived from the time history of the free vibration as illustrated in Figure 4-27 right.

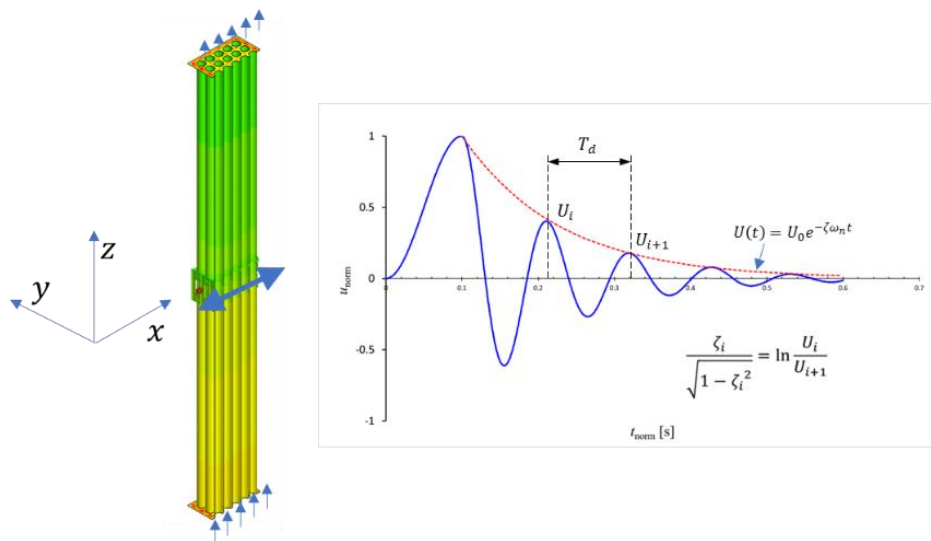


Figure 4-27: Simplified single-span FSI model for evaluation of fluid dynamic parameters.

Simulations were performed for different flow rates (axial velocities), different frequencies and different initial displacements. The corresponding results for added mass coefficient, which represents the ratio between the effective added mass and the fluid displaced mass corresponding to the volume occupied by the structure, obtained in stagnant water are shown in Figure 4-28. A value of 2.6 for the added mass coefficient is derived from the simulations results. The corresponding results for fluid damping as function of axial velocity obtained and for different frequencies are shown in Figure 4-29. The fluid damping shows strong dependency on both flow velocity and vibration frequency. With increasing axial velocity, the fluid damping increases. On the other hand, with increasing vibration frequency the fluid damping decreases.

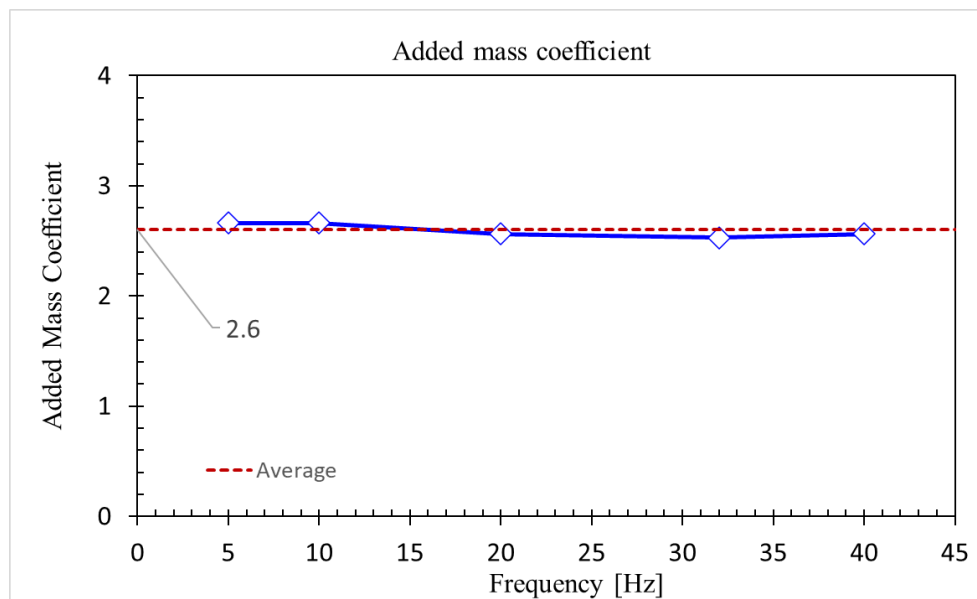


Figure 4-28: Added mass of the single span in stagnation water as function of frequency.

The results correspond to initial displacement of 0.5 mm. Simulations performed with different initial displacements did not show any substantial difference indicating that the results are independent of the initial displacement, which is consistent with conclusions in [34].

The results obtained from coupled FSI simulations with a simplified model of a fuel assembly span are implemented in the fuel assembly mechanical model. Added mass is considered as additional lumped mass at each calculation node. The added mass is considered by increasing the structural mass of each node by 29%, which corresponds to the effective added mass derived from the FSI simulations with the simplified span model.

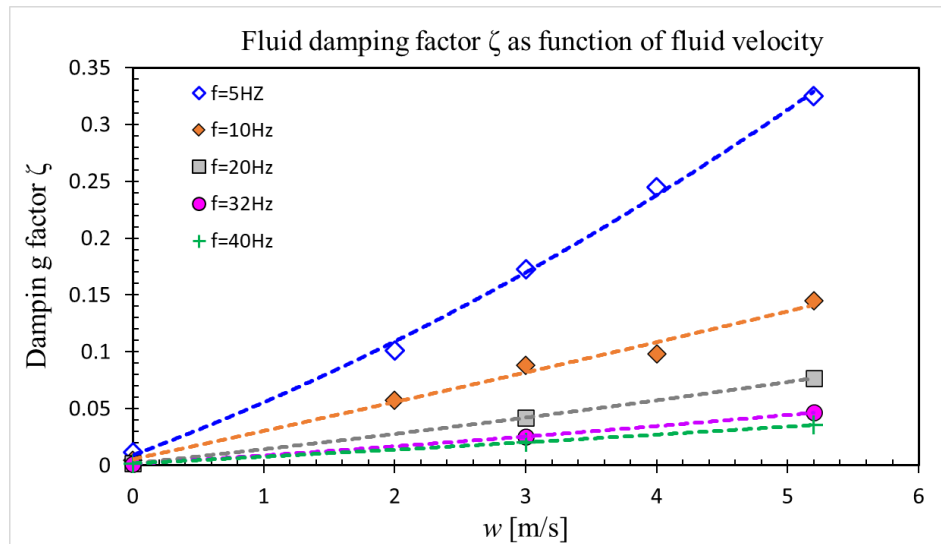


Figure 4-29: Fluid damping as function of frequency and axial velocity (right).

Fluid damping is integrated in the Rayleigh damping matrix defined by equation (4.11). The corresponding α and β parameters are calculated based on the assumption of 37% critical damping for the frequency of 4 Hz and 3.3% critical damping for the frequency of 45 Hz. The effective damping, as function of frequency, is illustrated in Figure 4-30.

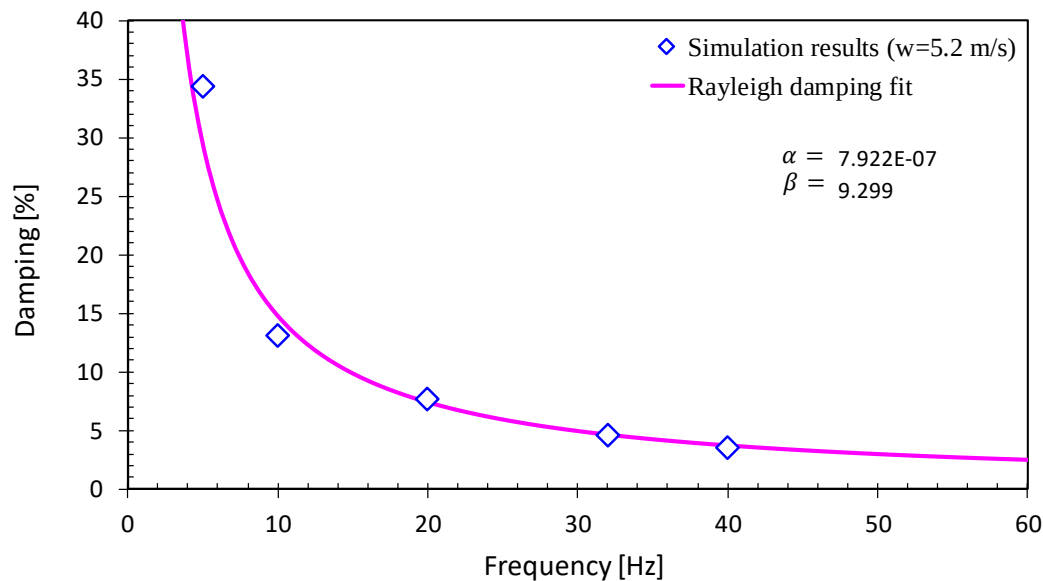


Figure 4-30: Rayleigh damping parameters implemented in mechanical model based on the fluid damping evaluation using FSI simulation for single span.

4.4 FIV simulations

Based on the parameters of the mechanical model described above, that consider structural properties and fluid effects, as well as on the fluid forces calculated by CFD simulations, corresponding structural dynamic simulations are performed to calculate the vibration response of the fuel assembly to given flow-induced excitation. The dynamic component (static component subtracted) of the fluid forces calculated by CFD is applied directly as external load in the dynamic equation of motion (4.10). Although the static force component would result in a static displacement of the fuel assembly, it would not affect the dynamic response which is of the main interest. Furthermore, since in the present model the fluid structure interaction is not included, the resulting static displacement would be overestimated.

The results of the FIV simulation with the structural dynamic model are shown in Table 4-2 and in Figure 4-31 in terms of vibration amplitude (RMS) and PSD of the lateral displacements respectively.

From the RMS values it is evident that the vibration amplitude obtained by simulation is in the range around 10 μ m which is in the same range as the experimental results. Nevertheless, the experimental results are generally overestimated by 10% to 85% relative to experimental results, with the highest deviation for displacement u_x at the spacer grid position 3. The only exception where the vibration amplitude from the experiments is underestimated is the displacement u_y at spacer grid position 4.

Table 4-2: Fuel assembly vibration response – RMS vibration amplitude.

	Lateral displacement RMS amplitude [μ m]				Δ	
	Experimental results		Simulation results		Simulation / Experimental	
Position	u_x	u_y	u_x	u_y	u_x	u_y
SG-2	5.25	4.67	8.35	6.55	59.0%	40.2%
SG-3	5.96	7.44 (R)	10.99	8.23	84.3%	10.7%
		5.16 (L)				59.5%
SG-4	6.42	9.14	8.24	6.88	28.4%	-24.8%

Contrary to simulation, where the RMS vibration amplitude is consistent with the 1st bending mode for both lateral displacements (highest displacement in the middle), in the experimental results the highest vibration amplitude appears at the spacer grid position 4 for the displacement u_y . It is not clear what the reason for this behavior is. One possibility may be the mechanical boundary conditions for the fuel assembly such as softer clamping in the upper tie plate. This, however, was not observed in the impact tests. The results of impact tests suggest symmetric boundary conditions i.e. full clamping at both sides. Another possible reason may be the flow conditions. In the present analysis the fluid flow is calculated based on the perfectly straight, non-deforming geometry. For vibration response, only the dynamic component of the resulting fluid forces is considered. However, the static component in the fluid forces may produce an elastic bow of the fuel assembly and thus affect flow conditions. At the end this may lead to specific FSI effects which cannot be captured in the frame of a decoupled simulation.

One reason for overestimating the vibration amplitude may be the uncertainty in the fluid effects (added mass, fluid damping and possibly also fluid stiffening which was not considered at all). These are evaluated from a simplified model representing one span of the fuel assembly which moves in flowing fluid as rigid body. This simple model might not be enough to estimate all relevant effects with sufficient accuracy and a refined analysis that considers the full fuel assembly may be more appropriate. Another potential reason for the overestimation of the vibration amplitude may be the dynamic (turbulence-induced) fluid forces which are used as external excitation. The test simulations have shown that the selected modeling approach is capable to predict reasonably the flow fluctuations. Although quite good agreement with the LDA measurements is achieved, there is still no evidence how good the dynamic forces are reproduced by the CFD simulation. Since there are no force measurements available that can be used for direct validation, a refined analysis is needed to estimate the uncertainties in the fluid forces calculated with WM LES approach used in the present studies. It is suggested/recommended to perform CFD simulations on a small-scale model (e.g. one span) with systematic mesh refinement for this assessment.

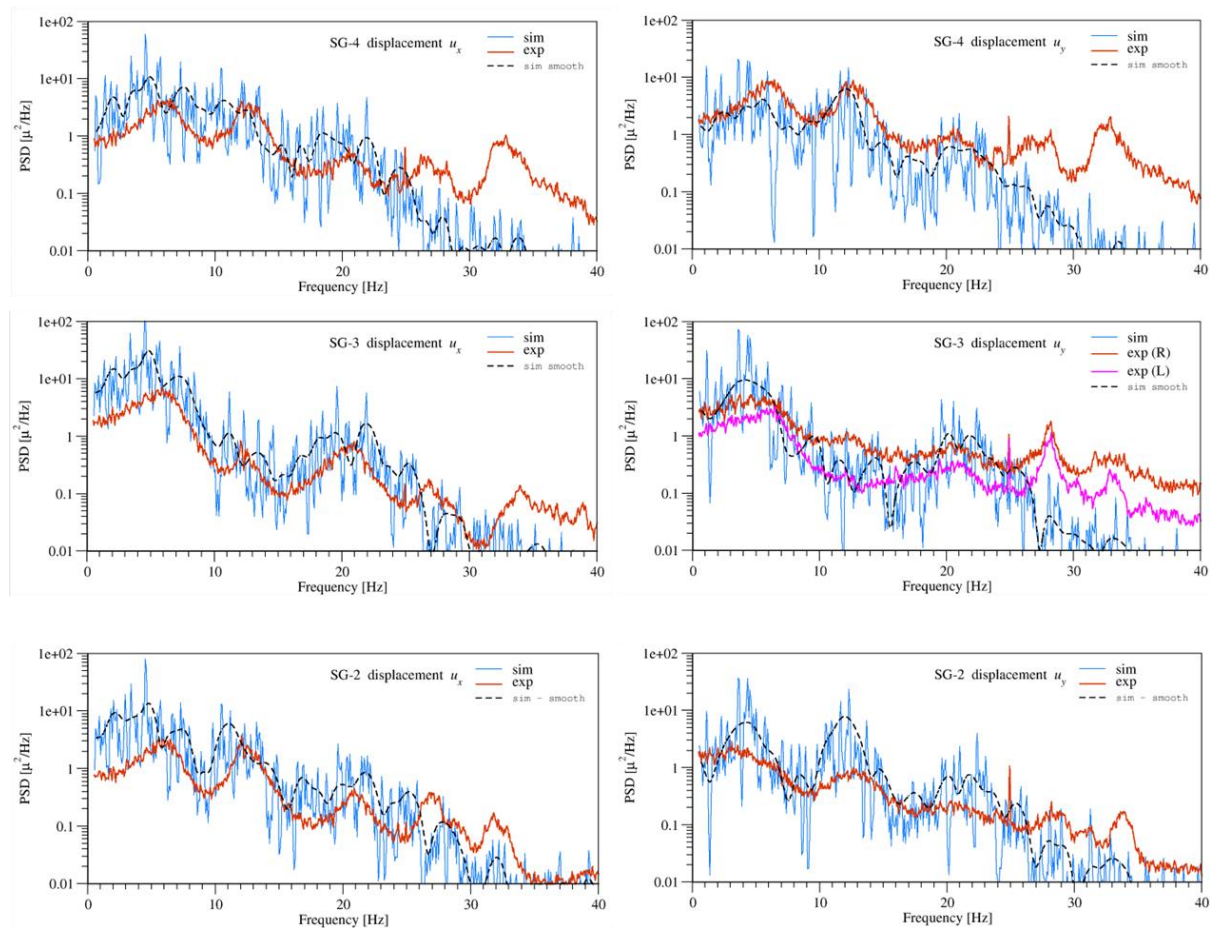


Figure 4-31: Fuel assembly vibration response PSD for lateral displacements u_x and u_y at spacer grids 2, 3 and 4 compared with experimental results.

Another interesting observation is that the vibration amplitude obtained by the simulation is different between two lateral directions u_x and u_y . Since the mechanical model has the same properties in both lateral directions, the only possible remaining cause for this difference are the fluid forces. However, from the analysis of the fluid forces presented in section 4.2.7.3 there is no evidence of any significant difference neither in the RMS values nor in the PSD. The

differences in static components do not play a role for present simulation. To understand this, a more detailed analysis of the time histories of the fluid forces needs to be performed.

From the PSD plots one can see that the vibration behavior is reproduced qualitatively in terms of frequency content. The dominant frequencies in the vibration response are captured well at all three spacer grid positions. All significant peaks in the PSDs from simulation are close to the relevant peaks from experimental results.

4.5 Summary and conclusions

In this Chapter, the results of the numerical simulations of the ALAIN Benchmark with a reduced 5x5 PWR fuel assembly performed by Framatome are presented. From the vibration response measured in the experiments, which is characterized by very small vibration amplitude in range of a few μm RMS, turbulence is assumed as the FIV mechanism. According to this, the FIV analysis is performed in the frame of a decoupled simulation approach. The whole analysis consists of two steps. First, unsteady CFD simulation is performed to calculate turbulence-induced fluid forces, and in the second step the vibration response of the fuel assembly is calculated by applying the fluid forces as external excitation.

For CFD simulations the commercial CFD code STAR-CCM+ is used. Since turbulence is assumed as the driving mechanism it is essential that turbulent fluctuations are resolved properly in the CFD simulation. Therefore, special attention is paid to the selection of an appropriate turbulence modeling approach and turbulence model for CFD simulation. Two turbulence modeling approaches capable of resolving turbulent flow structures, LES and DES, were tested on a small-scale model representing one span of the fuel assembly. These preliminary simulations revealed that LES predicts better the flow field profiles especially the flow fluctuations measured in experiments. Therefore, the LES approach in combination with WALE SGS model was selected for the final simulation with the full-scale geometry of the experimental setup. In all CFD simulations the wall modeling approach was used as the only feasible option for the given configuration which is very challenging for numerical simulation.

An important conclusion from the test CFD simulations on single-span model is that DES does not show any clear advantage over wall-modeled LES for given geometry and numerical setup. All test CFD simulations are performed with the same spatial and temporal discretization and wall-modeled LES showed better performance, especially in prediction of turbulent flow fluctuations. Hence, the advantages of DES typical for massively separated flows in aerodynamic application cannot be really confirmed for the strongly wall confined flow configuration within rod bundle. Therefore, the use of wall-modeled LES seems to be safer option for the simulation of turbulent flow inside fuel assemblies, especially if the numerical mesh used is not sufficiently fine, which is rather the case.

The vibration response of the fuel assembly is calculated using a mechanical model that incorporates the relevant mechanical properties of the fuel assembly and the effects of fluid (added mass and fluid damping) on the dynamic behavior of fuel assembly.

It was shown that the CFD simulation based on WM LES approach produces reasonable results in terms of fluid flow. The comparison of mean flow and flow fluctuations RMS profiles measured in the tests shows a reasonable agreement, which is within the presumed uncertainty of the experimental results. The simulation could also reproduce qualitatively the vibration behavior of the fuel assembly in terms of dominant vibration frequencies. However, the vibration

amplitude is overestimated by the simulation. The reasons for the overestimation of the vibration amplitude may be in the uncertainties in the calculation of fluid effects (added mass and fluid damping), which are evaluated on a simplified model representing one span that moves as a rigid body. Especially the potential effect of fluid stiffening needs to be evaluated further. Thus, additional refined analysis that considers the full fuel assembly is recommended to verify the values derived from the simplified model. Further refined studies are also recommended to evaluate the sensitivity of the fluid forces calculated by the current CFD approach. This can be done on the small-scale model.

5 Simulation performed by CEA

In this chapter, the numerical simulations of the ALAIN benchmark performed by CEA and the results of these simulations are described and presented in detail.

5.1 Modeling approach

5.1.1 Governing equations

Direct numerical simulation of the Navier-Stokes equations, which describe the motion of an incompressible, isothermal and Newtonian fluid has a prohibitively high cost for the Reynolds number encountered in the present application. Computationally less-demanding alternatives are employed in the present study, namely Reynolds Averaged Navier Stokes (RANS) equations and Large Eddy Simulation (LES).

5.1.1.1 RANS equations

Reynolds Averaged Navier-Stokes equations are obtained by applying Reynolds decomposition of the velocity and pressure fields into the instantaneous Navier-Stokes equations. The resulting equations describe the spatial-temporal evolution of the mean properties of the flow field. For a Newtonian, isothermal and incompressible flow, the mass and momentum conservation equations become:

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (5.1)$$

$$\frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} = \frac{-1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\nu S_{ij} - \overline{u'_i u'_j} \right) \quad (5.2)$$

Where U_i is the average velocity, P the average pressure and ρ and ν are the fluid density and kinematic viscosity, respectively. The mean rate of strain tensor, S_{ij} depends on the gradients of the mean flow

$$S_{ij} = \frac{1}{2} \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (5.3)$$

The averaging procedure leading to the RANS governing equations generates an additional right-hand-side term, $\overline{u'_i u'_j}$ also referred to as Reynolds stresses tensor and contains 6 additional unknowns. The choice of a model to determine $\overline{u'_i u'_j}$ is called the closure problem and several options are available (Wilcox, 2006 [17]). In the present study, the standard $k - \varepsilon$ model of Launder and Spalding, 1974 [35] is employed.

This model is based on Boussinesq eddy viscosity concept, which assumes that Reynolds stresses are aligned with the main strain rates

$$\overline{u'_i u'_j} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij} \quad (5.4)$$

Where δ_{ij} is Kronecker's delta, k is the Turbulent Kinetic Energy (TKE) defined as:

$$k = \frac{1}{2} (\overline{u'^2_x} + \overline{u'^2_y} + \overline{u'^2_z}) \quad (5.5)$$

and ν_t is the turbulent viscosity (also referred to as the eddy viscosity) defined as:

$$\nu_t = C_\mu \frac{k}{\varepsilon} \quad (5.6)$$

with ε being the turbulent kinetic energy dissipation rate. In the $k - \varepsilon$ model, $C_\mu = 0.09$ is a scalar coefficient and k and ε are determined by the solution of the following two transport equations

$$\frac{\partial k}{\partial t} + U_i \frac{\partial k}{\partial x_i} = \overline{u'_i u'_j} \frac{\partial U_j}{\partial x_i} - \varepsilon + \frac{\partial}{\partial x_i} \left[\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] \quad (5.7)$$

$$\frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i} = C_{\varepsilon 1} \frac{\varepsilon}{k} \overline{u'_i u'_j} \frac{\partial U_j}{\partial x_i} - C_{\varepsilon 2} \frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_i} \left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] \quad (5.8)$$

with the five model constants being defined as:

$$C_{\varepsilon 1} = 1.44, C_{\varepsilon 2} = 1.92, C_\mu = 0.09, \sigma_k = 1.0, \sigma_\varepsilon = 1.3 \quad (14)$$

5.1.1.2 Large eddy simulation

The equations for Large Eddy Simulation (LES) are obtained by applying a low-pass filtering operation (indicated by a $\sim$) to the instantaneous Navier-Stokes equations (Sagaut, 2002 [36])

$$\frac{\partial \tilde{u}_i}{\partial x_i} = 0 \quad (5.9)$$

$$\frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \tilde{p}}{\partial x_i} + \frac{\partial}{\partial x_j} (\nu \tilde{S}_{ij} - \tilde{u'_i u'_j}) \quad (5.10)$$

The resulting equations have the same structure as the RANS governing equations but describe the flow on a shorter timescale (Heinz, 2006 [37]). The eddy-viscosity concept is employed to obtain an expression for the filtered stress tensor

$$\tilde{u'_i u'_j} = 2\nu_{SGS} \tilde{S}_{ij} - \frac{2}{3} k_{SGS} \delta_{ij} \quad (5.11)$$

Which depend on the sub-grid-scale (SGS) eddy viscosity, ν_{SGS} and on the rate of strain tensor of the filtered velocity field

$$\tilde{S}_{ij} = \frac{1}{2} \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) \quad (5.12)$$

In the present study, ν_{sgs} is computed with the wall adapting local eddy-viscosity model (WALE) by Nicoud and Ducros, 1999 [38]

$$\nu_{sgs} = (C_w \Delta)^2 \frac{(\tilde{S}_{ij}^d \cdot \tilde{S}_{ij}^d)^{3/2}}{(\tilde{S}_{ij} \cdot \tilde{S}_{ij})^{5/2} + (\tilde{S}_{ij}^d \cdot \tilde{S}_{ij}^d)^{5/4} + 10^{-6}} \quad (5.13)$$

where C_w is a model constant ($C_w = 0.55$) and $\Delta = V_{cell}^{1/3}$ is the mesh cell size. The tensor $\tilde{S}_{ij}^d$ depends on the filtered rate of strain tensor, $\tilde{S}_{ij}$ and of the filtered rate of rotation tensor, $\tilde{\Omega}_{ij}$

$$\tilde{S}_{ij}^d = \tilde{S}_{ik}\tilde{S}_{kj} + \tilde{\Omega}_{ik}\tilde{\Omega}_{kj} - \frac{1}{3}(\tilde{S}_{mn}\tilde{S}_{mn} - \tilde{\Omega}_{mn}\tilde{\Omega}_{mn}) \quad (5.14)$$

where

$$\tilde{\Omega}_{ij} = \frac{1}{2} \left(\frac{\partial \tilde{u}_i}{\partial x_j} - \frac{\partial \tilde{u}_j}{\partial x_i} \right) \quad (5.15)$$

Finally, the SGS turbulent kinetic energy is estimated by the formula from Schumann [39]:

$$k_{SGS} = \left(\frac{\nu_{SGS}}{0.094\Delta} \right)^2 \quad (5.16)$$

5.1.2 The arbitrary Lagrange- Euler formalism

For the sake of generality, the governing equations are presented here for the standard Navier-Stokes equations and the same methodology applies to the RANS and LES governing equations. The system governing the motion of a Newtonian, incompressible and isothermal fluid is

$$\nabla \cdot \mathbf{v} = 0, \quad \text{in } \Omega(t), \quad (5.17)$$

$$\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} + \frac{1}{\rho} \nabla p - \nu \Delta \mathbf{v} = \mathbf{0}, \text{ in } \Omega(t), \quad (5.18)$$

$$\mathbf{v} = \mathbf{v}_d, \quad \text{at } \Gamma_D, \quad (5.19)$$

$$(\nu \nabla \mathbf{v} - p \mathbf{I}_d) \cdot \mathbf{n} = 0, \quad \text{at } \Gamma_N, \quad (5.20)$$

$$\mathbf{v} = \mathbf{v}_b, \quad \text{at } \Gamma_B(t), \quad (5.21)$$

$$\mathbf{v} = \mathbf{0}, \text{ at } \partial\Omega(t) \setminus \{\Gamma_D \cup \Gamma_N \cup \Gamma_B(t)\}, \quad (5.22)$$

Where $\mathbf{v}$ is the velocity field, p the pressure field and ρ and ν are properties of the fluid as in the previous sections. The fluid domain is denoted by $\Omega(t)$ and is bounded by $\Gamma_B(t)$, a boundary moving with velocity $\mathbf{v} = \mathbf{v}_b$, a fixed inlet, Γ_D , where the velocity $\mathbf{v} = \mathbf{v}_d$ is imposed, and by a fixed outlet, Γ_N , where a zero-pressure condition is imposed. The equations are completed with no-slip and non-penetration conditions on the remaining boundaries, $\partial\Omega(t) \setminus \{\Gamma_D \cup \Gamma_N \cup \Gamma_B(t)\}$.

In the arbitrary Lagrange-Euler approach, the domain is deformed to follow the motion of the fluid-solid interfaces. This interface corresponds to the bundle surface, noted $\partial\mathbf{B}$ in the present study. The mesh is treated as a frame that moves with arbitrary velocity $\mathbf{v}_{ALE}$. For moderate deformations $\mathbf{v}_{ALE}$ is usually defined as the solution of an auxiliary Laplace problem

$$\Delta \mathbf{v}_{ALE} = \mathbf{0}, \text{ in } \Omega(t), \quad (5.23)$$

$$\mathbf{v}_{ALE} = \mathbf{0}, \text{ at } \partial\Omega(t) \setminus \Gamma_B(t), \quad (5.24)$$

$$\mathbf{v}_{ALE} = \mathbf{v}_b \text{ at } \Gamma_B(t) \quad (5.25)$$

Where $\mathbf{v}_b$ is the displacement of the bundle boundary, and from which the kinematics of the numerical mesh is updated. Equations (5.23) to (5.25) are solved using a finite element discretization (Ern and Guermond, 2004 [40]). In this new frame of reference, the Navier-Stokes equation rewrites

$$\frac{\partial J\mathbf{v}}{\partial t} + J \left((\mathbf{v} \cdot \nabla) \mathbf{v} - (\mathbf{v}_{ALE} \cdot \nabla) \mathbf{v} + \frac{1}{\rho} \nabla p - \nu \Delta \mathbf{v} \right) = \mathbf{0}, \quad (5.26)$$

With J the Jacobian of the transformation between the ALE and the Lagrange description. The numerical resolution is carried out in TrioCFD whose description can be found in Panunzio *et al.*, 2022 [41].

5.1.3 Boundary conditions

The governing equations are completed by boundary conditions representative of the studied setup. At the inlet and outlet section, the velocity is imposed, while a zero-pressure-gradient boundary condition is imposed at the outlet section. For both RANS and LES, the inlet velocity profiles are determined by computing the flow in a recirculation box (corresponding to the flow in a domain of infinite length). A detailed analysis of the flow in the recirculation box can be found in Michel *et al.*, 2025 [42].

The steady-state velocity profile obtained in the recirculation box is imposed as the boundary condition for RANS simulations (Figure 5-1). For LES, two simulations are carried out. First, the flow in the recirculation box is simulated until it reaches a statistically steady state. The recirculation box is then coupled to the domain and the flow in the two domains is computed concurrently. The velocity field at the outlet of the recirculation box is used as the inlet boundary condition for the domain. This method ensures that the velocity field at the inlet of the computational domain is updated at each timestep and accounts for the unsteady effects in the turbulent flow.

At solid boundaries a no-slip no-penetration boundary condition is employed, and wall functions (Reichardt, 1951 [43]) are used everywhere in both RANS and LES simulations. This approach models the momentum exchange between the wall region and the fluid. When the boundaries are static, Reichardt's law expresses the dimensionless velocity as a function of the wall distance

$$U^+ = \frac{U}{U_\tau} = \frac{1}{\kappa} \ln(1 + \kappa + y^+) + 7.44 \left[1 - e^{(-y^+/11)} - \frac{y^+}{11} e^{(-y^+/3)} \right] \quad (5.27)$$

Where U^+ is the wall-shear velocity and y^+ is the dimensionless wall distance, expressed in wall units (Pope, 2000 [16]).

Additional boundary conditions for k and ε are required at the walls and at the free-stream boundaries (inlet at outlet sections) for the $k - \varepsilon$ model. At solid walls, they are derived from Reichardt's law of the wall

$$k^+ = \frac{k}{U_\tau^2} = 0.07y^{+2}e^{-y^+/9} + \frac{1}{\sqrt{C_\mu}}[1 - e^{y^+/20}]^2 \quad (5.28)$$

$$\epsilon^+ = \frac{\epsilon}{\nu U_\tau^4} = \frac{1}{\kappa(y^{+2} + 15^4)^{1/4}} \quad (5.29)$$

Where $\kappa = 0.415$ is von Kàrmàn constant.

At the free stream boundaries, standard definitions of k and ε (Pope, 2000 [16]) are employed

$$k = \frac{2}{3}(U_{bulk}I)^2, \quad \varepsilon = C_\mu \frac{k^{(3/2)}}{l} \quad (5.30)$$

Where I is the turbulent intensity (estimated to be 5% in the present study) and $l = 0.038d_h$ is a turbulent length scale.

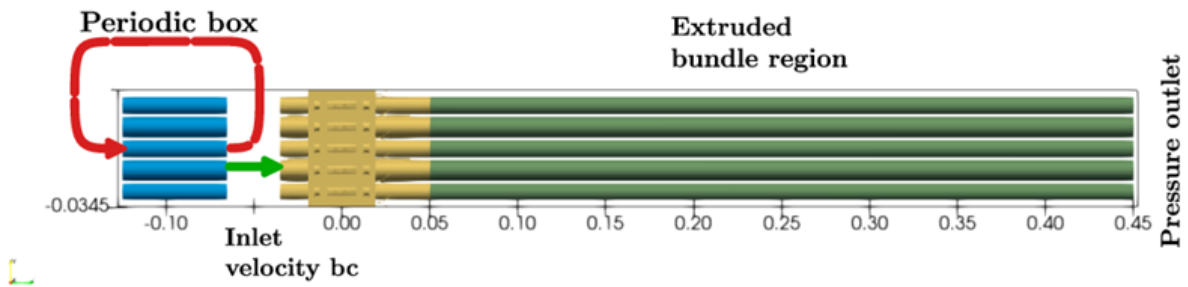


Figure 5-1: Sketch of a single span showing the recirculation box employed to determine the inlet velocity condition (in blue), the meshed grid region (in yellow) and the bundle region downstream of the grid obtained by extrusion (in green).

5.1.4 Numerical simulation approach

5.1.4.1 CFD code used

Numerical calculations are carried out using TrioCFD, an open-source software developed to study thermal hydraulics of incompressible and quasi-compressible flows [44]. The implementation relies on the message passing interface (MPI) to take advantage of massively parallel HPC machines.

The solver relies on a finite volume element approach on tetrahedral cells. The discretization is based on the Crouzeix-Raviart element [45]. The velocity components are defined at the center of the element faces while the pressure is defined at the elements center of mass and nodes. The conservation equations are integrated in conservative form over the control volumes [46]. The time and space discretization as well as the methods employed to solve the pressure problem are standard and summarized in Table 5-1.

Table 5-1: Summary of the numerical methods.

Governing equations	URANS	LES
Closure model	$k - \varepsilon$	WALE
Time scheme	1 st order Euler implicit	3 rd order explicit Runge-Kutta
Predictor step	GMRES	GMRES
Corrector step	Conjugate gradient	Conjugate gradient
Convection operator	MUSCL	MUSCL
Diffusion operator	2 nd order centered	2 nd order centered
Pressure gradient	1 st order	1 st order

5.1.4.2 Description of the numerical mesh

The SALOME open-source software has been employed to partition the domain and generate fully tetrahedral meshes at different refinement levels. As a first step, the mesh was generated for the RANS simulations. The rationale behind this choice was to carry out low-cost computations first to gain insight into the flow properties and to build upon this knowledge to determine the mesh characteristics required for LES computations.

For the RANS simulations, several triangular meshes were first generated in the inlet section. A uniform target mesh size was employed, whose value was varied between 0.4 and 0.8 mm. The resulting 2D mesh was then extruded into 3 layers of 1 mm long prisms in counter-flow directions to generate the volume of the recirculation box. Each prism was cut into 3 tetrahedrons to obtain the final mesh. The result is several meshes with a low total number of elements, which were used to assess the dependence of the flow in the recirculation box to the mesh cell size. The results have been summarized in (Michel *et al.*, 2025 [42]).

Following this step, a mesh was built for the RANS simulations including the mixing grid. From the results obtained in the bare bundle, no accuracy gains are obtained for mesh cells of size <0.6 mm. This mesh cell size was therefore chosen to generate the initial RANS meshes. Mesh generation proved a challenging task in the grid region, due to the complex geometry and the highly detailed model. Our first attempts to mesh the grid region resulted in the generation of several very small and highly skewed elements near the guide tubes in the grid region which impaired computational accuracy severely. To improve the mesh quality, the model was slightly simplified in this region as shown in Figure 5-2. The quality of the resulting mesh was significantly improved, and the modification is not expected to deteriorate the quality of the solution.

The mesh around the spacer grid was extruded for 500 mm in the flow direction to generate the single span domain. Several extrusions were employed to gradually increase the axial size of the cells in the flow direction when moving further away from the spacer grid.

For the whole assembly, the domain was split into three regions, that is the inlet, the first spacer grid, and the outlet. Each part was meshed using the same parameters as previously

described. Mesh conformity was ensured at the interfaces. To generate the mesh for the central part of the test channel, a single span was generated by extruding the mesh in a section behind the first grid, in a manner like that employed for the single span domain. This domain was then duplicated and translated at the position of the other grids to generate the model of the whole ALAIN experiment setup. The complete domain was obtained by merging the inlet, central part and outlet meshes.

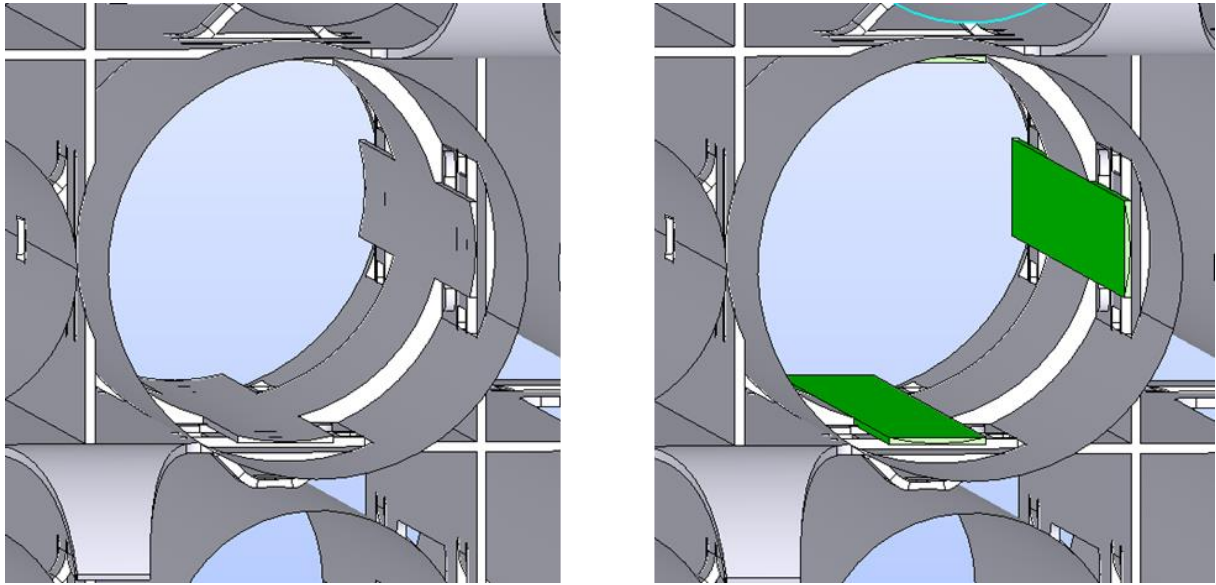


Figure 5-2: Original geometry of the grid region (left panel) and simplifications made to improve the mesh quality (right panel).

For LES computations, only the single span domain was employed. The meshing strategy is the same as the one employed to generate the mesh for RANS simulations. The mesh cell size was however estimated from the preliminary RANS simulations (Figure 5-3). The aim was to ensure that the resolved scales of the turbulent flow contain a sufficient fraction of the turbulent kinetic energy (Celik *et al.*, 2005 [47]; Pope, 2000 [16]). The mesh cell size criterion employed is based on the ratio $\Delta < 25 \eta_k$, where η_k is Kolmogorov's length scale, which is calculated from the results of RANS simulation as:

$$\eta_k = \left(\frac{v^3}{\varepsilon} \right)^{1/4} \quad (5.31)$$

Following RANS simulation results, a first mesh was built with cell size $0.2 < \Delta < 0.36$ mm. smaller cells were located near the solid boundaries, where the value of ε is small due to the high dissipation. A second LES mesh with $\Delta = 0.12$ mm near the solid boundaries was also generated for the comparison.

The number of control volumes for the RANS and LES simulations is presented in Table 5-2 and range between 40M for the RANS simulation of the single span to 650M for a refined LES. The large number of control volumes can be explained by the tetrahedral shape of the elements and by the small elements needed to capture all the turbulent length scales in LES. Further on, the high confinement of the studied geometry induces many near-wall cells. Carrying out LES computations remains a computationally challenging task in the context of the ALAIN experiment, even if wall functions were used everywhere. This is due to the geometric

complexity, the high Reynolds number and the need to average the solution over several periods to converge statistics. As a useful indicator of the current capabilities of TrioCFD, the average cell size at solid boundaries, $\langle y^+ \rangle$ is presented in Table 5-2 for the simulated configurations. For the most refined mesh employed $\langle y^+ \rangle \approx 14$ which extends over the viscous sub-layer and half of the buffer layer. Small scale phenomena in the near-wall region are therefore unresolved in our simulations. A resolved LES would require $\langle y^+ \rangle \approx 1$ near solid wall boundaries, which would require a mesh of several billion elements. Such a simulation is out of reach for the operational conditions of the ALAIN experiment.

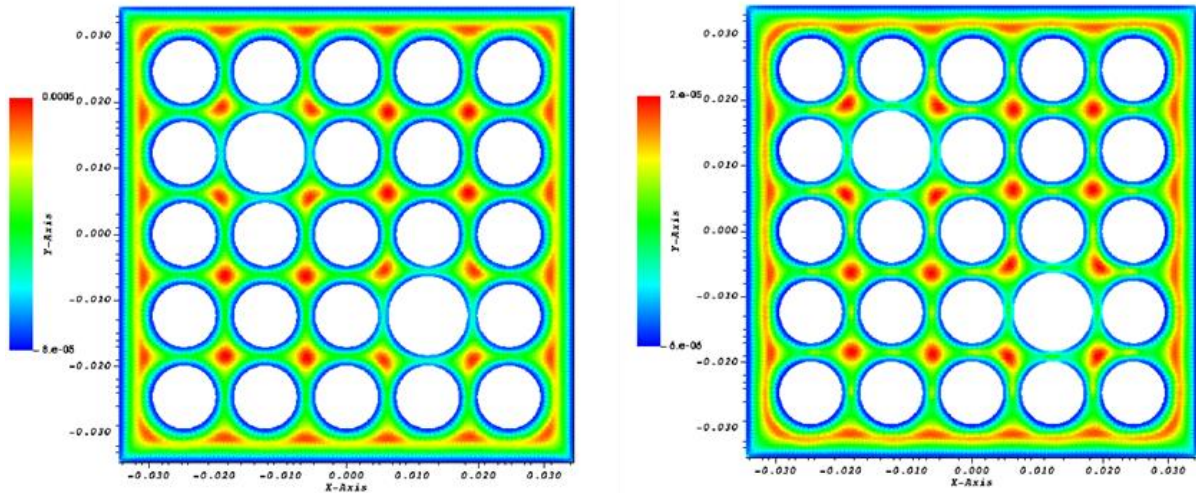


Figure 5-3: Average value of the Taylor microscale (λ , left panel) and Kolmogorov's lengthscale (η_k , right panel), estimated from a RANS simulation of a recirculation box with a mesh cell size $\Delta = 0.6$ mm.

Table 5-2: summary of the mesh characteristics. The target mesh cell size given as well as the number of control volumes are indicators of the computational cost. The average value of the wall mesh cell size $\langle y^+ \rangle$ is also provided for reference.

Turbulence model	Domain	Number of control volumes	Target cell size	$\langle y^+ \rangle$ (post treated value)
RANS	Single span RANS	40 M	0.60 mm	61.0
	Full domain RANS	200 M	0.60 mm	56.5
LES	Periodic box (2, 4, 6 and 10 d_h)	19.4, 38.8, 58.2, 97 M	0.2-0.36 mm	19.5
	Periodic box (6 d_h)	72.4 M	0.12-0.36 mm	15.2
	Single span	400 M	0.2-0.36 mm	18-25
	Single span	660 M	0.12-0.36 mm	12-16

5.1.4.3 Post processing setup

Post-processing was set up beforehand and carried out during the simulations. In TrioCFD, this step is performed by defining fields which can then be visualized or extracted along lines and

planes. The computed fields included averages (for the velocity, pressure, turbulent kinetic energy, dissipation rate, turbulent viscosity or sub-grid scale viscosity) and rms (root mean square; for the velocity and pressure). Additionally, line probes were defined to extract the velocity profiles along the traverses to compare them with experimental results.

The viscous and pressure force fields were also computed for analysis and were extracted at several locations downstream the mixing grids. In TrioCFD, this is done by extracting surfaces on which the forces are then integrated.

5.1.5 Calculated configuration

Simulations were carried out for the single span configuration using RANS and LES turbulence models, and for the whole domain using the RANS model. Additionally, calculations with imposed vibrations using the ALE module of TrioCFD were carried out. These were performed for the whole domain and the single span. Vibrations in quiet fluid were studied using the RANS and LES meshes, since the cost is affordable due to the low velocities. Vibrations under the flow in experimental conditions were however only performed using the RANS turbulence model. The impossibility to carry out this task with a finer mesh was due to the need to collect statistics over large durations. This is extremely costly when coupled with LES, due to the small cells (and thus small timestep) and the large number of control volumes used to discretize the domain.

5.2 Simulations results

5.2.1 Fluid flow profiles

This section presents the comparison of the velocity profiles from the numerical simulations. Two modeling aspects are examined, namely the computational domain and turbulence modeling. The former relies on the simulations carried out in the RANS formalism and emphasizes the difference in the flow predicted when the whole experimental setup or a single span is considered. The latter is carried out for the single span domain and highlights the effect of RANS and LES modelling approaches on the predicted flow.

For both cases, comparison of the numerical and experimental average axial and transverse velocity profiles (as well as their rms) are presented at five distances downstream the mixing grid. The difference between the numerical prediction and the experimental data are also provided as a quantitative tool to assess the more reliably the predictive capability of each model.

5.2.1.1 Influence of computational domain on RANS velocity profiles

The influence of the computational domain on the velocity profile at the inlet of the grids is presented in Figure 5-4 since it is rarely covered in the literature. For the single span configuration, the flow is like that in a bare bundle of infinite length, and it does not exhibit remnants of a disturbance induced by an upstream grid. At the inlet of the first grid of the whole assembly (Figure 5-4, center panel), the velocity profile is strongly disturbed by the presence of the

support plate. The mass flow rate distribution among the subchannels varies significantly at this location. These differences of the inlet velocity profile appear to be partly smoothed by the grid and a moderate effect on the velocity field is visible in their wake (along traverses T1 and T2 where comparison to experimental data is possible, as attested by the results presented in the following sections).

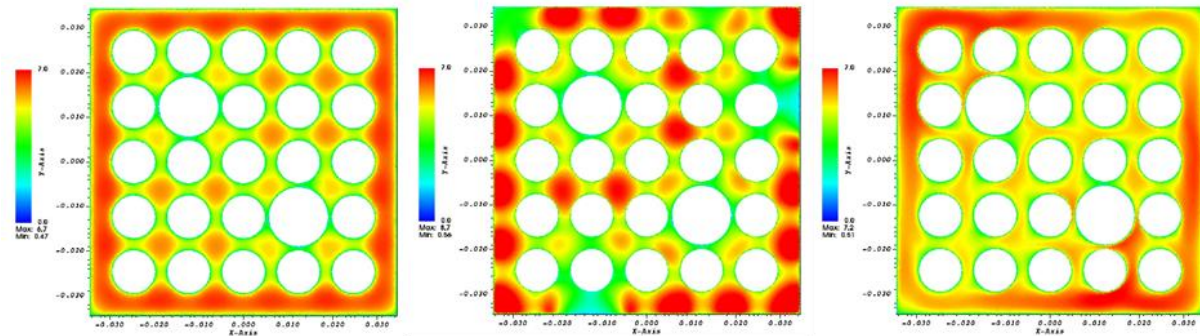


Figure 5-4: Average axial velocity predicted by a RANS model 10 mm upstream of the mixing grid for the single span (left panel), the first grid of the whole assembly (center panel) and the second grid of the whole assembly (right panel). The discrepancies in the inlet velocity field influence the flow downstream of the grids.

At the inlet of second grid of the assembly (Figure 5-4, right panel), the disturbance induced by the presence of the anterior grid is still visible and the flow pattern differs significantly compared to the one in the single span configuration. This observation is consistent with previously published results indicating that the velocity profile is still affected by the presence of the grids $34 d_h$ downstream of the grid. A comparison of the velocity profiles at different distances downstream of the mixing grids is presented in the next section.

5.2.1.1.1 Axial velocity profiles

This section presents the profile of the mean axial velocity in the near wake (10 and 50 mm downstream) of the mixing grids. Close to the grid, 10 mm downstream, the axial velocity profile exhibits a complex pattern with a strongly varying amplitude. For instance, comparatively to the bare bundle case, the velocity extrema located in the center of the subchannels are no longer visible and are replaced by local minima. This is a direct consequence of the presence of mixing vanes, which deflects the axial flow and reduce its intensity at this location. The position of the axial velocity decrease corresponds to the subchannels centers, where maxima of the transverse velocity profile that can be observed in Figure 5-10 to Figure 5-14. Axial velocity minima in the fuel rods and guide-tube gaps are nonetheless still present. As the distance from the mixing grid increases, the velocity profile smoothens, corresponding to a weakening of the disturbance induced by the mixing vanes. The perturbation of the axial velocity field induced by the mixing vanes leaves place to a wake region where the flow pattern rearranges. One notable observation is the influence of the guide-tube on the velocity profile. In the narrow gap next to the guide tube the velocity can decrease substantially, and this phenomenon is not symmetrical (most visible for the traverse 2 than for the traverse 1).

There is a good overall agreement between the numerical prediction and the experimental measurements of the axial velocity profiles for the considered locations. Some discrepancies are nonetheless visible. For instance, the axial velocity is overestimated in the subchannel

along the traverse T1, and in the subchannel along the traverse T2 at the measurement's positions (Figure 5-5). Additionally, the axial velocity minima in the guide-tube gap at $x = 12.32$ for the traverse T2 is not accurately captured by the RANS model (Figure 5-6, left panel). These observations are valid for the different computational domains selected. Nonetheless, variations of the predicted axial velocity magnitude are observed for the different domains. This is quantified by computing the average absolute error between the CFD-predicted velocity and the experimental measures, presented in Table 5-3. The average absolute error is computed as

$$Err = \frac{1}{N_p} \sum_{i=1}^{N_p} |U_{\text{CFD}}^i - U_{\text{exp}}^i| \quad (5.32)$$

where U_{CFD}^i and U_{exp}^i are the CFD-computed and the experimental velocity at the i -th measurement point of the N_p points of the traverses, respectively. The average absolute error varies between the different computational domains, and it is generally the lowest for grid2 case. This conclusion agrees with the location of the experimental measurements which were taken in the wake of the second grid of the fuel assembly. It is also noteworthy that the results for the single span configuration are closer to the one observed downstream of the second grid than downstream of the first grid, and this is ascribed to the velocity field at the grid inlet. In the single span configuration, the velocity field taken from the recirculation box is nearly unidirectional and has maximum velocity in the gap between the bundle and the square channel (Figure 5-4). This is somewhat close to the flow observed at the inlet of the second grid, although remnants of the transverse velocities induced by the first grid are still visible in Figure 5-4 (right panel). At the inlet of the first grid, the velocity field significantly differs, with jets induced by the presence of the support plate at the entrance of the experimental apparatus (Figure 5-4, center panel).

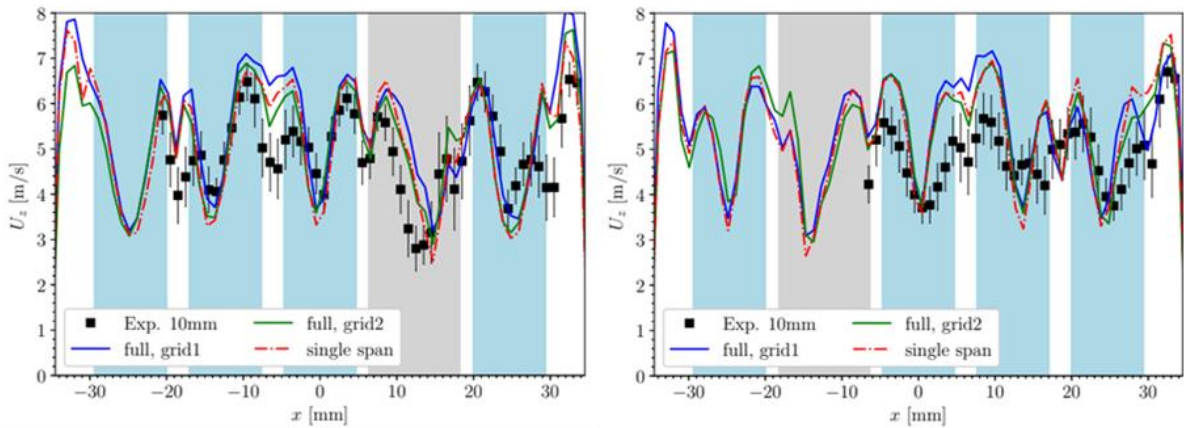


Figure 5-5: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 10 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

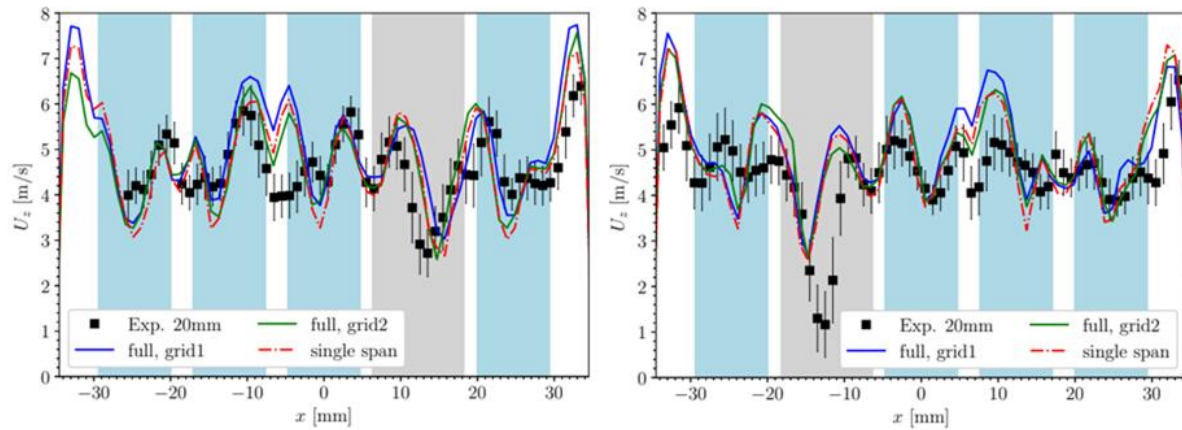


Figure 5-6: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 20 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

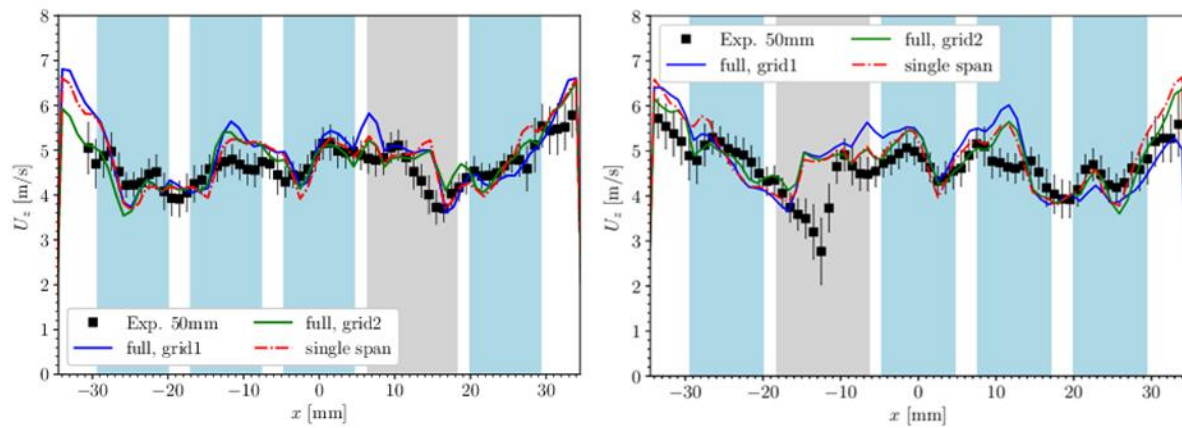


Figure 5-7: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 50 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

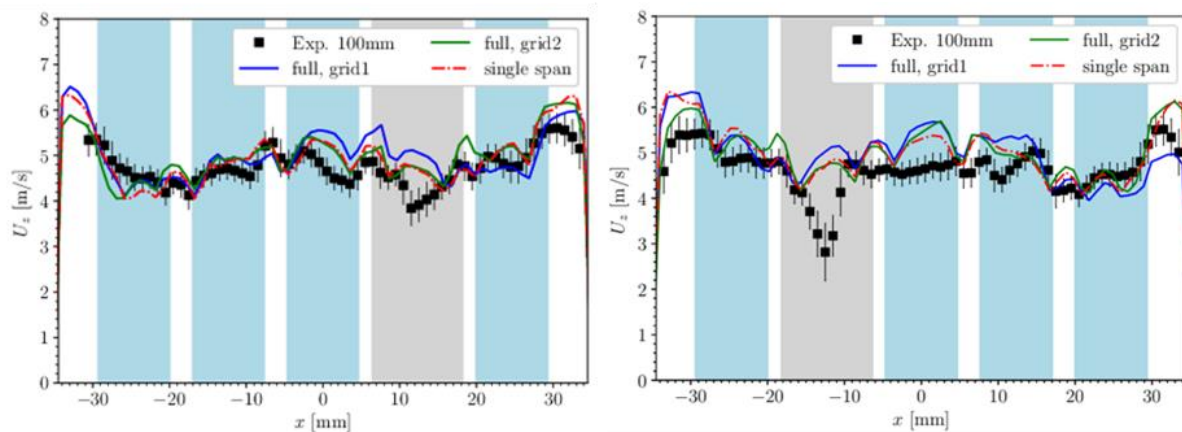


Figure 5-8: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 100 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

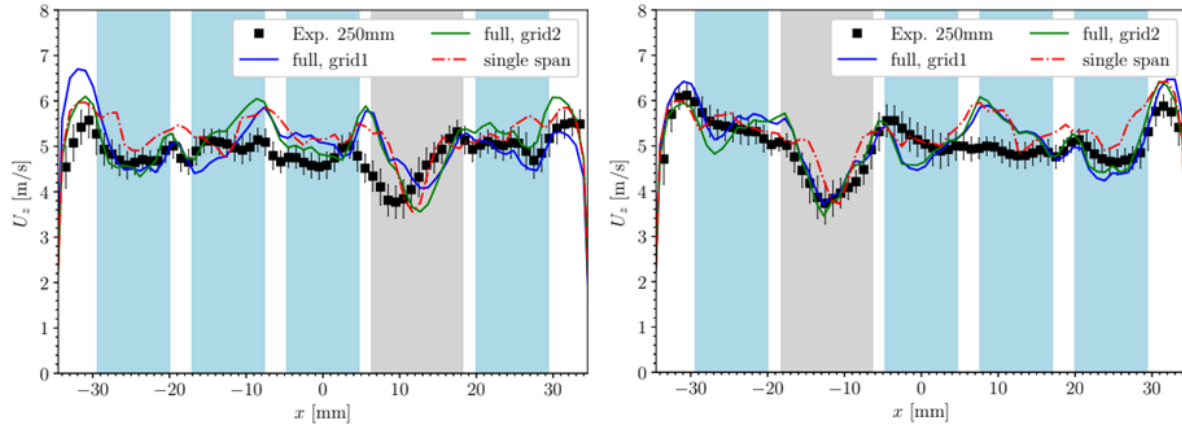


Figure 5-9: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel), 250 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

Table 5-3: Average of the absolute error (in SI units) on the axial velocity profiles obtained from RANS simulations at several locations downstream of the grid. Results are given for the traverses T1 and T2, for the single span domain, and for the whole domain behind the first and second grid (Grid 1 and Grid 2, respectively).

Traverse	Profiles reference	Profile line axial position z				
		10 mm	20 mm	50 mm	100 mm	250 mm
T1	Grid 1	0.839	0.704	0.368	0.366	0.450
	Grid 2	0.700	0.622	0.305	0.337	0.419
	Single span	0.790	0.673	0.349	0.312	0.475
T2	Grid 1	0.899	0.787	0.529	0.549	0.340
	Grid 2	0.813	0.697	0.375	0.464	0.339
	Single span	0.905	0.766	0.447	0.452	0.383

5.2.1.1.2 Transverse velocity profiles

The RANS prediction of the transverse velocity is also generally decent over the measurement data range, and experimental trends are captured. The value of the error on the transverse velocity profiles is reported in Table 5-4. The error is the highest 10 mm away from the grid and decreases further downstream, where the disturbance induced by the vanes is less important. These observations hold for the different setups considered (for the single span domain and behind the two first grids of the whole assembly). The error nonetheless differs between the different setups, being higher for the measurements taken behind the first grid and lower for the measurements taken behind the second grid. We ascribe this difference to the velocity profile at the entrance of the grid.

A closer examination of the results (Figure 5-10 to Figure 5-14) reveals that the transverse velocity can be locally strongly underestimated (up to 60%) in the grid wake. This shortcoming

might be a consequence of using the $k - \varepsilon$ model, which is known to perform poorly for strongly rotational flows such as the one in the wake of the grids. We nonetheless stress that a review of several studies about the prediction of the transverse flow in fuel assemblies with mixing grids indicate that only slight improvements were reported using more sophisticated RANS closure model (Kang and Hassan, 2016 [48]; Angeli and Peybernes, 2018 [49]). The analysis presented by Xiong *et al.*, 2020 [50] suggests that the failure of URANS closure models is due to their inability to predict the turbulence anisotropy in the wake of the mixing grids. On our side, we suggest that the discrepancies between the experiments and numerical results might come from the use of Reichardt law of the wall. This model is not expected to properly model the complex flow on the curved extrados of the mixing vanes. This point might be worth considering for further investigation, since most of the numerical studies using the RANS model have employed wall functions and reported difficulties predicting transverse velocities (Michel *et al.* 2025 [42] for a complete list).

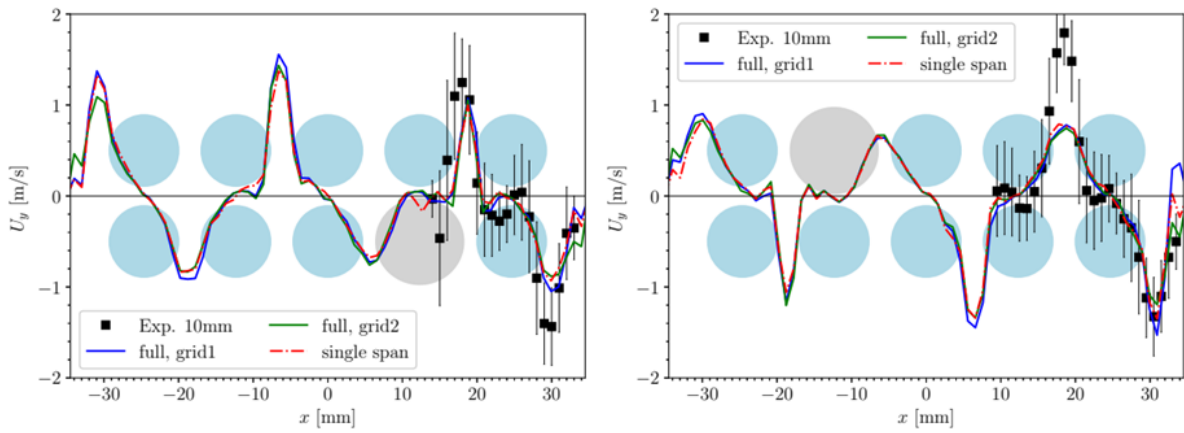


Figure 5-10: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 10 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

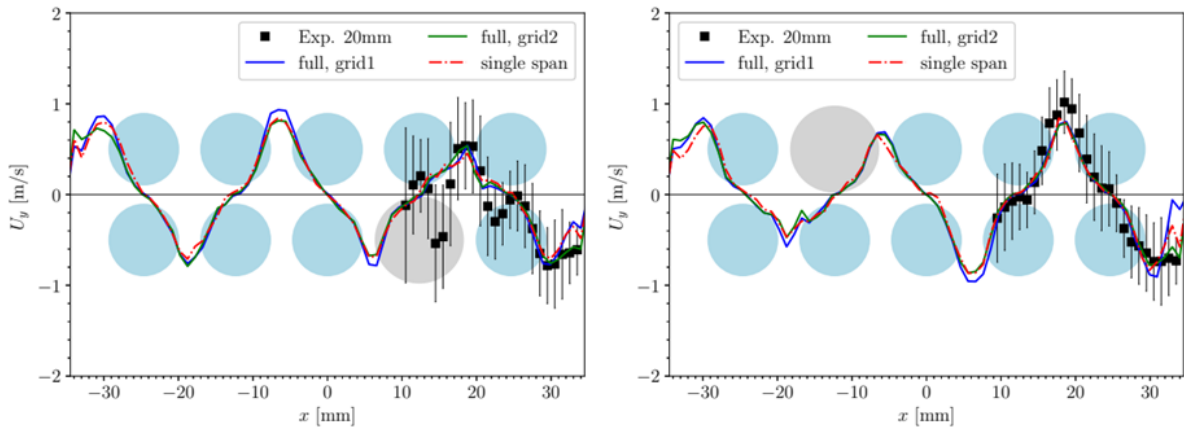


Figure 5-11: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 20 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

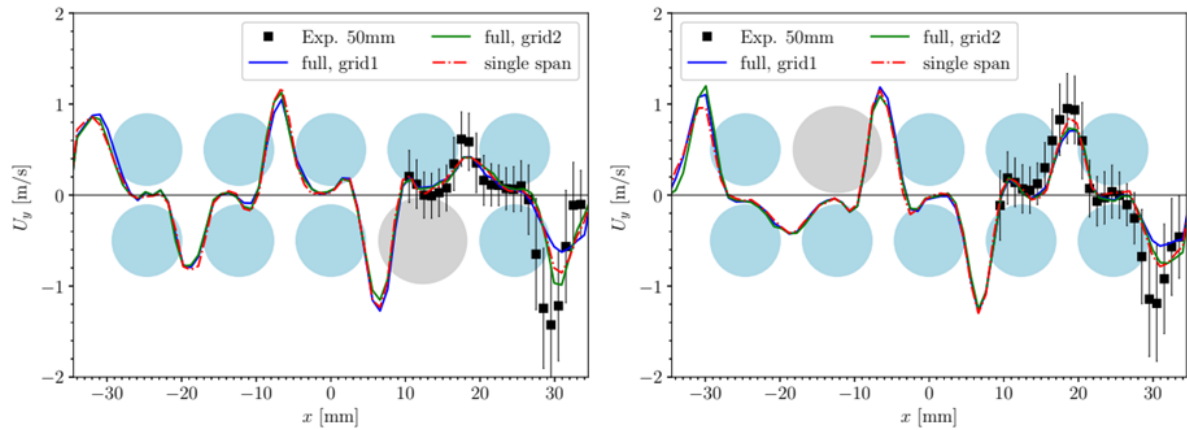


Figure 5-12: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 50 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

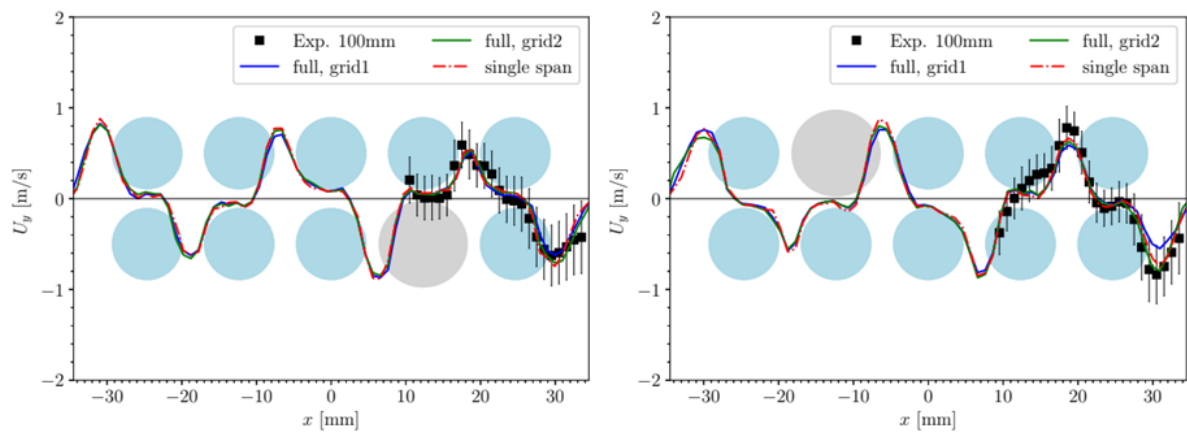


Figure 5-13: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 100 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

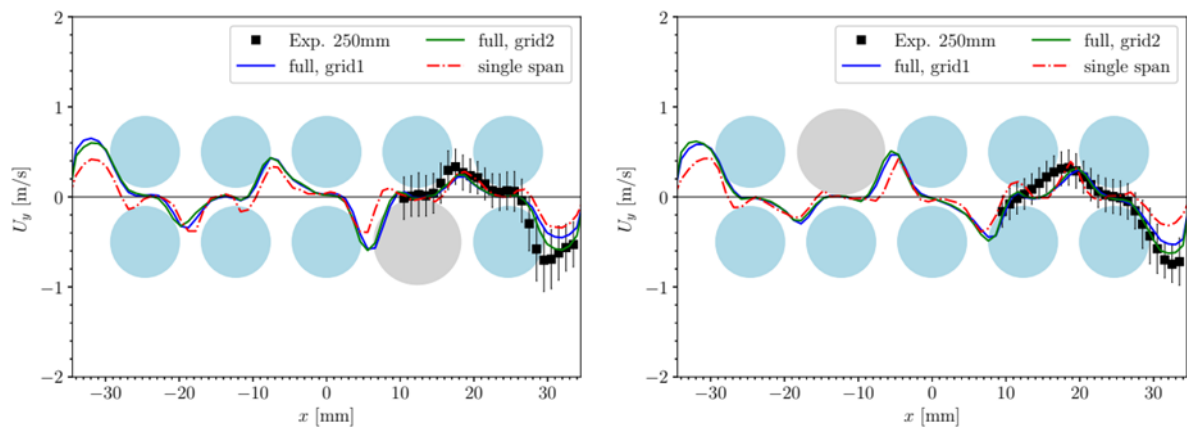


Figure 5-14: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel), 250 mm downstream of the mixing grid. Solid lines: whole domain simulated, results behind the first (blue line) and second grid (green line); dash-dotted line: single span simulated. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

Table 5-4: Table 4.2.1.4: average of the absolute error (in SI units) on the transverse velocity profiles obtained from RANS simulations at several locations downstream of the grid. Results are given for the traverses T1 and T2, for the single span domain, and for the whole domain behind the first and second grid (Grid 1 and Grid 2, respectively).

Traverse	Profiles reference	Profile line axial position z				
		10 mm	20 mm	50 mm	100 mm	250 mm
T1	Whole domain Grid 1	0.266	0.181	0.206	0.087	0.114
	Whole domain Grid 2	0.301	0.162	0.191	0.081	0.080
	Single span	0.292	0.199	0.194	0.089	0.144
T2	Whole domain Grid 1	0.258	0.143	0.180	0.141	0.093
	Whole domain Grid 2	0.228	0.112	0.173	0.100	0.066
	Single span	0.234	0.129	0.157	0.103	0.164

5.2.1.2 Influence of the turbulence modeling approach for the single span configuration

This section presents the comparison of the results obtained for the RANS and LES turbulence models for the single-span configuration. The models describe the turbulent flow at different levels of complexity. We emphasize the fact that RANS simulations are significantly more affordable than LES. The latter requires a higher level of mesh refinement to resolve the largest scales of the turbulent motion, as well as a long computational time to converge statistics. In the following, the flow in the ALAIN experiment has been computed by LES on two different meshes to assess mesh independence.

5.2.1.2.1 Axial velocity profiles

The influence of the turbulence model on the average velocity profiles in the wake of the mixing grids is first examined. Figure presents the velocity profiles 10 and 250 mm downstream of the mixing grid. The axial and transverse velocity profiles from URANS simulations and LES approach are similar in the near wake (Figure 5-15 and Figure 5-16). Quantitative analysis of the error (Table 5-5) nonetheless reveals that LES substantially improves the prediction of the axial velocity in the near wake (10 and 20 mm from the grid). In this region, the profiles strongly depend on the disturbance induced by the mixing grid. The better prediction is due to the resolution of the flow features achieved by the fine LES spatial-temporal resolution. More discrepancies between URANS and LES results are visible in the far-wake. In Figure 5-18 and Figure 5-19, the axial velocity appears to be better captured by LES than by URANS between the fuel rods ($-34.5 < x < 6$ mm). A different trend is noticed near the guide tube gap, near $x = 10$ mm. A minimum of U_z is visible in the LES results, but it is not as pronounced as the one visible in the experimental results. Although the distribution of the error along the traverse differs, the average error over the profile is similar (Table 5-5). Finally, a small decrease of the error on the axial velocity profiles is observed with the refinement of the mesh used for LES, at all locations downstream the mixing grids.

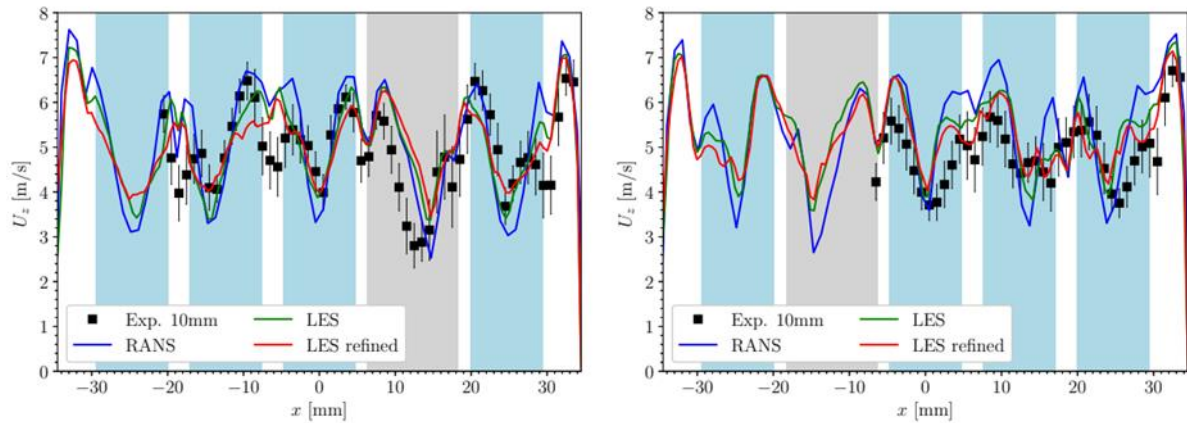


Figure 5-15: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 10 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

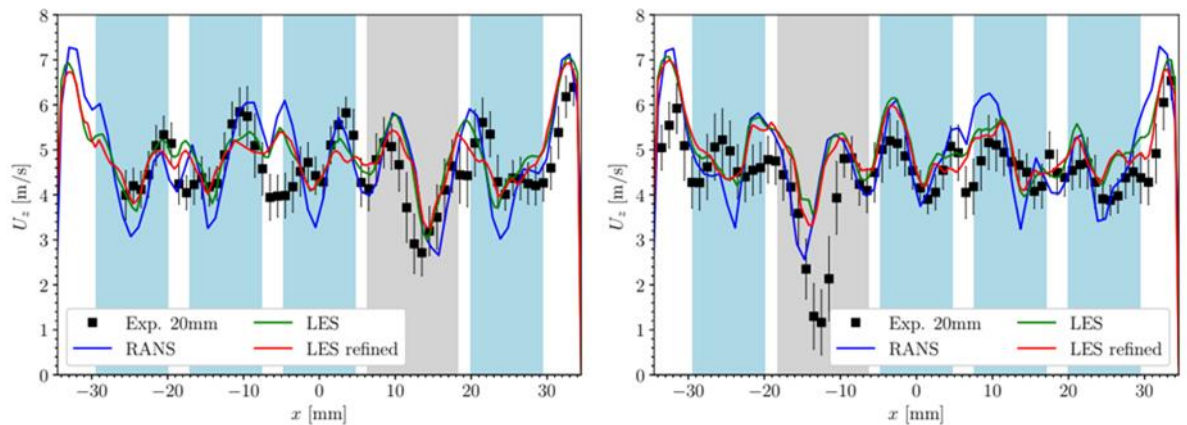


Figure 5-16: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 20 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

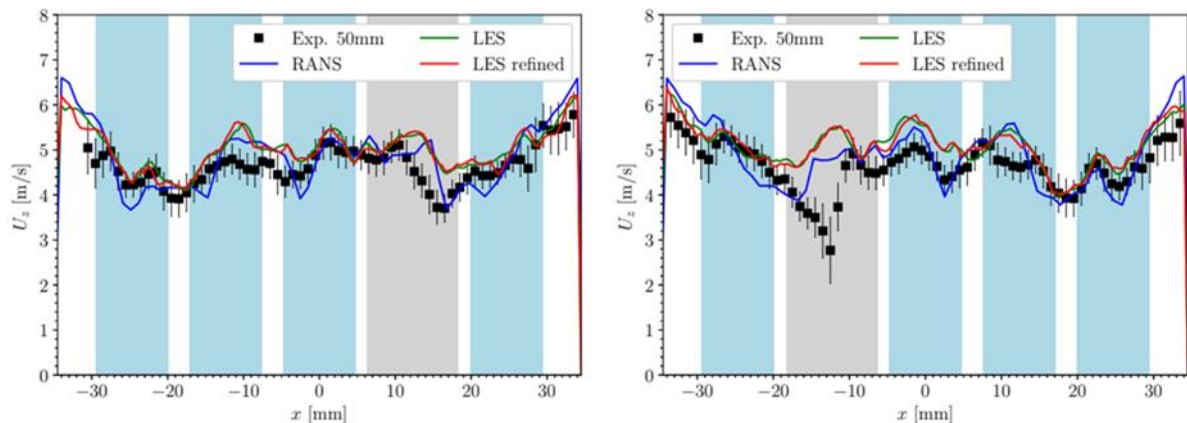


Figure 5-17: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 50 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

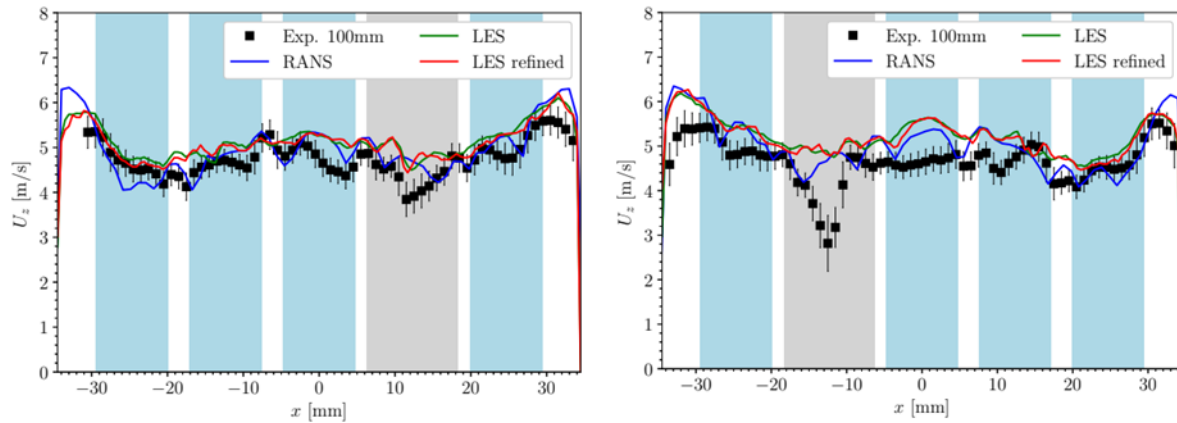


Figure 5-18: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 100 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

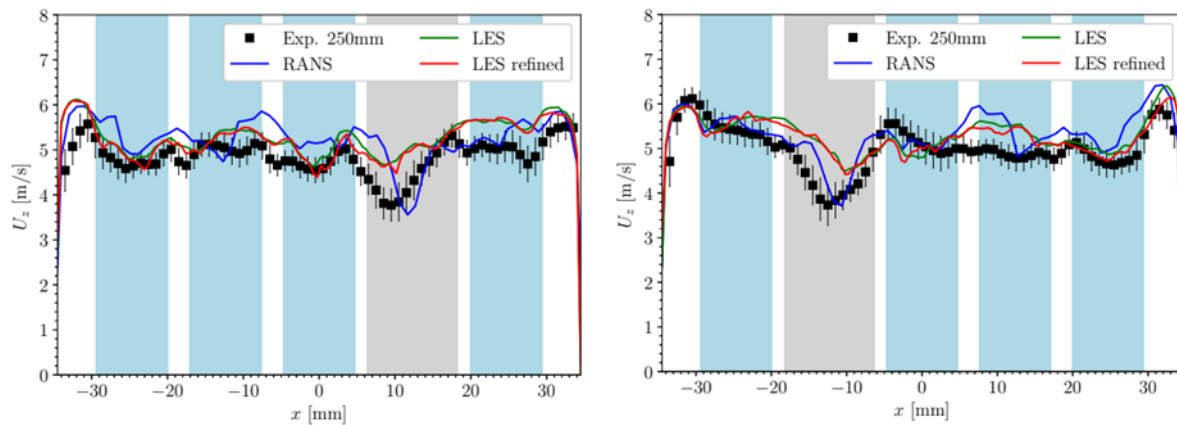


Figure 5-19: Average axial velocity profiles along the traverse T1 (left panel) and T2 (right panel) 250 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

Table 5-5: Average of the absolute error (in SI units) on the average axial velocity profiles at several locations downstream of the grid. Results are shown for the RANS and LES models (at different refinement) and for the two traverses.

		10 mm	20 mm	50 mm	100 mm	250 mm
T1	RANS	0.791	0.673	0.349	0.312	0.475
	LES	0.658	0.520	0.364	0.392	0.447
	Ref. LES	0.633	0.505	0.365	0.353	0.401
T2	RANS	0.905	0.766	0.447	0.452	0.383
	LES	0.668	0.730	0.485	0.591	0.416
	Ref. LES	0.542	0.655	0.481	0.568	0.377

5.2.1.2.2 Transverse velocity profiles

The profiles of the transverse velocity are presented in Figure 5-20 to Figure 5-24 and the average error made on the profiles in Table 5-6. In a general manner, the qualitative profiles of the mean transverse velocity agree with the one reported experimentally. The accurate prediction of the extrema proves challenging in the near wake (10 mm downstream of the grids, Figure 5-20) for both the URANS and LES models. Further on, the experimental profiles feature strong velocity gradients and sign changes near the guide tube 20 mm downstream of the mixing grid (Figure 5-21), which are not well reproduced numerically. At further distance downstream, agreement between the experimental and numerical velocity profiles improves. One noteworthy observation is that the RANS model fails to properly estimate the extrema at several locations (figures Figure 5-22 and Figure 5-24), while the LES model performs better. The main interest of the LES model lies in the prediction of the fine features of the transverse velocity far downstream of the mixing grids.

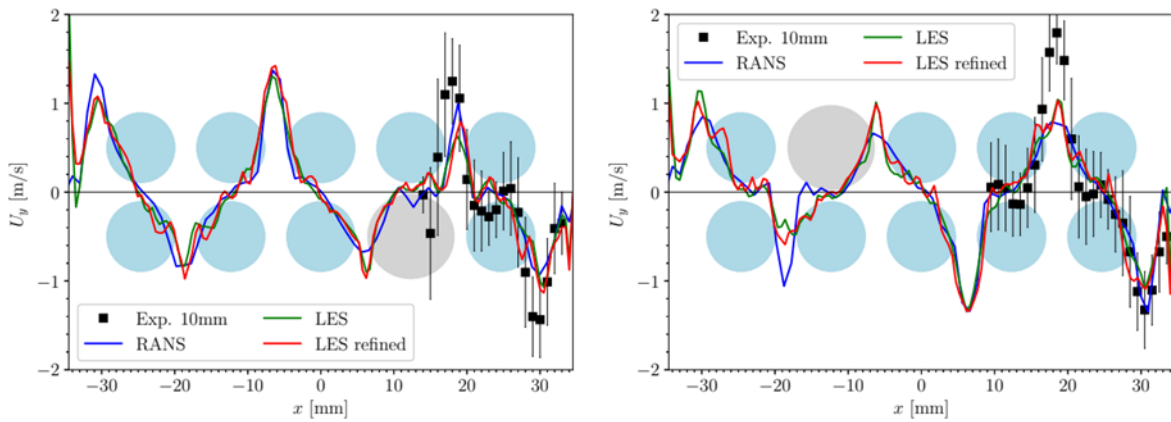


Figure 5-20: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 10 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

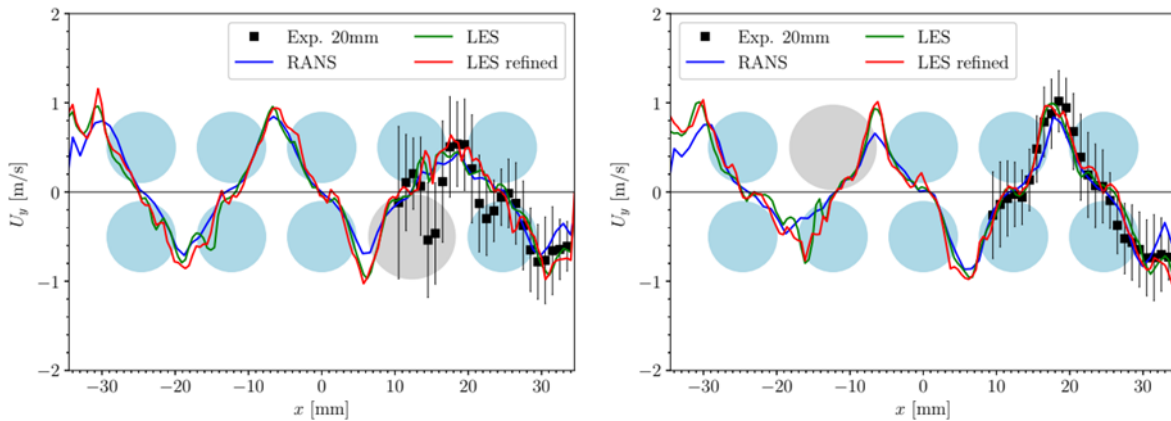


Figure 5-21: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 20 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

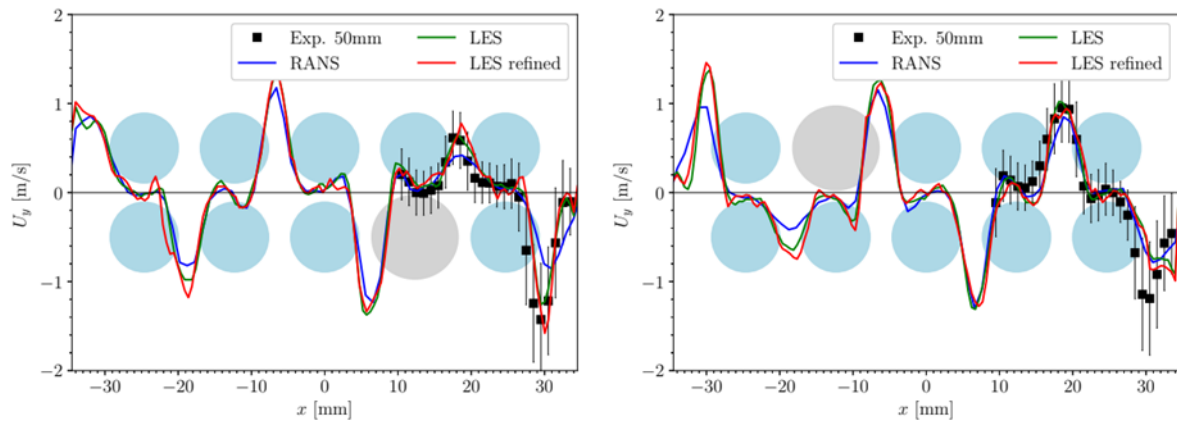


Figure 5-22: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 50 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

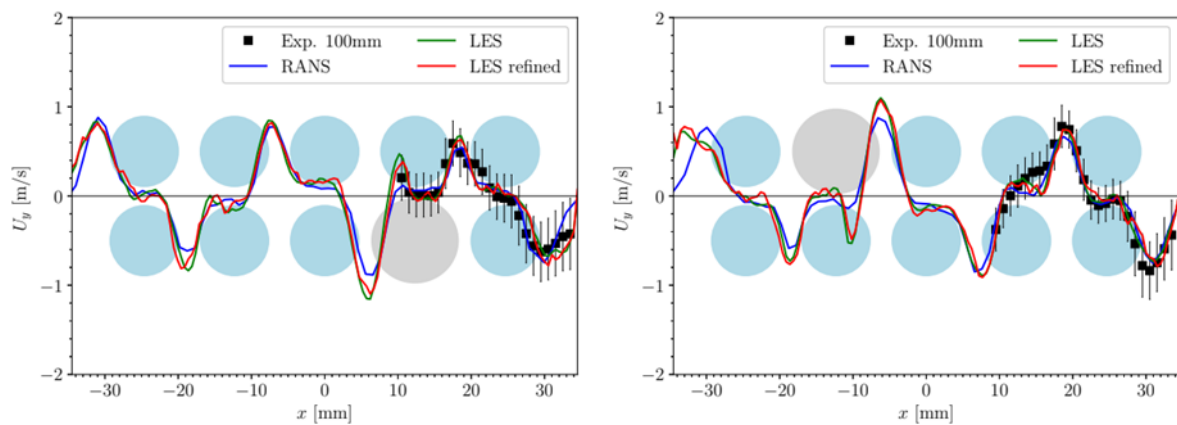


Figure 5-23: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 100 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

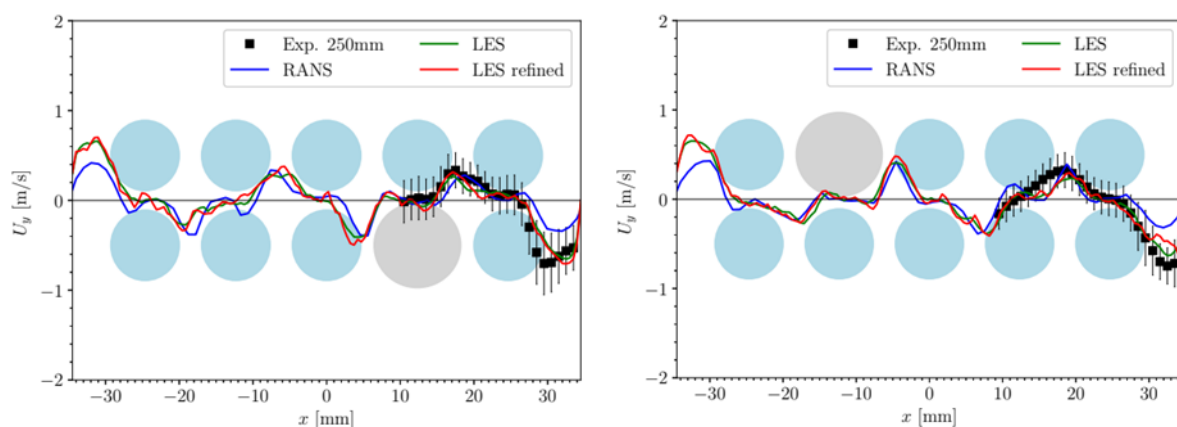


Figure 5-24: Average transverse velocity profiles along the traverse T1 (left panel) and T2 (right panel) 250 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

Table 5-6: Average of the absolute error (in SI units) on the average transverse velocity profiles at several locations downstream of the grid. Results are shown for the RANS and LES models (at different refinement) and for the two traverses.

		10 mm	20 mm	50 mm	100 mm	250 mm
T1	RANS	0.292	0.200	0.194	0.089	0.144
	LES	0.340	0.213	0.115	0.107	0.079
	Ref. LES	0.345	0.235	0.151	0.112	0.080
T2	RANS	0.234	0.129	0.157	0.103	0.164
	LES	0.277	0.104	0.179	0.083	0.074
	Ref. LES	0.279	0.111	0.182	0.104	0.083

5.2.1.2.3 Velocity fluctuations

The RMS of the axial and transverse components of the velocity predicted by the URANS and LES models are presented in Figure 5-25 to Figure 5-34. For the URANS simulations, these quantities were estimated under the assumption that the turbulent kinetic energy is equally distributed between the three velocity components, so that $\text{rms}(U_i) = \sqrt{\frac{2k}{3}}$. At short distance from the grid (10 mm, Figure 5-25 to Figure 5-26 and Figure 5-30 to Figure 5-31), the profiles from both URANS simulation and LES are in qualitative agreement with experiments. For instance, the peak of $\text{rms}(U_i)$ at $x = 16$ mm, near the guide tube gap, is visible with both methods. At further distance from the grid (100 and 250 mm), LES provides a better qualitative prediction of the axial velocity rms with the experiment (Figure 5-28 to Figure 5-29 and Figure 5-33 to Figure 5-34). This conclusion stems from the ability of LES to reproduce the peaks (although not at the right location) of $\text{rms}(U_z)$ inside of the rod network and the increase of $\text{rms}(U_z)$ on the periphery of the bundle. In contrast, $k - \varepsilon$ URANS predicts a nearly flat profile of $\text{rms}(U_z)$ and $\text{rms}(U_y)$ downstream of the grid. Prediction of the velocity rms in the far wake of the grid is challenging for CFD models, since neither of the two considered models can capture the peak of $\text{rms}(U_y)$ in the outer subchannel. One possible explanation is the use of turbulence models based on the eddy-viscosity hypothesis, which assume isotropy of the small-scale turbulence. Such models cannot accurately reproduce the anisotropy of the velocity fluctuations inherent to wall-bounded flows. Quantitative comparison of the average absolute error on the profiles of the velocity rms reveals that the error between numerical and experimental results is similar for the two turbulence models considered (Table 5-7 and Table 5-8).

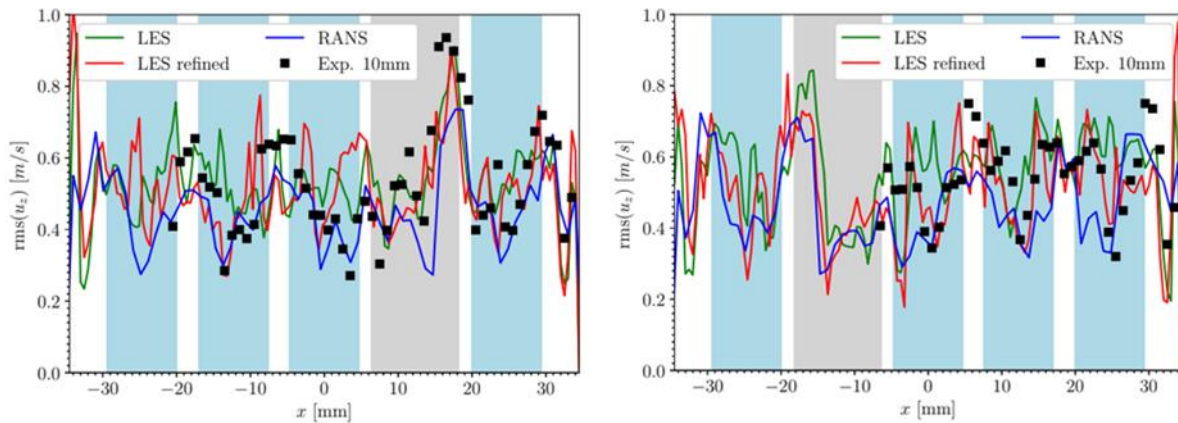


Figure 5-25: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 10 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

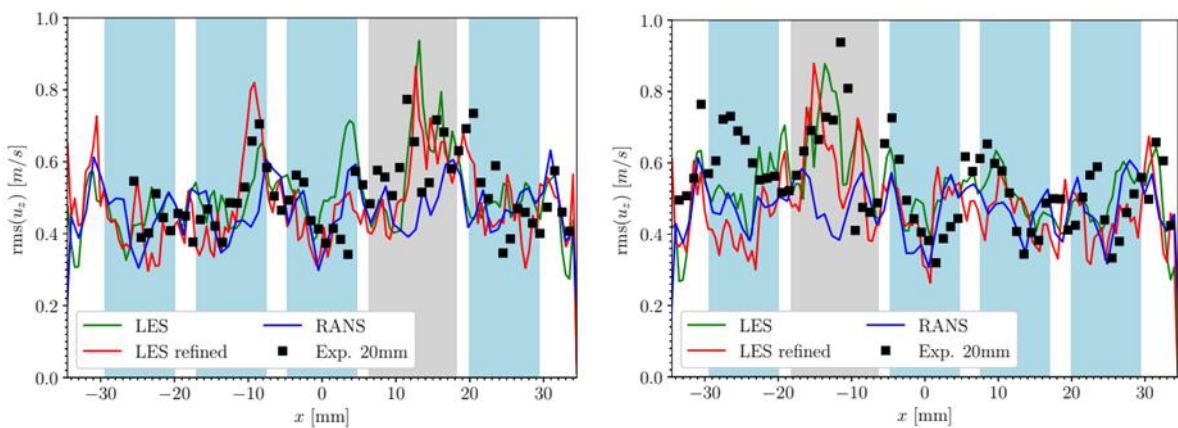


Figure 5-26: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 20 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

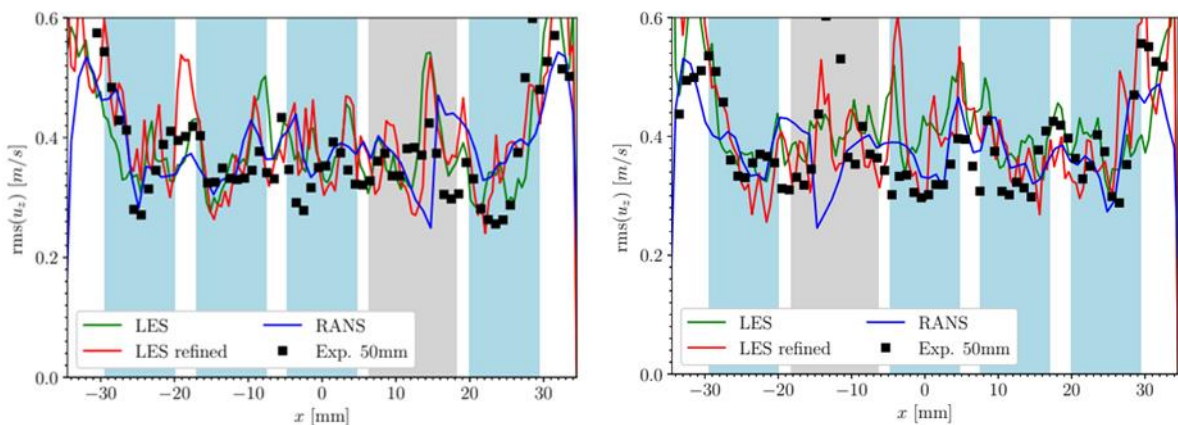


Figure 5-27: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 50 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

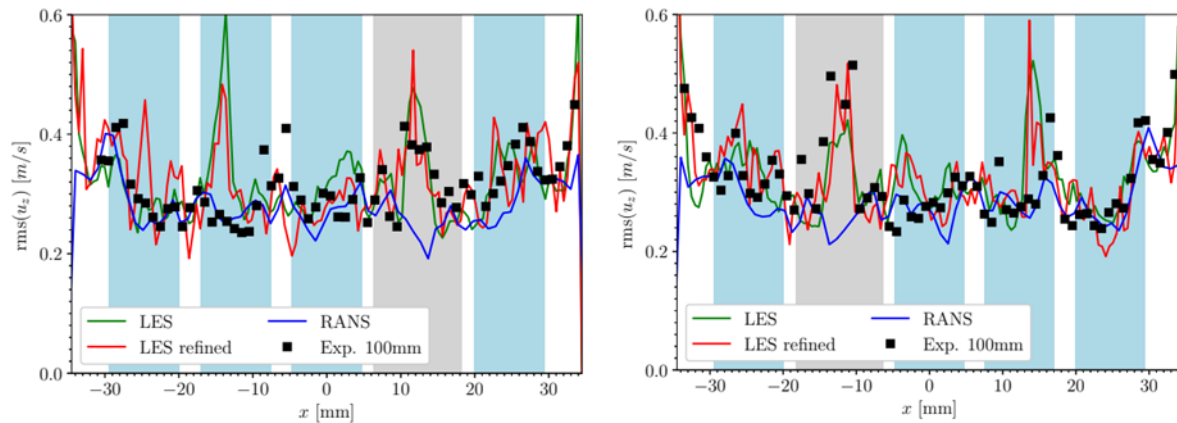


Figure 5-28: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 100 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

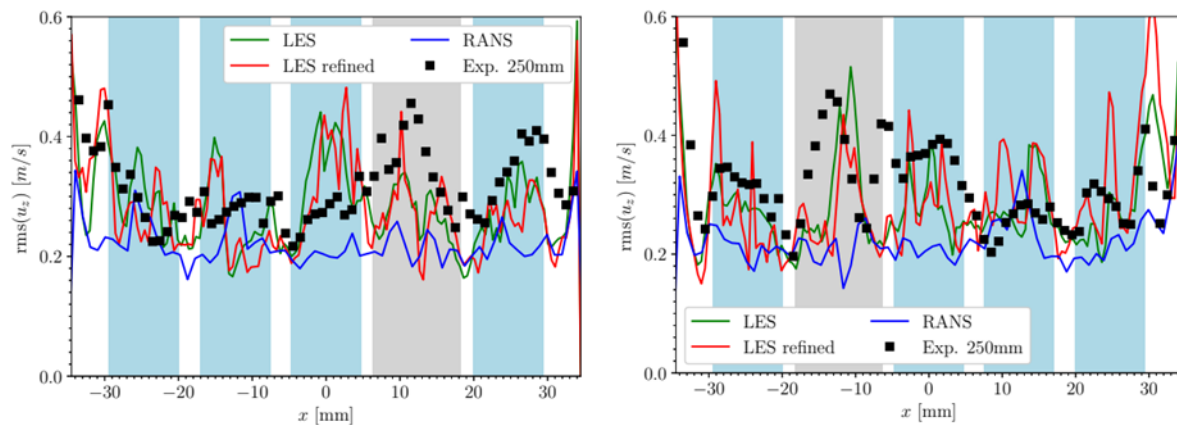


Figure 5-29: RMS of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 250 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

Table 5-7: Average of the absolute error (in SI units) on the axial velocity rms profiles at several locations downstream of the grid. Results are shown for the RANS and LES models (at different refinement) and for the two traverses.

		10 mm	20 mm	50 mm	100 mm	250 mm
T1	RANS	0.094	0.088	0.054	0.045	0.095
	LES	0.104	0.088	0.055	0.047	0.068
	Ref. LES	0.124	0.081	0.061	0.049	0.073
T2	RANS	0.113	0.116	0.065	0.052	0.099
	LES	0.117	0.101	0.070	0.049	0.064
	Ref. LES	0.106	0.127	0.072	0.050	0.086

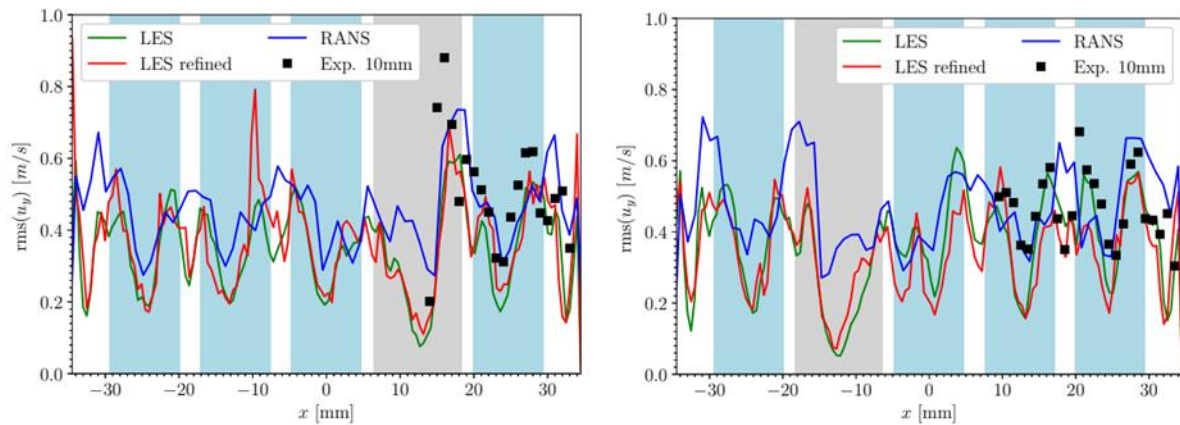


Figure 5-30: RMS of the transverse velocity along the traverse T1 (left panel) and T2 (right panel) 10 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

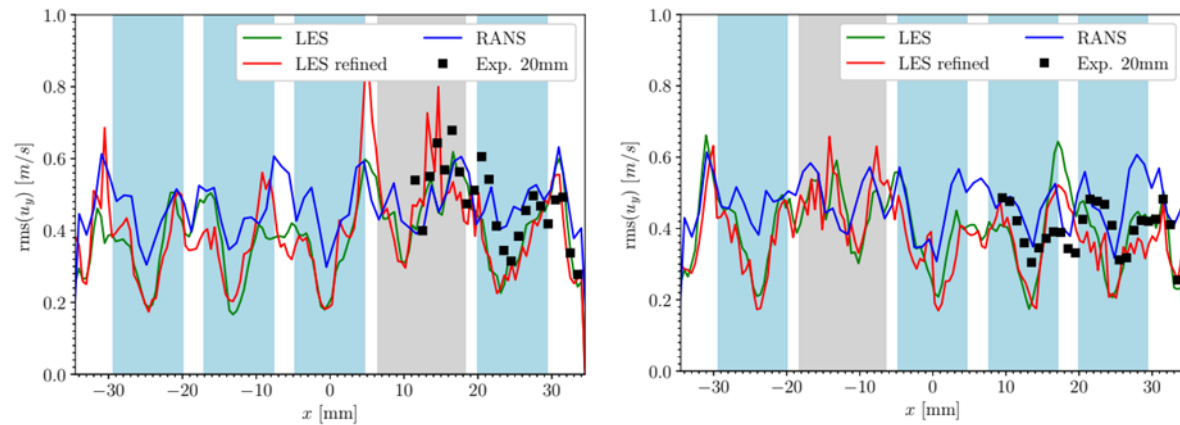


Figure 5-31: RMS of the transverse velocity along the traverse T1 (left panel) and T2 (right panel) 20 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

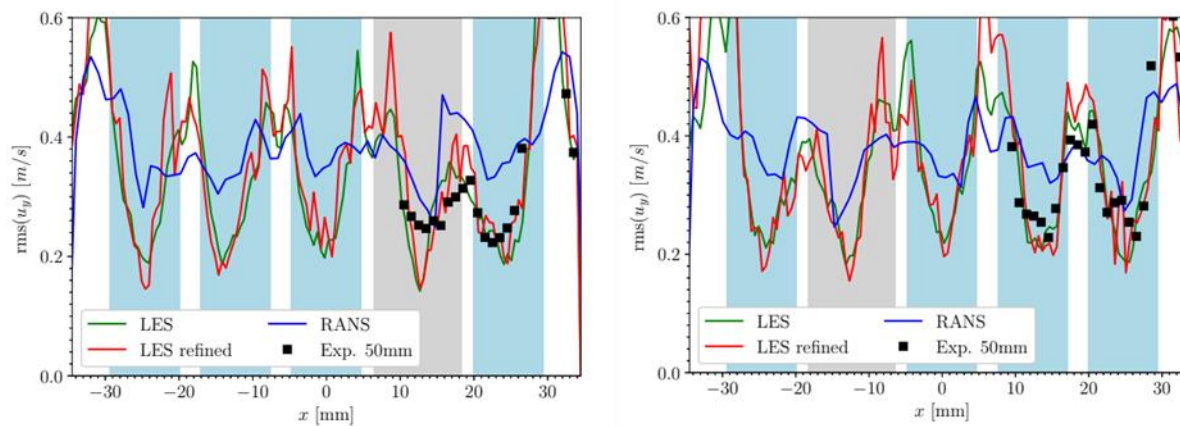


Figure 5-32: RMS of the transverse velocity along the traverse T1 (left panel) and T2 (right panel) 50 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements.

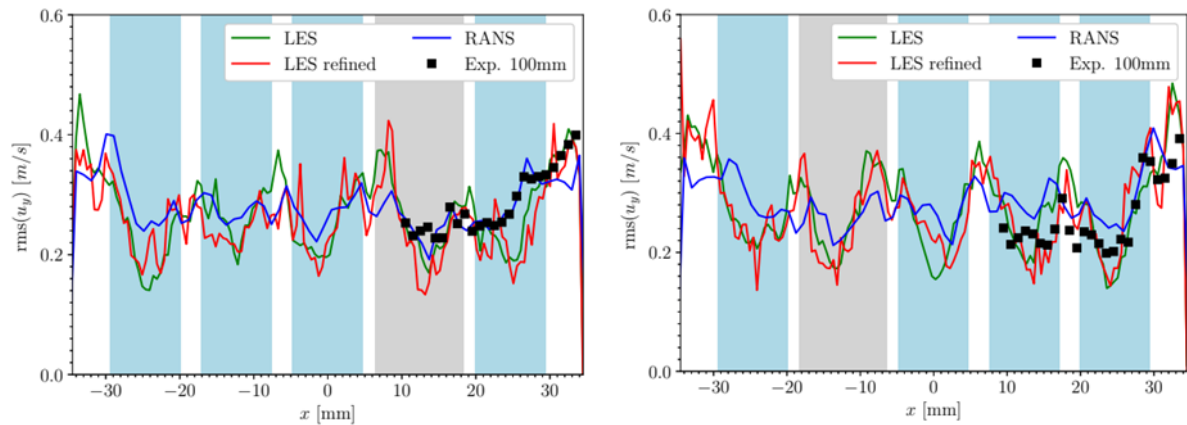


Figure 5-33: RMS of the transverse velocity along the traverse T1 (left panel) and T2 (right panel) 100 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements.

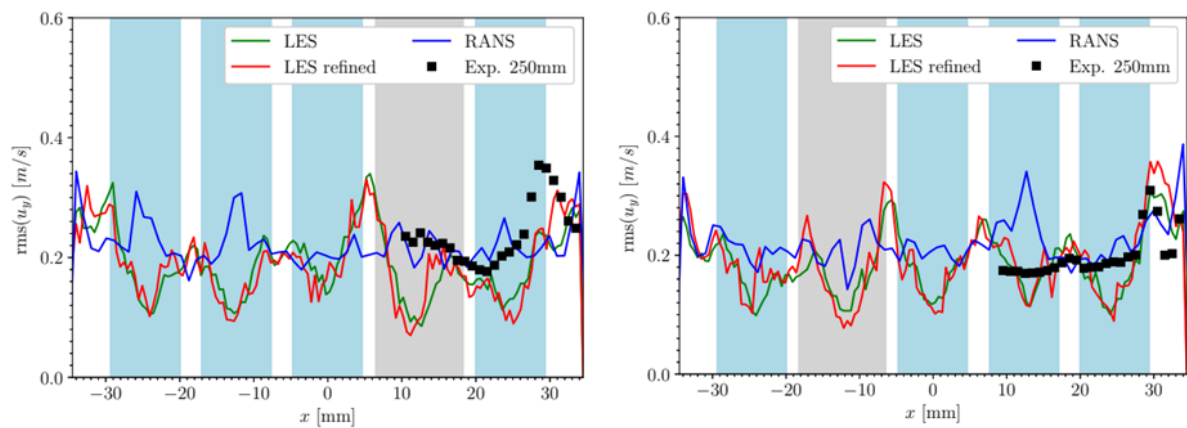


Figure 5-34: RMS of the transverse velocity along the traverse T1 (left panel) and T2 (right panel) 250 mm downstream of the mixing grid for the single span domain. Black squares denote experimental measurements.

Table 5-8: Average of the absolute error (in SI units) on the transverse velocity rms profiles at several locations downstream of the grid. Results are shown for the RANS and LES models (at different refinement) and for the two traverses.

		10 mm	20 mm	50 mm	100 mm	250 mm
T1	RANS	0.109	0.095	0.110	0.018	0.043
	LES	0.147	0.096	0.046	0.031	0.069
	Ref. LES	0.137	0.106	0.064	0.048	0.067
T2	RANS	0.114	0.082	0.072	0.043	0.044
	LES	0.086	0.094	0.050	0.046	0.030
	Ref. LES	0.126	0.085	0.061	0.041	0.041

5.2.2 Conclusion on the flow simulations in the ALAIN experiment

The flow in a 5×5 rod bundle representative of the ALAIN experiment has been simulated using TrioCFD. The experimental apparatus features 23 fuel rods, two guide tubes and five mixing grids to represent a reduced-scale pressurized water reactor fuel assembly. Two turbulence models with standard closure models were employed to simulate the fluid flow, namely URANS ($k - \varepsilon$) and LES (WALE) both associated with wall functions. The flow in a periodic domain made of a bare rod bundle was first simulated to realize a mesh independence study and to examine the turbulence properties. Starting from URANS simulations, the turbulent length scales (Taylor and Kolmogorov) were computed and the LES mesh requirements were estimated based on the Kolmogorov's length scale criterion proposed by Celik *et al.*, 2005 [47] for wall-bounded flows. A posteriori comparison revealed a good agreement between the URANS-estimated turbulent length scales and the one post-treated from LES, supporting a sufficient mesh resolution for the present application. Further analysis of the results showed that the ratio ν_{sgs}/ν was lower than 20 everywhere in the domain, supporting the adequacy of the LES mesh. Finally, the LES velocity statistics were found to be weakly dependent on the axial extent of the domain.

The influence of the computational domain on the flow in the wake of mixing grids was then examined using the URANS approach. Two domains were considered for numerical simulations, that is, a single span made of a unique mixing grid and 500 mm of axial extent, and a domain representative of the whole ALAIN experiment comprising 5 mixing grids (250 mm axial extent). The experimentally measured flow properties were qualitatively well reproduced in both setups, indicating that the mixing grids reduce the dependence of the downstream flow to the inlet conditions. In the present study, the minimal error between CFD-predicted velocity and experimental measurements is obtained when the simulated domain corresponds to the full experimental setup. A noticeable influence of the domain on the transverse velocity in the far wake of the mixing grids is reported and it may be an interesting point for further investigation. The characteristics of the flow pattern far downstream the mixing grids remains are poorly understood (Bieder *et al.*, 2014 [51]; Muller *et al.*, 2017 [52]), especially when guide tubes are present in the bundle lattice. The velocity statistics predicted by URANS and LES for the single span domain were then compared to experimental measurements. Agreement between the CFD-computed and experimental profiles of the axial and transverse components of the velocity are close for both simulation methods. Nonetheless, prediction of the average axial velocity in the near wake of the mixing grids (10 and 20 mm downstream), and of the average transverse velocity in the far wake (100 and 250 mm downstream) is significantly improved when LES is employed. In contrast, only a slight improvement of the prediction of the rms of the axial and transverse velocity components by LES is noticed, and only at distances larger than 50 mm downstream of the grid.

The discrepancy between CFD and experimental data can be partly explained by the usage of turbulence modeling approaches and the use of wall functions, which do not perfectly reproduce the complex small-scale features of the turbulent flow behind the grids. Further on, manufacturing and measurement uncertainties also influence the experimental data. Uncertainty quantification could help determine the influence of these parameters in the future. Direct extension to the present study involves assessing the forces exerted by the fluid on the grid and fuel rods.

5.3 Fluid forces and assembly vibrations

5.3.1 Modeling of the assembly vibration and modal coefficients

5.3.1.1 Mode shape: beam theory and finite elements model

Impact tests have been conducted to determine the natural vibration frequencies of the skeleton and bundle. As a first step, these results are compared to the ones from a finite element model (FEM) to gain insight into the dynamics of the vibrating bundle. This model is based on a simplified bundle geometry, where the fuel rods and guide-tubes are modeled as Euler-Bernoulli (EB) beam elements, and the grids are modeled as shell elements without their complex geometrical features. The EB beam elements are standard elements employed in mechanical studies and are described in Logan, 2007 [53]. The model has been implemented in Cast3M (CEA, 2020), an open-source software developed for thermo-mechanics applications. For the guide-tubes, the mass and inertia moment are determined from the materials and dimensions specified in [1]. For the fuel rods, the quadratic moment of the cladding tube was used, and density was adapted to account for the mass of the molybdenum pellets mimicking the nuclear fuel in the experiments. Finally, a rigid contact was specified between the grids and guide-tubes, while the contact between the grids and fuel rods was modeled by springs. This latter point is a sensitive aspect of the model since the predicted vibration frequencies of the bundle significantly vary with the spring stiffness. It was found that a stiffness of 50 N/mm yielded a good agreement with the experimentally measured frequencies. A summary of the material properties employed is provided in Table 5-9.

Table 5-9: Material properties employed for the EB beam and shell elements used in the finite element model of the bundle

	Fuel rod	Guide tube	grid
Density [kg/m ³]	8690	6600	8200
Young Modulus [GPa]	98.1	98.1	198

Comparison between the natural frequencies measured experimentally and predicted by the FEM is presented in Table 5-10. The numerical model underpredicts the first and third natural frequencies of the skeleton and overpredicts the second natural frequency of the bundle. The finite element model is nonetheless able to predict the other frequencies with good accuracy and provides a first information about the structural dynamics of the assembly. The first and second modes of the bundle vibration predicted from the numerical model are presented in Figure 5-35. The mode shape is nearly symmetrical and is similar in shape to that expected from beam theory. The presence of the grids nonetheless visibly influences the second mode and induces a rigidity-like effect. The mode shape from the finite element model is compared to the prediction of the EB beam theory in Figure 5-36. As expected, the results are close, but the presence of the grids visibly affects the mode shape in the numerical model. The main effect of the grids is to decrease the bending near the grid's locations. This is especially visible around grids 2 and 4 for the first mode, and grid 3 for the second mode. The mode shape predicted from Euler-Bernoulli beam theory is a good first approximate to the fuel assembly.

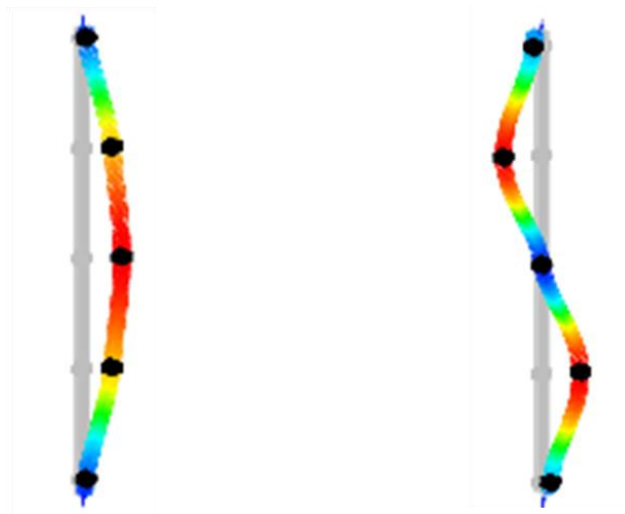
Table 5-10: Natural frequency (in Hz) of the first modes of the skeleton (guide-tubes and grids) and the bundle measured in ALAIN experiment impact test and predicted by the Cast3M finite element model.

		f_1	f_2	f_3	f_4	f_5	f_6
Skeleton	Experiment	9.1	15.8	23.6	34.1	43.9	50.8
	Numerical model	7.7	16.7	21.1	34.3	40.6	50.6
	Deviation from experiment (%)	15.4	5.69	10.5	0.59	7.51	0.39
Bundle	Experiment	6.0	14.5	23.9			
	Numerical model	5.9	16.5	24.3			
	Deviation from experiment (%)	1.67	13.8	1.67			

Following this preliminary study, simulations of the flow in the ALAIN experiment were carried out using CFD. To reduce the computational cost, the bundle displacement is not determined at each iteration by solving the equations governing the structural dynamics but is imposed as a boundary condition. The imposed displacement is of the form:

$$u(x, t) = \sum_{k=0}^m a_k \Phi_k(x) q_k(t) \quad (30)$$

where a_k is the amplitude of the k -th harmonic. From experimental measurements (see [1]) the first and second modes dominate the harmonic response of the bundle and have an approximate amplitude of $5 \mu\text{m}$. Their frequency is 6 and 12.5 Hz, respectively. As a first approximation, the modal displacement, $\Phi_k(x)$, is computed from the EB beam theory and the analytical formula is modified to include the effect of grid rigidity on the displacement by enforcing a constant displacement at the location of the grids (corresponding to the case *modif. EB* in Figure 5-36).

**Figure 5-35: First and second modes for the whole assembly computed by Cast3M.**

Our objective was to carry out RANS simulations of the single span and of the whole experimental apparatus with imposed vibrations. The former proved challenging, due to the need

to impose the displacement between two arbitrary bounds. For preliminary investigation, the displacement corresponding to a pinned-pinned EB beam was used for the single span domain. The whole ALAIN setup is more straightforward since the guide tubes are pinned at their extremities in the experiment. LES of the single span with imposed vibration was also undertaken, but it was not possible to complete a simulation time corresponding to several periods to analyze the flow behavior. The computational resources needed to perform a LES, even for the single span, and with the simplifications employed (use of the law of the wall) remains challenging in the operational conditions of ALAIN experiment. This is a complex multifactorial problem due to the size of the cells near the walls, the high flow velocity, and the cost induced by the Arbitrary Lagrange-Euler algorithm.

Attempts at higher vibration frequencies (30 Hz) were carried out to reduce the period, and thus, the physical time to be simulated, but the calculation remained too costly, nevertheless. It was possible to carry out a comparison between the results obtained for the RANS and LES meshes for the case of the single span vibrating in a quiet fluid, from which added modal coefficients were extracted.

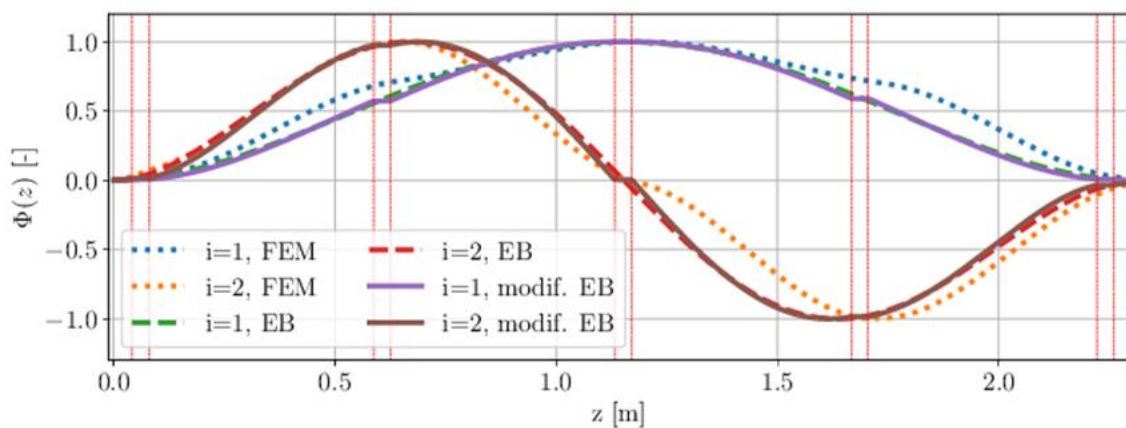


Figure 5-36: First and second mode of the bundle vibration. FEM: cast3m model. EB: Euler-Bernoulli beam theory. modif. EB: Euler Bernoulli beam theory modified to rigidify the bundle at the grid location. The grids positions are represented by thin vertical red lines.

5.3.2 Simulations with imposed vibration

Axial velocity profiles for the single span and the whole domain are presented in Figure 5-37 to Figure 5-39 (10, 50 and 250 mm downstream the mixing grid, respectively). Including the bundle vibration has a minor effect on the average velocity field, regardless of the position. This is likely explained by the low vibration amplitude (5 μm), which is 2 orders of magnitude lower than the spacing between two rods. The same observations hold for the transverse velocity profiles and the rms of the velocity (we therefore choose not to include these figures for brevity). With the present $k - \varepsilon$ RANS model and mesh employed, there is no improvement to the prediction of the velocity by including the bundle vibration. It should nonetheless be noted that a different outcome might occur using a different approach where the mesh cell size near solid boundaries is of the same order of magnitude as the imposed displacement. Such simulations are possible with other RANS turbulence models, or with wall-resolved LES but involve a significantly higher computational cost due to the exponential increase of the mesh size.

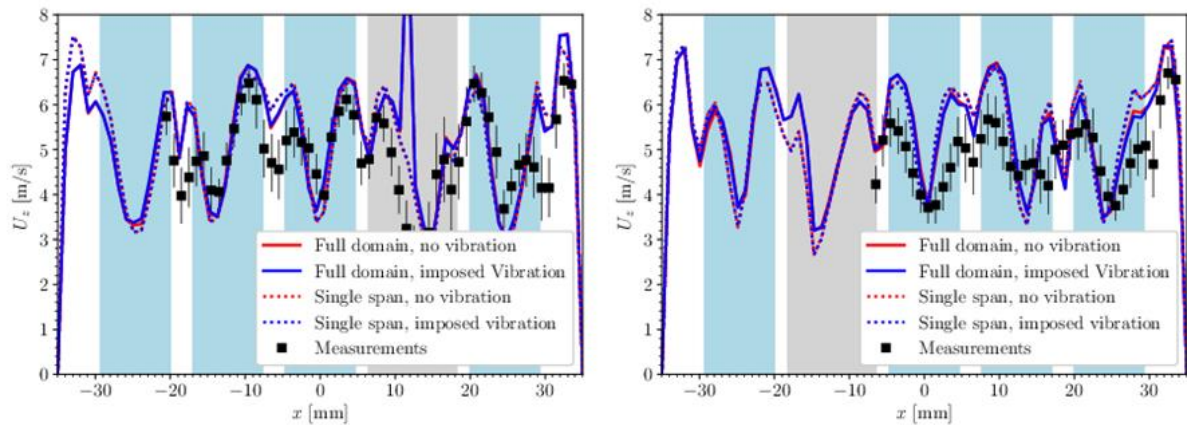


Figure 5-37: Average of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 10 mm downstream of the second grid of the full domain and for the single span computed with the RANS model. The results are shown for the static structure, and with imposed vibration. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

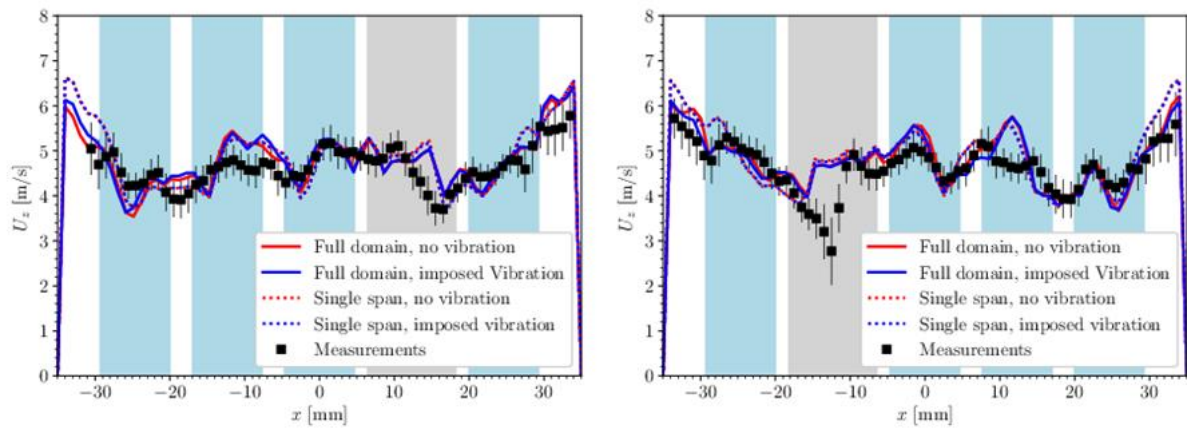


Figure 5-38: Average of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 50 mm downstream of the second grid of the full domain and for the single span computed with the RANS model. The results are shown for the static structure, and with imposed vibration. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

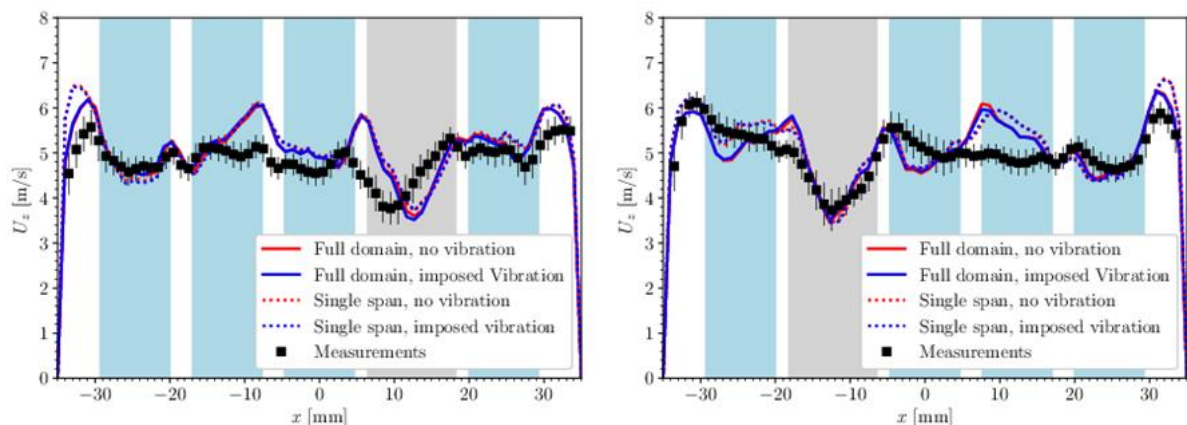


Figure 5-39: Average of the axial velocity along the traverse T1 (left panel) and T2 (right panel) 250 mm downstream of the second grid of the full domain and for the single span computed with the RANS model. The results are shown for the static structure, and with imposed vibration. Black squares denote experimental measurements and error bars materialize $\pm$ one standard deviation.

5.3.2.1 Force acting on the bundle

5.3.2.1.1 Force coefficients for the immobile structure

The force coefficients for the average force acting on two rods of the assembly are presented in Figure 5-40 for the LES model. These rods are the one in the center of the assembly, and one on the side. The force coefficient represents the ratio between the predicted force and the force acting on a hypothetical sphere of diameter d_h (hence of projected area $S = \pi d_h^2/4$), immersed in a fluid of density ρ flowing with axial velocity U_z . Both the total (sum of the pressure and viscous forces) and the pressure forces are represented. Since they do not significantly differ, the contribution of the pressure is significantly larger than the viscous one. There are significant differences between the force coefficients depending on the considered rod and on the axial position. The center rod undergoes a relatively even force over its span, and a slight increase in the coefficient and of its fluctuations is visible in the wake of the grid. In contrast, the mean force coefficient on the side rod exhibits much stronger variations, with two peaks located upstream and downstream of the grid. This difference is ascribed to the asymmetry of the flow around the rod. Upstream of the grid, the force is generated by transverse currents, induced by the grid presence while downstream of the grid, the mixing vanes deflect the flow generating transverse currents and vortices. It is also noteworthy that the magnitude fluctuations of the force do not significantly differ for the two rods, and most of the impact of the rod location is visible for the mean force.

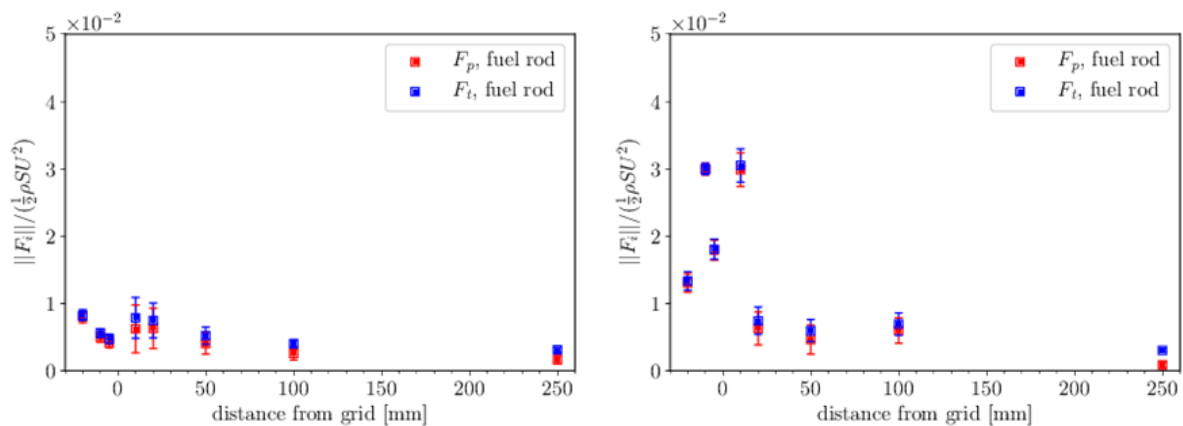


Figure 5-40: Normalized norm of the average force acting on the center (left panel) and an adjacent (right panel) fuel rod when the flow is computed by LES. Several points were added upstream of the grid for informative purpose. Red squares represent the pressure force and blue squares the total force (sum of pressure and viscous forces). Error bars materialize $\pm$ one standard deviation around the average.

The same results for the URANS turbulence model are shown in Figure 5-41 and show that the average force coefficient along the rods is equal to nearly zero. A closer examination reveals that the value predicted by the URANS model is two orders of magnitude lower than the one predicted using LES. This is an unexpected and significant discrepancy, and it was expected that the URANS model was able to capture at least the average component of the force predicted by LES.

One possible explanation is that the size of the mesh cells near the walls is too large in the RANS simulations, leading to a poor prediction of the velocity gradients and pressure near the

walls and thus of the force. In the same vein, the RANS model results in weak rms of the force coefficient in the wake of the grid. This is ascribed to the resolution of the averaged equations, which implies that only temporal fluctuations of the mean field are captured. The instantaneous fluctuations of the velocity and pressure field are not resolved, and their effect is modeled by the closure of the Reynolds stress tensor. Similar results were obtained for the vibrating assembly using the URANS model and are not shown to avoid redundancy.

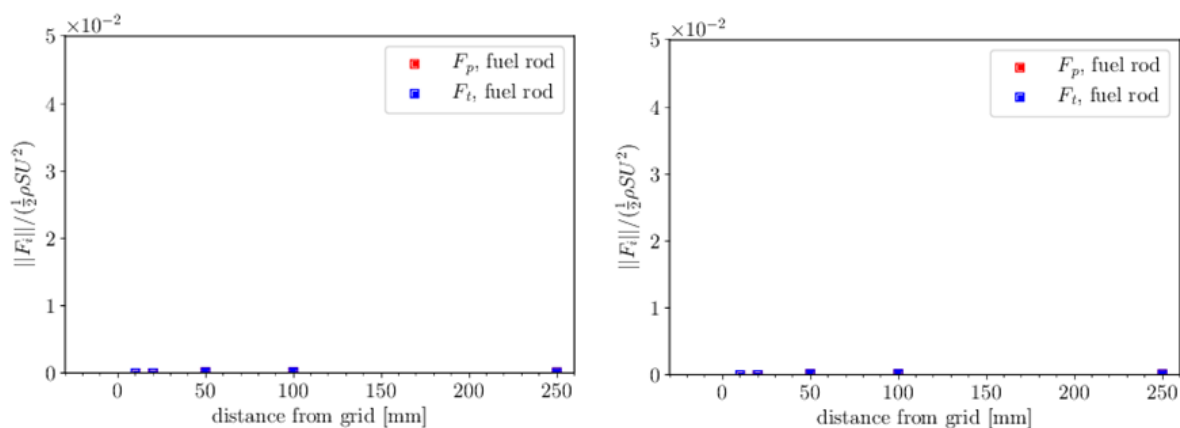


Figure 5-41: Normalized norm of the average force acting on the center (left panel) and an adjacent (right panel) fuel rod when the flow is computed by RANS. Red squares represent the pressure force and blue squares the total force (sum of pressure and viscous forces). Error bars materialize $\pm$ one standard deviation around the average.

In the future, it would be valuable to obtain experimental measurements of the force acting on the assembly rods, which could serve to assess the reliability of different turbulence models and of the mesh cell size near solid boundaries on their numerical prediction.

5.3.2.1.2 Spectral force density for the vibrating assembly

As a complement to the average force coefficient, Figure 5-42 presents the power spectral density of the force acting on the center rod of the assembly, 250mm downstream of the mixing grid. The results are first shown for the assembly vibrating in a quiet fluid for which both the RANS and LES meshes were successfully employed. For such simulations, the turbulence models are turned off and the standard Navier-Stokes equations are appropriate due to the small Reynolds number of the vibration-induced flow. The imposed frequency is 30 Hz and a peak at such a frequency is clearly visible. The peak at the imposed vibration frequency is more pronounced with a fine mesh (LES case) than with a coarse mesh (RANS case). This is ascribed to the smaller size of the cells near the boundaries, which seems more appropriate to capture the effect of the small amplitude displacement on the force. Overall, the signal obtained with the RANS mesh exhibit a larger noise level, and a spurious frequency peak in the low frequency domain. This observation is more pronounced in the transverse than in the axial direction.

Figure 5-43 and Figure 5-44 present the PSD of two components of the force acting on the central rod, 10 and 250 mm downstream of the second grid for the whole assembly domain. The results are shown for the assembly vibrating in quiet fluid, as well as in axial flow at 5.2

m/s. For the relevant cases, the imposed frequency corresponds to the one measured experimentally, namely 6 and 12.5 Hz. The first important result is the imposed vibration frequencies do not stand out clearly in the PSD. A small peak is nonetheless visible around 12.5 Hz, corresponding to the frequency of the second mode, for the cases with imposed vibration. This might be a consequence of a too small computation time (the cumulative simulation time corresponds to two periods at 6 Hz), or due to the large mesh cell size near the boundaries. We believe that the second option is more likely, considering the results already shown in Figure 5-42. Nonetheless, a noticeable coupled effect of the flow and vibration on the magnitude of the PSD (and hence of the force) is visible. The magnitude of the PSD is the lowest when the assembly vibrates in a quiet fluid. It is generally higher for the axial flow and further increases under the coupled effect of imposed vibration and axial flow.

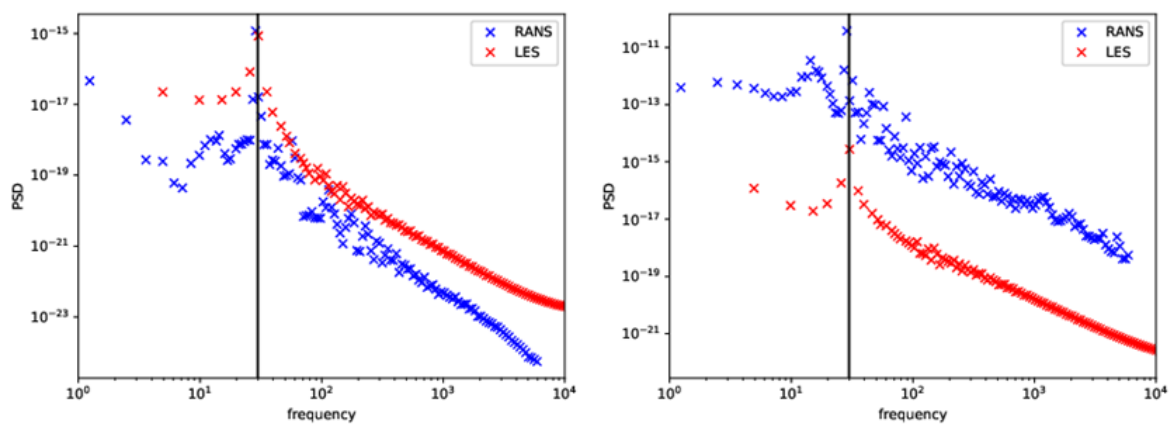


Figure 5-42: Spectral density of the axial (left panel) and transverse (right panel) component of the force acting on the central rod of the single span assembly in quiet fluid, 250 mm downstream of the grid for the single span domain. RANS and LES refer to the mesh employed for the computation. A vertical black line materializes the 30 Hz frequency of the imposed vibrations.

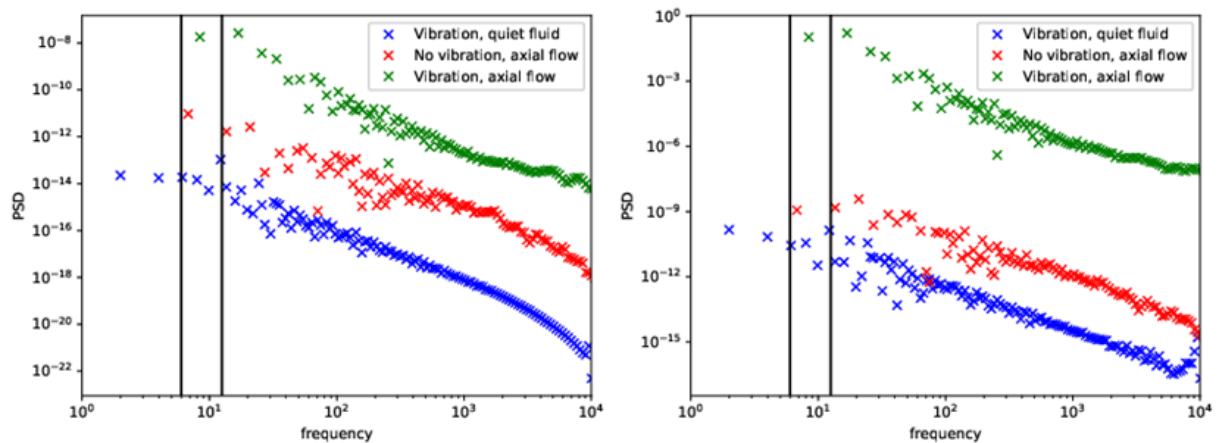


Figure 5-43: Spectral density of the axial (left panel) and transverse (right panel) component of the force acting on the central rod of the assembly in quiet fluid, 10 mm downstream of the second grid for the whole domain. RANS and LES refer to the mesh employed for the computation. Vertical black lines materialize the 6 and 12.5 Hz frequencies of the imposed vibrations.

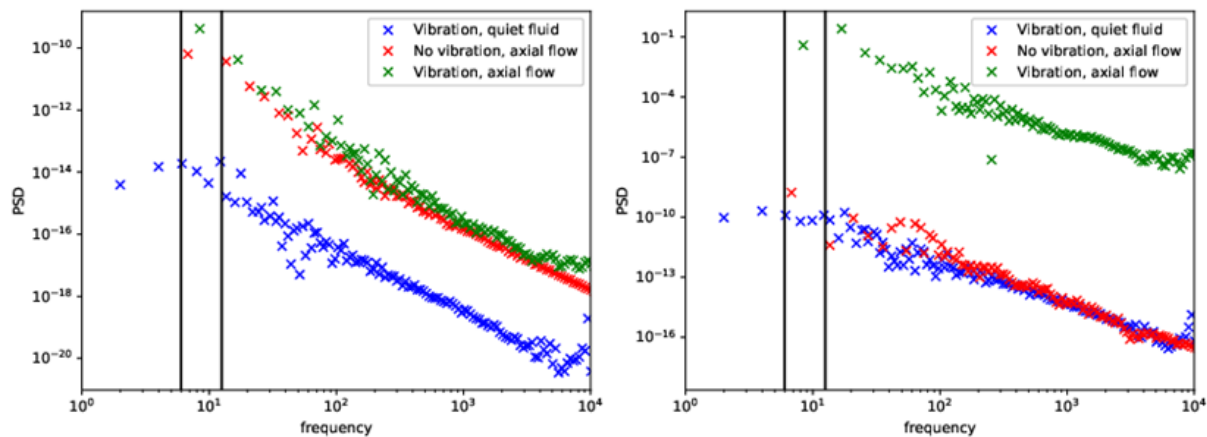


Figure 5-44: Spectral density of the axial (left panel) and transverse (right panel) component of the force acting on the central rod of the assembly in quiet fluid, 250 mm downstream of the second grid for the whole domain. RANS and LES refer to the mesh employed for the computation. Vertical black lines materialize the 6 and 12.5 Hz frequencies of the imposed vibrations.

5.3.2.1.3 Modal added coefficients

As a last element, we present the value of the modal added coefficients (added mass, damping and rigidity) that can be used to model the assembly vibration. The method to determine the coefficients assumes that the force acting on the assembly can be split as the sum of a turbulent contribution, and a linear force due to the vibration. The latter can be expressed as a linear combination of the modal acceleration, velocity and displacement. A detailed procedure employed to determine the expressions of the added coefficients is provided in Appendix 5-1.

The table below summarizes the values of the added modal coefficients (normalized using the domain length, the fluid density, the hydraulic diameter and the vibration frequency) for the two meshes considered and for the single span domain. The value of M and C_0 , determined from simulations in the fluid at rest, exhibit a mesh dependency. The finer (LES) mesh results in a lower value of the coefficients, with an important effect on the added mass. This is likely tied to the mesh cells size near the moving boundaries, which influences the numerical prediction of the force acting on the assembly. It is also noteworthy to mention that the coefficients are the same for the x and y direction for the LES mesh while a slight asymmetry occur with the RANS mesh. The added damping and stiffness coefficients were determined for the assembly in experimental conditions using only the RANS turbulence model. A significant asymmetry of added damping is visible, and added stiffness is two orders of magnitude higher. Unfortunately, it is hard to comment on the reliability of these values since there is only limited experimental knowledge on the subject. The theoretical framework developed in Ricciardi, 2022 [54] for fuel assemblies under axial flow nonetheless predicts that the added-mass and added-stiffness coefficient differ by several orders of magnitudes. This observation supports the validity of the present results but does not guarantee their validity. The values extracted from the present simulations should be taken with caution, as it appears that the mesh convergence has not been achieved with the coarse mesh, which results in an uncertain prediction of the force acting on the assembly.

Table 5-11: Value of the first added modal coefficient computed for the single span domain. The results are given for the two different meshes.

	LES mesh		RANS mesh	
	x_1	y_1	x_1	y_1
M	0.085	0.085	0.533	0.469
C_0	0.285	0.285	0.674	0.641
$C_{\square}$	×	×	6.691	46.00
K	×	×	18.9k	16.6k

5.4 Conclusions

RANS and LES simulations have been carried out to examine the behavior of the flow in the ALAIN experiment with imposed assembly vibration. As a first step, an analysis of the dynamical behavior of the vibrating assembly was undergone. The results reveal that a simple finite element model can be employed to reproduce the experimentally measured vibration frequencies of the assembly with a good accuracy. The mode shape of the vibrating assembly predicted numerically is close to the one predicted by the Euler-Bernoulli beam theory, but the presence of the grids induces a rigidity-like effect. In contrast, the vibration frequency is more difficult to predict, and this effect is attributed to the complex spring-dimple system connecting the guide-tubes, fuel rods and mixing grids. The vibration frequency of the assembly, in contrast, cannot be predicted by assuming that the system behaves as a beam.

When accounted for in CFD simulations of the ALAIN experiment, the assembly vibrations weakly influence the mean flow field predicted by the RANS turbulence modeling approach. This is most likely explained by the small amplitude of the vibration. Some additional constraints in the simulation design, such as the large cell size near solid boundaries, due to the coupling of a $k - \varepsilon$ turbulence model with a law of wall could also have an impact, which could not be precisely assessed. A specific example is the discrepancy between the force acting on the assembly in a quiet fluid predicted for the different meshes. This aspect is expected to influence the prediction of the added modal coefficients and should be examined in detail in the future.

Finally, it was not possible to carry out simulations of the vibrating assembly in experimental conditions using the LES approach due to the prohibitive computational cost involved. There are still significant limitations for generalized use of scale-resolving simulations in an industrial context and for complex multiscale simulations such as the one of the ALAIN. Experiments designed to assess the predictive capability of such turbulence models should therefore be preferentially realized at small Reynolds number to ensure they can be reproduced numerically.

Appendix 5-1: Computation of the added modal coefficients

This section provides more detail on the computation of the modal added mass, damping, and rigidity coefficients from numerical fluid forces. In the following, the structure (namely, the bundle) is represented by B . Using the Fourier decomposition of the displacement, the imposed displacement on B is of the form :

$$u(x, t) = q(t)\Phi(x) \quad (1)$$

where the $u(x, t)$ is the displacement, $q(t)$ the harmonic part and $\Phi(x)$ the mode. Note that to improve clarity, the indices referring to the mode have been omitted in the following. The formulas imply summation of the contribution of all modes relevant to the considered application. We denote by F the fluid forces acting on B and by $\bar{F}$ the modal fluid forces, i.e., the fluid forces F projected onto vibration mode Φ , according to the scalar product

$$\bar{F}(t) = \langle \Phi, F \rangle = \int_{\Gamma_S} \Phi(x) F(x, t) dS \quad (2)$$

We suppose that the modal fluid forces $\bar{F}$ can be expanded as follows

$$\bar{F} = \bar{F}_T + \bar{F}_{lin} \quad (3)$$

where $\bar{F}_T$ refers to a turbulent (random) fluid force, considered as independent of the structure vibrations. This hypothesis implies that turbulence in the vicinity of the boundaries is not affected by the disturbance induced by the moving structure. The term $\bar{F}_{lin}$ refers to the modal linear fluid force acting on B . This latter is assumed to be a linear combination of the modal acceleration d^2q/dt^2 , velocity dq/dt , and position q , i.e.

$$\bar{F}_{lin} = -M \frac{d^2q}{dt^2} - C \frac{dq}{dt} - Kq, \quad (4)$$

With M , C , and K the modal added-mass, damping and rigidity coefficients. It is postulated that the modal added-mass M does not depend on flow velocity, whereas C and K are expanded as

$$C = C^{(0)} + \hat{C}, \quad K = K^{(0)} + \hat{K}. \quad (5)$$

The term $C^{(0)}$ (resp. $K^{(0)}$) is the modal added-damping (resp. stiffness) in a still fluid and $\hat{C}$ (resp. $\hat{K}$) is a complementary damping (resp. stiffness) due to fluid flow velocity. Usually, surface stiffness effects in still fluid are neglected, i.e. $K^{(0)} = 0$, so that Eq. (4) rewrites

$$\bar{F}_{lin} = -M \frac{d^2q}{dt^2} - (C^{(0)} + \hat{C}) \frac{dq}{dt} - K. \quad (6)$$

The aim of the following is to determine the added modal mass M , damping C , and rigidity K coefficients from the fluid force computed from numerical experiments. This can be achieved by following three steps.

First, static simulations (without vibration) are carried out to determine the velocity and pressure field around the structure. This step allows to extract the modal fluid forces F_T . As there is no displacement $u(x, t) = 0$ and therefore $F_{lin} = 0$. In this configuration, the turbulent fluid force is $F_T = F$. The **turbulent modal fluid force** $\bar{F}_T$, is then computed by

$$\bar{F}_T(t) = \int_{\partial S} \Phi(x) F(x, t) dS \quad (7)$$

Second, simulations including structure vibration in a still fluid are carried out. Therefore, the turbulent force is zero, $F_T = 0$, and the numerical modal linear fluid force is $\bar{F}_{lin} = \bar{F}$. A harmonic displacement is imposed, of the form

$$u(x, t) = a \sin(\omega t) \Phi(x), \quad (8)$$

With a the displacement amplitude. Since the fluid is initially at rest, $\hat{C} = 0$ and $K = 0$, and Eq. (6) rewrites

$$\overline{F_{lin}} = -M \frac{d^2 q}{dt^2} - C^{(0)} \frac{dq}{dt}. \quad (9)$$

From Eq. (8) and Eq. (9), the modal linear fluid force is $\overline{F_{lin}}$ writes

$$\overline{F_{lin}} = a \omega (\omega M \sin(\omega t) - C^{(0)} \cos(\omega t)). \quad (10)$$

To determine the modal added coefficients, the Fourier inner product, $[\cdot, \cdot]$, is introduced

$$[f, g] = \frac{2}{T} \int_0^T f(t) g(t) dt \quad (11)$$

With f and g T-periodic functions. Applying the Fourier inner product to modal linear fluid force $\overline{F_{lin}}$ on sine and cosine functions gives

$$\begin{aligned} [\overline{F_{lin}}(t), \sin(\omega t)] &= \frac{2}{T} \int_0^T \overline{F_{lin}}(t) \sin(\omega t) dt \\ &= a \omega^2 M \underbrace{\frac{2}{T} \int_0^T \sin^2(\omega t) dt}_{-1} - a \omega C^{(0)} \underbrace{\frac{2}{T} \int_0^T \sin(\omega t) \cos(\omega t) dt}_{-0} \end{aligned} \quad (12)$$

and

$$\begin{aligned} [\overline{F_{lin}}(t), \cos(\omega t)] &= \frac{2}{T} \int_0^T \overline{F_{lin}}(t) \cos(\omega t) dt \\ &= a \omega^2 M \underbrace{\frac{2}{T} \int_0^T \sin(\omega t) \cos(\omega t) dt}_{-0} - a \omega C^{(0)} \underbrace{\frac{2}{T} \int_0^T \cos^2(\omega t) dt}_{-1} \end{aligned} \quad (13)$$

The modal added-mass M can then be computed using Eq. (12) as

$$M = \frac{1}{a \omega^2} [\overline{F_{lin}}(t), \sin(\omega t)] = \frac{1}{a \omega^2} \frac{2}{T} \int_0^T \overline{F_{lin}}(t) \sin(\omega t) dt \quad (14)$$

And using Eq. (13), the **modal added-damping** $C^{(0)}$ in still fluid can be determined

$$C^{(0)} = \frac{1}{a \omega} [\overline{F_{lin}}(t), \cos(\omega t)] = -\frac{1}{a \omega} \frac{2}{T} \int_0^T \overline{F_{lin}}(t) \cos(\omega t) dt \quad (15)$$

Third, one simulation is carried out where both the fluid flow and structure displacement are accounted for. Building on the previous step. The linear modal force can be calculated by subtracting the turbulent modal force from the total modal force, i.e.

$$\overline{F_{lin}} = \overline{F} - \overline{F_T}. \quad (16)$$

where $\overline{F_T}$ is given by Eq. (7). From Eq. (6) and Eq. (8), $\overline{F_{lin}}$ writes

$$\overline{F_{lin}}(t) = (a \omega^2 M - a K) \sin(\omega t) - a \omega (C^{(0)} + \hat{C}) \cos(\omega t) \quad (17)$$

Once again, the Fourier inner product is applied to the modal linear fluid force $\overline{F_{lin}}$ on sine and cosine functions

$$[\overline{F_{lin}}(t), \sin(\omega t)] = (a\omega^2 M - aK) \underbrace{\frac{2}{T} \int_0^T \sin^2(\omega t) dt}_{-1} - a\omega(C^{(0)} + \hat{C}) \underbrace{\frac{2}{T} \int_0^T \sin(\omega t) \cos(\omega t) dt}_{-0} \quad (18)$$

$$[\overline{F_{lin}}(t), \cos(\omega t)] = (a\omega^2 M - aK) \underbrace{\frac{2}{T} \int_0^T \sin(\omega t) \cos(\omega t) dt}_{-1} - a\omega(C^{(0)} + \hat{C}) \underbrace{\frac{2}{T} \int_0^T \cos^2 dt}_{-0} \quad (19)$$

The modal added-rigidity K , and the modal added-damping C are then computed from Eq. (18) and Eq. (19)

$$K = \omega^2 M - \frac{1}{a} [\overline{F_{lin}}(t), \sin(\omega t)] = \omega^2 M - \frac{1}{a} \frac{2}{T} \int_0^T \overline{F_{lin}}(t) \sin(\omega t) dt \quad (20)$$

$$C = \hat{C} + C^{(0)} = -\frac{1}{a\omega} [\overline{F_{lin}}(t), \cos(\omega t)] = -\frac{1}{a\omega} \frac{2}{T} \int_0^T \overline{F_{lin}}(t) \cos(\omega t) dt, \quad (21)$$

Where M is given by Eq. (14).

6 Simulations performed by GRS

In this chapter the numerical simulations of the ALAIN benchmark performed by GRS and results of these simulations are described and presented in detail.

6.1 Setup of CFD simulations

GRS performed CFD simulations of one span of the fuel element which contained one spacer grid and half of the length of the fuel element between two spacer grids in both directions up-stream and down-stream of the spacer grid (see Figure 6-1).

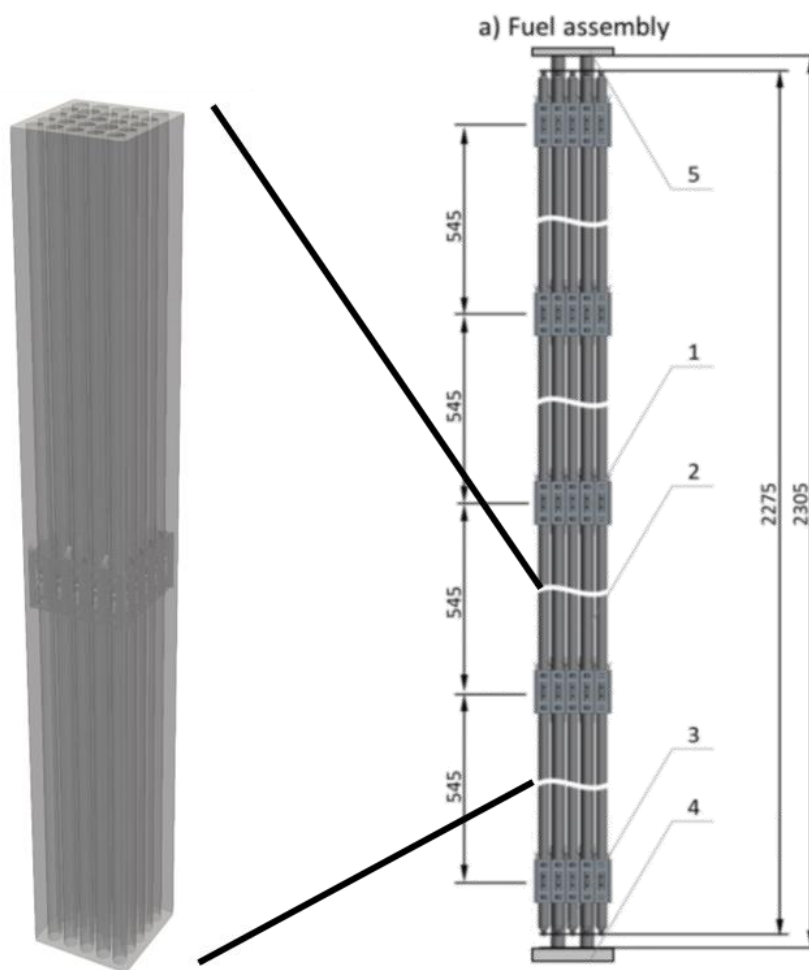


Figure 6-1: Sketch of a single span used in the GRS CFD simulations.

Two different meshes were used for the CFD simulations. The first one was generated by GRS using the OpenFOAM (version 11) tool snappyHexMesh. The second one was provided by Framatome generated using the STAR-CCM+ meshing tool. Table 6-1 summarizes the number of cells and their distribution over the different cell types. The Framatome mesh contained about six times more cells than the GRS mesh. As will be shown later, no significant differences were found between the results of the two meshes, regarding the investigated quantities like

mean velocities and RMS values at the positions where values were measured in the experiment. Therefore, for most of the simulations, the smaller GRS mesh was used. For both meshes the OpenFOAM tool checkMesh showed a satisfying quality.

Table 6-1: summary of the mesh characteristics as output by the checkMesh tool

GRS mesh		Framatome mesh	
Cells:	5,080,663	Cells:	33,058,325
Overall number of cells of each type:		Overall number of cells of each type:	
hexahedra:	4,457,567	hexahedra:	30,710,134
prisms:	169,392	prisms:	1,139,563
wedges:	11	wedges:	31,561
pyramids:	0	pyramids:	3,538
tet wedges:	667	tet wedges:	3,222
tetrahedra:	75	tetrahedra:	33,751
polyhedra:	452,951	polyhedra:	1,136,556

All simulations were preformed using OpenFOAM 11 for an incompressible fluid with a kinematic viscosity of $\nu = 6.5803 \cdot 10^{-7} \text{ m}^2/\text{s}$.

All simulations used in comparisons/the benchmark were transient simulations using a fixed time step size of 0.25 ms, resulting, for the GRS mesh, in a mean Courant number of 0.8 and a maximal value of about 66. A comparison with a simulation with a reduced time step of size 0.01 ms showed no influence on the results.

The results were statistically analyzed for the last 0.25 s of simulations taking at least 1 s to compute mean and root mean square values. The length of the flow domain was 0.545 m and the mean flow velocity 5.18 m/s resulting in a mean throughput time of $\approx 0.1 \text{ s}$, so enough time was given in the simulation to reach quasi-steady conditions.

Table 6-2: boundary conditions of the CFD simulations

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

The flow in the domain was flowing upwards. The simulations used a cyclic setup meaning the outflow of the computational domain at the top was used as inflow at the bottom of the domain (see Table 6-2). To match the fluid flow rate measured in the experiment, a special boundary condition was implemented for the pressure at the bottom of the flow domain. This new boundary condition imposed pressure values at the bottom using the pressure values at the top and adjusted an additional pressure difference in a way that the resulting flow showed the correct flow rate. For this additional pressure increase the same value was used across the whole bottom inlet.

Different unsteady Reynolds-averaged Navier-Stokes (URANS) type simulations using different 2 and 6 equation turbulence models were performed. Also, a “coarse-mesh” LES was performed, using the GRS mesh for it.

For the turbulent quantities boundary conditions were used which adaptively modelled the flow for both, the laminar and the turbulent zones of the boundary layer. The y^+ -values in the URANS simulations for two equation models were typically in the range of 0.5 to 190. These values are consistent with using URANS simulations with wall functions. However, these values clearly show that the LES simulation was done using a mesh which was too coarse compared to the theoretical requirement, which is $y^+ < 1$.

For the pressure and momentum equations, second order numerical schemes were selected, for the turbulence quantities - first or second order depending on the simulation (see Table 6-3).

Table 6-3: numerical schemes

```

ddtSchemes
{
    default          CrankNicolson 0.3;
}
gradSchemes
{
    default          cellLimited Gauss linear 0.5;
    grad(U)          cellLimited Gauss linear 1;
}
divSchemes
{
    default          none;
    turbulence        linearUpwind default; // upwind;
    div(phi,U)        Gauss linearUpwindV grad(U);
    div(phi,k)         Gauss $turbulence;
    div(phi,epsilon)   Gauss $turbulence;
    div(phi,omega)     Gauss $turbulence;
    div(phi,v2)        Gauss $turbulence;
    div((nuEff*dev2(T(grad(U)))) Gauss linear;
}
laplacianSchemes
{
    default          Gauss linear limited 0.5;
}
interpolationSchemes
{

```

```

    default      linear;
}
snGradSchemes
{
    default      limited 0.5;
}

```

All URANS simulations showed no transient behavior, but the velocity and the turbulence quantities reached steady values. This is to be expected if the flow shows only turbulent, e.g. random fluctuations. Then all these fluctuations should be represented by the quantities of the turbulence model, in this case the turbulent kinetic energy. Only if the flow shows macroscopic fluctuations like in a configuration with vortex shedding, the URANS model might be able to reproduce this in the time-dependent mean quantities.

To compare the measured RMS values of the velocity with the simulation, an effective RMS velocity was calculated using the turbulent kinetic energy at the measurement positions using equation (6.1):

$$v_{RMS,k} = \sqrt{\frac{2}{3}k} \quad (6.1)$$

6.2 CFD simulation results

A set of different CFD simulations were performed to determine the influence of different settings as mentioned in the previous subchapter. The results were compared to the experimental data which consisted in mean and RMS values of the velocity at different measurement positions. The velocities were measured along the flow direction and in one direction perpendicular to the flow (see section 3.1).

6.2.1 Mesh sensitivity study

To assess a possible mesh dependency of the results, simulations using the SST turbulence model with first order discretization were compared. In Figure 6-2 and Figure 6-3 the mean velocities 10 mm downstream of the spacer grid are compared. They show a very good agreement of the simulations with the measurements for both grids. There are some differences visible between the two simulations, but these differences are small compared to the turbulent fluctuations observed in the measurements. Comparisons of the traces further downstream of the spacer grid show even smaller differences. Therefore, the small grid was used for further simulations.

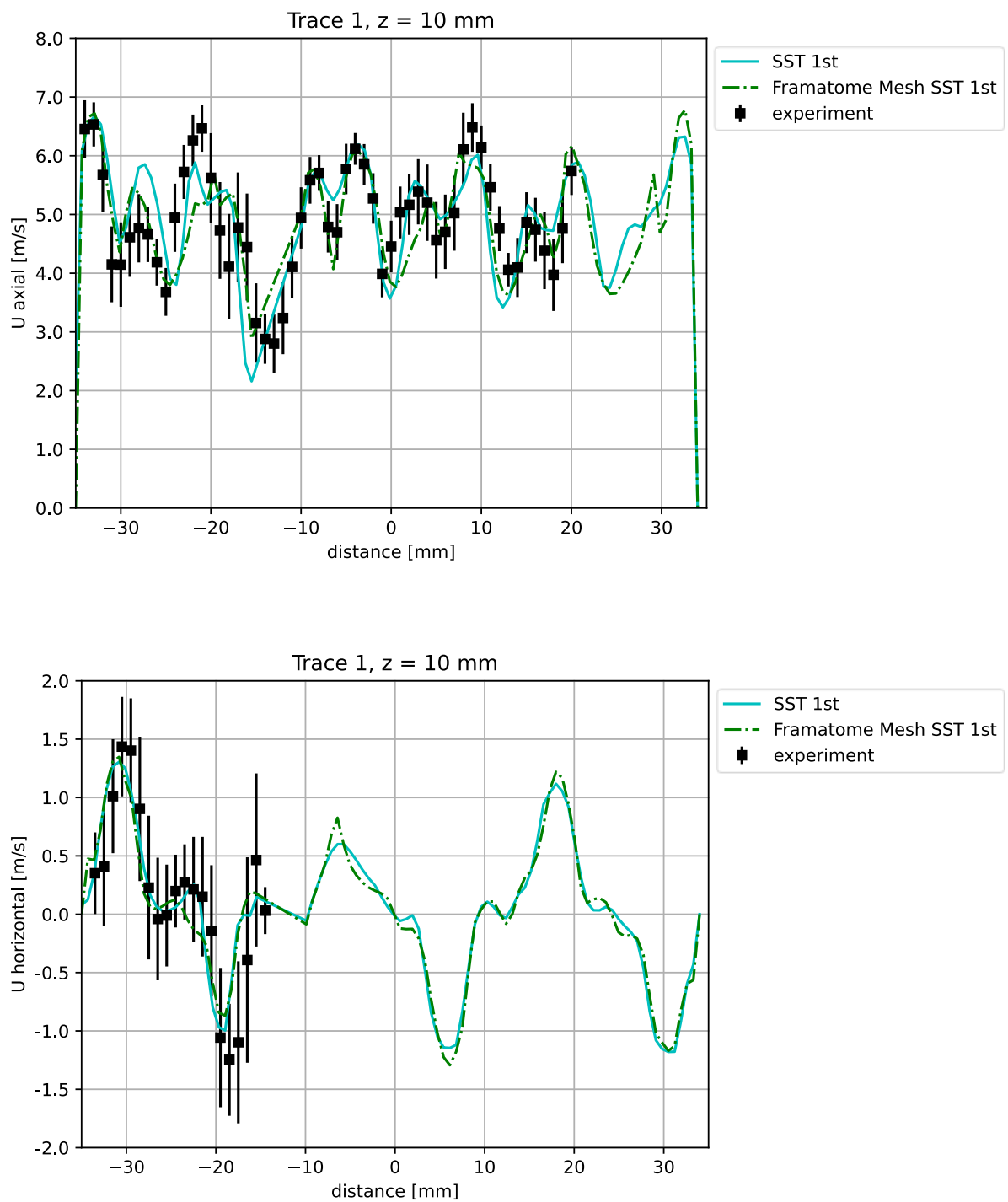


Figure 6-2: Mean velocities on trace 1 at 10 mm downstream of spacer grid using different meshes.

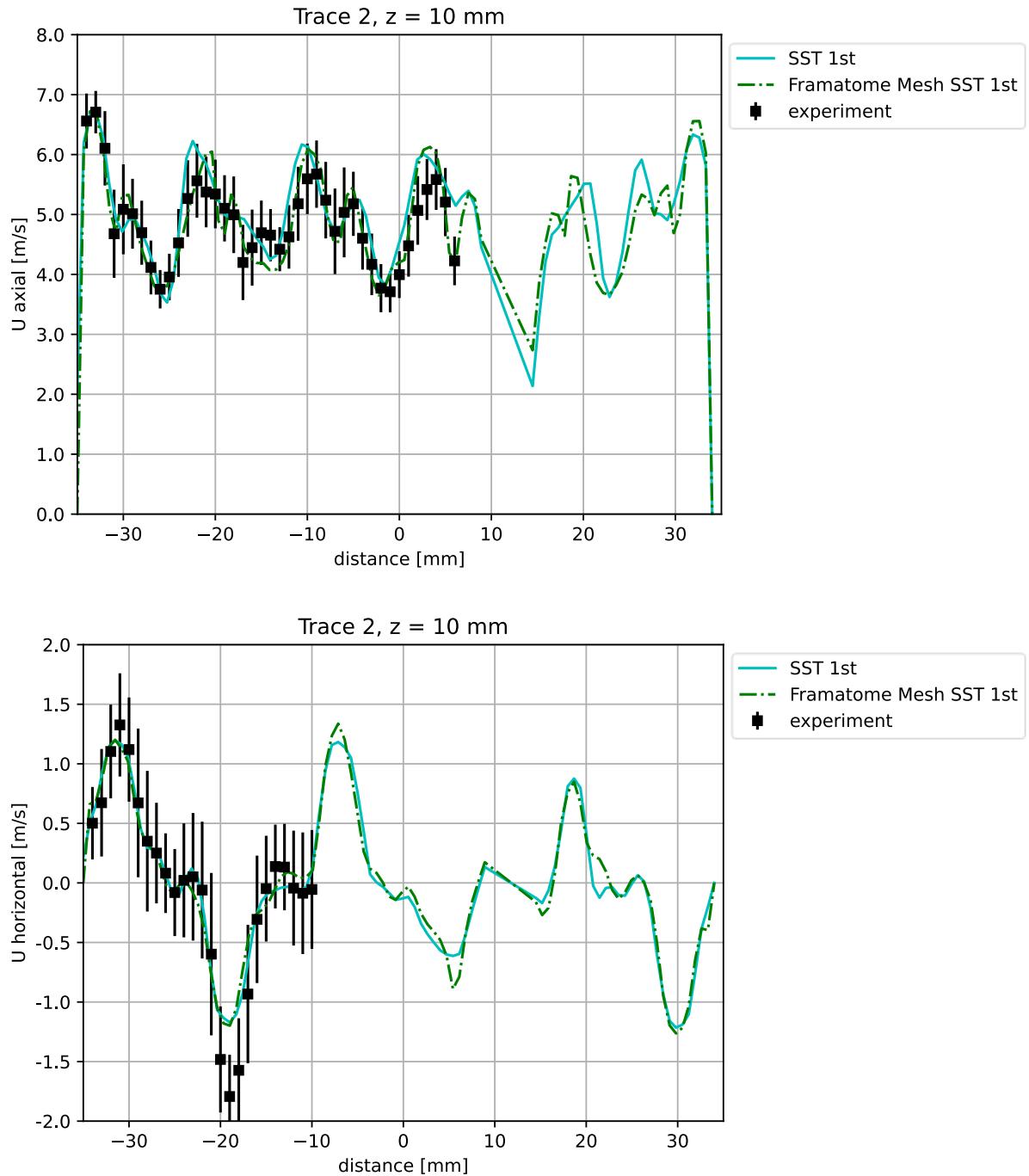


Figure 6-3: Mean velocities on trace 2 at 10 mm downstream of spacer grid using different meshes.

6.2.2 Discretization sensitivity study

To assess the influence of the discretization, the velocities of two simulations using the SST turbulence model with 1st and 2nd order discretization are compared in Figure 6-4. The simulation results show nearly perfect agreement. The same was observed for the other measurement positions. This also demonstrates some kind of mesh independence because otherwise there should be differences between the two simulations. For most of the remaining simulations, first order discretization was used, because it requires less computational resources.

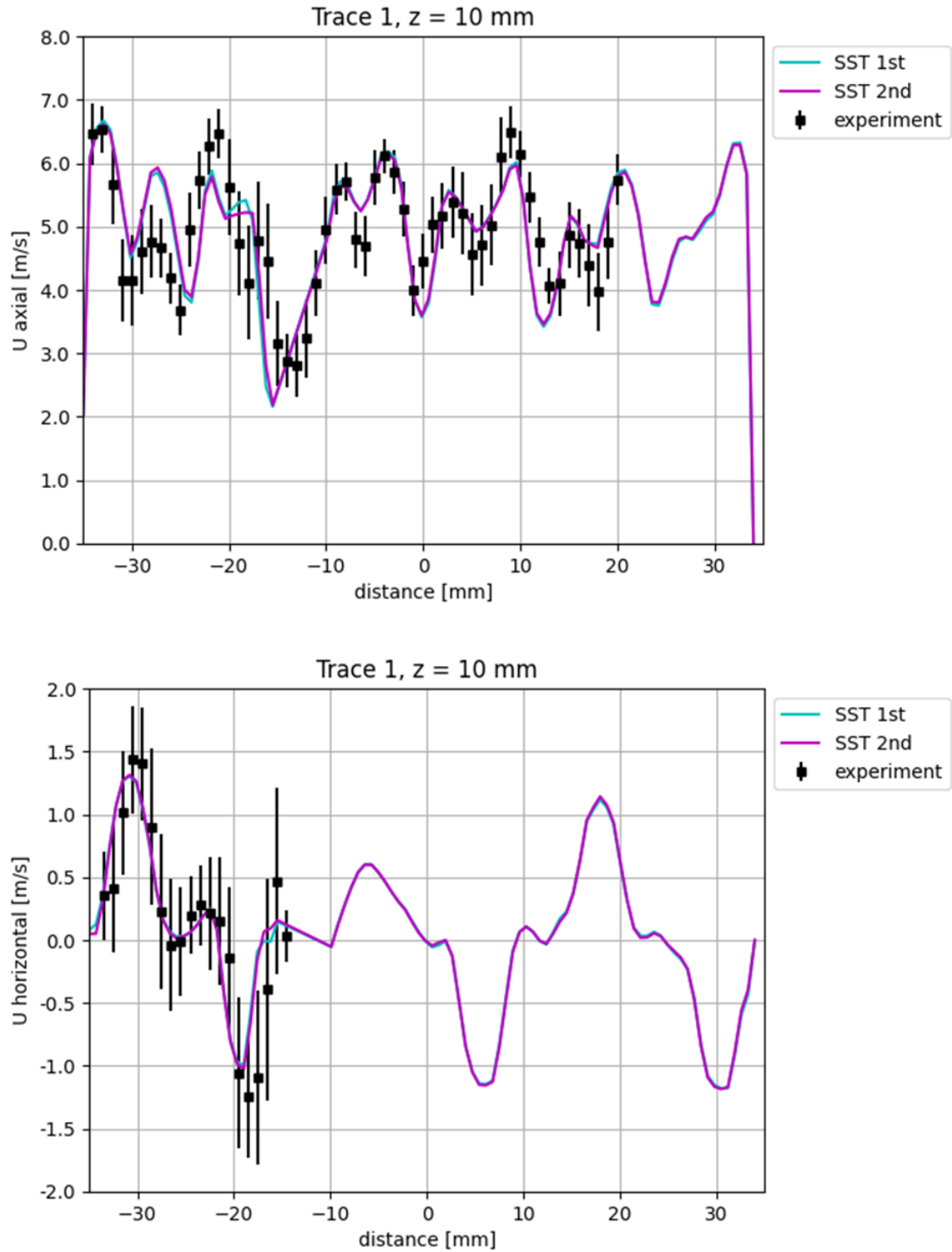


Figure 6-4: Mean velocities on trace 1 at 10 mm downstream of spacer grid using different discretization.

6.2.3 Time step size sensitivity study

Figure 6-5 shows three simulations using the time step size, specified in section 6.1 (0.25 ms), a fifth of it (0.05 ms) and a 25th of it (0.01 ms). There is a nearly perfect agreement between the three simulations. The same was observed for the other measurement positions. This demonstrates that the selected time step size has no influence on the results.

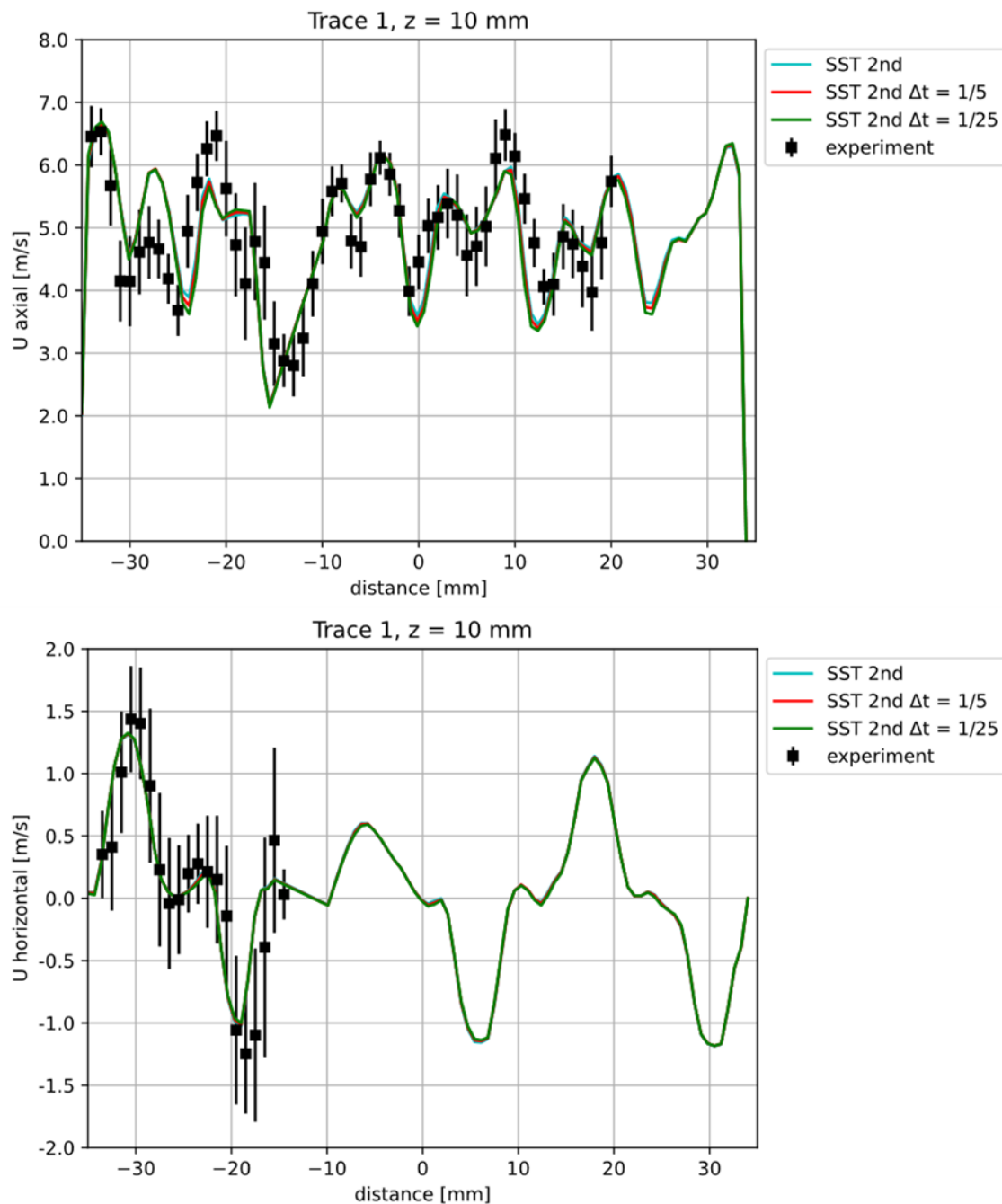


Figure 6-5: Mean velocities on trace 1 at 10 mm downstream of spacer grid using different turbulence models

6.2.4 Turbulence model sensitivity study

Figure 6-6 shows five simulations using different two and six equation turbulence models: The two linear models $k-\omega$ Shear Stress Transport (SST) model and the realizable $k-\epsilon$ model (realizableKE), the non-linear Lien cubic $k-\epsilon$ model (LienCubicKE), the Reynolds stress model Launder, Reece and Rodi (LRR) and a simulation using the coarse LES approach described above. There is a nearly perfect agreement between the four simulations using two/six equation turbulence models. The results of the coarse grid LES model show some differences to the other simulation results. Still, those differences are within the size of the fluctuations observed in the experiments. The same was observed for the other measurement positions.

This demonstrates that the selected turbulence model has little to no influence on the results of the mean quantities.

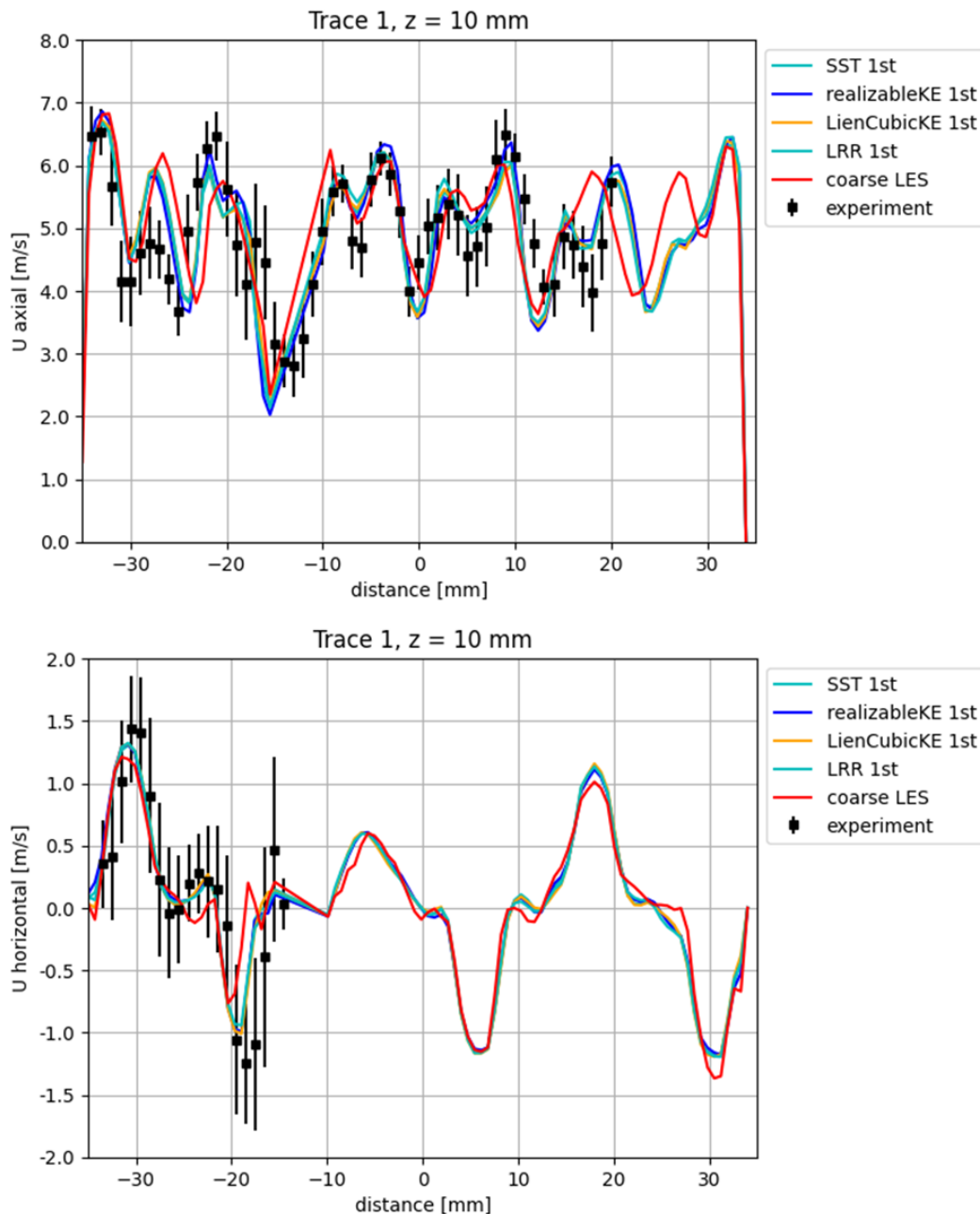


Figure 6-6: Mean velocities on trace 1 at 10 mm downstream of spacer grid using different turbulence models.

6.2.5 Turbulent quantities/velocity fluctuations

The fluctuating pressure values integrated on the surface of the fuel rods, guide tubes and spacer grids together with the wall shear stresses are the main input provided by the CFD simulations for the mechanical simulations (see section 6.3). To validate those values, the measured velocity fluctuations at the given measurement positions were used.

Figure 6-7 shows the RMS values calculated from the velocity fields in the same five simulations with different turbulence models, described in section 6.2.4. For the two-equation models, the RMS values were nearly zero. For the LRR Reynolds stress model, velocity fluctuations could be seen in the plots, but their values were at least one order of magnitude too low. The results provided by the coarse mesh LES simulation showed the correct order of magnitude, but the local features could not be resolved correctly. The same observations were found in the comparison of the simulation results for the other measurement positions.

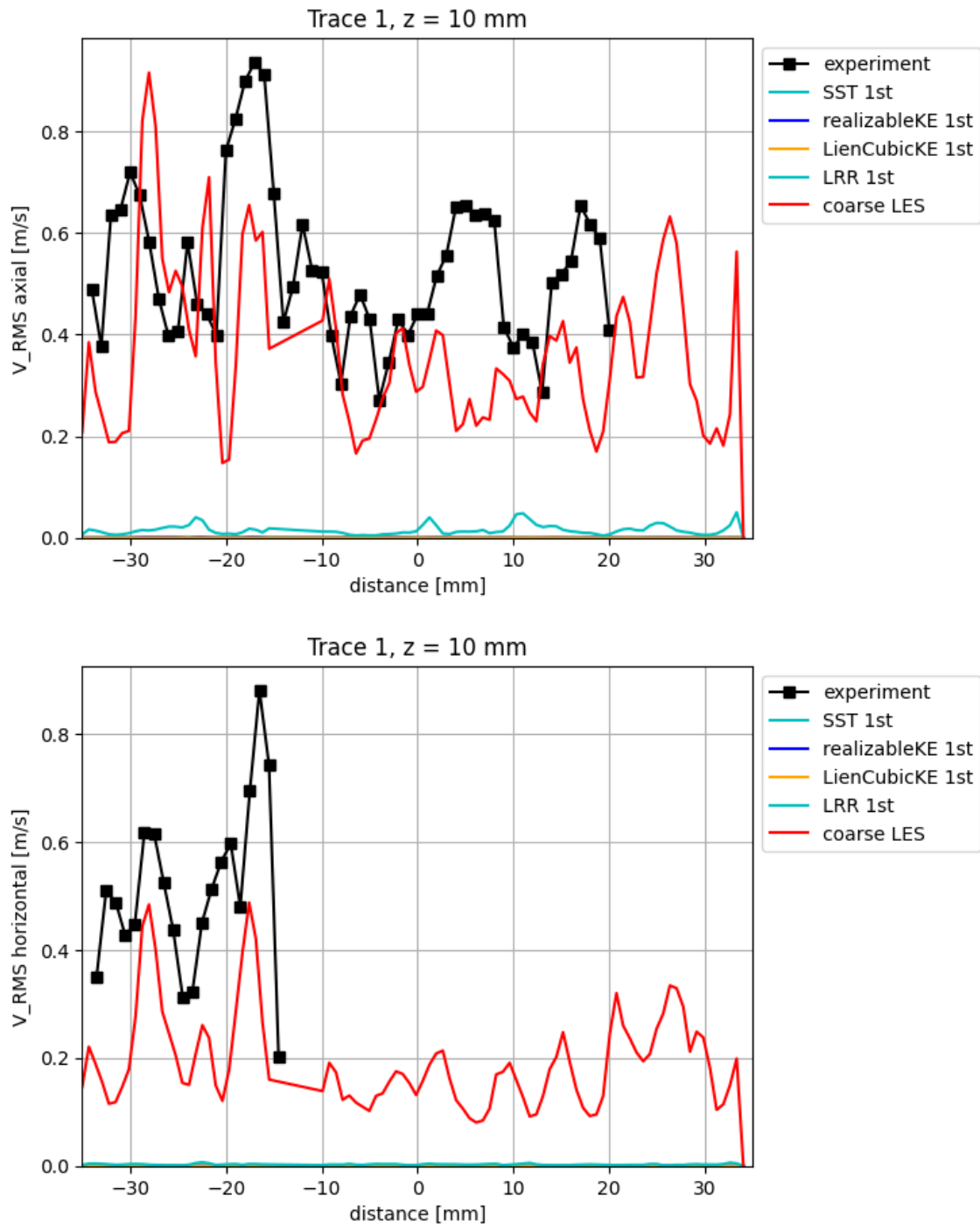


Figure 6-7: RMS values of velocities on trace 1 at 10 mm downstream of spacer grid using different turbulence models.

In URANS simulations, the fluctuations of the velocities should be represented by the turbulent kinetic energy. Therefore, RMS values calculated by equation (6.1) were compared with the experimental results for the URANS simulations in Figure 6-8. The results of the SST and the realizable k- ϵ model are very similar, whereas the Lien Cubic k- ϵ results in values, about half the values of the other two models. The first two models underestimate the velocity fluctuations by a factor of about one half, the latter model by a factor of one quarter, consequently. So, all URANS models result in too low velocity fluctuations.

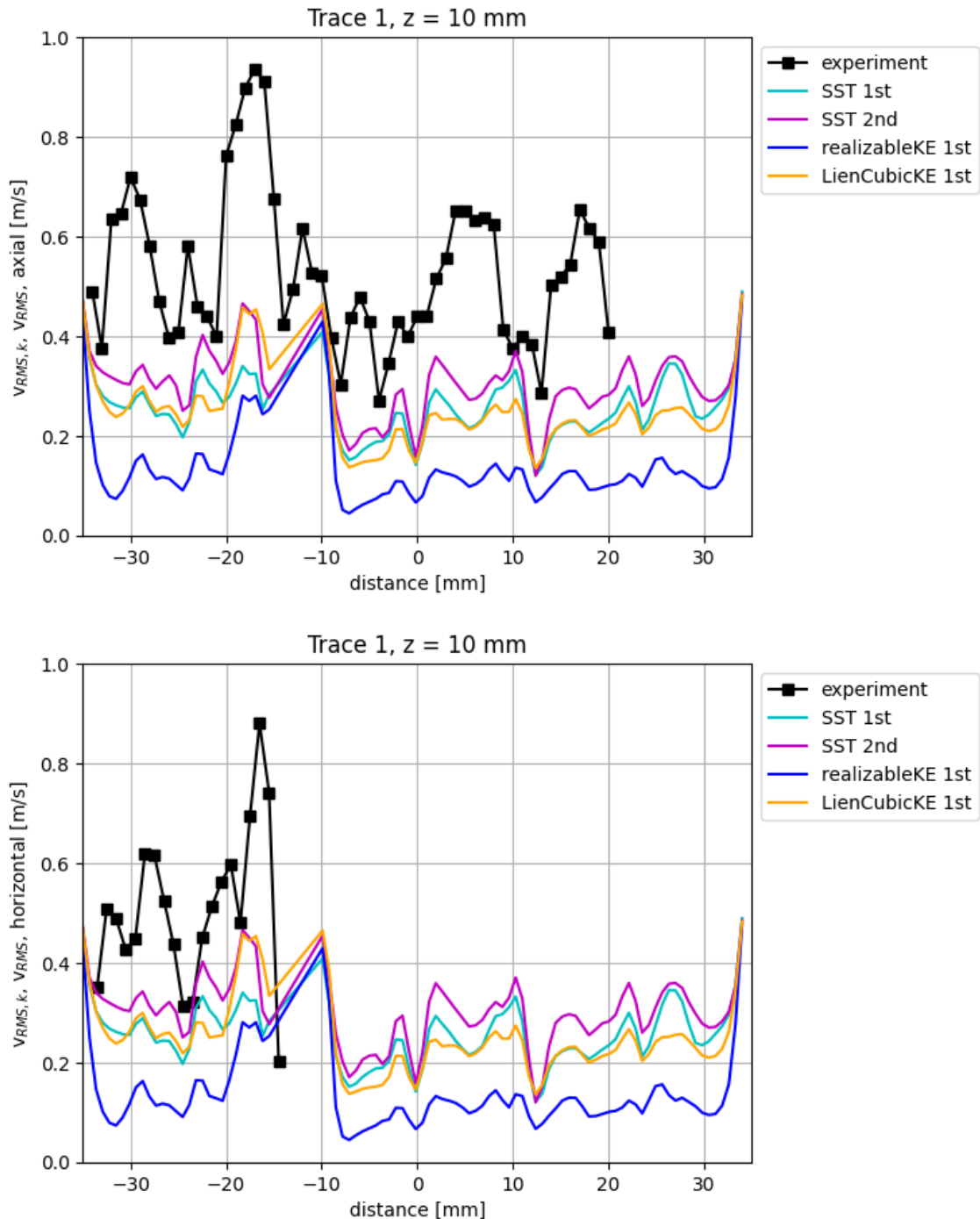


Figure 6-8: Calculated RMS values of velocities on trace 1 at 10 mm downstream of spacer grid using different URANS models compared with the experiment.

The agreement between the models and the experiment seemed to get better further downstream (see Figure 6-9). The level of the velocity fluctuations averaged along the traces seemed to be predicted in the correct order of magnitude by the simulations at the measurement positions the most downstream. Still differences were observable, showing that local features were not predicted correctly.

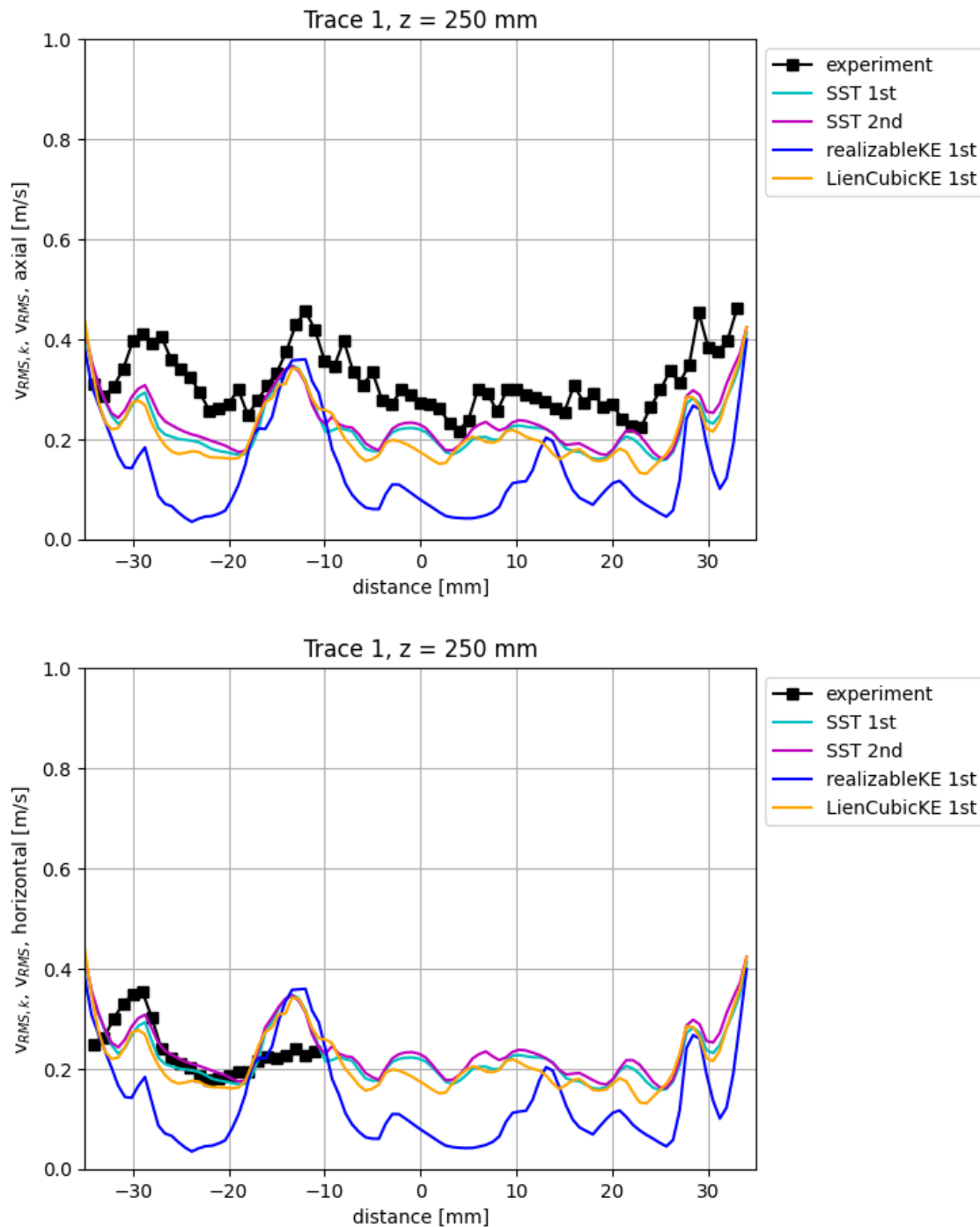


Figure 6-9: Calculated RMS values of velocities on trace 1 at 250 mm downstream of spacer grid using different URANS models compared with the experiment.

6.2.6 Model selection for benchmark/input for mechanical simulations

For the benchmark (see section 8) the results of the SST model with 2nd order model numerics and the coarse grid LES simulations (1st order) were provided.

For the mechanical simulations, the coarse grid LES model was selected. No coupled simulations were performed, because the expected mechanical oscillation amplitudes were several orders of magnitude smaller than the grid resolution, so no effect of the oscillations were expected on the results of the fluid simulations.

The OpenFOAM function object `forcesIncompressible` was used to calculate the forces and momenta on the structures. This function object uses the local pressure values on the surface and also the wall shear stresses to calculate the force and momentum. The values were stored as comma separate values (CSV) and provided for use in the mechanical simulations.

6.3 Mechanical simulation

6.3.1 Model description

To compute the mechanical reaction of the test fuel assembly, a mechanical model is implemented in the FE code ANSYS Mechanical (Figure 6-10). Based on a prior model developed in [55], it consists of a single beam, representing the whole fuel assembly. As idealized boundary conditions, the lower end is fixed, while the upper end is guided laterally. At the height of each spacer grid, pairs of torsion springs and damper elements are implemented in X- and Y-direction using modified COMBIN14 elements, mimicking the stiffening effect of the spacer grids as well as the fluidic damping. The beam is discretized into 220 elements and discretization studies were successfully performed.

Mechanical parameters of the beam are determined in consecutive steps. A first set of parameters is calculated based on geometry and material data. Cross section area and moments of inertia are calculated by adding corresponding values for the guide tubes and fuel rods. Parallel axis terms are not considered, as the fuel rods are movable within the spacer grids and the fixation of the guide tubes ends is not perfectly stiff. Density is calculated by dividing the total mass by the product of beam length and the calculated cross section area. Torsion stiffness is set arbitrarily high.

In a second step, the model is iteratively fitted to the first three natural frequencies and mode shapes according to the experiment by tuning the moment of inertia and the individual torsion springs using a linear eigenmode analysis (Table 6-4).

In a next step, damping effects are considered. Structural damping originates from material damping, bolting and the friction hysteresis. To obtain the structural damping ratio, impact tests in air were recalculated. In the simulation, an excitation is provided by an initial deflection, which is tuned to match the level of the deflections. A best fit is achieved for a damping ratio of $D = 0.005$ (logarithmic decrement $\Lambda = -0.03$) (Figure 6-11). This is in line with similarly low values reported in [56], [57], [58], [59] and [60], especially for small deflections.

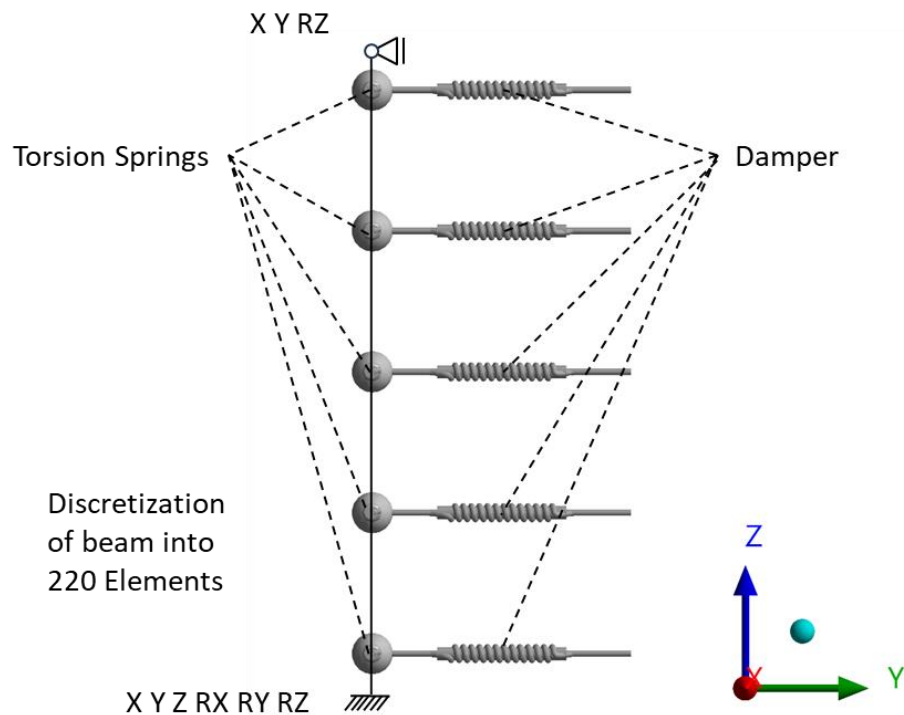


Figure 6-10: Schematic representation of the mechanical model

Table 6-4: Natural frequencies and mode shapes after fitting

	1st mode	2nd mode	3rd mode
Natural frequency experiment, Hz	6	14.5	23.9
Natural frequency model, Hz	5.97	14.9	23.55
Mode shape			

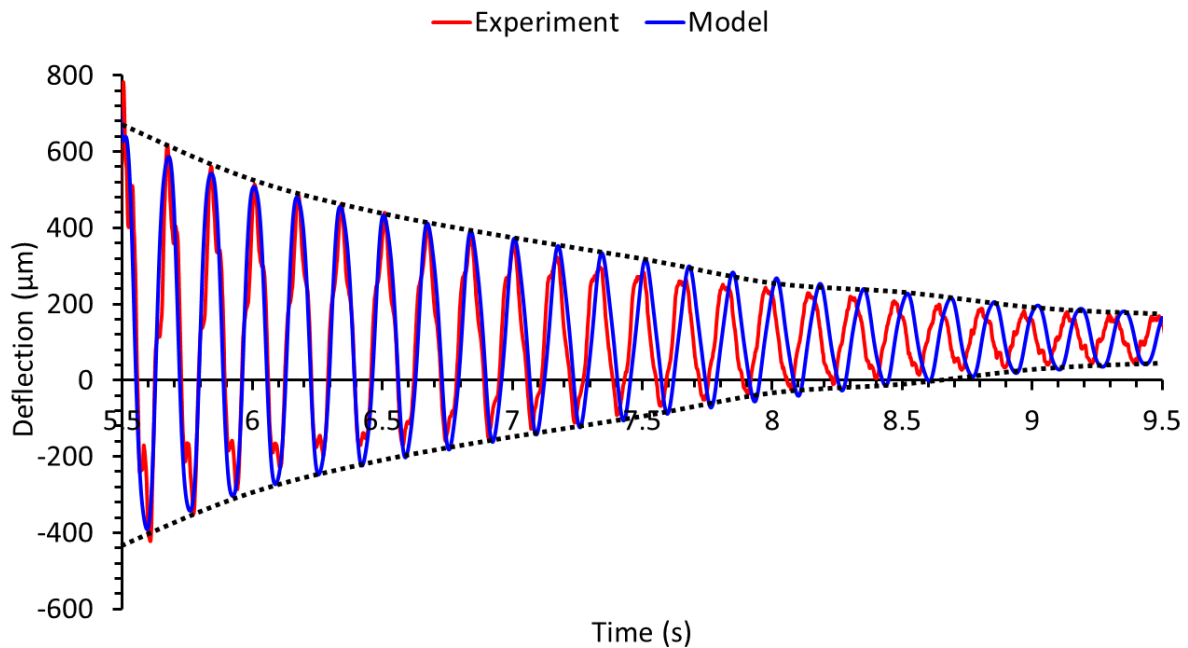


Figure 6-11: Recalculation of the lateral deflection of SG2 in the impact test.

However, fluidic damping is expected to dominate the overall damping ratio. The main components of fluidic damping are viscous damping of the surrounding fluid and flow-dependent damping. While the first one is caused by drag effects and can be also observed in still fluid, the latter arises from a lift phenomenon due to axial flow velocity relatively to the assembly lateral velocity [58]. As experimental data for a fitting of the fluidic damping are not available and fluidic damping is not considered through bidirectional coupling, values for the damping ratio are taken from experiments on real and mock-up fuel assemblies under axial flow on shaker tables, which were performed in the framework of seismic analyses (Figure 6-12). To be able to evaluate sensitivity from damping, two values $D = 0.5$ and $D = 0.8$ are taken. The model is iteratively adapted in a damped modal analysis to meet these values. As the damping is concentrated in the spacer grids and the damping ratio is slightly different for every vibration mode, the first mode is taken as reference. Finally, the beam stiffness is adapted once more, as the additional damping has a slight effect on the natural frequencies. Further influences of the fluid in terms of added mass and added stiffness are neglected as the above mentioned tests on shaker tables suggest moderate effects [58], [59], [61], [62], [63], [64], [65].

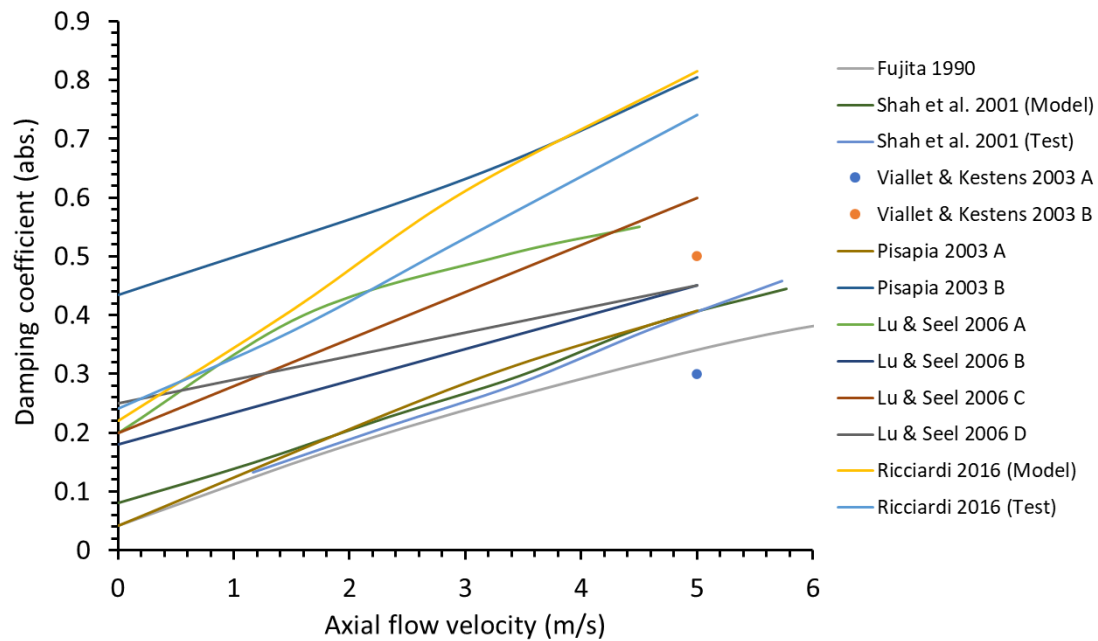


Figure 6-12: Values for fluid damping measured on shaker table tests for real fuel assemblies under different axial velocities in [58], [59], [60], [64], [66] and [67].

6.3.2 Load application

Since results from the CFD analysis are available only for one section and about one second, data must be extended before it can be applied to the mechanical model. As the calculated fluidic forces concentrate at the spacer grid, they are added to one sum force. The static part of the force, i.e. the average value, is subtracted to eliminate a transient effect at the beginning. The calculated force is then split into shorter intervals (cut at zero crossing), recomposed and distributed to the full mock-up for 10 seconds based on random patterns (Figure 6-13).

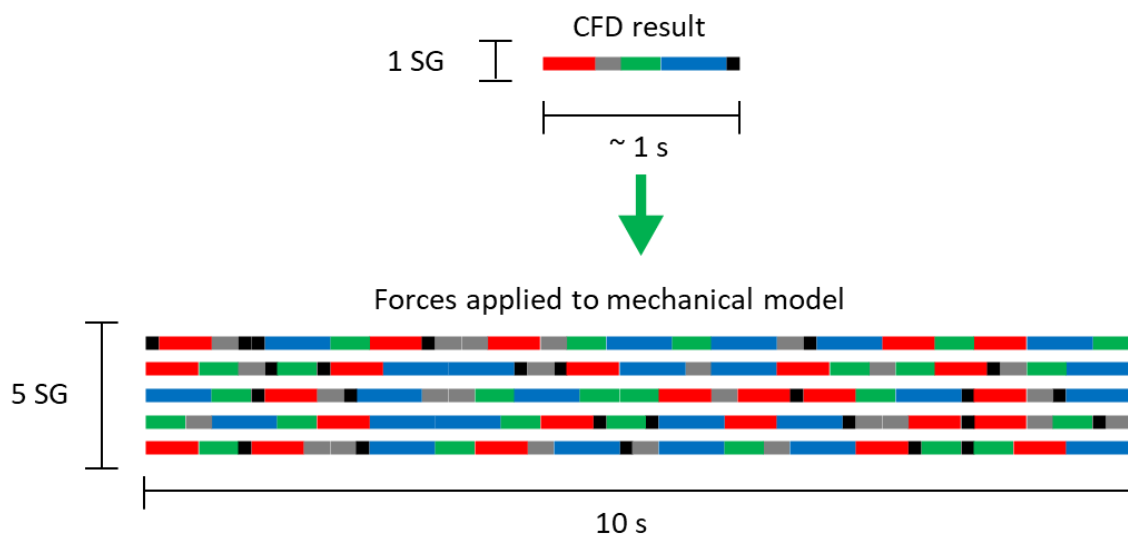


Figure 6-13: Visualization of the data extension process

In a separate static mechanical calculation using the average force values, a deflection of 0.013 mm in X-direction and -0.428 mm in Y-direction can be found at the height of spacer grid 2. The maximum deflection amounts for 0.014 mm and -0.457 mm respectively.

6.3.3 Results

APSDs of lateral deformations of SG2 in X/Y-direction are shown in Figure 6-14 for a damping of $D = 0.5$ and in Figure 6-15 for a damping of $D = 0.8$. The general level and shape of the APSD match quite well for the simulation with damping $D = 0.5$. A peak can be clearly identified for the first natural frequency. Higher natural frequencies are not distinctly identifiable. The peak, which can be seen around 33 Hz in the experiment is most likely not associated to bending vibration and cannot be resolved by the current model. The peaks in the interval 0-4 Hz, which are not found in the experiment, could be artefacts from the data extension approach applied. The RMS values amount for 6.66 μm /7.04 μm in X/Y direction for the simulation with damping $D = 0.5$ and 2.95 μm /3.1 μm for the simulation with damping $D = 0.8$.

Comparing the RMS values with values obtained from fuel assemblies in actual plants, they seem reasonable considering the smaller dimensions of the experiment and additional fluid forces in the real application. [68] obtained an RMS of 50 μm by comparing the neutron flux fluctuations of intact and broken fuel assemblies. [69] and [70] obtained 35 μm using actual accelerometer measurements with miniature in-core probes in a KWU 350 MWe class PWR. Lastly, [71] obtained 3.47 μm by reconstructing fuel assembly oscillation from ex-core detector neutron noise at Sequoyah 1 PWR.

In summary, the analyses show that the presented simulation chain can reproduce the main vibration phenomena observed in the experiment, such as mean displacement amplitude, APSD shape and lower natural frequencies. The low deflection amplitudes confirm the premise that a unidirectional uncoupled analysis is applicable here. However, to resolve additional phenomena, such as the internal vibration modes of the fuel assembly, further developments could be necessary in the future. Improvements could be also achieved in the fluid domain by applying finer, LES-grade meshes and larger models (not just a single span) that will avoid the artificial data extension. All this, of course, will increase the computational effort dramatically.

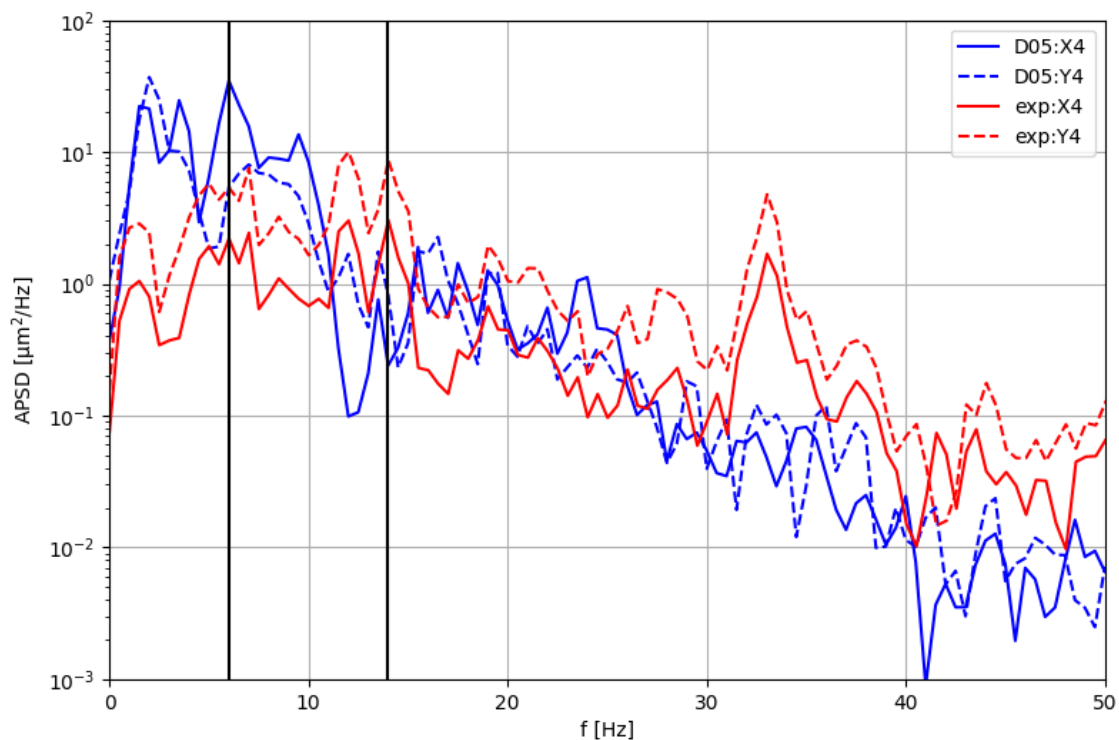


Figure 6-14: APSD of lateral displacement at spacer grid 2 for damping $D = 0.5$

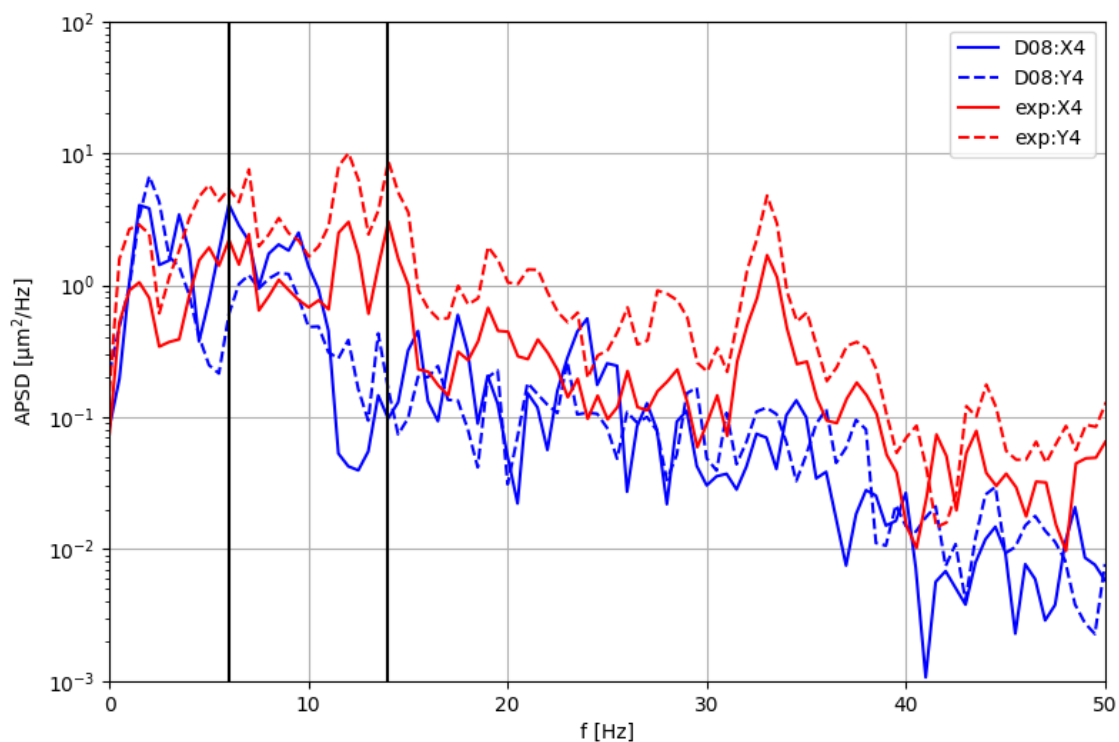


Figure 6-15: APSD of lateral displacement at spacer grid 2 for damping $D = 0.8$

7 Simulations performed by IPP

In this chapter the numerical simulations of the ALAIN benchmark performed by IPP and results of these simulations are described and presented in detail.

7.1 Introduction

Vibrations caused by the coolant flowing around the fuel assemblies are one of the destructive factors in the safe operation of the core. For example, the start of the BN-800 reactor [72] was delayed for almost a year due to the occurrence of high-amplitude vibrations in the fuel assemblies upon reaching steady-state parameters, which could have caused significant damage. However, it is important to understand that vibrations in fuel assemblies are a natural phenomenon but vary depending on the exciter and its magnitude. The problematic ones are precisely those oscillations whose nature is associated with the propagation and development of a vortex structure in the core of the flow. Such a vortex structure is generated based on Tollmien-Schlichting waves [73] when the laminar boundary layer loses stability. Herewith, if the geometry contributes to the emergence of an "unfavorable" pressure gradient, a separated flow appears, which determines the large forces acting on the surface. In general, these two factors significantly increase the transverse unevenness of the flow and contribute to the emergence of significant non-stationary hydrodynamic forces. In this case, the maximum deformation amplitudes can reach several millimeters. In reality, fuel assemblies are designed using spacer grids (including mixing type) in such a way that the main source of forces are the fluctuation properties of the flow (turbulent pulsations), and the geometric features affect only the effective diffusion properties of the coolant. In this case, the vibration amplitudes of the structural elements are several orders of magnitude smaller and relatively safe for operation.

One important feature is worth noting. In both cases of oscillation excitation, turbulent flow is present. However, the fundamental difference is that at the moving boundary, with an amplitude greater than the displacement thickness, so-called induced velocity and pressure pulsations [74] arise in the boundary layer, which can interact with the turbulent pulsations. Therefore, it is important to correctly account for such interactions, which essentially requires separate closing equations. In this case, induced pulsations can have a significantly stronger effect on the average flow than turbulence. The simplest way is to assume the additivity of the mean field, the fluctuating turbulent field and the induced pulsations of the moving surface. In the case of small oscillations of the moving boundary, the order of turbulent fluctuations is significant and, therefore, in the transverse direction, often decisive. In this case, for large displacements the URANS approach is acceptable, while for the assessment of small oscillations (of the order of turbulent noise) it is often not applicable. Thus, it is quite justified to use explicit or sequential schemes for analyzing coupled oscillations, which generally simplifies the organization of calculations. The only difficulty is the reproduction of turbulent fluctuations, as well as their anisotropy, which determines the possibility of obtaining qualitative and quantitative results consistent with experiment.

Therefore, from a scientific and engineering point of view, predicting such oscillations is an important task. Below we will focus on the development of engineering approaches based on URANS with elements of vortex synthetics [75].

7.2 CFD model of the ALAIN experimental assembly

7.2.1 Geometry of the flow section of the experimental assembly

To simulate the process of water flow through a 5x5 square-section fuel assembly (FA), a geometric model of the flow part of the ALAIN experimental assembly was prepared in [1]. The assembly consists of 23 fuel rods with an outer diameter of 9.5 mm and a length of 2275 mm, as well as two guide tubes with an outer diameter of 11.96 mm and a length of 2305 mm. Spacer grids with a height of 38 mm, excluding the upper ends, and a cross-sectional width and height of 66 mm are evenly installed along the length of the fuel assembly with a distance of 545 mm. In addition to the flow segment of the fuel assembly, the test section design also includes inlet and outlet parts. The width and height of the cross-section of the flow part is 69 mm. All elements of the spacer grids are modeled precisely with their points of contact with the fuel rods and guide tubes. The metal structure elements and Plexiglas walls of the test assembly are not included in the geometric model. A general view of the geometric model of the ALAIN experimental assembly flow part is shown in Figure 7-1.

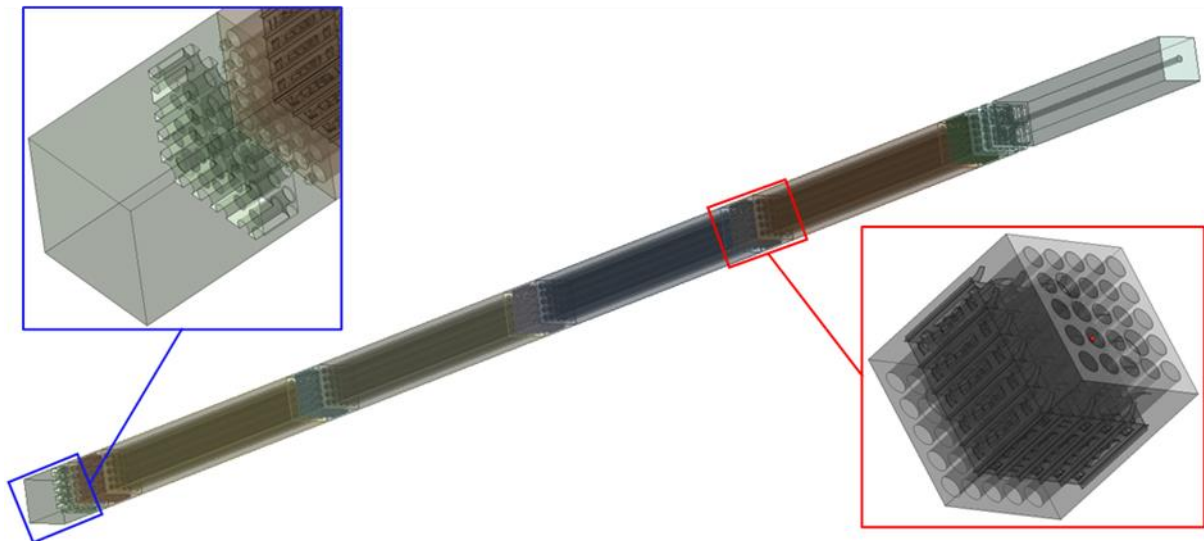


Figure 7-1: General view of the modeled flow part of the ALAIN test assembly

7.2.2 Computational meshes of the ALAIN FA CFD model

While performing the numerical simulation of vertical water flow through the experimental fuel assembly to assess the influence of mesh discretization density on solution convergence, three types of computational meshes were prepared:

- a mesh with lower discretization density and fluid-fluid interfaces (**FFI_L**);
- a mesh with higher discretization density and fluid-fluid interfaces (**FFI_L**);
- a conformal mesh with medium discretization density (**LNK**).

The general view of the prepared computational meshes used for the analysis of water flow through the experimental square fuel assembly with complex spacer grids is represented in Figure 7-2.

In regions with complex geometry, a triangulation method was applied using the «Curvature» sizing function, which accounts for the curvature of surfaces and geometric edges. For other, geometrically simpler regions, the «Multizone» or «Sweep» methods were applied to generate the volume mesh by extruding the mesh from surface.

To ensure a more accurate representation of velocity gradients near the walls, a boundary layer was generated. The main mesh quality characteristics, such as average orthogonality and element skewness, as well as the total number of elements, are listed in Table 7-1.

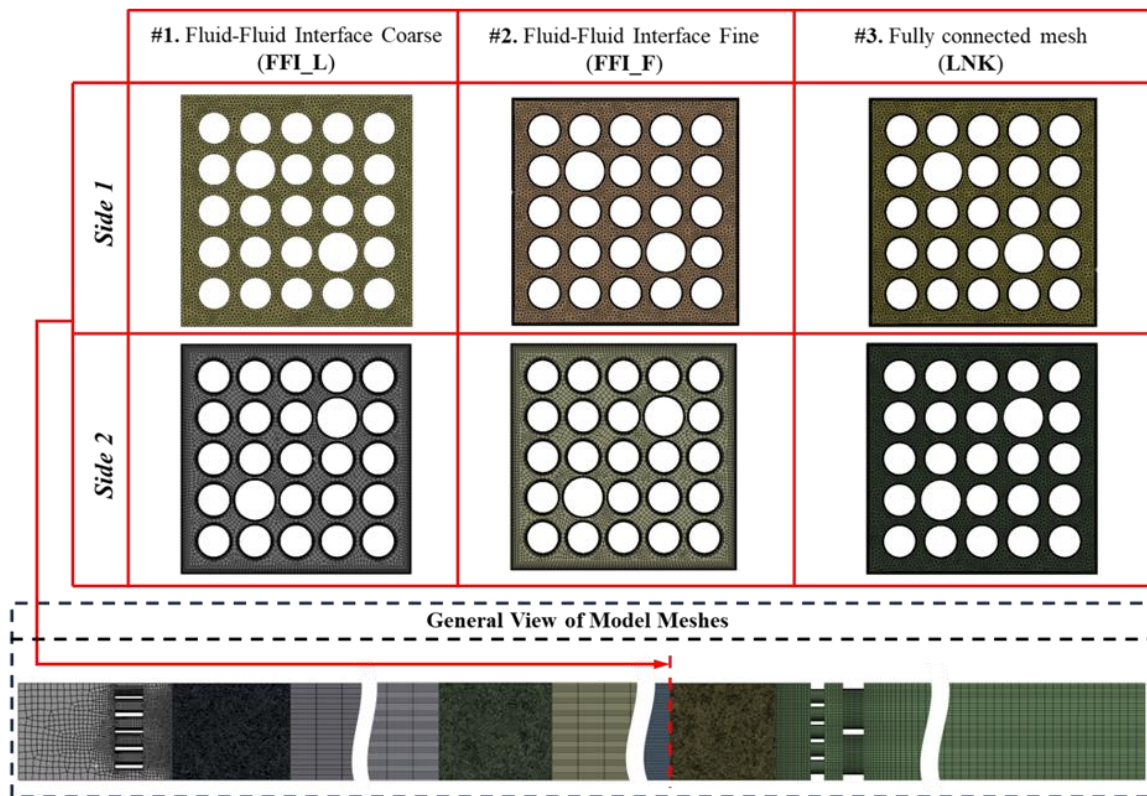


Figure 7-2: General view of the prepared computational meshes for the ALAIN FA CFD model

Table 7-1: The main characteristics of the prepared computational meshes

Mesh	Nodes	Elements	Orth. Quality	Skewness
FFI_L	$\approx 3.87 \cdot 10^6$	$\approx 12.96 \cdot 10^6$	0.835	0.260
FFI_F	$\approx 8.02 \cdot 10^6$	$\approx 22.58 \cdot 10^6$	0.850	0.246
LNK	$\approx 5.66 \cdot 10^6$	$\approx 16.86 \cdot 10^6$	0.828	0.270

7.2.3 CFD model and applied boundary conditions

In the experiment, single-phase liquid flow is investigated without accounting for heat transfer, so the isothermal formulation of the problem is assumed. The coolant temperature is set to 40 °C, and the reference pressure is 4 bar.

As boundary conditions (Figure 7-3), an outlet mass flow rate and zero reference pressure at the inlet were specified, in accordance with the experimental setup. The experimental mass

flow rate was 15 kg/s, corresponding to an average water velocity of 5.2 m/s between the fuel rods. No-slip boundary conditions were applied to the external surfaces of the structural elements of experimental assembly, as well as to the outer vertical surfaces of the computational domain. In cases where a split-domain model was used, flow continuity was ensured through the implementation of fluid–fluid interfaces.

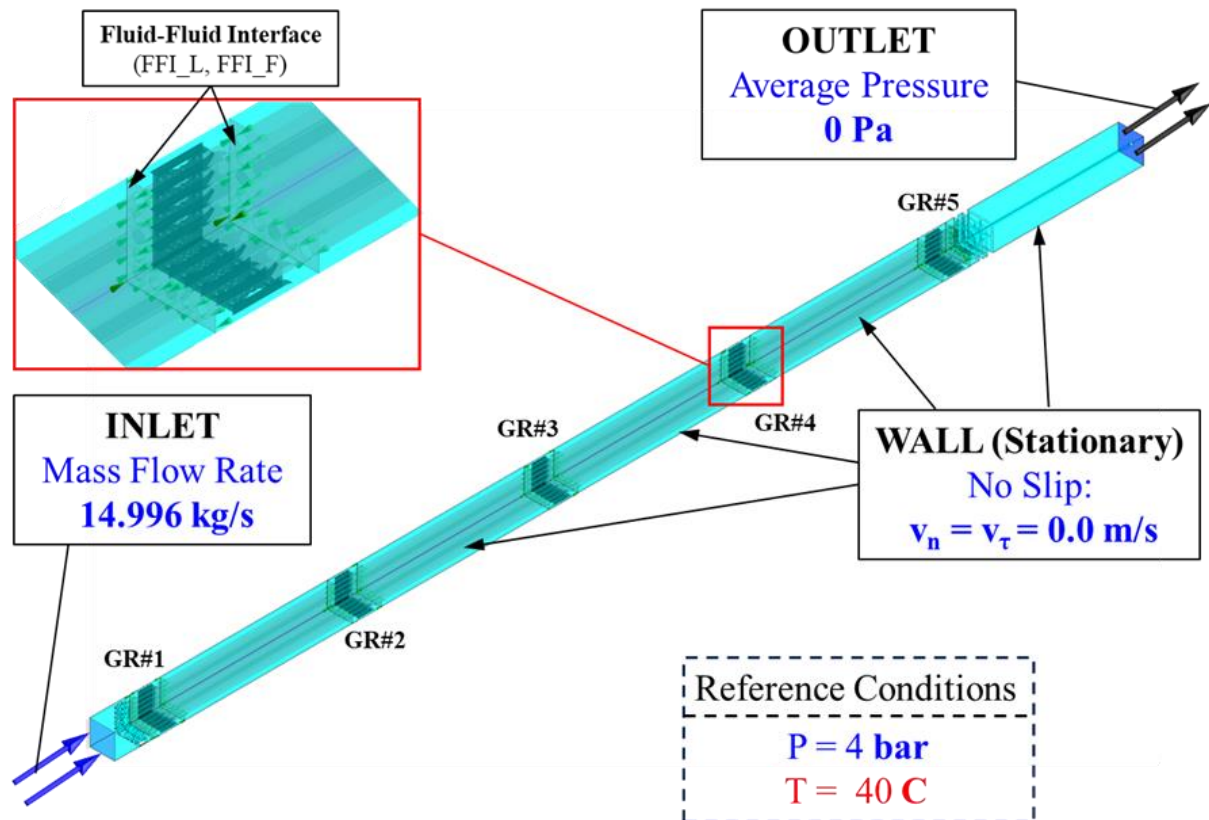


Figure 7-3: Boundary conditions specified to the ALAIN FA CFD model

The basic system of isothermal hydrodynamic equations without taking into account the compressibility of the medium does not differ from the classical one for computational fluid dynamics (the ANSYS CFX v.18 calculation code is used) [76]. To form the initial state of the velocity field, the SST turbulence model with improved processing of wall functions for low values of y^+ is applied.

7.3 Analysis of the computational mesh and initial state

A preliminary analysis was performed in a steady-state setting using the SST turbulence model for each of the computational mesh options.

Comparative calculations of the hydrodynamics of water flow in the inner element space of the experimental fuel assembly were performed. In the experiment, measurements were carried out using the laser Doppler anemometry (LDA) method at axial distances of 10, 20, 50, 100, and 250 mm downstream of the upper edge of the second spacer grid.

In the transverse (cross-sectional) plane, measurements were conducted along the Transverse 1 and Transverse 2 directions, as shown in Figure 7-4.

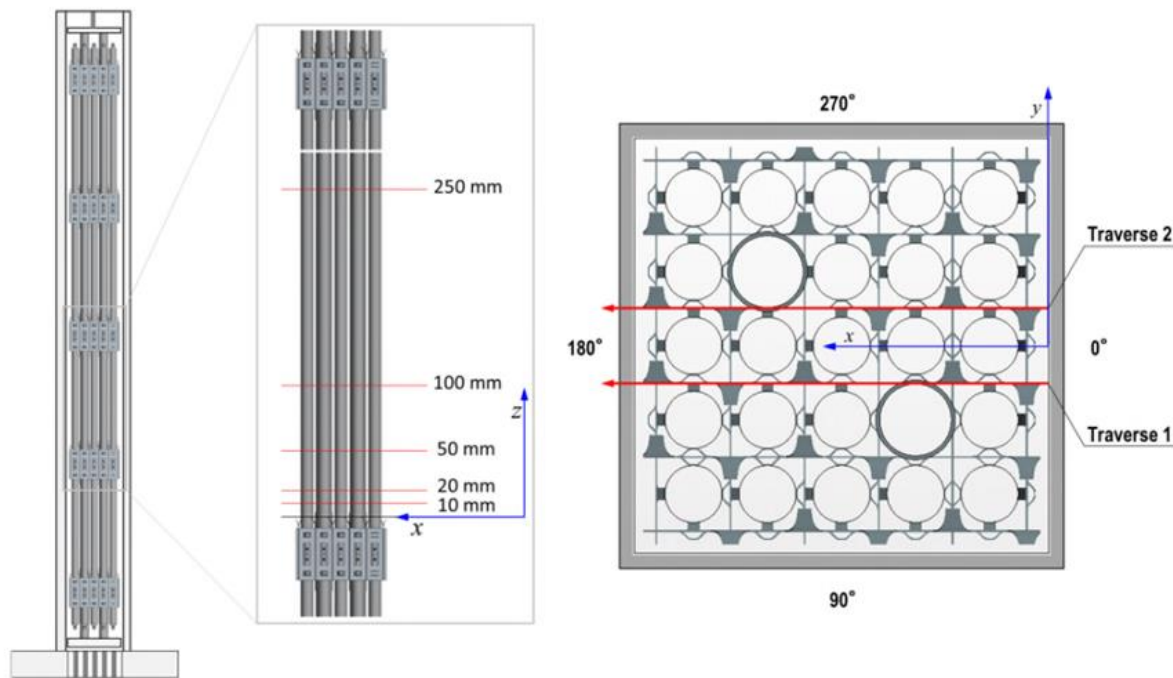


Figure 7-4: Directions for the LDA velocity measurements in the ALAIN experiment

In the experiment, both axial and transverse velocity components of the flow were measured. The comparative CFD results for the axial velocity component at each measurement elevation and direction are shown in Figure 7-5. The calculated results compared with the experimental data for the transverse velocity component are shown in Figure 7-6.

The developed models demonstrate generally good quantitative and qualitative predictive capability for the transverse velocity component of the fluid flow. Regarding the axial velocity component, the results are satisfactory. However, better agreement with the experimental data is observed at axial distances starting from the 20 mm downstream from the upper edge of the spacer grid.

In general, the steady-state evaluation results show that the obtained profiles in the transverse direction are in quite good agreement with the experimental data. The main deviations are concentrated behind the grids, indicating a poor reproduction of the turbulent flow structure caused by local geometric irregularities. This generally confirms the assumption that the turbulent structure is the primary source of the disturbing forces. This assumption will then be tested by analyzing a transient process taking into account the deformation of the mesh based on a special semi-empirical model of the displacements of characteristic assembly points.

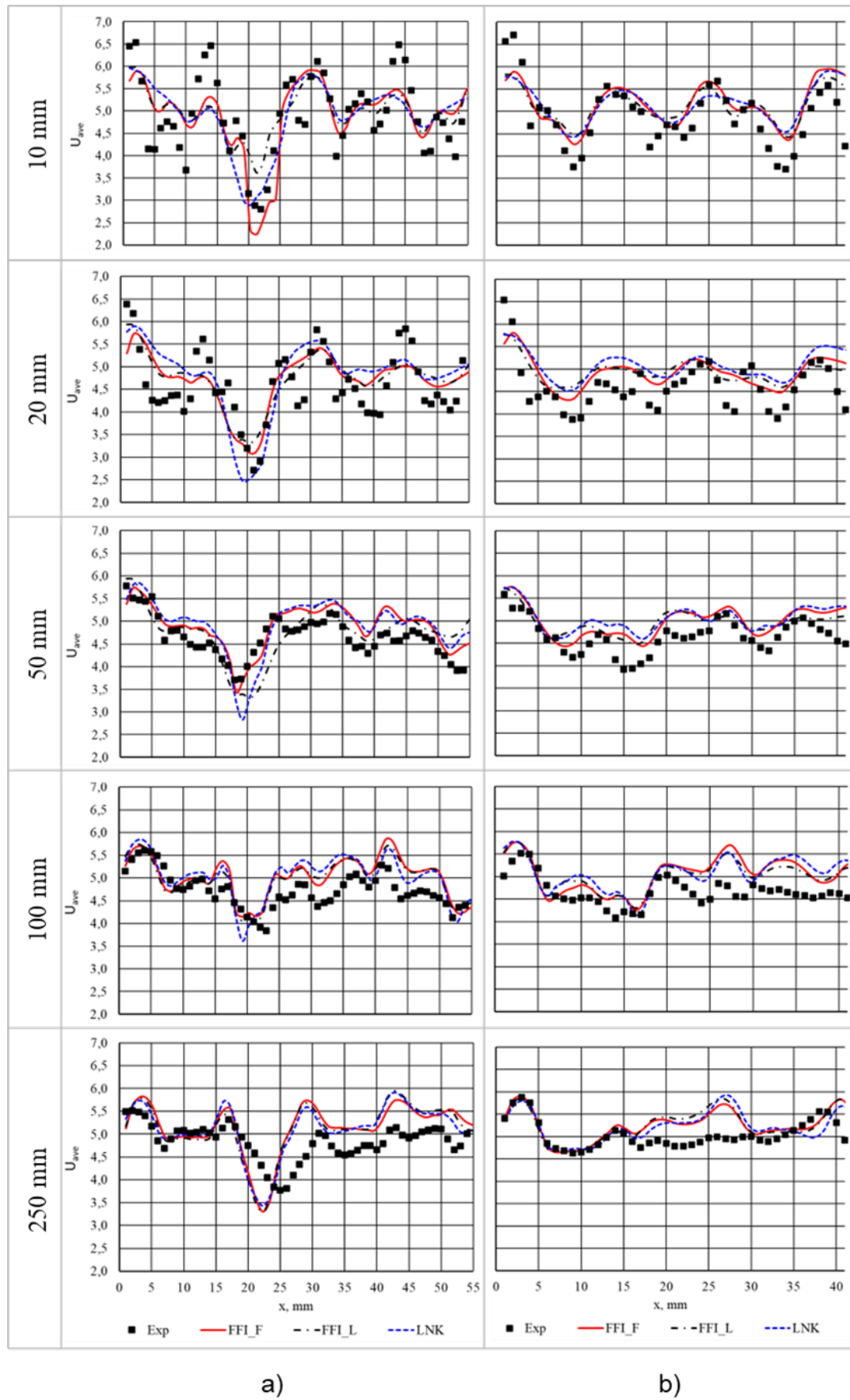


Figure 7-5: Axial velocity profiles in the Transverse 1 (a) and Transverse 2 (b) directions.

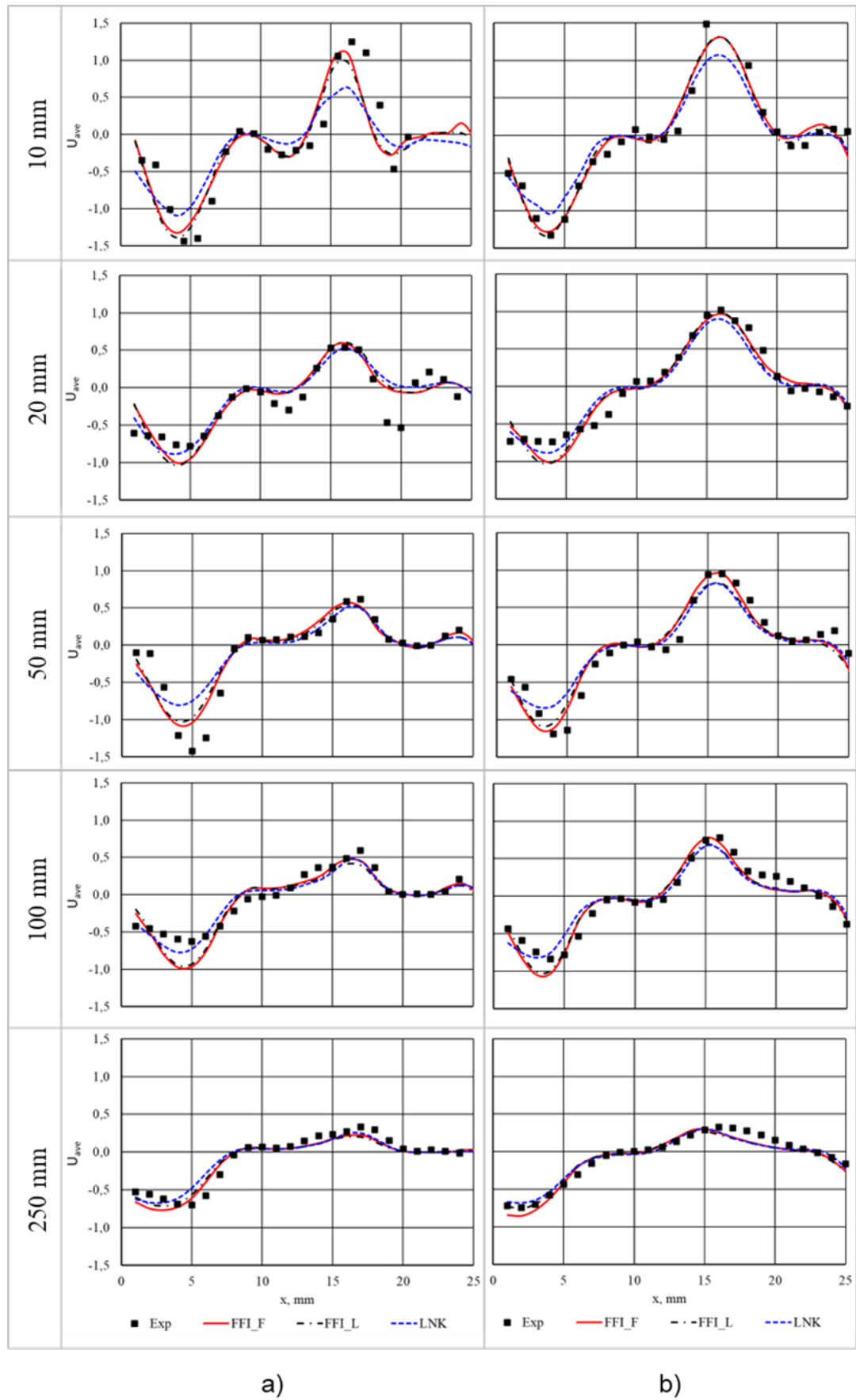


Figure 7-6: Transverse velocity profiles in the Transverse 1 (a) and Transverse 2 (b) directions.

7.4 Three-point mechanical model of fuel assembly displacement

First let's estimate the vibration level for our configuration, to get understanding what kind of simulation and model we can use. For this purpose, an empirical equation from [77] is used:

$$\frac{\delta_v}{D} = 5 \times 10^{-4} \alpha_1^{-4} K \left[\frac{u^{1.6} \varepsilon^{1.8} Re^{0.25}}{1 + u^2} \right] \left(\frac{D_h}{D} \right)^{0.4} \frac{\beta^{\frac{2}{3}}}{1 + 4\beta} \quad (7.1)$$

Here:

δ_v - vibration amplitude, [m]

K - factor accounting for the effect of upstream disturbance and mechanically transmitted vibration (= 5 for realistic disturbance levels) [-]

α_1 - eigenvalue of the first mode (π for pinned ends and $3\pi/2$) for clamped ends, [-]

u - dimensionless flow velocity $u = \sqrt{M/EI} \times VL$, [-]

M - hydrodynamic mass per unit length $M = \left[1 + \frac{D}{2(P-D)} \right] \pi D^2 \frac{\rho_w}{4}$, [kg/m]

P - rod pitch [m]

E - Young's modulus, [Pa]

I - moment of inertia of the rod cross section, $I = \frac{\pi}{64} (D_y^4 - D_i^4)$, [m⁴]

V - axial velocity, $V = \frac{m_f}{\rho_w A}$, [m/s]

m_f - mass flow [kg/s]

A - flow area [m²]

L - the rod effective length [m]

Re - Reynolds number, $Re = VD_h \frac{\rho_w}{\mu}$ [-]

D - rod diameter, [m]

D_h - hydraulic diameter of flow, [m]

m - rod mass per unit length (linear density) [kg/m]

ε - reduced length $\varepsilon = L/D$, [-]

β - the added mass ratio $\beta = \frac{M}{M+m}$, [-]

μ - fluid viscosity, [kg/m/s]

ρ_w - fluid density, [kg/m³]

So, substituting the values to eq. (7.1) we get the $\delta_v = 0.06241$ [mm], a very low value of vibrations. Thus, this assessment confirms from a mechanical point of view that the disturbing forces are determined exclusively by the turbulent fluctuation field, and the influence of pulsations caused by the moving surface can be neglected. In principle, for engineering practice this allows to examine the problem in a decoupled formulation, where the hydrodynamics is analysed on a stationary mesh, followed by the evaluation of mechanical vibrations based on the resulting velocity fields.

7.4.1 Model definition

There are several options for the FA modelling (see Figure 7-7):

- 3-DOF system – the simplest way to model dynamic behavior with well-developed procedures for effective mathematical description. Also, the benefit of such a system is that we can directly obtain vibrational characteristics from experimental measurements.
- Single beam model – a more complicated way to describe the dynamic, more adaptive iterations are required (or more experimental data) to adequately describe the FA
- Advanced one [78] (if we are talking about the simplified representation) – here the skeleton and fuel rod bundle are described by separate beams, connected by the special springs.

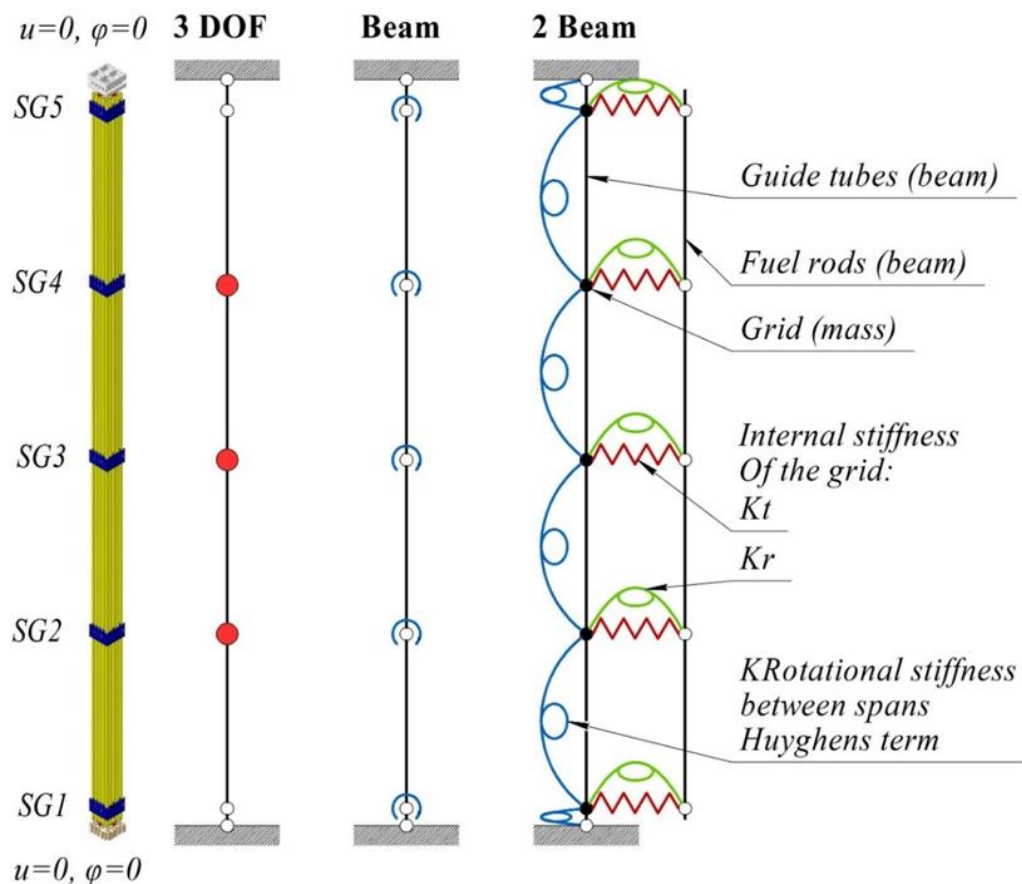


Figure 7-7: Possible representation of the FA model.

In our case, based on the preliminary analysis and availability of the experimental data, the first option with 3-DOF system was selected. However, in this case, an inverse vibration problem should be solved [79], see below.

7.4.2 Restoring system characteristics from experimental data

To establish the baseline dynamic characteristics of the fuel assembly in air, the raw time-domain displacement signals obtained from the impact tests at the 2nd, 3rd, and 4th spacer grids were processed using frequency-domain techniques.

First, the signals were transformed into the frequency domain using the Fast Fourier Transform (FFT). To ensure robust identification of global resonances, an averaged amplitude spectrum was computed across all three measurement locations. The natural frequencies f_n of the first three bending modes were identified by locating the dominant peaks in the averaged spectrum.

The structural damping ratios ζ_n associated with each mode were estimated using an enhanced Half-Power Bandwidth method. Given the transient nature of the impact tests, the discrete frequency resolution of the FFT can lead to significant overestimation of the peak width if standard index-based searching is used. To overcome this and achieve high accuracy, linear interpolation was applied between the discrete frequency bins to precisely pinpoint the lower f_1 and upper f_2 half-power frequencies—the exact points where the spectral amplitude drops to $1/\sqrt{2}$ of the resonant peak. The damping ratio was then calculated as:

$$\zeta_n = \frac{f_2 - f_1}{2f_1} \quad (7.2)$$

Finally, the experimental mode shapes X_{exp} were extracted to capture the spatial deflection of the assembly. To avoid vulnerabilities associated with arbitrary phase shifts in the time domain, the true vibration amplitudes were extracted directly from the magnitude of the complex FFT output at each natural frequency. The directional signs of the nodal displacements were determined by evaluating the cross-phase angles relative to the reference node (the node with the maximum amplitude). Nodes exhibiting a phase difference near 0° were assigned a positive relative direction (in-phase), while those near 180° were assigned a negative direction (out-of-phase). The resulting vectors were then amplitude-normalized to formulate the fundamental mode shapes (Figure 7-8).

To construct a reduced-order model of the fuel assembly, a 3-Degree-Of-Freedom (DOF) lumped-parameter system was developed based on the experimental "impact test" data obtained in air. The measurement points correspond to the 2nd, 3rd (central), and 4th spacer grids.

The dynamics of the system are governed by the classical equation of motion:

$$M \frac{d^2x}{dt^2} + C \frac{dx}{dt} + Kx(t) = 0 \quad (7.3)$$

where M , C and $K \in \mathbb{R}^{3 \times 3}$ are the physical mass, damping, and stiffness matrices, respectively, and $x(t)$ is the displacement vector of the spacer grids.

The inverse eigenvalue problem aims to reconstruct M , C and K using the known total mass of the system M_{tot} and the experimentally identified modal parameters: the natural frequencies f_n , damping ratios ζ_n , and raw mode shapes $X_{exp} \in \mathbb{R}^{3 \times 3}$.

The angular frequencies and spectral matrices are defined as:

$$\omega_i = 2\pi f_i \quad (7.4)$$

$$\Omega^2 = \text{diag}(\omega_1^2, \omega_2^2, \omega_3^2) \quad (7.5)$$

$$\Delta = \text{diag}(2\zeta_1\omega_1, 2\zeta_2\omega_2, 2\zeta_3\omega_3) \quad (7.6)$$

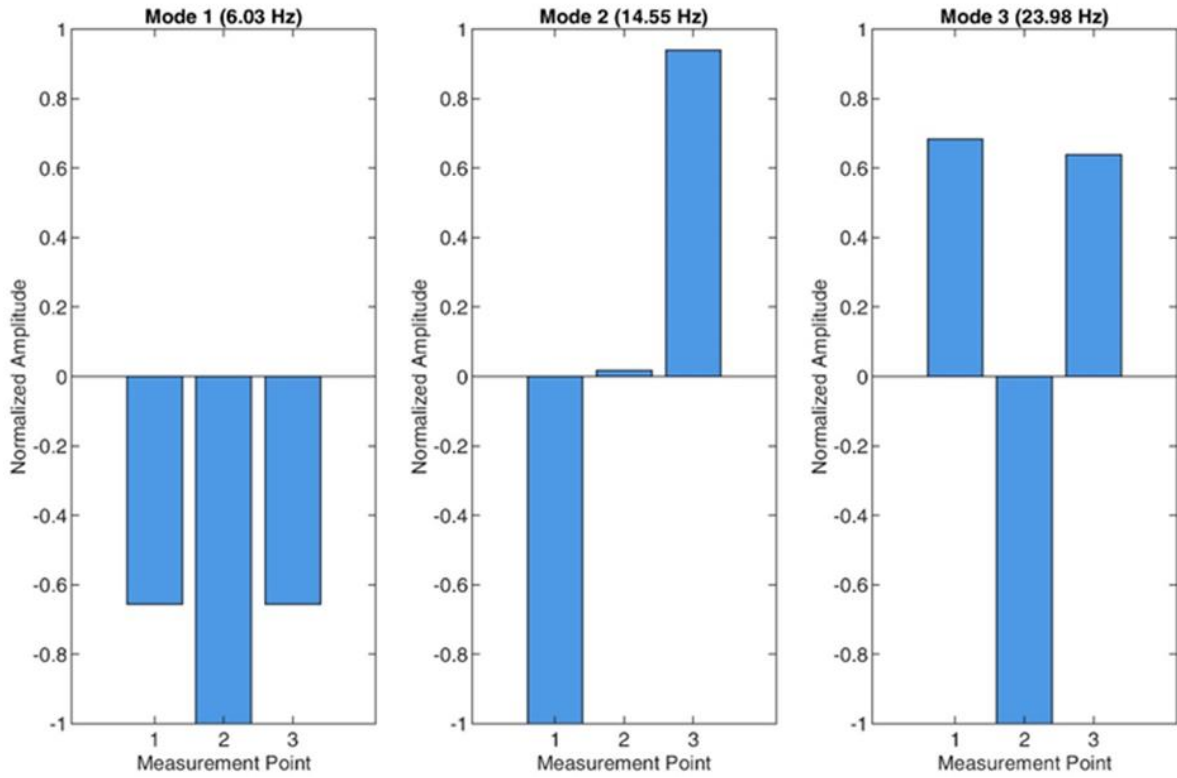


Figure 7-8: Restored mode shapes

Theoretical mode shapes Φ of a lumped-mass system must satisfy orthogonality with respect to the mass matrix ($\Phi^T M \Phi = I$). For a uniform fuel assembly, we introduce the assumption that the mass is distributed equally among the three modeled DOF nodes. Consequently, the mass matrix is proportional to the identity matrix:

$$M = mI, \text{ where } m = \frac{M_{total}}{3} \quad (7.7)$$

Because raw experimental mode shapes X_{exp} are rarely perfectly orthogonal due to measurement noise and the continuous nature of the real assembly, they must be orthogonalized to satisfy the classical spectral decomposition conditions. This is achieved via Singular Value Decomposition (SVD):

$$X_{exp} = U\Sigma V^T \quad (7.8)$$

The left singular vectors U provide the optimal orthogonal approximation of the mode shapes such that $U^T U = I$. We assign the structural mode shapes as $\Phi = U$.

Using the orthogonalized mode shapes Φ , the system matrices are reconstructed by mapping the diagonal modal matrices back to the physical domain.

For the stiffness matrix K , the standard modal transformation $\Phi^T K \Phi = \Omega^2$ is rearranged. Given that Φ is strictly orthogonal, $\Phi^{-1} = \Phi^T$, leading to:

$$K = m\Phi\Omega^2\Phi^T \quad (7.9)$$

Similarly, assuming proportional (Rayleigh) damping, the damping matrix C is reconstructed from the modal damping matrix Δ :

$$C = m\Phi\Delta\Phi^T \quad (7.10)$$

The final reconstructed matrices M , C and K yield a 3-DOF model that perfectly preserves the experimentally measured natural frequencies and structural damping ratios in air, while providing an optimally matched orthogonal set of mode shapes (see Figure 7-9). The fidelity of the forced orthogonalization is subsequently verified using the Modal Assurance Criterion (MAC) by comparing the eigenvectors of $M^{-1}K$ against the raw experimental matrix X_{exp} (see Figure 7-10).

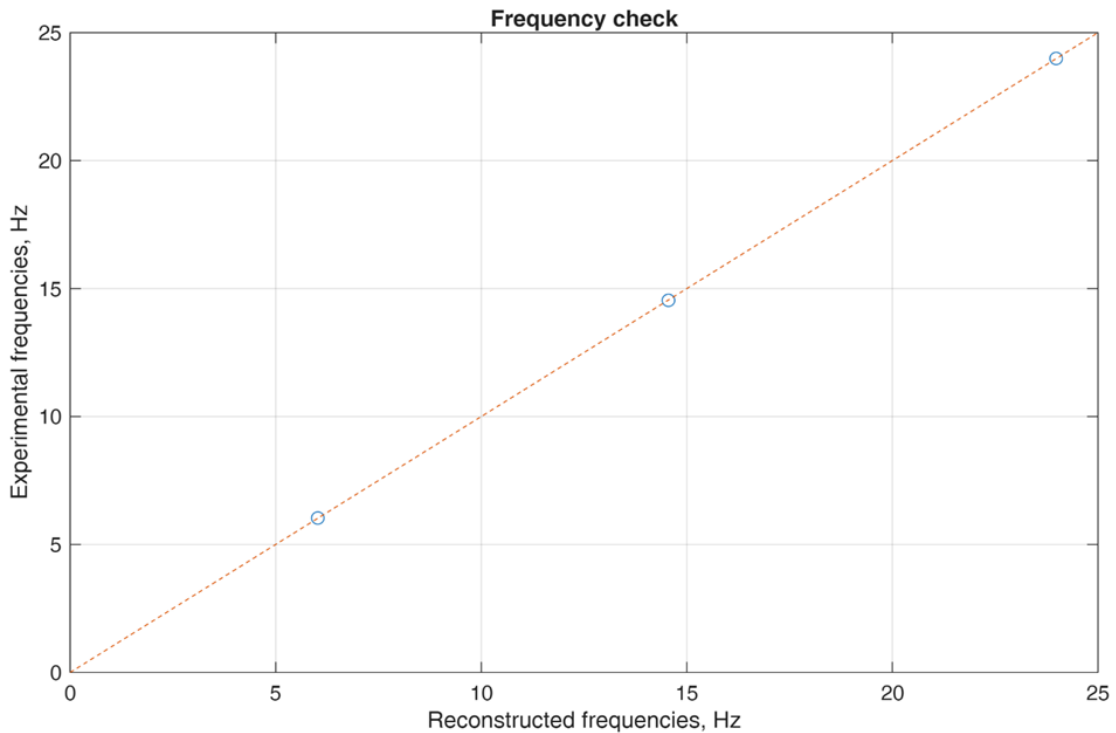


Figure 7-9: Comparison of restored versus experimental frequencies

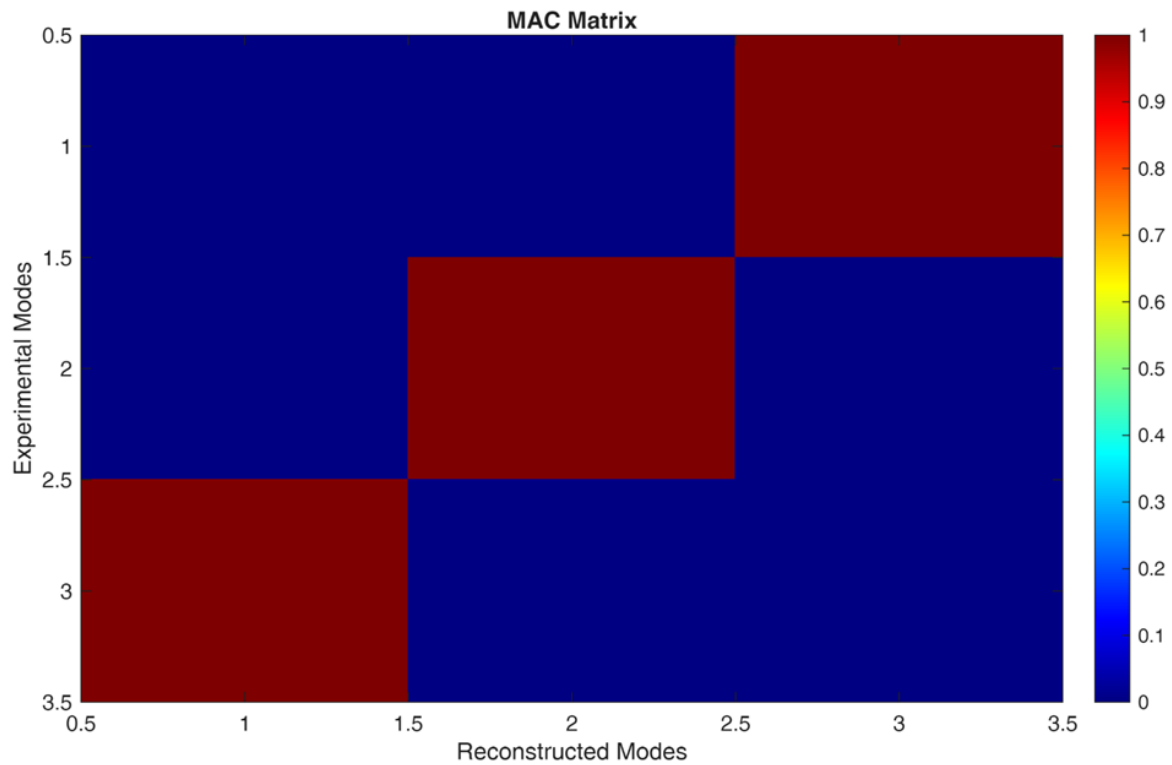


Figure 7-10: Modal Assurance Criterion

7.4.3 Fluid effects on the vibrational model

To incorporate the effects of the fluid environment into the 3-DOF mechanical model, the hydrodynamic added mass and viscous damping coefficients were evaluated [79]. Due to the nature of the forced oscillations, several assumptions can be formulated:

- Disturbing displacements of the wall with velocity δu do not affect the main turbulent flow.
- The hydrodynamic force generated by the turbulent flow acts as an independent disturbing force.
- A linear decomposition exists for the mean and fluctuating components of the turbulent velocity field, as well as for the velocity field δu associated with the moving boundary. This implies that δu satisfies the continuity equation.
- The moving boundary disturbance δu is small and practically is not transferred by advection into the core of the flow ($\delta Re \sim 1.0$).
- The boundary disturbance δu causes the perturbations in the static pressure field δp . It is assumed that the mean flow remains non-separated, therefore the interaction between δu and δp is similar to the mean flow.
- The unsteady term in the behavior of δp can be neglected. It is assumed that small wall perturbations do not generate acoustic noise, or their energy is negligible.
- A linear spectral decomposition exists for each term of the perturbation fields.

Taking into account all of the above, to estimate the response of the fluid mass in the assembly in the transverse direction, a linear theory of disturbance propagation based on the Stokes equations with decomposition of the velocity fields can be applied so that the spatial components δu satisfy the continuity equation:

$$i\omega\rho(\delta u) - \mu\nabla^2(\delta u) + \nabla(\delta p) = 0 \quad (7.11)$$

$$\nabla(\delta u) = 0 \quad (7.12)$$

where δu is the complex disturbing velocity vector, δp is the complex disturbing static pressure, ρ is the fluid density, μ is the dynamic viscosity, and $\omega = 2\pi f$ is the angular frequency of oscillation.

The fluid domain of the 5x5 rod bundle cross-section bounded by the square channel was modeled in 2D using the finite element solver FreeFEM [80], based on time-harmonic Stokes equations (7.11) and (7.12).

To compute the global hydrodynamic properties associated with the fundamental bending modes, a rigid bundle assumption was employed. All rods were constrained to move in-phase along the transverse axis with a uniform prescribed velocity amplitude $\delta u = (i\omega, 0)$, while a no-slip stationary boundary condition was imposed on the outer channel. For the disturbed pressure field, the conditions of zero normal gradient on the surface of each of the walls are adopted.

The problem was discretized using a Taylor-Hood [81] finite element scheme (P_2/P_1). To accurately resolve the steep velocity gradients within the thin Stokes boundary layer ($\delta \approx \sqrt{2\nu/\omega}$), an anisotropic mesh adaptation procedure was implemented. The mesh was iteratively refined based on the interpolation error metric of the real and imaginary components of the computed velocity field, ensuring high-fidelity evaluation of the wall shear stresses.

For the selected direction, the response of the medium to the moving wall can be evaluated, by analogy with the hydrodynamic force, by integrating the fluid stress tensor over the wetted perimeters of all inner rods:

$$F_x = \oint_{\Gamma} \left[-(\delta p)n_x + \mu \left(2 \frac{\partial}{\partial x} (\delta u_x) \frac{\partial}{\partial x} n_x + \frac{\partial}{\partial x} (\delta u_x) \frac{\partial}{\partial y} n_y + \frac{\partial}{\partial y} (\delta u_y) \frac{\partial}{\partial x} n_x + \frac{\partial}{\partial y} (\delta u_y) \frac{\partial}{\partial y} n_y \right) \right] d\Gamma \quad (7.13)$$

Subsequently, the unit-length added mass (m_{xx}) and viscous damping coefficient (c_{xx}) were extracted from the real and imaginary components of F_x , respectively:

$$m_{xx} = -\frac{\text{Re}(F_x)}{\omega^2} \quad (7.14)$$

$$c_{xx} = -\frac{\text{Im}(F_x)}{\omega^2} \quad (7.15)$$

To integrate these 2D cross-sectional properties into the discrete 3-DOF spatial model, a modal projection was applied. Assuming a fundamental sinusoidal mode shape $\phi(z) = \sin(\pi z/L_{fr})$ over the total length of the fuel rods L_{fr} , the generalized modal added mass is obtained by integrating the unit-length mass weighted by the squared mode shape:

$$\int_0^{L_{fr}} m_{xx} \sin^2(\pi z/L_{fr}) dz = \frac{m_{xx} L_{fr}}{2} \quad (7.16)$$

This projected hydrodynamic mass, and the corresponding damping were then superimposed onto the structural mass and damping matrices of the reduced-order model.

7.5 Results of the coupled FIV analysis

The computational model briefly described in section 7.2 is complemented by library using the User Fortran (Junction Box) [82], which employs the simplified procedure described above for estimating the displacements of reference points based on hydrodynamic force. The library is based on the following simplifying assumptions:

- All elements of the assembly move as a homogenized body in the X and Y directions, which are considered independent,
- The height displacement profile is formed as a linear function between the characteristic displacement points, which are selected at the centers of SG2, SG3 and SG4.

For each reference point of the displacement calculation, the hydrodynamic force (the selected projection) is calculated as the sum on the SG walls, as well as on the halves of the lower and upper sections between the SGs.

The displacement library implements two solvers, one of which is based on the Duhamel integral (DH) for an elastic dissipative system, and the second one is based on the Newmark method (NW). Both solvers are being tested further.

Two options for determining the hydrodynamic force are also considered: total (SF) by analogy with (7.13) or exclusively the contribution of static pressure without taking into account the hydrostatic component (PF).

The library is implemented in such a way that displacements can be calculated directly at the end of each coefficient loop or at the end of a time step, allowing for both implicit and explicit coupling. For further calculations, the explicit formulation is used.

The main calculations were performed using the SAS-SST turbulence model, as well as one using SST. The integration time step ranged from $5 \cdot 10^{-4}$ to $5 \cdot 10^{-3}$ seconds. The calculations were performed from the initial state obtained in Section 7.3. In the case of the SAS-SST turbulence model, the central difference (CD) was used for the advection scheme.

The resulting time-averaged transverse velocity fields in cross sections, as well as the characteristic deformations of the assembly in directions X and Y are shown in Figure 7-11 (visualization scale 1000.0) for the SST model, and in Figure 7-12 to Figure 7-15 for the SAS-SST.

Simulations using the SST turbulence model result in virtually no assembly oscillations. At the beginning of the transient, the assembly deviates relative to its initial position, but the resulting oscillations are damped even in the absence of damping. The parameters of discretization and time step size also have practically no effect on the displacements. Generally, it is in good agreement with the initial assumptions regarding the dominance of turbulent pulsations in the value of the disturbing hydrodynamic force.

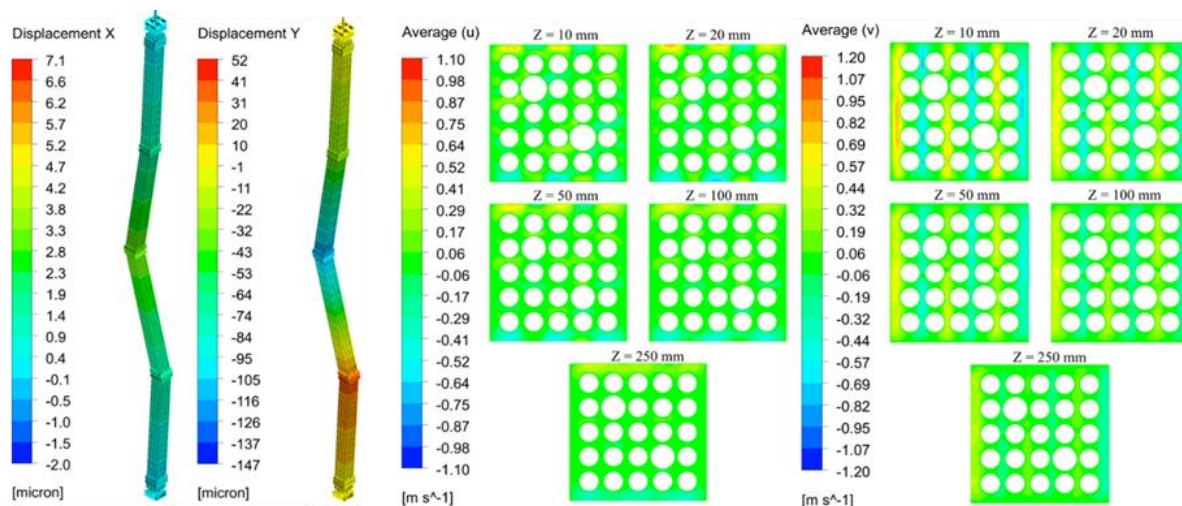


Figure 7-11: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SST, Force XY: PF, solver: NW)

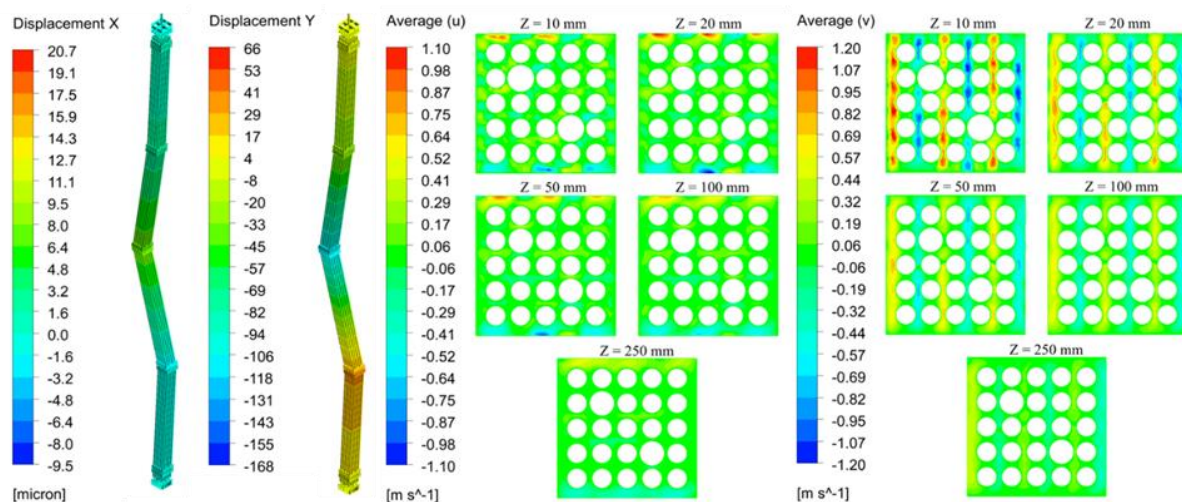


Figure 7-12: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SAS - SST, Force XY: PF, solver: DH)

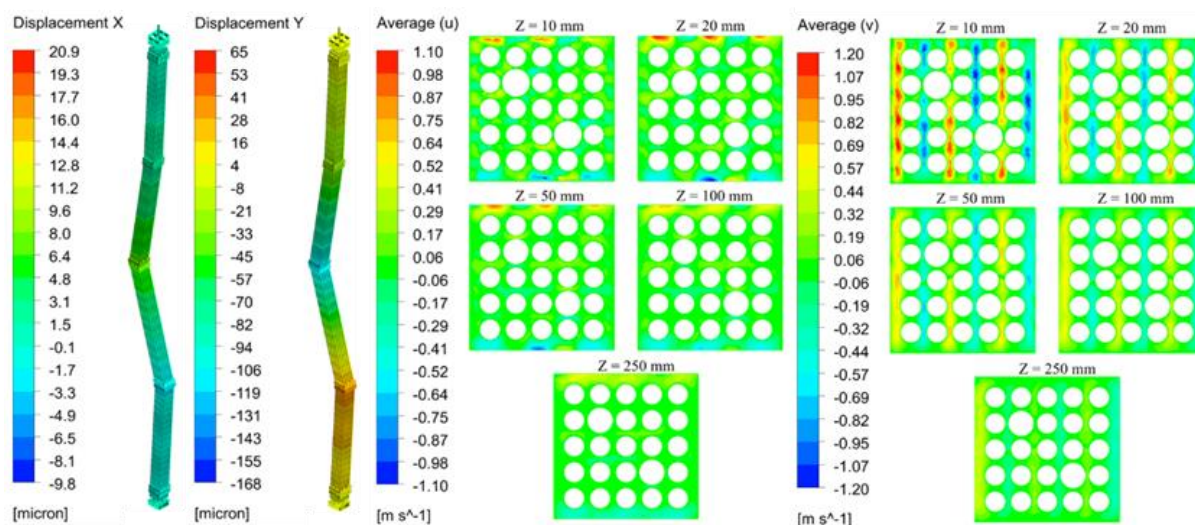


Figure 7-13: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SAS - SST, Force XY: SF, solver: DH)

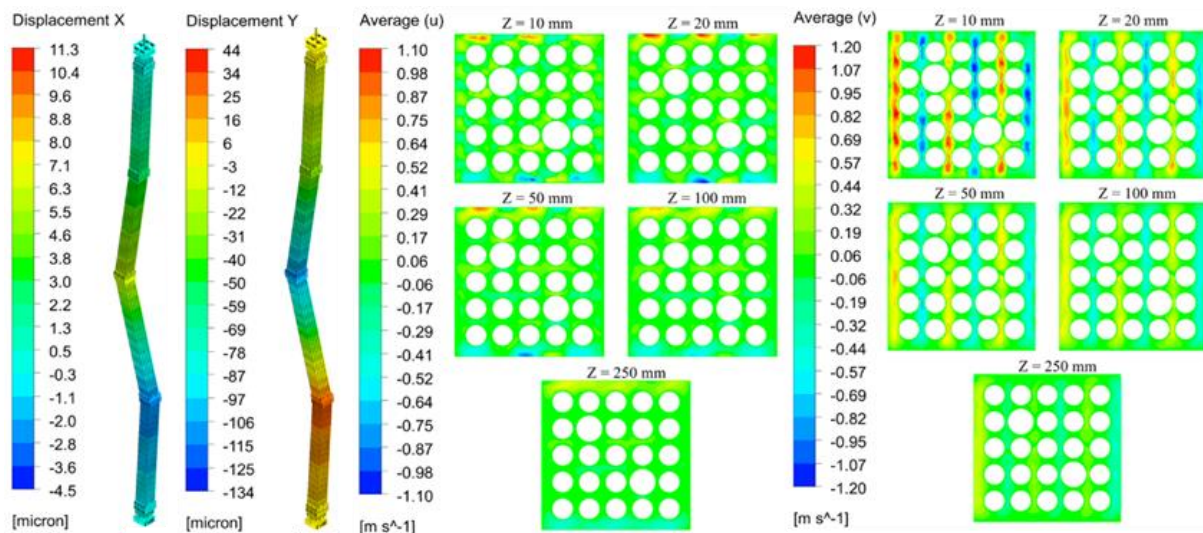


Figure 7-14: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SAS - SST, Force XY: PF, solver: NW)

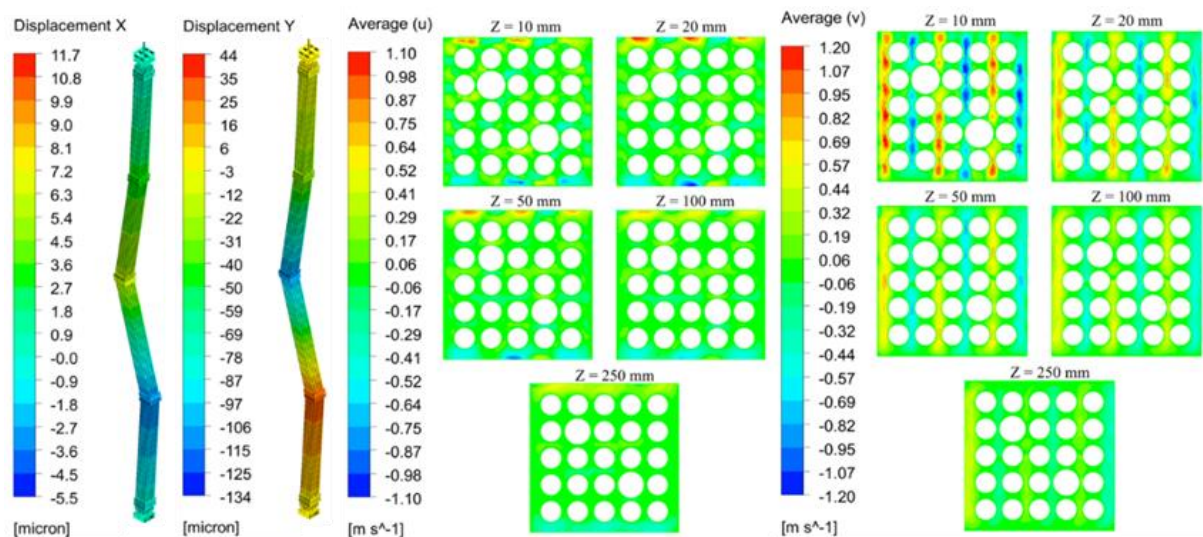


Figure 7-15: Typical displacements of the assembly and time-averaged transverse velocity fields in characteristic cross-sections (SAS - SST, Force XY: SF, solver: NW)

The results using SAS-SST (CD) are generally similar to each other, but fundamentally differ from SST in that the oscillations are not damped then in relation to the overall displacement (Figure 7-15). The amplitude of these oscillations is not significant (noticeable less than in the experiment) and is determined by the level of those fluctuations that the turbulence model is capable to reproduce. This is confirmed by a comparison of experimental and calculated velocity profiles behind the spacer grids. As the distance from the SG increases, the agreement between the calculation and the experiment improves, although significant discrepancies are observed directly behind the grids (Figure 7-17). In general, for axial velocity, this is more likely an effect of discretization and the use of domain decomposition near the grids (the presence of fluid-fluid interfaces), which negatively impacts the non-uniformity of the velocity field. However, in terms of forces, transverse velocities are more representative (Figure 7-18).

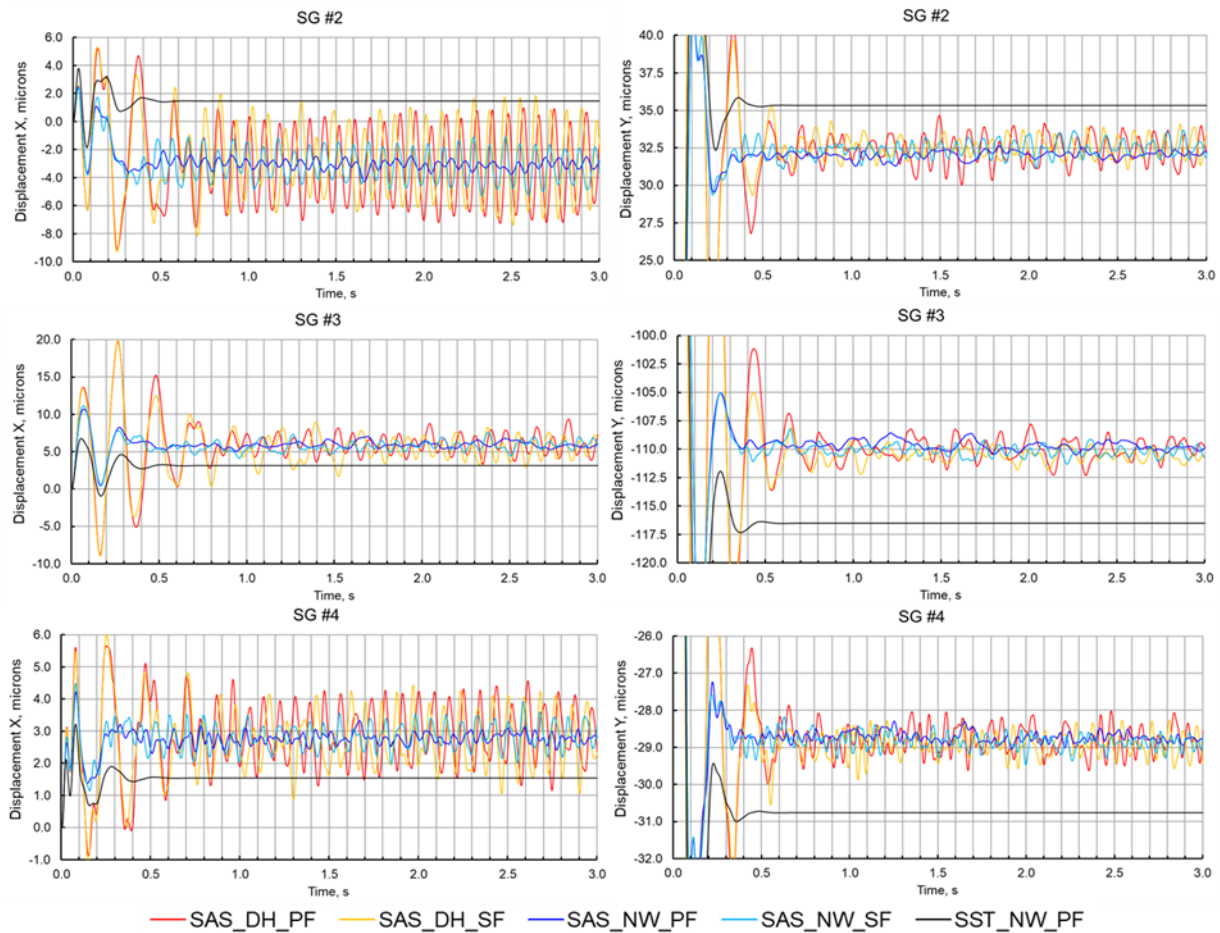


Figure 7-16: Oscillations of the assembly at first seconds of transient

Also, in each of the calculation options, the root-mean-square velocity (characterizes turbulent fluctuations) is significantly lower than in the experiment (Table 7-2), which indicates an underestimation of the unevenness and, therefore, the amplitude of the hydrodynamic force.

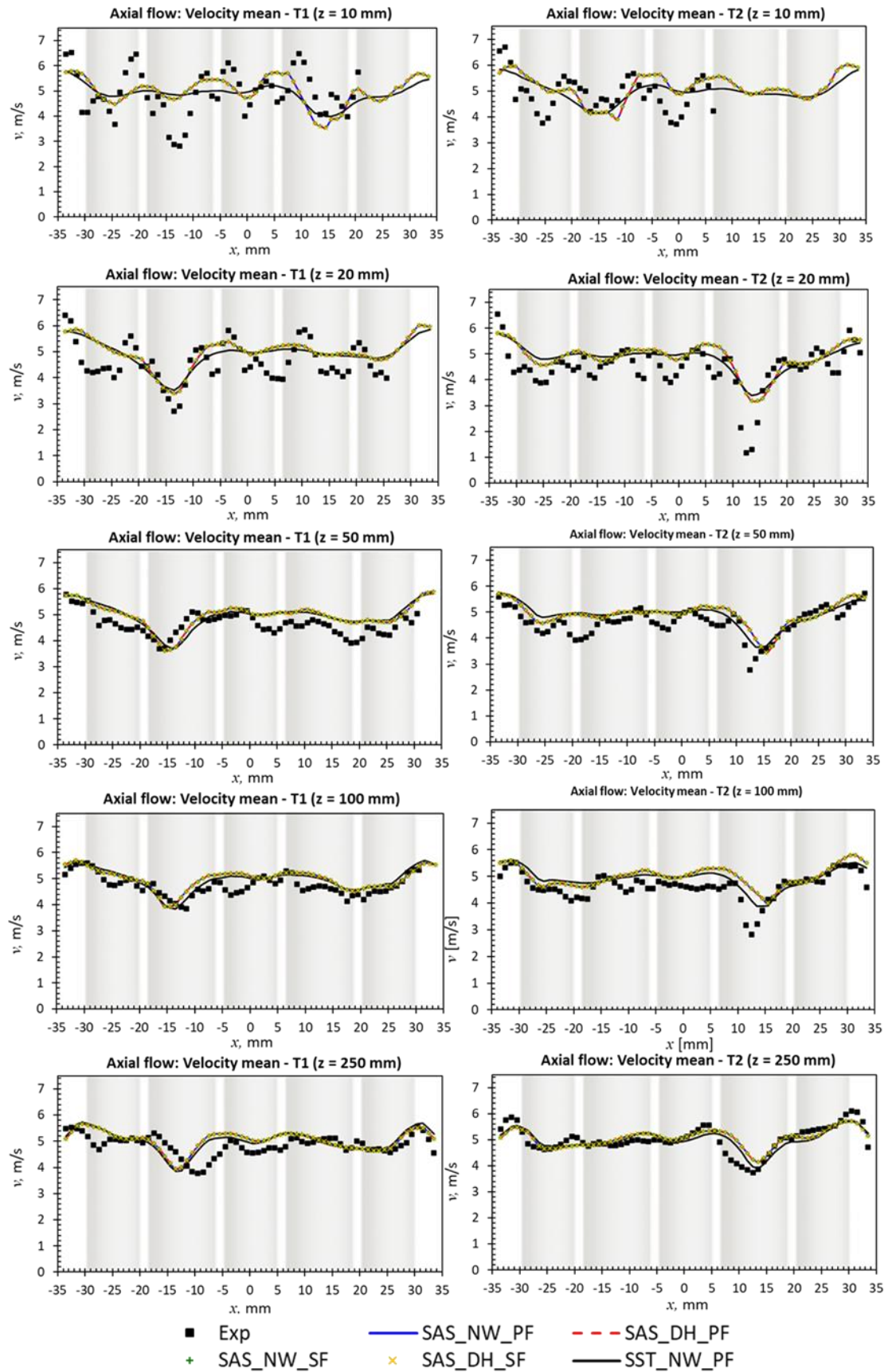


Figure 7-17: Distribution by height of the time-averaged axial flow velocity in characteristic cross-sections in two directions

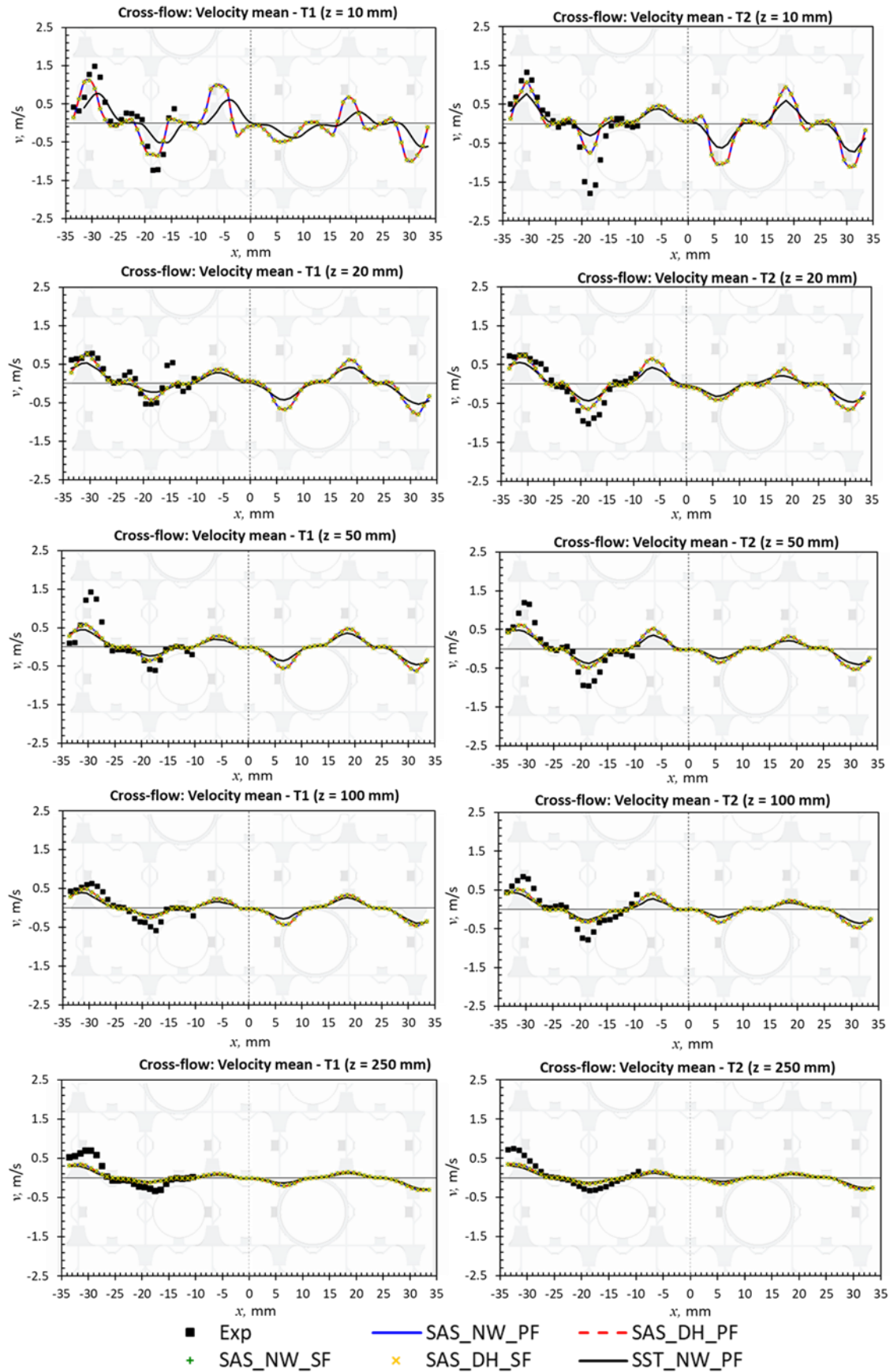


Figure 7-18: Distribution by height of the time-averaged transverse flow velocity in characteristic cross-sections in two directions

Table 7-2: Characteristic values of velocity RMS

Profile (Path)	T1 - Velocity RMS, [m/s]					T1 - Velocity RMS, [m/s]				
Maximum value										
Axial (z), mm	10	20	50	100	250	10	20	50	100	250
Ax. (Exp) ,m/s	0.936	0.773	0.599	0.450	0.461	0.750	0.938	0.733	0.629	0.556
Cr. (Exp), m/s	0.864	0.854	0.661	0.399	0.354	0.682	0.486	0.636	0.391	0.309
SST-NW-PF	0.324	0.301	0.302	0.303	0.313	0.331	0.306	0.310	0.314	0.321
SAS-NW-PF	0.344	0.319	0.311	0.304	0.305	0.346	0.314	0.312	0.311	0.313
SAS-DH-PF	0.344	0.318	0.311	0.303	0.306	0.346	0.314	0.311	0.310	0.311
SAS-NW-SF	0.341	0.319	0.311	0.304	0.305	0.346	0.313	0.310	0.310	0.313
SAS-DH-SF	0.344	0.320	0.312	0.304	0.305	0.346	0.315	0.311	0.311	0.314
Average value										
Ax. (Exp), m/s	0.531	0.507	0.373	0.313	0.314	0.540	0.545	0.394	0.326	0.315
Cr. (Exp), m/s	0.512	0.493	0.360	0.285	0.238	0.465	0.397	0.367	0.258	0.199
SST-NW-PF	0.321	0.280	0.256	0.251	0.254	0.328	0.277	0.254	0.251	0.253
SAS-NW-PF	0.321	0.280	0.256	0.251	0.254	0.328	0.277	0.254	0.251	0.253
SAS-DH-PF	0.321	0.280	0.256	0.251	0.254	0.328	0.277	0.254	0.251	0.253
SAS-NW-SF	0.321	0.280	0.256	0.251	0.254	0.328	0.277	0.254	0.251	0.253
SAS-DH-SF	0.299	0.262	0.237	0.236	0.255	0.306	0.258	0.235	0.233	0.252
Minimum value										
Ax. (Exp), m/s	0.271	0.343	0.256	0.235	0.217	0.320	0.320	0.288	0.233	0.196
Cr. (Exp), m/s	0.244	0.278	0.223	0.228	0.177	0.304	0.256	0.228	0.199	0.170
SST-NW-PF	0.277	0.241	0.214	0.205	0.214	0.279	0.241	0.209	0.197	0.209
SST-NW-PF	0.304	0.259	0.235	0.221	0.219	0.296	0.266	0.234	0.222	0.212
SAS-NW-PF	0.303	0.260	0.235	0.220	0.218	0.296	0.266	0.234	0.223	0.212
SAS-DH-PF	0.303	0.260	0.235	0.220	0.218	0.296	0.266	0.235	0.222	0.212
SAS-NW-SF	0.304	0.260	0.235	0.220	0.219	0.296	0.265	0.233	0.221	0.213

The corresponding vibration spectra are shown below:

- Figure 7-19 - Figure 7-20 show PSD for the X and Y directions of the SG2 grid.
- Figure 7-21 - Figure 7-22 show the PSD for the X and Y directions of the SG3,
- Figure 7-23 - Figure 7-24 show the PSD for SG4, respectively.

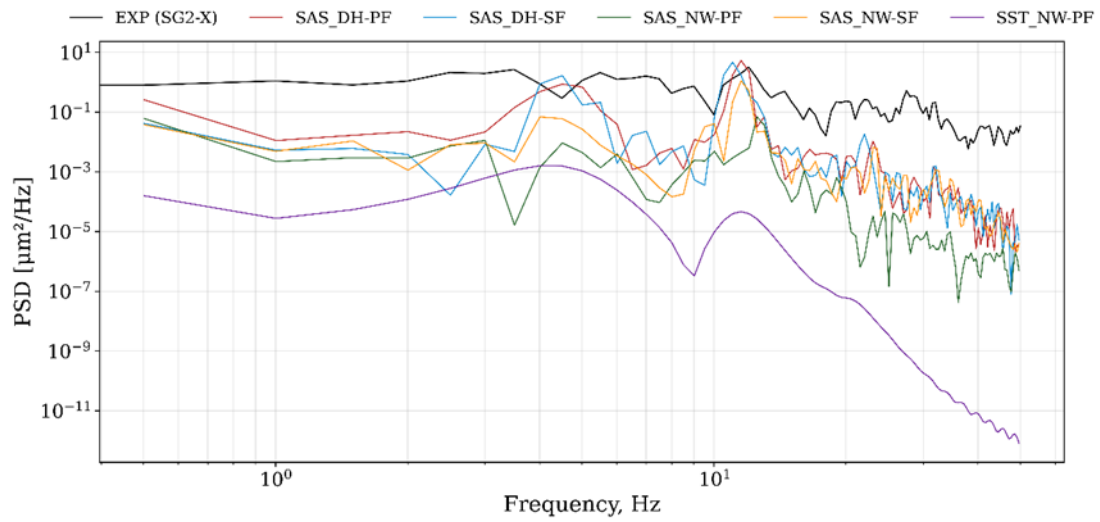


Figure 7-19: Vibration spectrum in X-direction for SG2

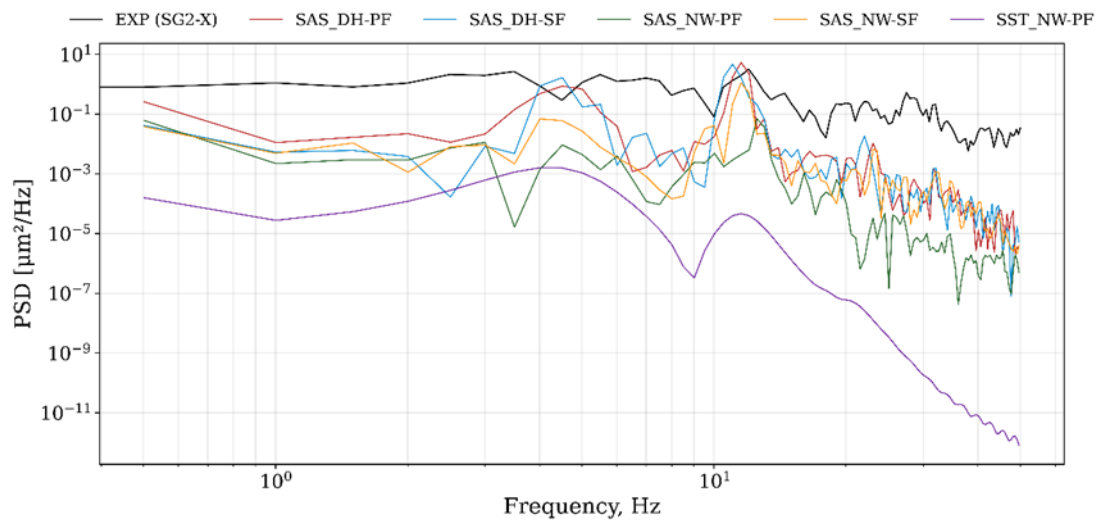


Figure 7-20: Vibration spectrum in Y-direction for SG2

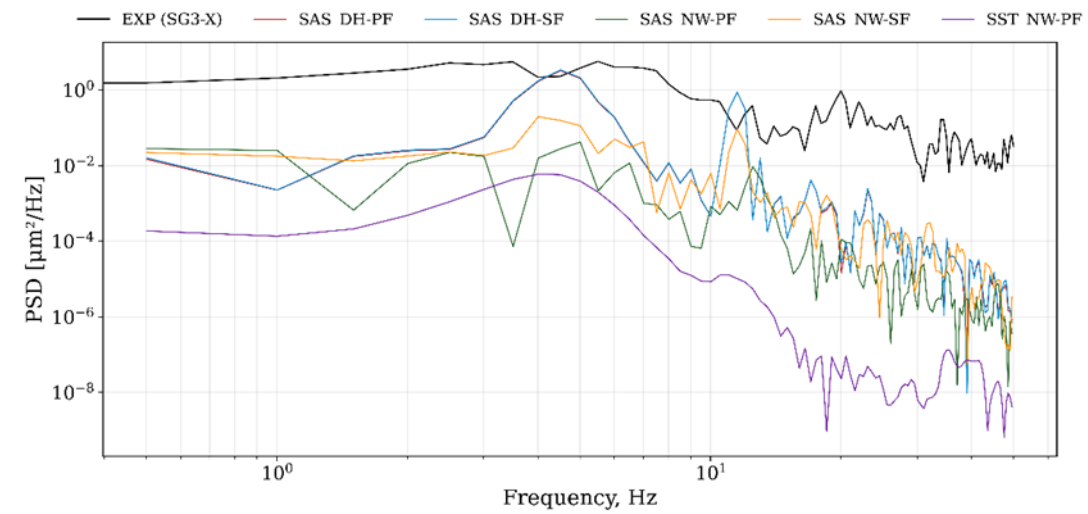


Figure 7-21: Vibration spectrum in X-direction for SG3

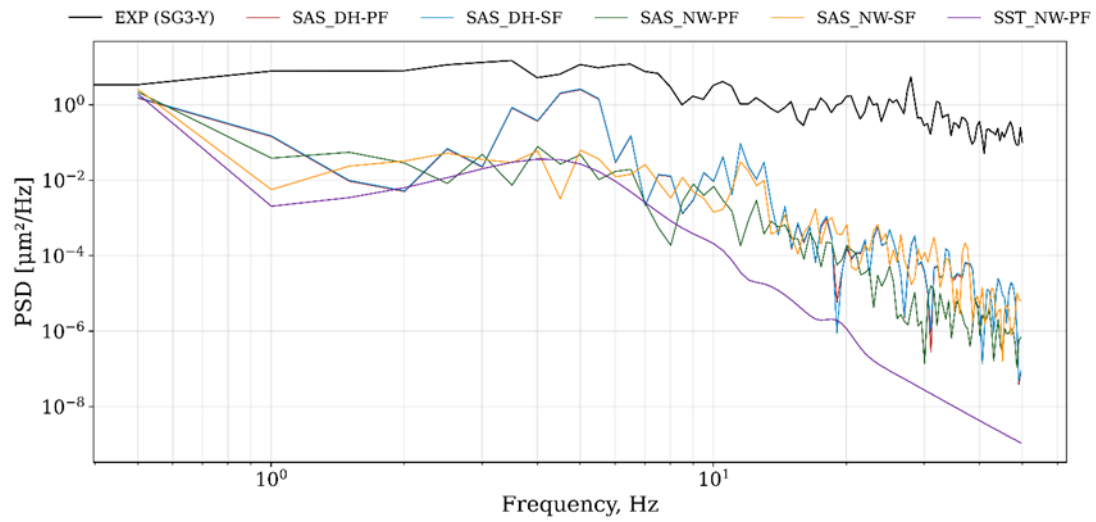


Figure 7-22: Vibration spectrum in Y-direction for SG3

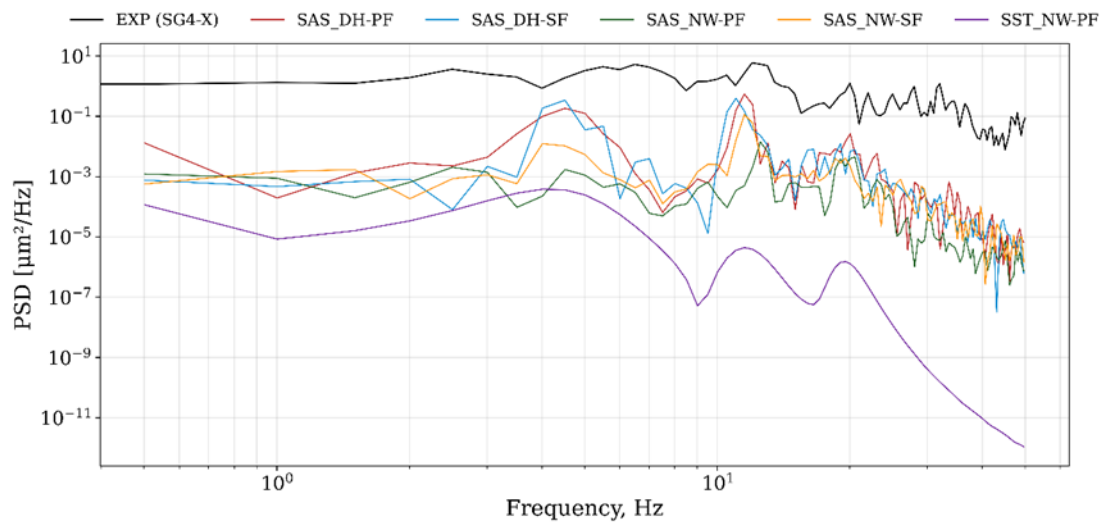


Figure 7-23: Vibration spectrum in X-direction for SG4

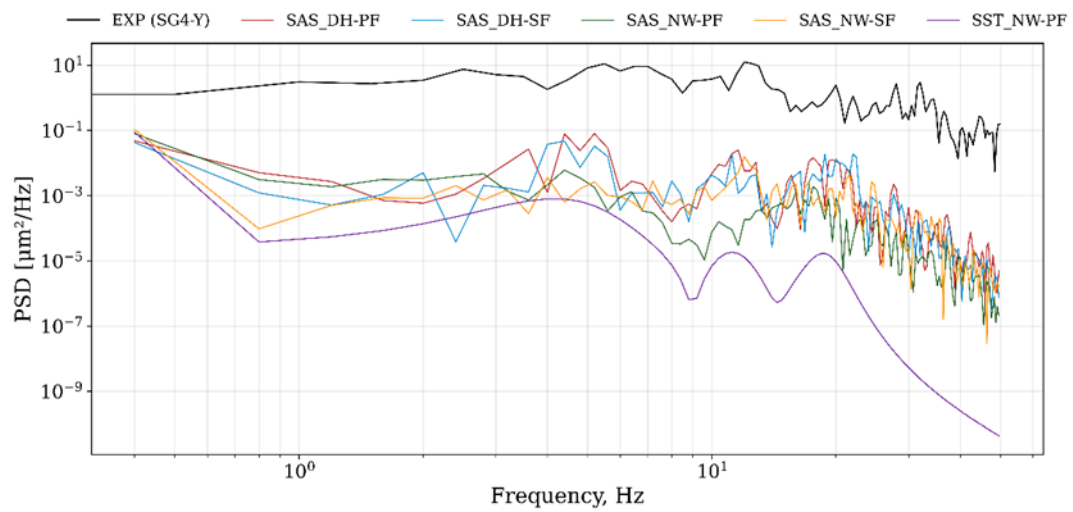


Figure 7-24: Vibration spectrum in Y-direction for SG4

In the low frequency ranges, the spectrum of vibrations is qualitatively similar to that observed in the experiment, but the amplitude of the vibrations is significantly lower. As noted earlier, this is related to the quality of reproduction of turbulent pulsations in the calculation model, as well as the ability to take into account local features and the individuality of each component in the mechanical model. There are two aspects worth noting here. From a scientific perspective, at low vibration amplitudes, it's important to qualitatively and quantitatively reproduce the turbulent structure that forms behind the spacer grid. In this case, the resulting hydrodynamic force largely depends on the prediction of the vortex structure that develops due to the loss of stability of the boundary layer and the transfer of this structure to the core of the flow. Due to the complexity of the spacer grids geometry, it is obvious that the flow structure is anisotropic and therefore can be more or less reproduced within URANS using Reynolds stress models, or even better, using SRS approaches. In this case, detailed discretization of the geometry (of the order of 10^8 cells) and the use of low-dissipation advection schemes that have a minimal effect on the eigenvalues associated with the flow are important. Such calculations require enormous computational resources, especially for practical purposes.

For engineering practice, such approaches are redundant and often unacceptable. For this reason, URANS turbulence models with one or two parameters on relatively coarse meshes remain preferred. As the results of ALAIN analysis show, the underestimation of the hydrodynamic force is associated with poor reproduction of the hydrodynamic picture behind the grids, and the spacer grid, in turn, does not allow for a relatively coarse discretization. From a practical standpoint, the simulated grid geometry should be simplified by introducing special additional angular momentum (empirical information) into the computational model. Such additional adjustments can be based on preliminary detailed calculations using SRS methods or on synthetic generators (e.g., physically calibrated Vortex Confinement [82]) with geometry (or flow swirl) correction. In a sense, SAS-SST is a model with a deterministic method for determining eddy additives, which made it possible to obtain vibrations (albeit smaller than in the experiment) in contrast to the SST model. Thus, supplementing the computational models of large-scale assemblies with empirical and synthetic information for both mechanics and turbulence can make it possible to estimate the vibration parameters of fuel assembly fragments in a reasonable time.

8 Benchmark

8.1 Introduction

In this chapter the simulation results from the partners participated to the benchmark are compared with each other and against experimental results.

As described in Section 3.5 the benchmark consists of two parts. First part is related to the CFD simulations with the objective to compare the simulation results for flow field profiles with the experimental data. The second part is related to the results of FIV simulations with the objective to compare the simulation results for vibration response of the fuel assembly with the experimental data.

From previous chapters, where the simulations performed by the partners participating in the benchmark and associated results achieved by these simulations are described and presented in detail, it is evident that all participants performed several simulations in combination with different models and simulation strategies. The aim of all these simulations was to test the performance of different models or to find the optimal setup for the simulations. Out of all simulation results, only one set of data from each participant is selected and used for the benchmark comparison.

Four contributing partners have participated to the benchmark:

1. Framatome GmbH (FRA-G),
2. Commissariat à l'énergie atomique et aux énergies alternatives (CEA),
3. Gesellschaft für Anlagen- und Reaktorsicherheit gGmbH (GRS) and
4. IPP Centre LLC (IPP) were involved in the benchmark. They performed numerical simulations using different simulation approaches and different tools.

Each partner used different simulation strategies and different simulation tools and models to simulate given benchmark case. All simulations consist of two principal parts:

- CFD simulation to calculate fluid flow,
- Structural dynamic simulation to calculate the response of the fuel assembly to the excitation induced by fluid flow.

Different strategies are used for the simulation, ranging from CFD simulation combined with imposed fuel assembly motion to test the effect of the structure motion on the fluid flow, over decoupled FIV simulation, in which CFD and structural dynamic simulations are performed separately, up to a fully coupled FSI simulation, in which CFD and structural models are coupled, and the vibration response is calculated simultaneously.

In Table 8-1, the CFD simulations performed by each partner involved in the ALAIN benchmark including some specific information about the CFD code used, model geometry considered, turbulence modeling approach as well as turbulence models are summarized. Figure 8-1 shows the strategies for mechanical modeling of the fuel assembly structure. All partners used some kind of reduced modeling of the fuel assembly structure. In the case of FRA-G, CEA and GRS the fuel assembly mechanical model is based on beam elements as the basic structure with additional elements added at the spacer grid positions to take into account the rigid connection between spacer grids and guide tubes and the clamping of the fuel rods in the spacer

grids. CEA used a detailed model in which both the skeleton (geode tubes) as well as the rod bundle (fuel rods) are modeled explicitly by beams and the spacer grids are modeled in detail featuring the fuel rod support. In the fuel assembly models of FRA-G and GRS, a single beam is used to represent the whole fuel assembly and additional elements are applied at spacer grid positions to increase rotational stiffness because of the fuel rod clamping in the spacer grids as well as the connection between spacer grids and guide tubes. On the other hand, IPP used a simplified 3DOF model consisting of three mass nodes accompanied with spring elements to reproduce the dynamic characteristic of the fuel assembly measured in the mechanical impact tests.

Table 8-1: Strategies for CFD simulations followed by different benchmark participants.

	FRA-G	CEA	GRS	IPP
CFD code(s)	STAR-CCM+	TrioCFD	OpenFOAM	ANSYS CFX
CFD geometry	One span Full-scale	One span Full-scale	One span	Full-scale
Turbulence modelling approach	LES DES	LES URANS	LES URANS	SAS URANS
Turbulence / SGS models	Smagorinsky WALE Spalart-Almaras Elliptic Blending SST	WALE k-e	Smagorinsky k-e realizable k-e qubic k-w SST RSM LLR	SAS-SST k-w SST

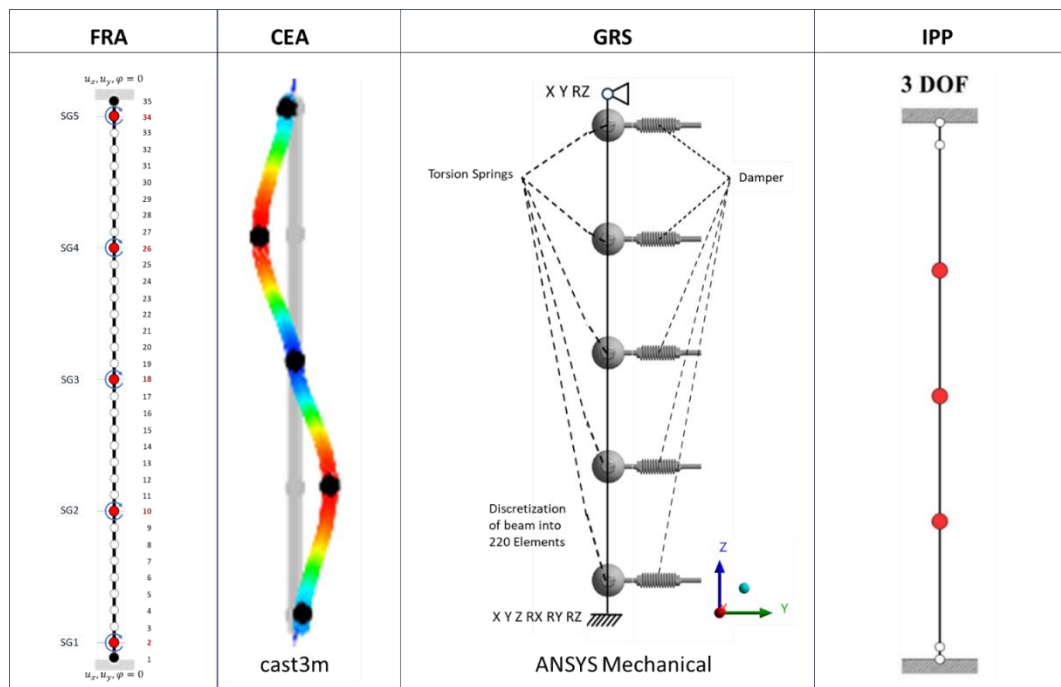


Figure 8-1: Strategies for mechanical modelling of fuel assembly structure.

In the following sections the comparison of selected simulation results for the flow field and fuel assembly vibration response produced by each partner with the experimental data is presented and discussed in detail.

8.2 Flow field

As stated above only one data set per organization is selected for final comparison. An overview of the results selected for the flow field benchmark is provided in Table 8-2, with additional details about performed simulations.

Note that only the results based on scale-resolving approaches for turbulence modeling are selected for the benchmark. These are LES in the case of FRA-G, CEA and GRS and SAS in the case of IPP. As described in Chapters 5 and 6 in the analyses presented by CEA and GRS, the approaches based on the use of URANS are not capable to reproduce unsteady turbulent fluctuations as a direct result of the CFD simulations, i.e., the turbulent velocity fluctuations are not resolved by the simulation. Therefore, the comparison of flow fluctuations with the experimental results was based on the recalculation of the flow fluctuations from the turbulent kinetic energy, which is calculated by the RANS models (section 5.2.1.2.3 and 6.1). Since these fluctuations are not resolved by the simulation, they cannot be used directly to generate the turbulent fluid forces to excite the structure. To do so, additional means are necessary such as special methods to calculate turbulence and to add it on the results of a RANS simulations. An example of such an approach is given in [28], where the pressure fluctuation equation is utilized to obtain turbulent pressure fluctuation from RANS simulations. The poor capability of RANS models to resolve turbulent fluctuations was confirmed by FSI simulations performed by IPP. The SST model has not produced any fuel assembly vibration (see Section 7.5). For this reason, the RANS models were not considered at all by FRA-G (see Section 4.2.1).

Table 8-2: CFD simulations used for the flow field benchmark.

	FRA-G	CEA	GRS	IPP
Domain	Full scale	Single span	Single span	Full scale
Turbulence modeling	LES	LES	LES	SAS
Turbulence / SGS model	WALE	WALE	Smagorinsky	SST
Resolution (number of cells)	245M	660M	5M	17M
Mesh/cell type	Trim-mesh*	Tetrahedral	Hexahedral	Hexahedral
Wall treatment	WM	WM	WM	WM
*) predominantly hexahedral mesh with prism layers at wall boundaries and so-called trim cells in the area between boundary layer and core mesh				

The flow velocity profiles from selected CFD simulations are compared directly with the experimental profiles in Figure 8-8 and Figure 8-9.

To facilitate further the analysis of the results and to provide a kind of quantitative comparison the equation (4.7) from section 4.2.7.1.1 is recalled here again:

$$\Delta = \left(\sum_i^N |\phi_{i,\text{sim}} - \phi_{i,\text{exp}}| \right) / \left(\sum_i^N |\phi_{i,\text{exp}}| \right) \quad (8.1)$$

Note that equation (8.1) is principally the same as equation (5.32) with the difference that the result of equation (8.1) is normalized by an appropriate norm calculated from the experimental results thus providing a non-dimensional value, while the equation (5.32) provides a physical values in form of an average deviation from experimental results.

8.2.1 Axial mean flow profiles

The comparison of the axial flow mean velocity profiles is shown in Figure 8-2, for traverse T1 on the left side and traverse T2 on the right side. As described in previous chapters, a good qualitative agreement between CFD and experimental results can be observed. The shape of the velocity profiles is generally well reproduced, with smaller or larger local deviations. At the position $z = 10\text{mm}$ behind the space grid the axial mean flow profiles calculated by FRA-G show a tendency to best follow the shape of the experimental profiles. The shape of the whole profile is captured quite well. Similar behavior can be observed also for the profiles from CEA and GRS. The profiles from IPP show the tendency to smoothen the profile peaks. At some locations the shape of the experimental profiles is captured quite well, but at many other locations the profiles are quite smooth compared to the experimental results that show stronger formed peaks. Similar behavior can be observed also for the next position $z = 20\text{mm}$ downstream the spacer grid. All simulation results capture quite well the significant peaks around the guide tubes. The profiles from IPP show again for the remaining part of the profile a smoothening tendency, while other three results follow generally the shape of the experimental profiles although not all peaks are predicted correctly. At axial position $z = 50\text{mm}$ the profiles from IPP, FRA-G and GRS predict reasonably the pronounced peaks around guide tubes, while the profiles from CEA do not show any special change in the form at these locations. This tendency of the CEA profiles can be observed also at positions $z = 100\text{mm}$ and $z = 250\text{mm}$. At position $z = 50\text{mm}$ the profiles from IPP still show some smoothening compared to other profiles which show more pronounced wavy form at this position. At axial positions $z = 100\text{mm}$ and $z = 250\text{mm}$ all profiles show similar and more flat form. Although all simulations results show a tendency to overestimate the axial velocity profiles at axial positions farther from the spacer grid the IPP results seem to be closest to the experimental results at these positions.

A quantitative representation of the deviation of the simulation results from the experimental data for the mean axial flow is provided in Figure 8-6.

8.2.2 Cross mean flow profiles

The comparison of the cross flow mean velocity profiles is shown in Figure 8-3, for traverse T1 on the left side and traverse T2 on the right side. Like for the axial flow mean velocity profiles, generally a reasonable qualitative agreement can be observed also in this case with local deviations especially at locations closer to the spacer grid. As for axial mean flow profiles, also for the mean cross flow profiles the results from IPP show a tendency for smoothening the profiles. This is especially the case at axial positions farer from the spacer grid $z = 50, 100$ and 250mm . Other results look similar with some local exceptions like e.g. at axial position $z=20\text{mm}$ where the profiles from GRS show significantly larger peaks on the left side channel wall. A quantitative comparison is provided in Figure 8-7 which principally confirms the analysis provided above.

8.2.3 Axial flow fluctuations profiles

Figure 8-4 shows the comparison of the axial flow RMS velocity profiles. A general tendency that can be observed is that the IPP results show quite flat profiles at all positions and underestimate the experimental results at axial positions close to the spacer grid. This is consistent

with the smoothening effect observed for mean flow profiles. All other results show certain similarity with the experimental results although could not predict correctly the whole shape. The results from GRS also show a tendency to underestimate the flow fluctuations close to the spacer grid. At the positions farther from the spacer grids the prediction gets better. It looks like the profiles from RFA-G follow the best the experimental profiles. Especially interesting are the correct predictions of significant peaks around guide tube at the profiles along T2 at positions $z = 50\text{mm}$ and $z = 100\text{mm}$. Such peaks exist in the CFD results also on the traverse T1 close to the guide tube, but they do not exist in experimental results. The question that arises at this place is whether there is an uncertainty in the experimental results. The results from CEA predict quite good the level of fluctuations but the profiles have a much wavier form compared to other simulation results and to experimental data.

A quantitative comparison for the axial velocity fluctuations is provided in Figure 8-8. It reflects generally the analysis provided above.

8.2.4 Cross flow fluctuations profiles

The profiles plot showing the comparison of the cross-flow velocity fluctuations are shown in Figure 8-5. The trends observed for axial velocity fluctuations can also be observed in the case of cross-flow velocity fluctuations with an exception that the IPP results in this case predict very well the level of fluctuations, although the profiles are still quite flat as in the case of axial flow fluctuations. Other results are very similar to the results obtained for axial flow fluctuations. A quantitative comparison is provided in Figure 8-9. The observation related to the results of IPP can be clearly seen in these plots.

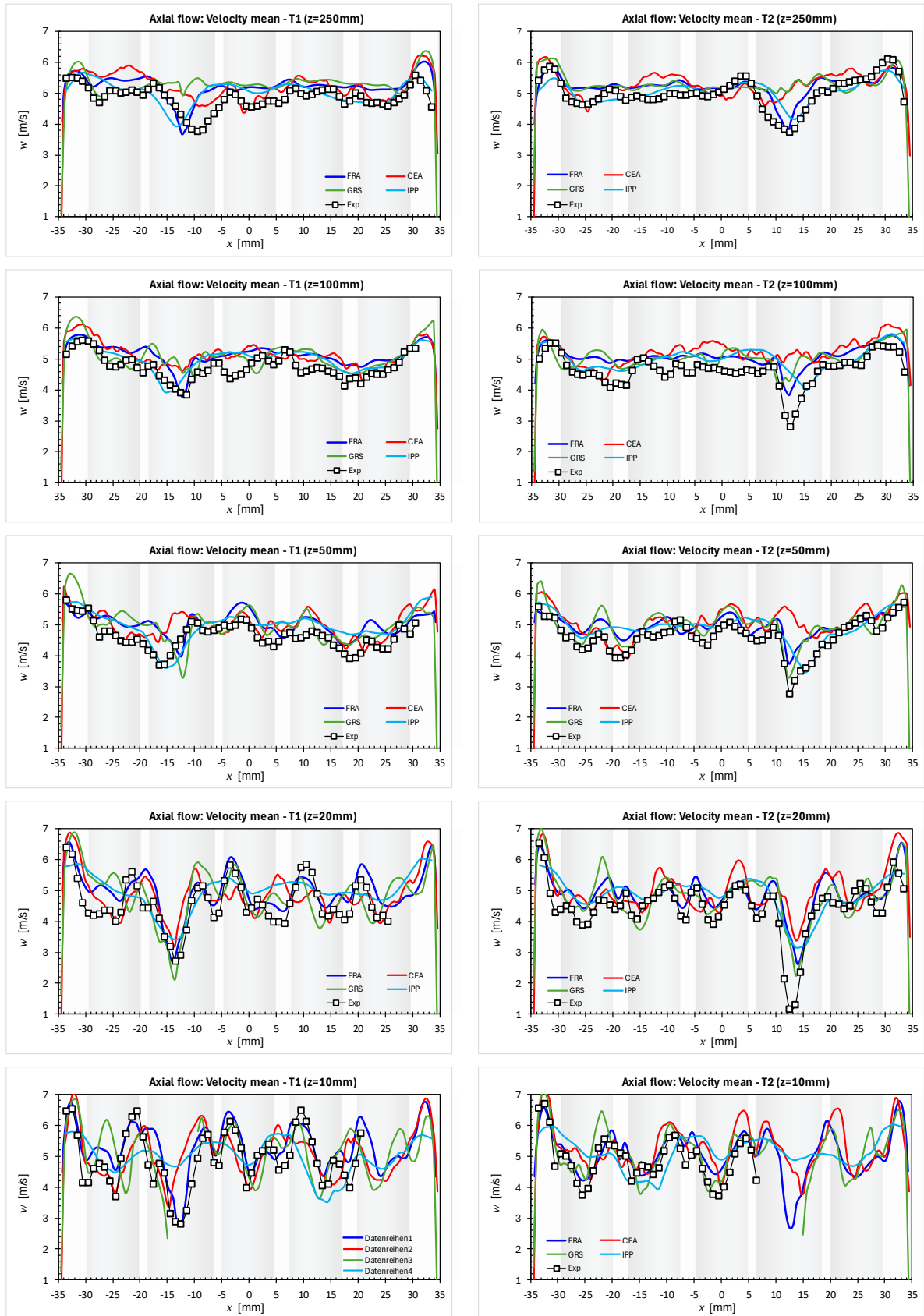


Figure 8-2: Benchmark - axial flow mean velocity profiles - traverses T1 and T2.

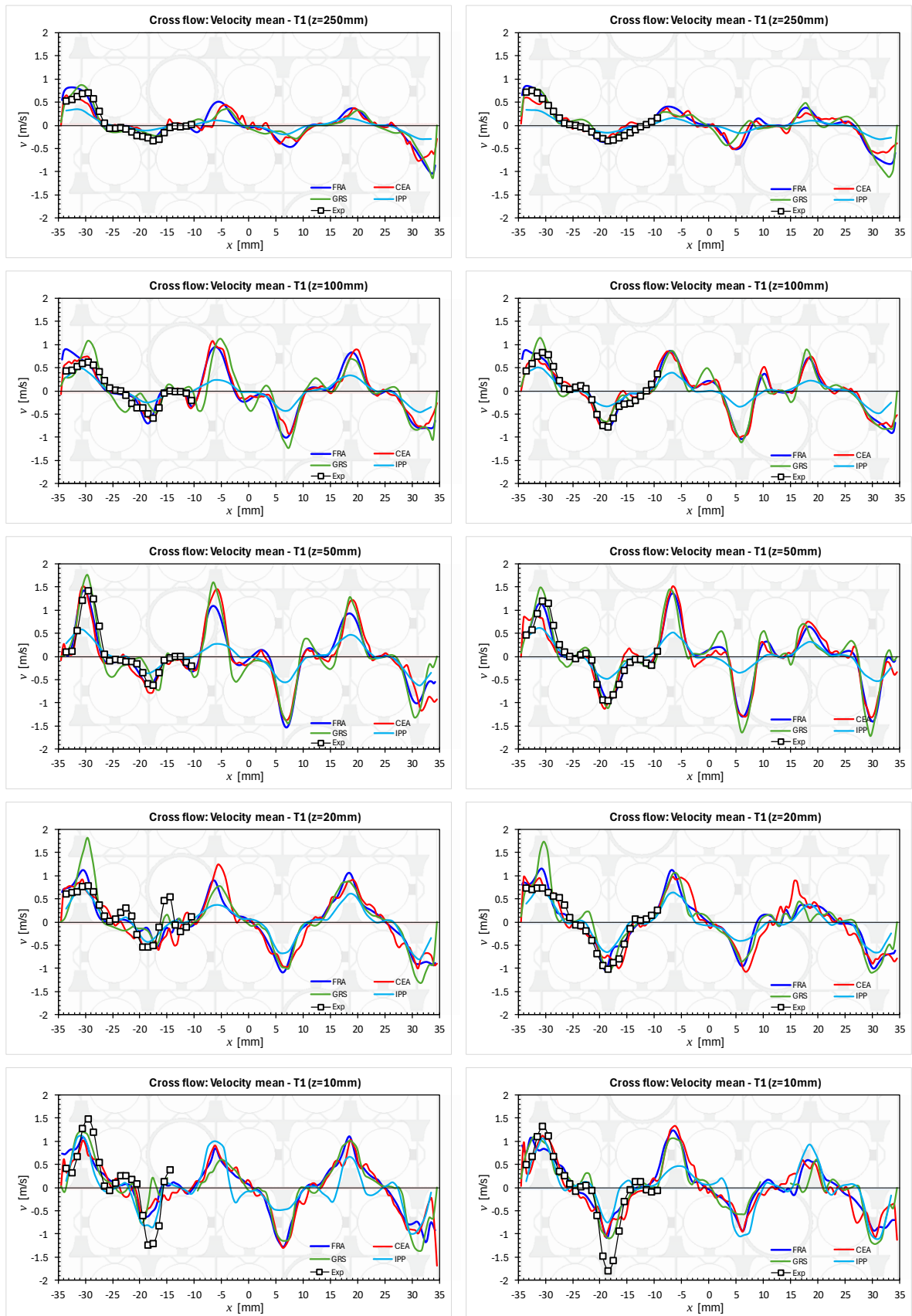


Figure 8-3: Benchmark - cross flow mean velocity profiles - traverses T1 and T2.

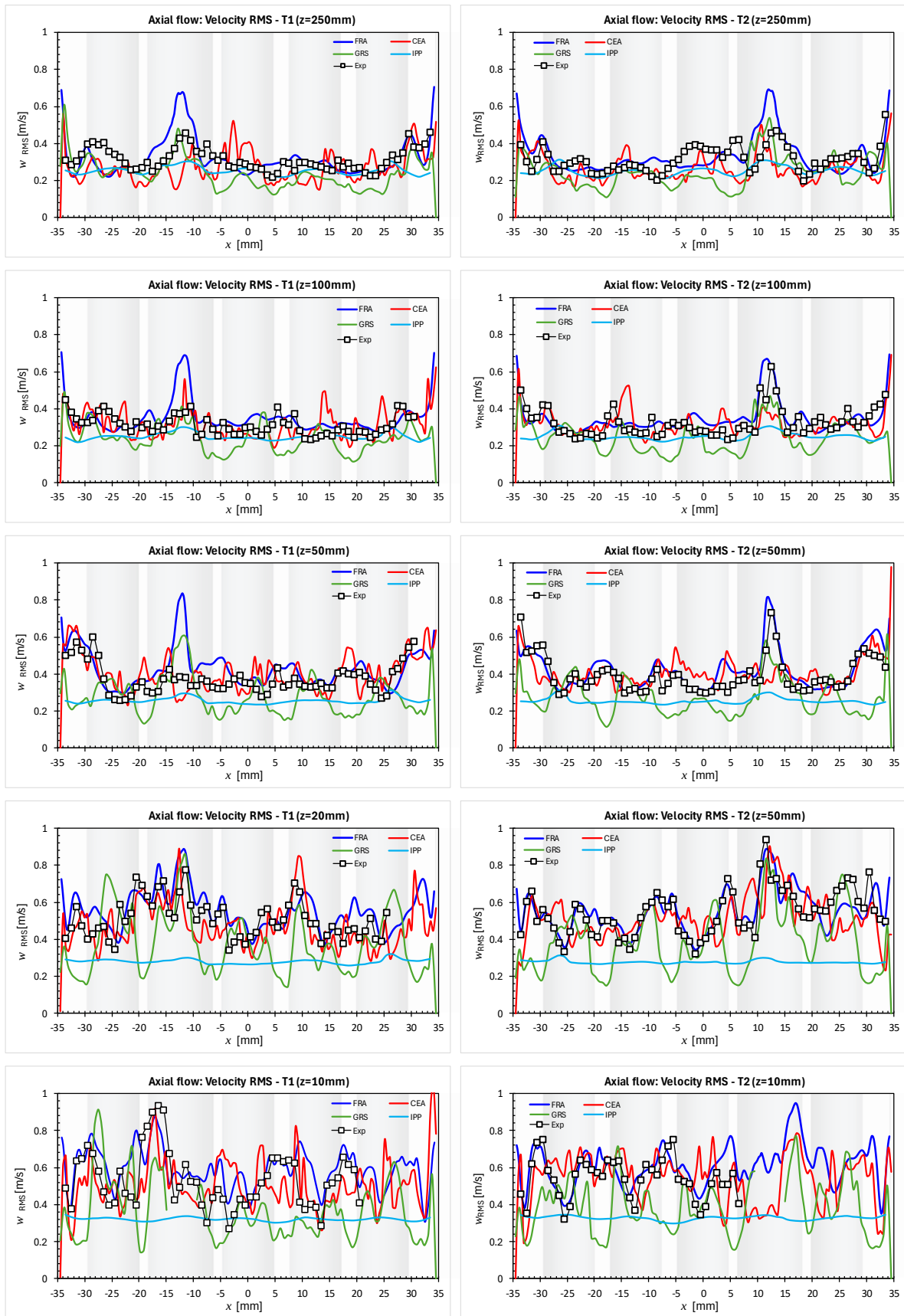


Figure 8-4: Benchmark - axial flow RMS velocity profiles - traverses T1 and T2.

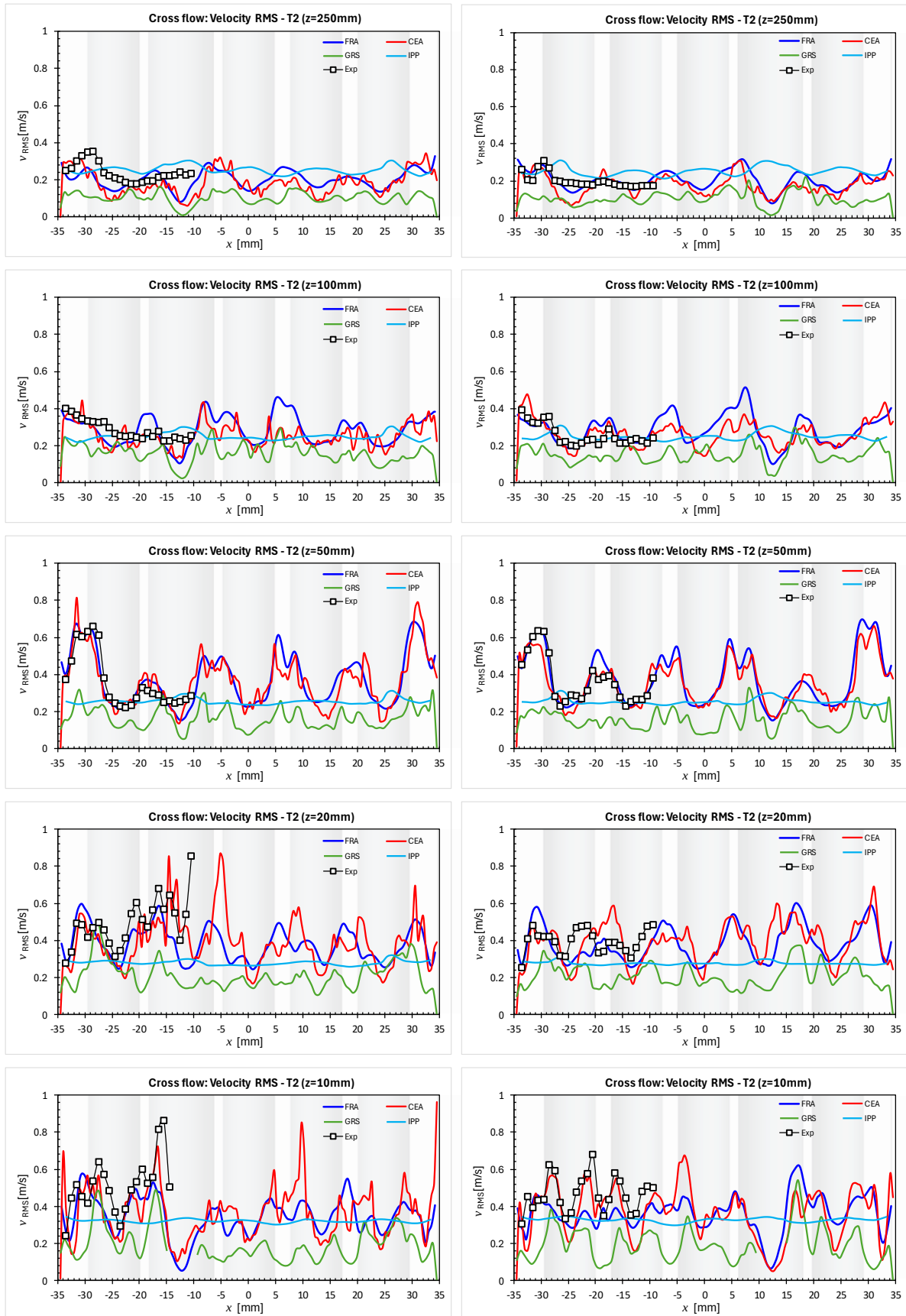


Figure 8-5: Benchmark - cross flow RMS velocity profiles - traverses T1 and T2.

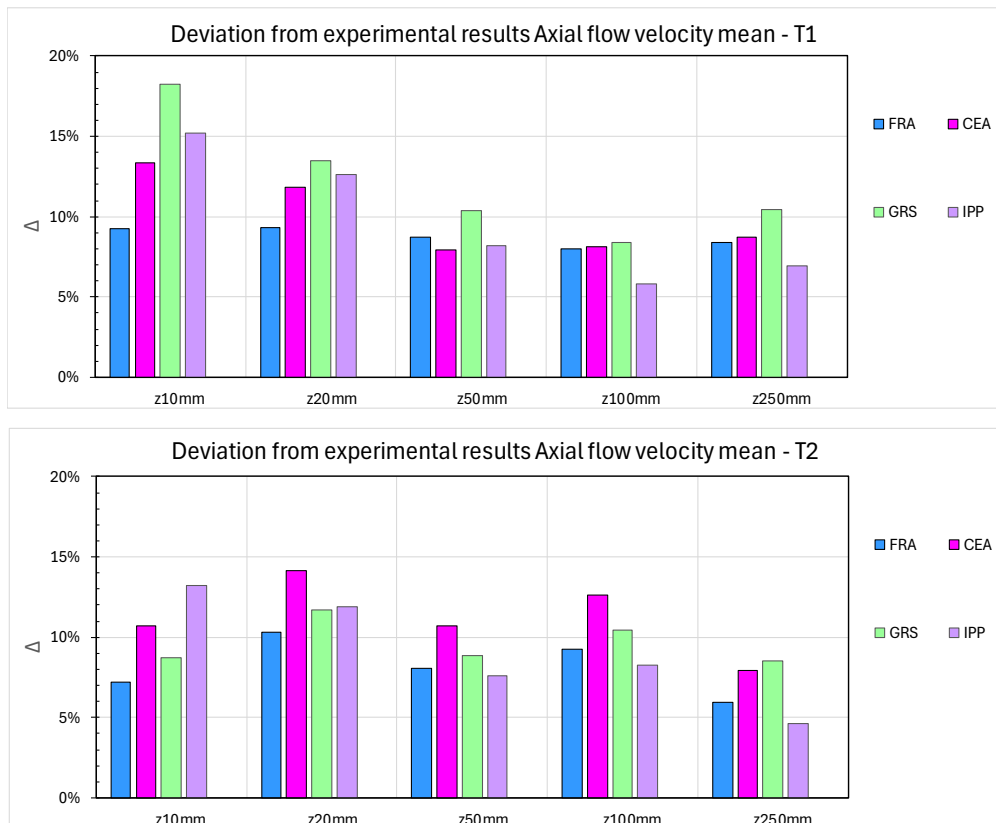


Figure 8-6: Benchmark: Axial flow field mean velocity profiles - deviation form experimental results.

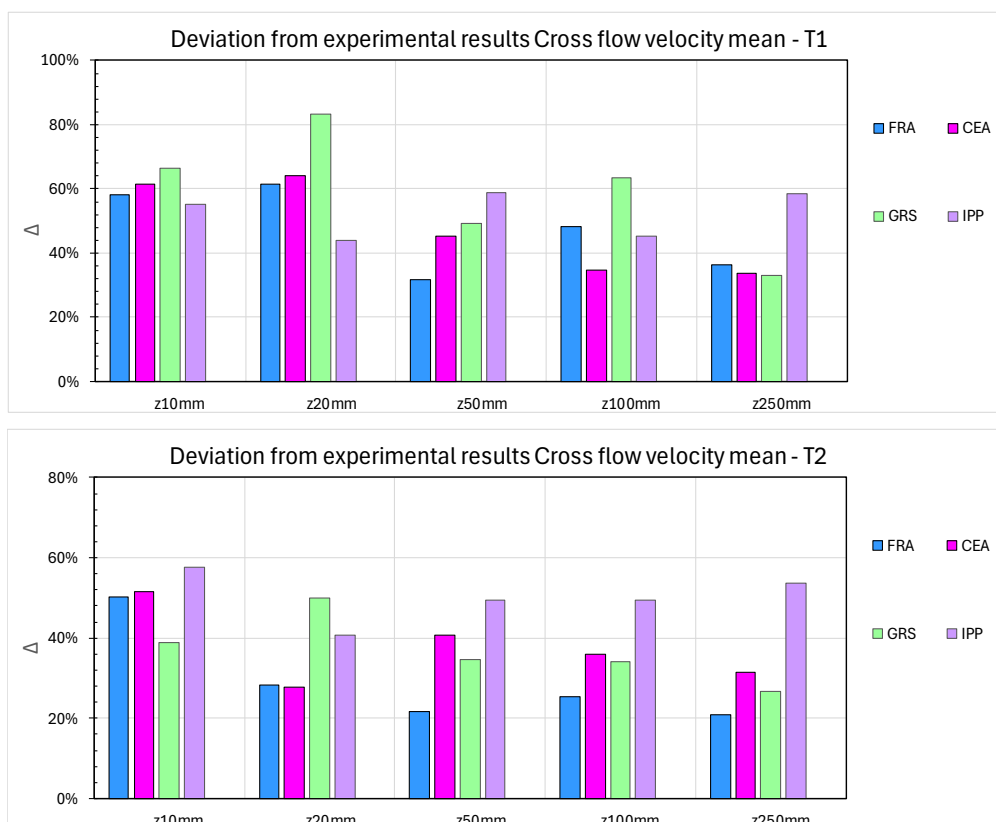


Figure 8-7: Benchmark: Cross flow field mean velocity profiles - deviation form experimental results.

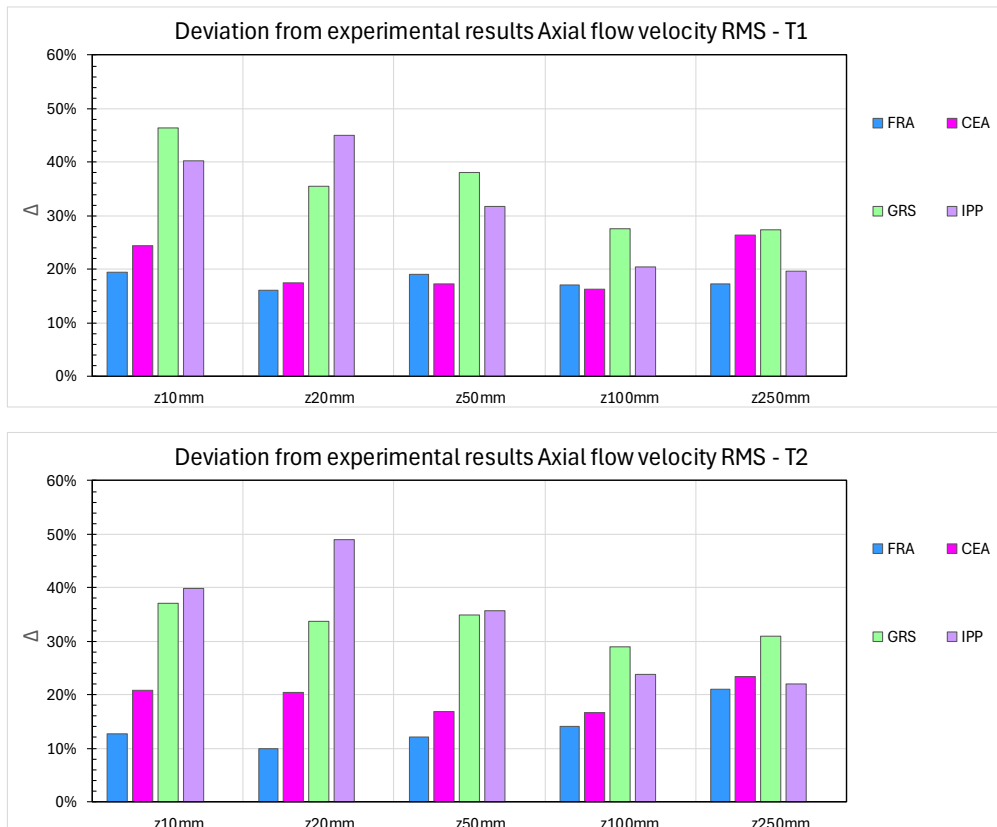


Figure 8-8: Benchmark: Axial flow field RMS velocity profiles - deviation form experimental results.

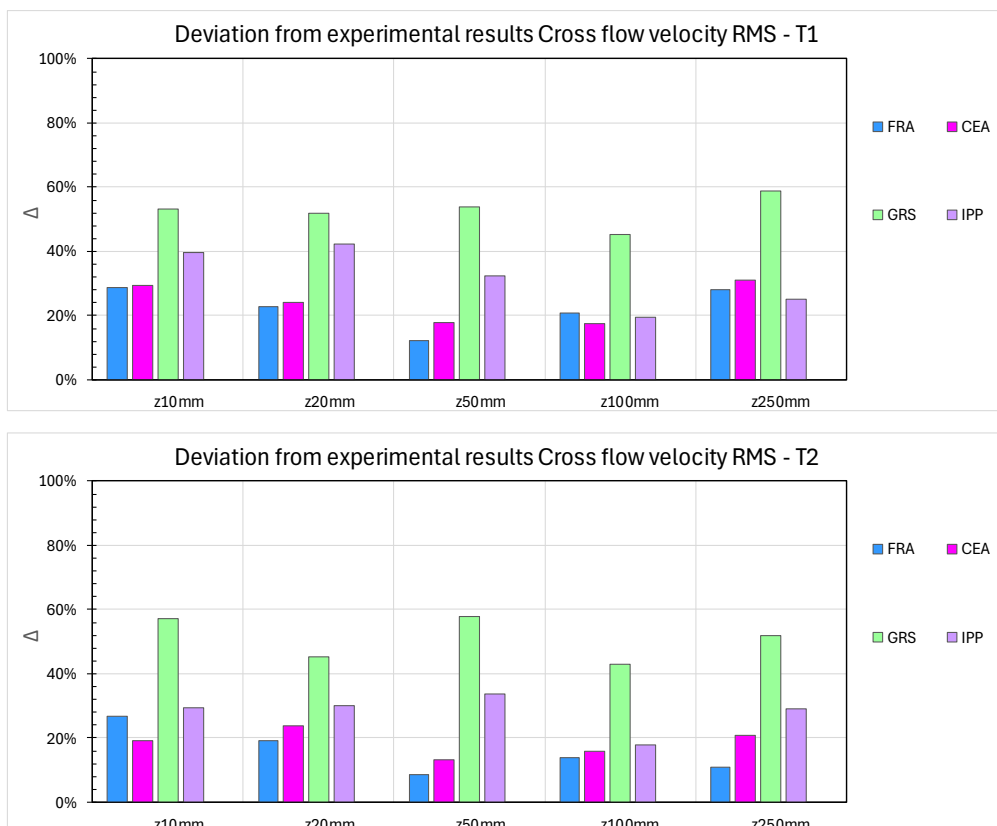


Figure 8-9: Benchmark: Flow field RMS velocity profiles - deviation form experimental results.

8.3 FIV fuel assembly response

In this section the second part of the benchmark is presented. The results related to the vibration response obtained by different partners are compared with each other and with experimental results. Note that CEA has not performed the FIV simulations i.e. they have not predicted the FIV response of the fuel assembly. Therefore, the results from FRA-G, GRS and IPP are included in this part of the benchmark. It should also be noted that CEA performed simulations with imposed fuel assembly motion, where the motion of the fuel assembly was prescribed based on the experimental results. The purpose of these simulations was to check whether the fuel assembly motion at given level affects the flow field and to derive the fluid parameters relevant for the fuel assembly dynamics. In this context the effort done by CEA is greatly acknowledged although they do not contribute directly to this second part of the benchmark.

As stated in Section 3.5 the parameters of interest for this part of the benchmark are related to the vibration response of the fuel assembly. These are reflected through vibration amplitude and vibration frequencies.

Note that FRA-G and GRS performed decoupled FIV simulations, while IPP performed coupled FSI simulations. As in the case of the CFD results discussed in Section 8.2, only one set of results is considered for the benchmark comparison. This applies particularly to IPP since they performed several FIV simulations using different models and settings. For the benchmark the data set that provides the best representative result is selected. It is the case denoted as “SAS-DH-PF”. For more detailed about all results provided by IPP the reader is referred to Chapter 7.

GRS describes also two sets of results from their simulations. One result obtained with a fluid damping coefficient of $D = 0.5$ and the other result obtained for the fluid damping coefficient $D = 0.8$. For the benchmark comparison only the results corresponding to $D = 0.5$ are used.

Table 8-3 shows the RMS values of the lateral displacements obtained from simulations and including also the experimental results. The RMS values correspond to dynamic part of the response, which means that the static component is removed. Further, for the experimental results for the spacer grid level 2 and displacement u_y an averaged value of the experimental results provided for sensors “L” and “R” that were installed at this position (see reference [1] for details of the experimental measurements and results). The deviations to the experimental results are provided in Figure 8-10.

The following conclusions can be drawn from the RMS displacements results:

- The results from FRA-G and GRS are quite similar. They both overestimate in general the vibration amplitude by up to 85%. There are some differences in the resulting displacement which can be attributed to different boundary conditions applied to the mechanical model at the top side of the fuel assembly. In the model of GRS, the upper end of the fuel assembly is guided laterally, while in the model of FRA-G fuel assembly is fixed at both ends.
- The results from IPP underestimate the vibration response by up to 85%. This underestimation is most likely to be attributed to the turbulence modeling approach and associated turbulence model used in the CFD simulations. As was seen on the flow field profiles the

SAS-SST model used by IPP tends to smoothen the flow field, especially the flow fluctuations, which are the essential part of the CFD simulation. Underprediction of the flow fluctuations leads to too low fluid forces that excite the structural vibration. It is not clear what is exactly the reason for this underperformance of the selected SAS model. Is it due to insufficiently fine numerical mesh used in the CFD model or is it due to some other reasons, e.g. model settings which are not optimal for given case. The numerical mesh used in the CFD model is probably rather coarse than fine. Further refinement is however beyond the reach at present time for this configuration, since the computational effort would not be manageable anymore.

Table 8-3: Benchmark: Comparison of the RMS of lateral displacements u_x and u_y .

	Displacement u_x [μm]			Displacement u_y [μm]		
	SG2	SG3	SG4	S2	SG3	SG4
FRA-G	8.35	10.99	8.24	6.55	8.23	6.88
GRS (D05)	7.04	10.49	11.15	6.66	9.09	9.87
IPP (SAS-DH-PF)	1.59	1.32	0.95	2.03	3.71	1.90
Experimental data	5.25	5.96	6.42	4.67	6.3	9.14

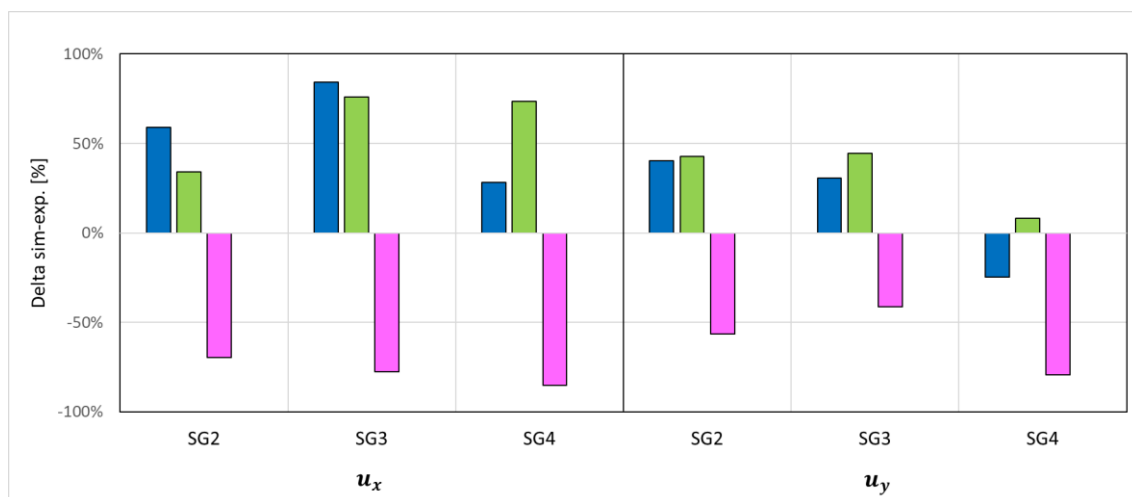


Figure 8-10: Benchmark: RMS of lateral displacements u_x and u_y – deviation from experimental results.

Figure 8-11 shows the time histories of the lateral displacements obtained by different simulations. The time history plots reflect the RMS results presented above. It can also be seen on the time history plots that the results from FRA-G and GRS look similar, while the displacement time histories calculated by IPP show significantly lower amplitude.

Finally, in Figure 8-12 the PSD plots of the lateral displacements calculated from the time histories shown in Figure 8-11 are presented. Due to limiting length of the time signals it was difficult to calculate the PSD of high quality. Nevertheless, some general trends and conclusions can still be drawn from these plots. The discrepancies in the amplitude described above can also be seen very well from the PSD plots. Again, the results from FRA-G and GRS look quite similar. Some significant differences can be observed in the higher frequency range, which is of less interest. It is hard to identify exactly the peaks in the frequency spectrum which

are associated to corresponding frequencies of the 1st, 2nd and 3rd vibration modes. The corresponding frequencies are indicated in the plots for the experimental results. One can see how the simulation results match these frequencies. The corresponding frequencies are not matched exactly, but they are in most of cases close to the corresponding experimental results. Thus, a qualitative reproduction of the vibration response can be generally confirmed.

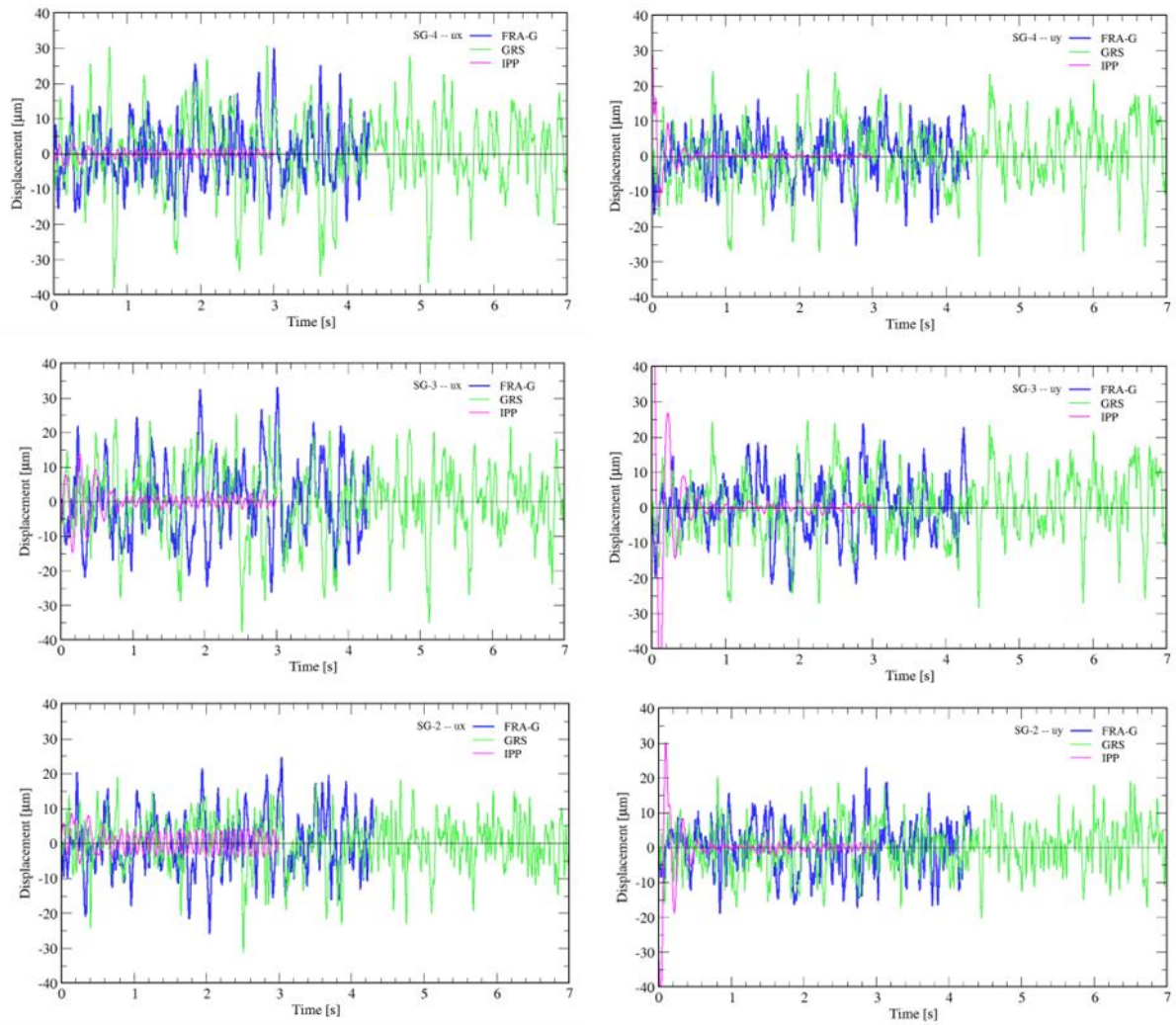


Figure 8-11: Benchmark: Comparison of time histories of the lateral displacements u_x and u_y .

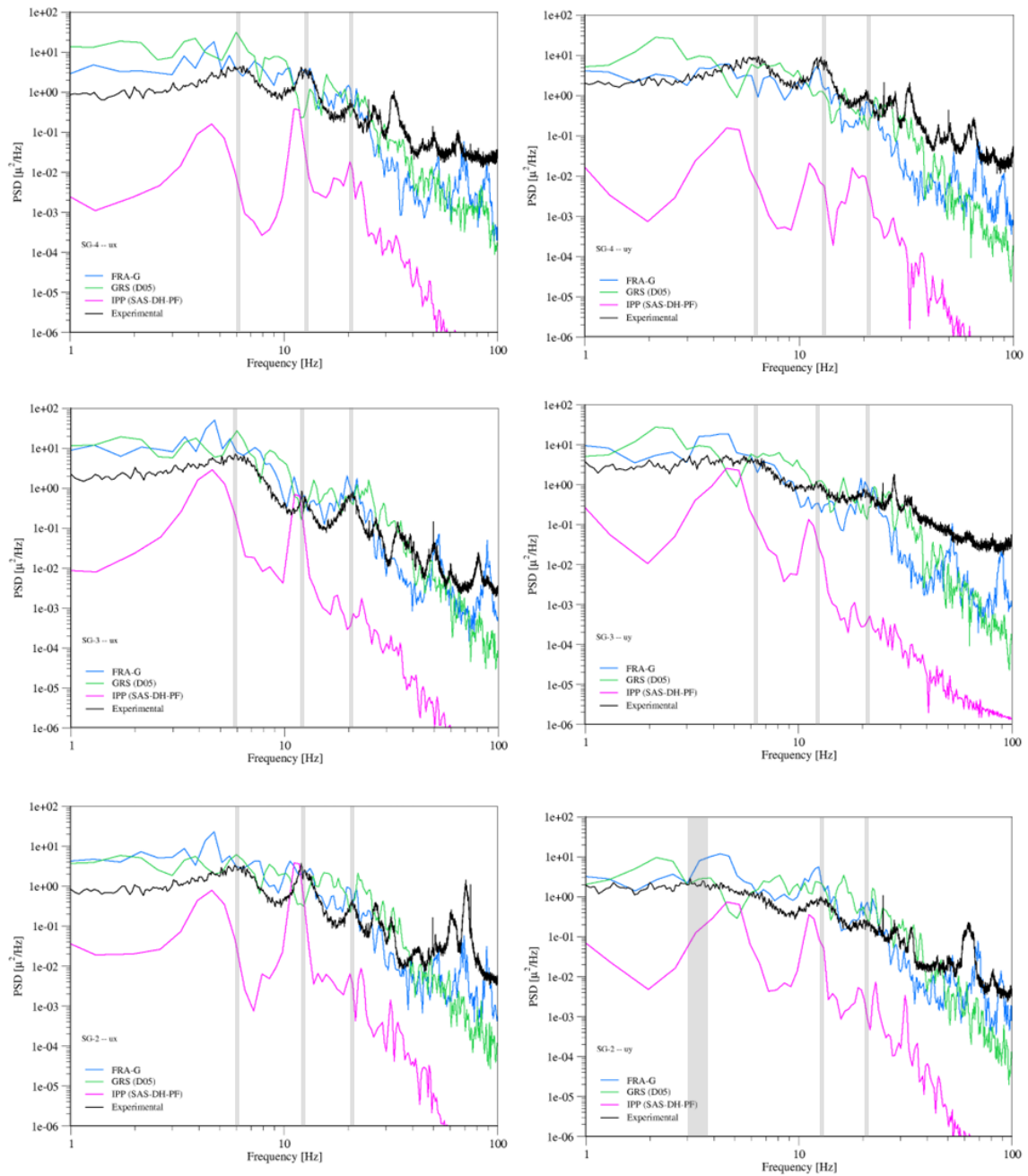


Figure 8-12: Benchmark: Comparison of the PSD of lateral displacements u_x and u_y .

8.4 Benchmark summary and conclusions

In this chapter, the ALAIN benchmark case was discussed. Two aspects of the benchmark were considered: the flow field and the FIV response of the fuel assembly. Four project partners participated to the benchmark provided their results for flow field comparison: FRA-G, CEA, GRS and IPP, and three partners performed the FIV simulations and provided the results for the FIV part of the benchmark: FRA-G, GRS and IPP. Each participant used different modelling approaches and simulation strategies: pure CFD without or with imposed fuel assembly motion, decoupled FIV simulation and coupled FSI simulation. Further, different strategies were used with respect to the computational domain. In this context reduced-scale single span

model and full-scale model of the whole setup were used for different purposes. Also, different strategies for the mechanical modeling of the fuel assembly structure were used. Such a wide range of differences make the benchmark quite rich, but it also brings forth the challenge of identifying a single aspect of the simulation strategy that can be said to make or break the accuracy of the calculations.

All participants performed several simulations using different physical models and considering different geometrical domain. Nevertheless, only one data set per participant was selected for final comparison. For the flow field part of the benchmark the results from CFD simulations with scale resolving turbulence modeling approach were selected. These CFD simulations are typically used also in combination with mechanical models to calculate the FIV response.

Based on the compared results the following key comments can be made:

- The benchmark results presented in this chapter show that the vibration response of the reduced-scale fuel assembly analyzed in the ALAIN benchmark can be predicted qualitatively in terms of both vibration frequency and vibration amplitude. Quantitatively an accuracy of $\pm 85\%$ of the vibration amplitude measured in the FIV tests could be achieved. Considering a very small vibration amplitude measured in the tests, of just a few μm RMS, and the overall uncertainties of the measurements, which could not be quantified exactly, the achieved results are quite reasonable.
- Decoupled simulation approach seems to be sufficient to predict reasonably the vibration response for such small vibration amplitude. The results of the CFD simulation with imposed fuel assembly motion (not a part of the benchmark) have shown that there is no effect of this kind of motion with such a small amplitude on the flow field and therewith conclusively also on the turbulence-induced fluid forces, which justifies the decoupled simulation approach.
- It needs to be noted in the context of decoupled simulation approach that the effects of fluid in terms of added (hydrodynamic) mass, fluid damping and fluid stiffening play also an essential role for reliable prediction of the fuel assembly vibration response. The effect of these parameters is probably in the same order as the effect of turbulence-induced excitation forces. In this context effort needs to be made to evaluate these parameters properly.
- Reliable prediction of turbulence-induced forces seems to be of essential importance for good FIV prediction. This in turn means that the correct selection of turbulence modeling approach and associated turbulence models is key for a successful simulation. This, however, means that scale-resolving simulation approaches are required, which are computationally more demanding due to general requirement for finer meshes, compared to traditional industrial CFD approach based on RANS turbulence models. The use of LES in combination with wall modeling, which is considered as the only feasible option for this case, seems to be a good choice. Nevertheless, even with this significant reduction of the computational cost, these are still prohibitively high and hard to manage.
- The SAS-SST model showed strong damping tendency leading to a significant underprediction of the vibration amplitude in the coupled FSI simulations. It is not clear what is exactly the reason for this behavior, whether it is just insufficiently fine mesh or some other model features. This might be out in focus of some future investigations.

9 Overall summary and conclusions

Deliverable D2.4 successfully concludes the comprehensive numerical benchmarking campaign for the ALAIN Benchmark marking the achievement of Task 2.2 within the GO-VIKING project. This report not only evaluated the predictive capacity of the methodologies for prediction of FIV of fuel assemblies but also stands as a testament to the effective collaborative efforts of the contributing partners: FRA-G, CEA, GRS and IPP.

The core objective of the task was to compare the solutions obtained from different computational methods against the experimental data from deliverable D2.3, and to assess the capability of different modeling approaches and simulation strategies in their predictive capability of the complex flow field within a reduced-scale fuel assembly and of the vibration response of the fuel assembly to flow-induced excitation when subjected to axial water flow. This objective was achieved with the following takeaways from all the simulations performed by the participants including the benchmark results:

- The turbulence modeling approach used in the CFD simulation plays obviously a significant role for the predictability of the vibration response. Based on very small vibration amplitude measured in the FIV tests, turbulence is assumed as the main driving vibration mechanism. According to this assumption, it is essential to use turbulence modeling approaches and turbulence models that are capable to resolve relevant turbulent flow structures which are responsible for excitation of the fuel assembly structure.
- Scale-resolving simulation class of approaches turned out as the right choice. Thereof, LES was found to be the best performing one in combination with wall-modeling, which is the only feasible option in combination with LES for given configuration. Wall-resolved LES would be out of reach. Other similar approaches like DES were found to underperform in direct comparison with wall-modeled LES by performing CFD simulations with LES and DES on the same mesh. Thus, the advantages of DES in aerodynamic applications cannot be really confirmed for wall-confined flow in the rod bundle. The SAS approach used in combination with different CFD code and different mesh showed underperformance in prediction of flow fluctuations compared to LES. It is, however, not quite clear whether this underperformance is just associated to insufficiently fine mesh or is it because of the model itself. This issue is worthwhile further investigations to assess better the performance of the SAS modeling approach for application to FIV simulation.
- The modeling approaches based on RANS turbulence models show very poor performance in prediction of turbulent flow fluctuations and therewith also in prediction of FIV for given case. Therefore, this class of turbulence models is not suitable for application to FIV, especially in the cases when the FIV is driven by turbulence. In some other case, when other flow features such as vortex shedding are driving the vibration the application of RANS models could be successful.
- Computational effort turned out to be a significant limitation in the analysis of the ALAIN benchmark. The requirement for more advanced turbulence modeling approaches such as LES makes it even more difficult to manage a CFD simulation of this very challenging configuration. This was also a significant limitation in terms of performing coupled FSI simulations, which proved to be unfeasible for most participants.

- In the context of computational effort, a reduction of the computational domain may be a useful simplification. This is especially recommended for the purpose of assessment of different models or in general different simulation approaches. For given configuration a single span model represents such a useful geometrical reduction. The assessment of the performance of different models can be done more efficiently on single span configuration. When the assessment is completed the best performing model i.e. approach can then be applied to the full-scale model.
- Given the fact that the vibration amplitudes measured in the FIV tests were very small, just a few μm RMS, decoupled simulation approach seems to be sufficient for the simulation of FIV in this case. The results of the CFD simulation with imposed fuel assembly motion performed by CEA have shown that there is no effect of this kind of motion with such a small amplitude on the flow field and therewith conclusively also on the turbulence-induced fluid forces, which justifies the use of decoupled simulation approach. However, it needs to be noted in the context of decoupled simulation approach that the effects of fluid in terms of added (hydrodynamic) mass, fluid damping and fluid stiffening play also an essential role for reliable prediction of the fuel assembly vibration response. The effect of these parameters is probably in the same order as the effect of turbulence-induced excitation forces. In this context, an effort needs to be made to evaluate these parameters properly either by simulations or by other means like e.g. from the available test data. In each case one must be careful in evaluation. As it was shown by CEA the evaluation of these parameters by simulation may be a nontrivial task.
- With respect to mechanical modeling of the fuel assembly structure, the use of the beams as the basic structural elements in combination with additional elements (e.g. springs) was proven to be a good choice. This modeling approach keeps the computational costs of the mechanical simulation at very low level and practically negligible compared to the computational costs of a CFD simulation, while providing sufficient accuracy for the results of interest. However, this approach may affect the strategy for coupled FSI simulations. Most of standard tools for FSI coupling are based on the common sharing boundary surface between fluid and structural domain. This boundary does not exist explicitly when the fuel assembly is modeled using beam elements. Hence, an alternative strategy is needed to establish the interface between fluid and structural domain in a coupled FSI simulation according to discretization of the structural dynamic model. This, in turn, requires in general special non-standard solutions for FSI coupling.
- Finally, the uncertainties of the experimental measurements, which could not be evaluated exactly, may play a non-negligible role for the assessment of the simulation results. In future experimental investigations more attention should be paid to the assessment of the measurement uncertainties to have a better basis for the assessment of the simulation results. Some additional measurements may also be of big benefit. Some examples are the measurement of the fuel assembly lateral stiffness and measurements of the lateral static displacement under axial flow.

10 Bibliography

- [1] H. Hadžić. GO-VIKING deliverable D2.3: ALAIN 5x5 PWR rod bundle experiment and benchmark setup, Version 1.9, November 2022.
- [2] International Atomic Energy Agency (IAEA). Review of Fuel Failures in Water Cooled Reactors (2006-2015), An update of IAEA Nuclear Energy Series No. NF-T-2.1. IAEA, Vienna 2019.
- [3] J. Stabel, M. Ren, B. Ladouceur, New knowledge and experiences of flow induced fretting in PWR fuel assemblies, 18th International Conference on Structural Mechanics in Reactor Technology (SMiRT 18), Beijing, China, August 7-12, 2005.
- [4] K. H. Haslinger, P. F. Joffre, L. Nordström, S. Anderson, Flow Induced Vibration Testing of a PWR Fuel Assembly, SMiRT 16, Washington DC, August 2001.
- [5] J. Vallory, Methodology of PWR fuel rod vibration and fretting evaluation in HERMES facilities, Structural behavior of fuel assemblies for water cooled reactors, Cadarache, France, 22-26 Nov. 2004, pp 209-219, IAEA July 2005.
- [6] A. Hatman, G. Williams, M. Martin, T. Keheley, C. Lascar, K. Goodheart, A. Chatelain, CFD analysis of reactor core flow field in support of FIV diagnosis, The 15th International Topical Meeting on Nuclear Reactor Thermal - Hydraulics, NURETH-15, Pisa, Italy, May 12-17, 2013.
- [7] X. Zhang, S. D. Yu, Large eddy simulation of turbulent flows surrounding two simulated CANDU fuel bundles, Nuclear Engineering and Design, Vol. 241: 3553-3572, Elsevier 2011.
- [8] A. M. Elmahdi, R. Lu, M. E. Conner, Z. Karoutas, E. Baglietto, Flow Induced Vibration Forces on a Fuel Rod by LES CFD Analysis, The 14th International Topical Meeting on Nuclear Reactor Thermal Hydraulics NURETH-14, Toronto, Ontario, Canada, September 25-29, 2011.
- [9] M. A. Christon, R. Lu, J. Bakosi, B. Nadiga, Z. Karoutas, M. Berndt, Large-Eddy Simulation, Fuel Rod Vibration and Grid-To-Rod Fretting in Pressurized Water Reactors, Journal of Computational Physics, Vol 322: 142-161, Elsevier 2016.
- [10] K. Bengtsson. FSI-analysis on vibrations of a slender rod exposed to axial flow. Master thesis, Chalmers University of Technology, Göteborg, Sweden 2016.
- [11] S. Okui, Y. Kubo, S. Kakinoki, R. Y. Lu, Z. Karoutas, T. Ikeno. Flow-induced grid-to-rod fretting test and simulation for NFI PWR fuel assembly. Proceedings of the 2013 21st International Conference on Nuclear Engineering, ICONE21, Chengdu, China, July 29 - August 2, 2013.
- [12] W. Wagner et al., "The IAPWS Industrial Formulation 1997 for the Thermodynamic Properties of Water and Steam," ASME J. Eng. Gas Turbines and Power, 122, 150-182 (2000).
- [13] R. D. Blevins. Flow-induced vibration. Litton Educational Publishing, Inc. 1977.
- [14] S. Kaneko, T. Nakamura, F. Inada, M. Kato, Flow-Induced Vibrations; Classifications and Lessons from Practical Experience, Elsevier Ltd. 2008.
- [15] P. A. Durbin, B. A. Petersson. Statistical Theory and Modeling for Turbulent Flows. John Wiley & Sons, Ltd, 2011.
- [16] S. B. Pope. Turbulent Flows. Cambridge University Press, 2000.
- [17] D. C. Wilcox. Turbulence modeling for CFD. DCW Industries, 2006.

- [18] I. B. Celik. Introductory Turbulence Modeling; Lecture Notes. West Virginia University, Mechanical & Aerospace Engineering Department, 1999.
- [19] F. R. Menter. Best Practice: Scale-Resolving Simulations in ANSYS CFD, Version 1.0. ANSYS Germany GmbH, 2012.
- [20] M. J. Acton, G. Lenci, E. Baglietto. Structure-Based Resolution of Turbulence for Sodium Fast Reactor Thermal Striping Application. The 16th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, NURETH-16, Chicago, IL, August 30-September 4, 2015.
- [21] F. R. Menter, M. Kuntz, R. Bender, A scale-adaptive simulation model for turbulent flow predictions, AIAA Paper 2003-0767, American Institute of Aeronautics and Astronautics, 2003.
- [22] F. R. Menter, Y. Egorov, The Scale-Adaptive Simulation Method for Unsteady Turbulent Flow Predictions, Part 1: Theory and Model Description, Turbulence Combust 2010, Vol. 85: 113-138, 2010, DOI 10.1007/s10494-010-9264-5, <https://doi.org/10.1007/s10494-010-9264-5>.
- [23] E. Baglietto, G. Lenci, D. Concu. STRUCT: A second-generation URANS approach for effective design of advanced systems. Precedings of the ASME 2017 Fluids Engineering Division Summer Meeting, FEDSM2017. July 30-August 3, 2017, Waikoloa, Hawaii, USA, (SEDSM2017-69241).
- [24] S. Jakirlić et al. RANS-Based Sub-Scale Modelling in Eddy-Resolving Simulation Methods. ERCOFTAC Bulletin 121, December 2019.
- [25] P. R. Spalart, W.-H. Jou, M. Strelets, S. R. Allmaras. Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach. Advances in DNS/LES; Proceedings of the International Conference on DNS/LES, Louisiana, USA, August 4-8, 1997, pp. 137-147.
- [26] U. Piomelli, E. Balaras. Wall-Layer Models for Large-Eddy Simulations. Annual Review of Fluid Mechanics, Vol. 34:349-374, 2002.
- [27] S. A. Senthoooran. Computational Model to Calculate Flow-Induced Pressure Fluctuations on a Bluff Body. PhD Thesis, Texas Tech University, August 2002.
- [28] S. Kottapalli, S. Sharma, A. Shams, A. H. Zuijlen, M. J. B. M. Porquie. Numerical simulation of Turbulence Induced Vibrations from URANS models using the Pressure Fluctuation Model. The 17th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics, NURETH-17, Qujiang, September 3-8, 2017.
- [29] E. Baglietto. Applied Computational Fluid Dynamics and Heat Transfer. Lecture Notes, Spring Term 2013. Massachusetts Institute of Technology (MIT), Nuclear Science & Engineering at MIT.
- [30] J.H. Ferziger, M. Perić. Computational Methods for Fluid Dynamics. Third, rev. edition, Springer-Verlag 2002.
- [31] R.R. Craig Jr., A.J. Kurdila, Fundamentals of Structural Dynamics, John Wiley & Sons, Inc., second edition 2006.
- [32] R. J. Fritz. The Effects of Liquids on the Dynamic Motions of Immersed Solids. ASME Journal of Engineering for Industry, 94(1): 167-173, 1972
- [33] B. Dressel, U. Staude, S. Hofbeck, C.-J. Münch, Sz. Kovacs, Investigations on the FSI characteristic of a scaled and integral fuel structure under axial flow, Proceedings of the

- ASME 2017 International Mechanical Engineering Congress and Exposition, IMECE2017-70574, Tampa, Florida, USA, November 3-9, 2017.
- [34] J. Rigaudeau, B. d’Uston, “Fluid Effects in Seismic Response of PWR Fuel Assemblies”, ICME Conference, Shangai, Nov 20–22, 2000.
 - [35] B. E. Launder and D. B. Spalding. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering*, 3(2):269–289, March 1974.
 - [36] P. Sagaut. Large eddy simulation for incompressible flows. An introduction. Translated from the 1998 French original. With a foreword by Massimo Germano. *Sci. Comput.* Berlin: Springer, 2002.
 - [37] S. Heinz. Unified turbulence models for les and rans, fdf and pdf simulations. *Theoretical and Computational Fluid Dynamics*, 21(2):99–118, September 2006.
 - [38] F. Nicoud and F. Ducros. Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, Turbulence and Combustion*, 62(3):183–200, 1999.
 - [39] U. Schumann. Subgrid scale model for finite difference simulations of turbulent flows in plane channels and annuli. *Journal of Computational Physics*, 1975.
 - [40] A. Ern and J.-L. Guermond. *Theory and practice of finite elements*. Springer, 2004.
 - [41] D. Panunzio and M.-A. Puscas and R. Lagrange. FSI-vibrations of immersed cylinders. Simulations with the engineering open-source code TrioCFD. Test cases and experimental comparisons. *Comptes Rendus. Mécanique*. 350: 451-476, 2022.
 - [42] A. Michel, M. A. Puscas, and A. Bergeron. A comparative analysis of the flow in a rod bundle with two guide-tubes and mixing grids, URANS simulation and LES of the ALAIN experiment. *Nuclear Engineering and Design*, 441, 114115, 2025.
 - [43] H. Reichardt. Vollständige darstellung der turbulenten geschwindigkeitsverteilung in glatten leitungen. *ZAMM - Journal of Applied Mathematics and Mechanics / Zeitschrift für Angewandte Mathematik und Mechanik*, 31(7):208–219, January 1951.
 - [44] P.-E. Angeli, U. Bieder, and G. Fauchet. Overview of the TrioCFD code: Main features, VetV procedures and typical applications to nuclear engineering. August 2015.
 - [45] M. Crouzeix and P.-A. Raviart. Conforming and nonconforming finite element methods for solving the stationary stokes equations. *Revue française d’automatique informatique recherche opérationnelle. Mathématique*, 7(R3): 33–75, 1973.
 - [46] P.-E. Angeli, M.-A. Puscas, G. Fauchet, and A. Cartalade. FVCA8 Benchmark for the Stokes and Navier–Stokes Equations with the TrioCFD Code—Benchmark Session, pages 181–202. Springer International Publishing, 2017.
 - [47] I. B. Celik, Z. N. Cehreli, and I. Yavuz. Index of resolution quality for large eddy simulations. *Journal of Fluids Engineering*, 127(5):949–958, September 2005.
 - [48] S. K. Kang and Y. A. Hassan. Computational fluid dynamics (cfd) round robin benchmark for a pressurized water reactor (pwr) rod bundle. *Nuclear Engineering and Design*, 301:204–231, 2016.
 - [49] P.-E. Angeli and M. Peybernes. CFD analysis of the turbulent flow in a PWR 5x5 rod bundle with staggered structural grids. September 2016.
 - [50] J. Xiong, C. Lu, W. Qu, Y. Yang, and X. Cheng. Assessment of rans models in predicting mixing flow induced by split-type vanes in rod bundle. *Nuclear Engineering and Design*, 363:110615, July 2020.

- [51] U. Bieder, F. Falk, and G. Fauchet. LES analysis of the flow in a simplified PWR assembly with mixing grid. *Progress in Nuclear Energy*, 75:15–24, 2014.
- [52] F. Muller, B.-J. Grea, A. Burbeau, and P. Sagaut. Minimum enstrophy states and bifurcations in 2d Euler flows around a central obstacle. In *Congres Francais de Mecanique*, Lille, France, August 2017.
- [53] D. Logan. *A First Course in the Finite Element Method*, Fourth Edition. Thomson, 2007.
- [54] G. Ricciardi, Analytical model of added mass, damping and stiffness of a fuel assembly induced by axial flow, *Nuclear Engineering and Design*, Volume 389, 2022, 111670.
- [55] C. Bläsius, J. Arndt, T. Schimpfke, R. Wenke and J. Sievers, "Strukturmechanische Forschungen zur Entstehung von Brennelementverformungen in DWR und deren Einfluss auf die Auslegungsreserven im Normalbetrieb und bei Störfällen," GRS gGmbH, GRS-596, 2020.
- [56] M. J. Pettigrew, C. E. Taylor, N. J. Fisher, M. Yetisir and B. A. W. Smith, "Flow-induced vibration: recent findings and open questions," *Nuclear Engineering and Design*, Vol. 185, pp. 249-276, 1998.
- [57] E. Viallet, G. Bolsee, B. Ladouceur, T. Goubin and J. Rigadeau, "Validation of PWR Core Seismic Models with Shaking Table Tests on Interacting Scale 1 Fuel Assemblies," 17th SMiRT Conference, Prague, 2003.
- [58] E. Viallet and T. Kestens, "Prediction of Flow Induced Damping of a PWR Fuel Assembly in Case of Seismic and Loca Load Case," 17th SMiRT Conference, Prague, 2003.
- [59] S. Pisapia, B. Collard, S. Bellizi and V. Mori, "Modal Testing and Identification of a PWR Fuel Assembly," 17th SMiRT Conference, Prague, 2003.
- [60] R. Y. Lu and D. D. Seel, "PWR Fuel Assembly Damping Characteristics," ICONE 14 Conference, Miami, USA, 2006.
- [61] J. Rigadeau, D. Brochard and A. Benjedidia, "Fluid Structure Interaction in the Response of PWR Fuel Assemblies to Horizontal Seismic Loads," 12th SMiRT Conference, 1993.
- [62] B. Collard, "Flow Induced Damping of a PWR Fuel Assembly," Proceedings of a technical meeting held in Cadarache, France, IAEA-TECDOC-1454, 2004.
- [63] G. Ricciardi and E. Boccaccio, "Modelling of the flow induced stiffness of a PWR fuel assembly," *Nuclear Engineering and Design*, Vol. 282, pp 8-14, 2015.
- [64] G. Ricciardi, "Fluid–structure interaction modelling of a PWR fuel assembly subjected to axial flow," *Journal of Fluids and Structures*, Vol. 62, pp. 156-171, 2016.
- [65] G. Ricciardi, "Dynamical Nonlinear Modelling Of A Pressurised Water Reactor Fuel Assembly Subjected To An Axial Flow," 10th International Conference on Structural Dynamics, EUROLYN, 2017.
- [66] K. Fujita, "Flow-Induced Vibration and Fluid-Structure Interaction in Nuclear Power Plant Components," *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 33, pp. 405-418, 1990.
- [67] S. J. Shah, B. Brenneman and J. Rigadeau, "Comparison of Analytical and Experimental Damping Under Axial Flow for Different Fuel Assembly Types," 9th International Conference on Nuclear Engineering, Nice, 2001.
- [68] J. Pohlus und U. Paquée, „Untersuchung veränderter Neutronenflussschwankungen und Brennstab-Beanspruchungen in DWR-Anlagen im Rahmen der Sicherheitsforschung,“ TÜV Rheinland ISTec GmbH, ISTec - A - 3695, 2018.

- [69] J. Runkel, E. Laggiard, D. Stegemann und P. Heidemann, „In-Core Measurements of Reactors Internals Vibrations by Use of Accelerometers and Neutron Detectors,“ Institute of Nuclear Engineering and Non-destructive Testing, 1995.
- [70] E. Laggiard, J. Fiedler, J. Runkel, H. Starke, D. Stegemann, B. Lukas und D. Sommer, „Vibration Measurement in PWR Obrigheim by Use of In-Core Accelerometers,“ Progress in Nuclear Energy, Vol. 29, No. 3/4, pp. 229-238, 1995.
- [71] D. N. Fry, J. March-Leube und F. J. Sweeney, „Use of Neutron Noise for Diagnosis Of In-Vessel Anomalies in Light-Water Reactors,“ Union Carbide Corporation, NUREG/CR-3303, 1984.
- [72] IAEA-CN245-122. Experience of Commissioning of BN-800 Core Diagnosing System (SDRU).
- [73] Kachanov, Y.S., Kozlov, V.V. & Levchenko, V.Y. Occurrence of Tolman-Schlichting waves in the boundary layer under the effect of external perturbations. Fluid Dyn 13, 704–711 (1978). <https://doi.org/10.1007/BF01050234>
- [74] Abhishek Mishra, Lyazid Djenidi, Amit Agrawal; Dynamics of wall jet flow under external pulsation. Physics of Fluids 1 September 2021; 33 (9): 095103. <https://doi.org/10.1063/5.0057695>.
- [75] He, S. and Lau, R.W.H. (2013), Synthetic Controllable Turbulence Using Robust Second Vorticity Confinement. Computer Graphics Forum, 32: 27-35. <https://doi.org/10.1111/j.1467-8659.2012.03228.x>
- [76] ANSYS Inc., ANSYS CFX 18 -Solver Theory Guide, 2017.
- [77] M.P. Paidoussis. An experimental study of vibration of flexible cylinders induced by nominally axially flow, Nucl. Sci. Eng. 35 (1) (1969) 127138.
- [78] Validation of PWR Core Seismic Models with Shaking Table Tests on Interacting Scale 1 Fuel Assemblies, by Viallet et al, SMiRT 17, Prague, Czech Republic, August 17 –22, 2003.
- [79] G.M.L. Gladwell Inverse problems in vibration, second edition, 2004 472p.
- [80] Chen, S. S., Wambsganss, M. W., and Jendrzeczyk, J. A. (June 1, 1976). Added Mass and Damping of a Vibrating Rod in Confined Viscous Fluids. ASME. J. Appl. Mech. June 1976; 43(2): 325–329.
- [81] Hecht, F. (2012). New development in FreeFem++. Journal of Numerical Mathematics, 20(3-4), 251-266.
- [82] Yoon, J.-C., Kam, H.R., Hong, J.-M., Kang, S.J. and Kim, C.-H. (2009), Procedural Synthesis using Vortex Particle Method for Fluid Simulation. Computer Graphics Forum, 28: 1853-1859. <https://doi.org/10.1111/j.1467-8659.2009.01563.x>