

Numerical Simulation of Turbulent Wind Gusts and their Dynamic Effects on Flexible Structures

Doctoral Thesis

approved by the

Faculty of Mechanical Engineering

of the

Helmut-Schmidt-University

University of the German Federal Armed Forces

for obtaining the academic degree of

Doktor Ingenieur (Dr.-Ing.)

presented by

Khaled Boulbrachene

from Annaba, Algeria

Hamburg, 2026

Referees

Univ.-Prof. Dr.-Ing. habil. Michael Breuer

Professur für Strömungsmechanik (PfS)
Institut für Mechanik
Fakultät für Maschinenbau und Bauingenieurwesen
Helmut-Schmidt-Universität Hamburg
Holstenhofweg 85 / Postfach 70 08 22
22043 Hamburg / Germany

Univ.-Prof. Dr.-Ing. habil. Michael Manhart

Professur für Hydromechanik
Department of Civil and Environmental Engineering
Technische Universität München
Arcisstraße 21
80333 München / Germany

The day of submission: January 15, 2026

The day of completion of the oral examination: May 18, 2026

*To my parents,
for every sacrifice, every prayer,
and every moment of unwavering belief.*

Kurzfassung

Diese Dissertation untersucht numerische Strategien zur Einbringung diskreter Windböen in hochaufgelöste, wirbelaufösende Simulationen zur Analyse ihrer Wechselwirkung mit starren und flexiblen Strukturen. Die zentrale Anforderung besteht darin, die Böe physikalisch konsistent in ein bereits entwickeltes Strömungsfeld einzubringen, ohne Erhaltungsgesetze zu verletzen oder die numerische Stabilität zu beeinträchtigen. Dazu wird eine Hierarchie von Böen-Generationsverfahren mit unterschiedlichen Graden physikalischen Realismus in ein Large-Eddy-Simulations-Programm implementiert. Diese reichen von idealisierten, vorgeschriebenen Störungen bis hin zu mechanisch induzierten Böen, die durch numerische Gegenstücke experimenteller Böengeneratoren erzeugt werden. Die Hierarchie umfasst drei Klassen von Verfahren: Die vorgeschriebenen Geschwindigkeitsmethoden, die *Source-Term*-Formulierung sowie die Immersed-Boundary-Böen-Generatoren.

Der erste Teil führt eine Böe und ihre Bewegung als vorgeschriebene Störung mithilfe der *Field Velocity Method* (FVM) und der *Split Velocity Method* (SVM) ein. Beide Methoden werden konsistent neu hergeleitet, im Löser implementiert und ihre Verbindung zur *Arbitrary-Lagrangian-Eulerian*-(ALE)-Formulierung sowie zum geometrischen Erhaltungsgesetz wird herausgearbeitet. Ein neu eingeführter Testfall zeigt, dass der in der SVM enthaltene, in der FVM jedoch fehlende Rückkopplungsterm für die Vorhersage horizontaler Böenantworten entscheidend ist und die SVM als physikalisch konsistente Wahl etabliert. Aufbauend auf diesem Ergebnis wird die SVM auf ein elastisch gelagertes Profil angewendet, um dessen aeroelastisches Verhalten unter vertikalen Böen zu untersuchen.

Als weiterer Schritt hin zu einem physikalisch inspirierten Böen-Generationsansatz wird eine bestehende *Source-Term*-Formulierung eingesetzt, um eine lokalisierte Störung in der Nähe der interessierenden Struktur einzubringen, die sich anschließend frei durch das Rechengebiet ausbreitet. Die Methode wird auf starre sowie membranartige halbkugelförmige Strukturen in einer turbulenten Grenzschicht angewendet und ermöglicht die Auflösung der aerodynamischen Kräfte sowie im Fall der flexiblen Membran der daraus resultierenden Verformungen und Zugspannungen im Material während der Böe.

Der letzte Teil schreitet in Richtung physikalisch realisierter Böen-Generationsverfahren voran, indem der Betrieb von Windkanal-Böen-Generatoren numerisch mithilfe der *Immersed-Boundary*-Methode (IBM) nachgebildet wird. Zwei experimentelle Konfigurationen werden modelliert: Eine starre vertikal bewegte Platte sowie eine adaptive Düse mit rotierbarer oberer Kontur. In beiden Böen-Generatoren wird das durch die bewegte Geometrie erzeugte dreidimensionale, zeitaufgelöste Strömungsfeld detailliert erfasst, wodurch ein direkter Vergleich mit den verfügbaren experimentellen Messungen ermöglicht wird. Darüber hinaus wird die IBM mit einem ALE-basierten partitionierten Ansatz kombiniert, um die Fluid-Struktur-Interaktion einer flexiblen T-Struktur unter Böenlasten zu simulieren.

Insgesamt stellen die Beiträge dieser Arbeit verschiedene numerische Strategien zur Einbringung diskreter Windböen vor und ermöglichen hochaufgelöste Simulationen ihrer Wechselwirkung mit starren und flexiblen Strukturen. Diese Entwicklungen erweitern die Möglichkeiten numerischer Simulationen zur Analyse von Böenlasten und bilden eine Grundlage für zukünftige Untersuchungen komplexerer Strukturen und Böenbedingungen.

Abstract

This thesis investigates numerical strategies for imposing discrete wind gusts within high-fidelity eddy-resolving simulations to enable detailed analyses of their interaction with rigid or flexible structures. The central requirement is to introduce the gust into an already evolving flow field in a physically consistent manner, without violating conservation laws or compromising numerical stability. Toward fulfilling this requirement, the work implements a hierarchy of gust-generation techniques, spanning different levels of physical realism, into a large-eddy simulation fluid solver. These range from idealized prescribed disturbances to mechanically induced gusts generated by numerical counterparts of experimental gust-generation devices. This hierarchy comprises three classes of methods: The prescribed velocity methods (field and split velocity methods), the source-term formulation, and the immersed boundary gust generators.

The first part of the thesis imposes a gust and its motion as a prescribed disturbance using the field velocity method (FVM) and the split velocity method (SVM). Both methods are re-derived in a consistent manner, implemented within the numerical solver, and their connection to the Arbitrary Lagrangian–Eulerian (ALE) formulation and the geometric conservation law (GCL) is clarified. A newly introduced test case demonstrates that the feedback term present in the SVM but absent in the FVM becomes essential for accurately predicting horizontal gust responses, establishing the SVM as the physically consistent choice for subsequent applications. Building on this result, the SVM is applied to an elastically mounted airfoil to investigate its aeroelastic behavior under vertical gusts.

Moving one step closer to a physically inspired gust-generation approach, the thesis employs an existing source-term (ST) formulation to inject a localized disturbance near the structure of interest that subsequently propagates freely through the computational domain. The method is applied to rigid and membranous hemispherical structures in a turbulent boundary layer, enabling the simulations to resolve the aerodynamic forces and, for the flexible membrane, the resulting deformations and tensile stresses in the material during the gust impact.

The final part of the thesis advances toward physically realized gust generation procedures by numerically replicating the operation of wind-tunnel gust generators using the immersed boundary method (IBM). Two experimental configurations are modeled: The rigid vertically moving paddle and the adaptive nozzle with a rotatable upper contour. In both immersed-boundary gust generators (IBGG) developed here, the three-dimensional time-resolved flow field generated by the moving geometry is captured in detail, enabling direct comparison with the available experimental measurements. The IBM is further combined with an ALE-based partitioned approach to simulate the fluid–structure interaction of a flexible T-structure subjected to paddle-generated gusts.

Taken together, the contributions of this thesis present a set of numerical strategies for injecting discrete wind gusts across a wide range of configurations, enabling high-fidelity simulations of their interaction with rigid and flexible structures. These developments broaden the capability of numerical simulations to analyze gust encounters and provide a foundation for future studies involving more complex structures and gust conditions.

Acknowledgments

Pursuing a PhD is a long and demanding journey that tests not only one's scientific abilities, but also patience, resilience, and perseverance. While the final dissertation bears a single author's name, its completion would not have been possible without the support and contributions of many people. I would therefore like to express my gratitude to those who accompanied me throughout this journey.

First and foremost, my sincere thanks go to my advisor, Prof. Michael Breuer, for his supervision of this dissertation and for offering me the opportunity to conduct this research at the Department of Fluid Mechanics at Helmut Schmidt University. I would also like to extend my profound appreciation to my second examiner, Prof. Michael Manhart, for the time and effort devoted to evaluating this work.

I wish to thank my colleagues from the Fluid Mechanics group, especially Guillaume De Nayer, for generously sharing his experience, which made the transition to working with the in-house flow solver FASTEST-3D considerably smoother during the initial stages of my PhD. I am equally grateful for the many enriching technical discussions we shared throughout the course of my research. My appreciation also goes to Nick Wood for the valuable experimental input that contributed significantly to this work. Special thanks are extended to my office colleague and friend Surya Peravali, and to Ali Khalifa, whom I consider a brother, for bringing so much joy to this challenging journey.

I would like to thank my lifelong friends Yasin Mohamed, Sulaiman Fadlallah, Bishr Maradni, Mahmoud Ammar, Mahmoud Zidan and Mohammed Al-Harthi for their enduring friendship, despite the distance and the differing paths our lives have taken. I hold close to my heart the memory of my dear friend Younis, who always believed I would come this far. I hope this achievement brings a smile to your face in heaven.

To my older sisters and my childhood friends, Souhaila and Chahrazad, thank you for being a living example of inner strength and resilience. Your character has been a silent source of inspiration, and your care from afar has meant more than words can convey.

My heartfelt thanks go to my dear younger brother Abderrezak, and to my friends and loved ones who became like a second family and made Hamburg truly feel like home. Their companionship and support brought comfort during the taxing times, making this journey far more meaningful and memorable than it would have otherwise been.

No words could possibly express my gratitude to my parents, Prof. Messaoud Boulbrachene and Dr. med. Fouzia Saadni, for their unconditional love, unwavering support, and encouragement throughout the different phases of my life that ultimately led to this milestone. Despite being thousands of miles apart, their constant belief in me gave me the strength to overcome every challenge, exceed my own limits, and become the person I am today. To them, I dedicate this work.

I gratefully acknowledge the financial support of the German Research Foundation (DFG, grant no. BR 1847/17-1) and the computational resources provided by HSUper.

Contents

Kurzfassung	i
Abstract	iii
Acknowledgments	v
Contents	vii
List of Figures	xi
List of Tables	xvii

I	Numerical Methods and Summary of Results	1
1	Introduction	3
1.1	Motivation	3
1.2	Physical Background	8
1.2.1	Atmospheric Boundary Layer (ABL)	8
1.2.2	Wind Gusts	9
1.2.3	Microscale Gust Generation Mechanisms in the ABL	10
1.3	Numerical Injection of Discrete Wind Gusts	13
1.4	Objectives of the Thesis	16
1.5	Outline of the Thesis	18
2	Discrete Wind Gust Injection via Prescribed Velocity Methods	19
2.1	Numerical Methodology	19
2.1.1	Field Velocity Method (FVM)	20
2.1.2	Split Velocity Method (SVM)	26
2.2	Main Results of Publication I	29
2.2.1	Vertical Gusts	29
2.2.1.1	Temporal Evolution of Flow Fields for $L_g^\eta = 2c$	29
2.2.1.2	Effect of Gust Length on Force Coefficients	32
2.2.2	Horizontal Gusts	32
2.2.2.1	Temporal Evolution of Flow Fields for $L_g^\xi = 2c$	33
2.2.2.2	Time Histories of the Drag Coefficient	36
2.3	Main Results of Publication II	37
2.3.1	Comparison of CFD and FSI Cases	38

2.3.2	Gust-Induced Aeroelastic Behavior	38
2.3.3	Vorticity and Separation Dynamics during an LCO	39
2.3.4	Aerodynamic Hysteresis and Energy Analysis in LCO and Flutter Cases	40
2.4	Conclusions	42
3	Discrete Wind Gust Injection via a Source-term based Formulation	45
3.1	Source-term formulation	45
3.2	Smoothing the Gust Source Term	49
3.3	Main Results of Publication III	49
3.3.1	Flow Characteristics Prior to Gust Impact	51
3.3.2	Horizontal Gusts Impacting Rigid and Flexible Hemispheres	52
3.3.3	Structural Deformations and Material Response	54
3.4	Conclusions	56
4	Artificial Discrete Wind Gust Injection via Immersed Boundary Gust Generators	57
4.1	Immersed Boundary Method	59
4.1.1	Lagrangian Marker Tracking	60
4.1.2	IBM with a Direct Forcing Approach	62
4.2	Reproducing Kernel Particle Method based IBM	66
4.3	Moving Least Squares based IBM	69
4.3.1	Standard MLS Formulation	71
4.3.2	New Proposition for Curvilinear Block-Structured Grids	72
4.3.3	General Framework	73
4.4	Main Results of Publication IV	75
4.4.1	Flow Field Induced by the Paddle Wind Gust Generator	77
4.4.2	Comparison of Numerical and Experimental Results	77
4.5	Main Results of Publication V	80
4.5.1	Flow Field and Structural Displacements without Gust	81
4.5.2	Flow Field and Structural Displacements with Gust	83
4.6	Main Results of Publication VI	86
4.6.1	Flow Field Induced by the Adaptive Nozzle Wind Gust Generator	88
4.6.2	Comparison of Numerical and Experimental Results	89
4.7	Conclusions	92
5	Outlook	95
A	Definition of the Gust Shape	97
	Bibliography	101

II Publications	109
Publication I: Assessment of two wind gust injection methods: Field velocity vs. split velocity method	111
Publication II: Aeroelastic response of an elastically mounted 2-DOF airfoil and its gust-induced oscillations	135
Publication III: FSI Simulations of Wind Gusts Impacting an Air-Inflated Flexible Membrane at $Re = 100,000$	163
Publication IV: A numerical method to mimic an experimental wind gust generator: The immersed boundary gust generator	189
Publication V: Combination of two FSI methods and their validation based on artificial wind gusts impacting a flexible T-structure	213
Publication VI: A dynamic immersed boundary method for simulating an adaptive nozzle generating discrete wind gusts	235

List of Figures

1.1	Examples of wind-turbine failures caused by strong wind gusts in real-world environments: (a) Wind turbines at the Buffalo Ridge Wind Farm (Minnesota, USA, 2011) [4], (b) Wind turbines at a coastal wind farm (Shanwei, China, 2013) [5].	4
1.2	Failure of lightweight and temporary civil engineering structures due to gust-induced loading: (a) Collapsed Dallas Cowboys indoor practice facility (Texas, USA, May 2009) [6], (b) Uplift and destruction of a large event marquee near St Andrews, Scotland, during Storm Ali (September 2018) [7].	5
1.3	Schematic illustration of an aircraft encountering a microburst-induced wind shear during landing (adapted from [8]). The sequence comprises (i) initial headwind and increased lift, (ii) downdraft and descent, and (iii) tailwind with rapid loss of airspeed and lift.	6
1.4	Wreckage of Delta Air Lines Flight 191, which crashed in front of the runway at Dallas-Fort Worth Airport in 1985 due to microburst-induced wind shear during landing [9]. The photographs show severe structural breakup and exposed cabin interior following ground impact.	7
1.5	Schematic illustration of the diurnal evolution of the atmospheric boundary layer over land in high-pressure regions. Adapted from the canonical description of Stull [22].	9
1.6	Meteorological scales of motion. Adapted from Bos [30], who compiled them from Stull [22], Orlanski [32] and Steyn et al. [33].	11
1.7	Schematic of mechanical turbulence. Flow over topography, vegetation, and urban structures.	12
1.8	Schematic of thermal turbulence in the convective boundary layer. Adapted from Wyngaard [34], showing boundary layer circulation and entrainment across the entrainment zone. Here, V denotes the mean horizontal wind and θ the temperature.	13
2.1	Illustration of the field velocity method: A physical step change in the angle of attack α (a) can be equivalently represented either by a vertical displacement of the airfoil (plunge motion) (b) or by a pseudo-grid motion imposed on a fixed mesh using the FVM (c).	20
2.2	Illustration of the actual swept volume δV_e on the east face of a control volume, resulting from the mesh deformation between two consecutive time steps.	23
2.3	Illustration of apparent swept volume δV_e^g on the east face of a control volume, resulting from the field velocity method.	25

2.4	Illustration of the actual swept volume δV_e and the apparent swept volume δV_e^g on the east face of a control volume, resulting from mesh deformation between two consecutive time steps and from the field velocity method, respectively.	26
2.5	Schematic representation of the vertical gust configuration.	30
2.6	<i>Vertical gust passing around the fixed rounded flat plate</i> : Flow field predictions from the FVM (left) and SVM (right) at time instants 1, 2, 3, and 4. The gust has a length of $L_g^\eta = 2c$	31
2.7	<i>Vertical gust passing around the fixed rounded flat plate</i> : Instantaneous drag and lift coefficients. The gust shape is ECG with a gust amplitude of $A_g = 0.8$: FVM vs. SVM.	33
2.8	Schematic representation of the horizontal gust configuration.	33
2.9	<i>Horizontal gust passing around the fixed rounded flat plate</i> : Flow field predictions from FVM (left) and SVM (right) at time instants 1, 2, 3, and 4. The gust has a length of $L_g^\xi = 2c$	35
2.10	<i>Horizontal gust passing around the fixed rounded flat plate</i> : Time history of the drag coefficient C_D , of the pressure part of the drag coefficient C_D^p and of the shear part of the drag coefficient C_D^s for FVM and SVM. The gust has the length of $L_g^\xi = 2c$	36
2.11	Schematic representation of the investigated FSI case and the introduced vertical gust velocity profile.	37
2.12	Instantaneous lift and pitching moment coefficients of the fixed and elastically mounted airfoils subjected to a vertical gust with $L_g^{x_1}/c = 2$ and $A_g/u_\infty = 2$	38
2.13	Snapshots of (a) the total vertical velocity and (b) the pressure fields during the gust encounter ($L_g^{x_1}/c = 2$, $A_g/u_\infty = 2$) on the elastically mounted airfoil. From left to right: Gust center at leading edge, gust center when lift is maximum, gust center at trailing edge, gust completely left the airfoil.	39
2.14	Instantaneous heave and pitch DOFs when imposing a vertical gust with $L_g^{x_1}/c = 4$ and three different gust strength $A_g/u_\infty = 1, 2$ and 3.	39
2.15	Normalized y -vorticity ω_y (on the mid-span slice), skin-friction coefficient C_f (on the upper surface of the airfoil), and separation location (dashed line) for eight time instants (a-h).	40
2.16	Aerodynamic hysteresis curves within a single cycle for a LCO case (a, b) and a flutter case (c, d).	41
2.17	Dimensionless time histories of the instantaneous total energies of both DOFs and the total energy of the system for a LCO case (a) and a flutter case (b).	42
3.1	Adapted “1-cosine” vs. modified C^2 -“1-cosine” shapes and their derivatives for a given scale L_g^ϕ and central value ϕ_g	50

3.2	2D schematic sketch showing (a) the surface-mounted hemispherical geometry (diameter D) within a turbulent boundary layer, and (b) the definition of boundary conditions, the STIG injection window, and the gust injection volume.	51
3.3	Instantaneous snapshots in the symmetry plane ($y = 0$) showing the <i>medium</i> gust approaching the rigid hemisphere at $t^* \approx 0.95$. The top row depicts the case with gust injection, while the bottom row shows the corresponding flow without gust. The gust structure is indicated by a red iso-surface or a black iso-line of the streamwise velocity $u_1/u_\infty = 1.4$	52
3.4	Injected <i>medium</i> wind gust convecting downstream and interacting with a rigid hemisphere (top row) and an air-inflated flexible hemisphere (bottom row). The streamwise velocity u_1/u_∞ is visualized within a slice near the bottom wall and in the symmetry plane ($y = 0$). The gust structure is highlighted by a red iso-surface of the streamwise velocity $u_1/u_\infty = 1.4$. . .	53
3.5	Predicted streamwise and vertical force coefficients for the rigid and flexible hemisphere cases subjected to different gust strengths. The time-axis annotation (a), (b), and (c) correspond to the snapshots shown in Fig. 3.4.	54
3.6	Flexible FEM case: Maximum structural deflections in the symmetry plane for different gust strengths.	55
3.7	Flexible FEM case: Von Mises stress distributions for the maximally deflected structure at different gust strengths.	55
4.1	Side view sketches of the experimental wind gust generators: (a) the paddle retrofitted to a Göttingen-type wind tunnel, and (b) the adaptive nozzle contour installed in an Eiffel-type wind tunnel.	58
4.2	Two-dimensional schematic illustrating the overlapping Lagrangian and Eulerian discretizations.	59
4.3	(a) Two-dimensional sketch of the p-space showing a computational grid (black lines), cell centers (blue dots), $\mathbf{z}$ -grid (blue dashed lines), and a marker's trajectory (red line). (b) Corresponding c-space representation showing trilinear interpolation coefficients.	61
4.4	(a) Three-point kernel by Roma et al. [80]; (b) interpolation of Eulerian velocities to a Lagrangian location; and (c) spreading of Lagrangian forces to an Eulerian location.	63
4.5	Schematic showing the distribution of Lagrangian markers in the z -plane of the non-uniform Cartesian grid used for simulating the paddle gust generator.	66
4.6	Schematic showing the distribution of Lagrangian markers (red spheres) (a) in the z -plane of the curvilinear grid used for simulating the adaptive nozzle gust generator, and (b) near the wall boundaries in an i -plane of the grid within the nozzle region.	70
4.7	Schematic sketch showing two Lagrangian markers within a curvilinear block-structured grid. The sketch highlights the interpolation kernels employed along each local coordinate direction for markers near and far from the boundary of the computational domain.	74

4.8	Schematic of the paddle gust generator showing the monitoring points (in red) above the ground plate used for validation.	76
4.9	Kinematic characteristics of the gust <i>type 1</i> paddle movement pattern. Nozzle's blocking ratio (red line) annotated with visualization time instants and paddle's vertical velocity (blue line) during the downward and upward strokes.	76
4.10	Snapshots of the (a) streamwise velocity and the (b) pressure during the paddle's downward and upward strokes. The corresponding blocking ratios are: instant A: 15.8 %, instant B: 31.2 %, instant C: 38.3 %, instant D: 22 %.	78
4.11	Snapshots of the streamwise velocity u/u_∞ at the x -plane $x/w = 0.2$	78
4.12	Comparison between the ensemble-averaged experimental streamwise velocity u/u_∞ (black) by Wood et al. [52] and its numerical counterpart (red) of the gust generated by the paddle at the first seven points in Fig. 4.8.	79
4.13	Predicted ensemble-averaged pressure at the first seven points in Fig. 4.8.	80
4.14	Schematic of the paddle gust generator and the flexible T-structure mounted on the ground plate.	81
4.15	Comparison between measured and predicted data of the averaged flow field in the symmetry plane. Experimental data taken from Breuer and Neumann [90].	82
4.16	Comparison of predicted and measured time-averaged: (a) Streamwise velocity at four positions in the flow field ($0.3 \bar{u}/u_\infty + x/h_s$) (T-structure in red) and (b) deformation of the T-structure in the symmetry plane. Experimental data by Breuer and Neumann [90].	83
4.17	Time histories and fast Fourier transformation of the total displacements of the center of the upper rectangular part of the T-structure. Predicted data and experimental data by Breuer and Neumann [90] for the case without gust.	83
4.18	Comparison between measured and predicted data of the phase-averaged streamwise velocity in the symmetry plane at different time instants of the gust encounter. Experimental data taken from Breuer and Neumann [90].	85
4.19	Time histories of the total displacements of the center of the upper rectangular part of the T-structure: Phase-averaged data (black thick line) and 10 singular gusts (colored lines).	86
4.20	Schematic of the adaptive nozzle gust generator showing the monitoring points (in red) above and on the ground plate used for validation.	87
4.21	Nozzle's blocking ratio annotated with visualization time instants; rotational velocity of the upper contour during the downward and upward strokes.	88
4.22	Snapshots of the streamwise velocity (a) and the pressure (b) fields during the nozzle's downward and upward strokes. Instants correspond to those depicted in Fig. 4.21. For the instants 2 to 4, a line indicating the upper contour's idle position is depicted for reference.	90

4.23	Time histories of the phase-averaged experimental velocity magnitude [53, 92] and its numerical counterpart at the 11 monitoring points depicted in Fig. 4.22a.	91
4.24	Time histories of the phase-averaged experimental surface pressure coefficient [53] and its numerical counterpart at the 11 monitoring points depicted in Fig. 4.22b.	92
A.1	Local basis definition $\mathcal{B}_1 = (\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$ of the gust.	97

List of Tables

2.1	Vertical gust characteristics for the fixed flat plate with rounded edges. . .	30
2.2	Horizontal gust characteristics for the fixed flat plate with rounded edges. .	33
2.3	Vertical gust characteristics for the elastically mounted airfoil.	37
3.1	Gusts characteristics for ST formulation.	51

Part I

Numerical Methods and Summary of Results

1 Introduction

In recent decades, climate change has led to an observable rise in the frequency and intensity of extreme weather events worldwide [1]. This development is primarily attributed to the anthropogenic increase in greenhouse gas concentrations, which enhances the natural greenhouse effect and warms the Earth’s surface and lower troposphere [1]. According to the Clausius-Clapeyron relation [2], warmer air can hold approximately 7 % more water vapor per 1°C rise in temperature, thereby increasing the atmospheric moisture and the thermodynamic energy available to drive extreme events. As a result, phenomena such as heatwaves, heavy precipitation, and strong wind gusts have become more frequent and severe [1, 3]. Although large-scale climate drivers modulate their occurrence, wind gusts themselves are fundamentally short-duration, localized fluctuations in wind velocity. These rapid perturbations can impose large, transient aerodynamic loads on engineering structures and vehicles, loads that are not adequately captured by conventional design paradigms. Understanding the fundamental physics of gust-structure interaction is therefore essential for improving safety, performance, and structural reliability. These considerations define the scope of the present thesis, which focuses on the development of accurate and computationally efficient numerical methodologies for generating discrete gust events and analyzing their coupled interaction with rigid and flexible structures.

In this chapter, the motivation for studying wind gusts is first established by demonstrating their severe consequences for structures across multiple engineering sectors (Section 1.1). Section 1.2 then introduces the physical background of wind gusts from a meteorological perspective, including the gust scales relevant to this work and the atmospheric processes responsible for their generation within the atmospheric boundary layer. Section 1.3 reviews the numerical approaches used to model discrete wind gusts in computational solvers, including those employed or developed in this dissertation and detailed in subsequent chapters. The specific objectives driving this dissertation are defined in Section 1.4. Finally, Section 1.5 concludes the chapter with an outline of the thesis.

1.1 Motivation

Wind gusts are of significant concern in many branches of engineering because they generate rapid and intense fluctuations in aerodynamic loading that can critically affect structural performance. Unlike steady or slowly varying winds, gusts induce short-term peaks in velocity and pressure that may exceed design assumptions based on mean flow conditions. These impulsive load changes can lead to large-amplitude oscillations, excite natural vibration modes, trigger aeroelastic instabilities such as flutter, accelerate material fatigue, or completely destroy the structure.

Extreme wind events have been directly linked to numerous structural failures and operational disruptions across different engineering disciplines. Their effects, however, manifest differently depending on the scale, flexibility, and operating environment of the structure. The following examples demonstrate their destructive potential in real-world situations.

In the wind energy sector, wind turbines have become one of the fastest-growing sources of renewable electricity, driven by the need to reduce greenhouse gas emissions and reliance on fossil fuels. To increase power output and improve cost-efficiency, modern utility-scale turbines have grown substantially in size, with rotor diameters now exceeding 150 to 200 m and hub heights surpassing 100 m. However, this increase in scale and structural flexibility also makes turbine blades and towers more sensitive to unsteady aerodynamic loading. Among these loads, wind gusts represent one of the most critical design challenges, as they can induce rapid fluctuations in aerodynamic forces, extreme bending moments, and dynamic structural responses. A well-documented example occurred at the Buffalo Ridge Wind Farm in Minnesota in 2011, where several blades were torn off and one tower buckled due to intense gusts [4] (see Fig. 1.1a). A further well-documented catastrophic wind-turbine failure occurred during Super Typhoon Usagi in 2013 (see Fig. 1.1b), which caused the collapse of eight towers and the fracture of thirty-five blades at a coastal wind farm in Shanwei, China [5].



Fig. 1.1. Examples of wind-turbine failures caused by strong wind gusts in real-world environments: (a) Wind turbines at the Buffalo Ridge Wind Farm (Minnesota, USA, 2011) [4], (b) Wind turbines at a coastal wind farm (Shanwei, China, 2013) [5].

In civil engineering, lightweight thin structures such as membrane-covered and tensile-membrane structures have become widely used due to their functional and economic advantages. These structures can efficiently enclose large spans of space because they primarily resist loads through membrane tension rather than bending or compression. Their high formability allows architects and engineers to create complex geometries that would be difficult or impossible to achieve using conventional construction materials. In addition, such structures can be fabricated, transported, and erected relatively quickly and at lower

costs compared with conventional structural systems. However, these benefits come at the cost of lower resistance to transient aerodynamic loads. Wind gusts have repeatedly demonstrated their destructive potential for such structures. A well-documented example is the collapse of the Dallas Cowboys indoor practice facility in Irving, Texas, in May 2009 [6]. The tensioned-fabric roof and steel-frame structure failed during a thunderstorm producing strong downburst-like gusts, leading to partial structural collapse and multiple injuries (see Fig. 1.2a). Post-event investigations by the National Institute of Standards and Technology (NIST) revealed that the peak gust loads, although below the nominal design wind speed, produced highly transient suction and uplift forces that exceeded the structure's actual resistance due to design and connection deficiencies. A similar vulnerability of temporary lightweight constructions to gust-induced loading was observed during Storm Ali in September 2018. During this event, severe wind gusts struck parts of Scotland and caused a large marquee erected near St Andrews for a prestigious golfing event to be lifted from the ground and destroyed (see Fig. 1.2b), as reported by The Courier newspaper [7].

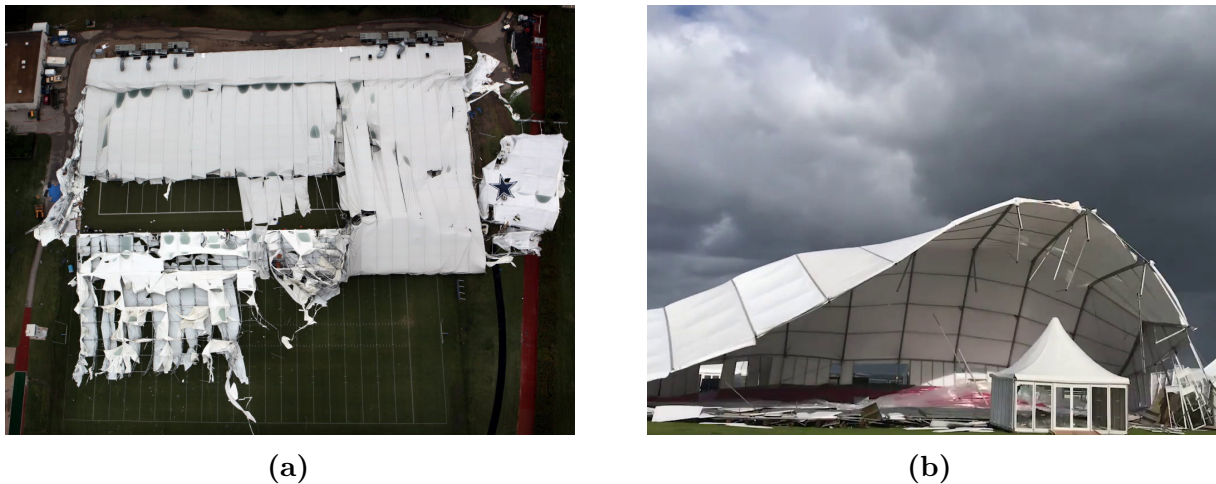


Fig. 1.2. Failure of lightweight and temporary civil engineering structures due to gust-induced loading: (a) Collapsed Dallas Cowboys indoor practice facility (Texas, USA, May 2009) [6], (b) Uplift and destruction of a large event marquee near St Andrews, Scotland, during Storm Ali (September 2018) [7].

In aviation, aircraft are particularly vulnerable to wind gusts during take-off and landing because they operate at low altitude, low airspeed, and with reduced aerodynamic safety margins. In these flight phases, lift, thrust, and control authority are only slightly above the minimum required for safe flight, leaving little time or altitude for recovery in the event of a sudden disturbance [8]. Among extreme wind events, microburst-induced wind shear has been responsible for several of the most catastrophic accidents in commercial aviation history, due to its ability to cause a sudden loss of lift during take-off or landing [8]. A microburst produces a sequence of rapidly changing wind conditions: An initial headwind that increases lift, followed by a strong downdraft, and finally a tailwind that causes a rapid loss of airspeed and lift. This sequence can lead to a sudden and unrecoverable loss of altitude if encountered close to the ground, as illustrated schematically in Fig. 1.3.

The consequences of such an event were tragically demonstrated by the crash of Delta Air Lines Flight 191 in 1985 near Dallas-Fort Worth Airport. During its final approach, the aircraft encountered a microburst, descended below the glide path, and impacted the ground in front of the runway, resulting in 136 fatalities [9] (see Fig. 1.4). This accident, along with others from the same period, led to significant advances in wind shear detection systems, pilot training, and airport weather monitoring [8, 10], underscoring the critical importance of understanding gust-induced aerodynamic loads in aviation safety.

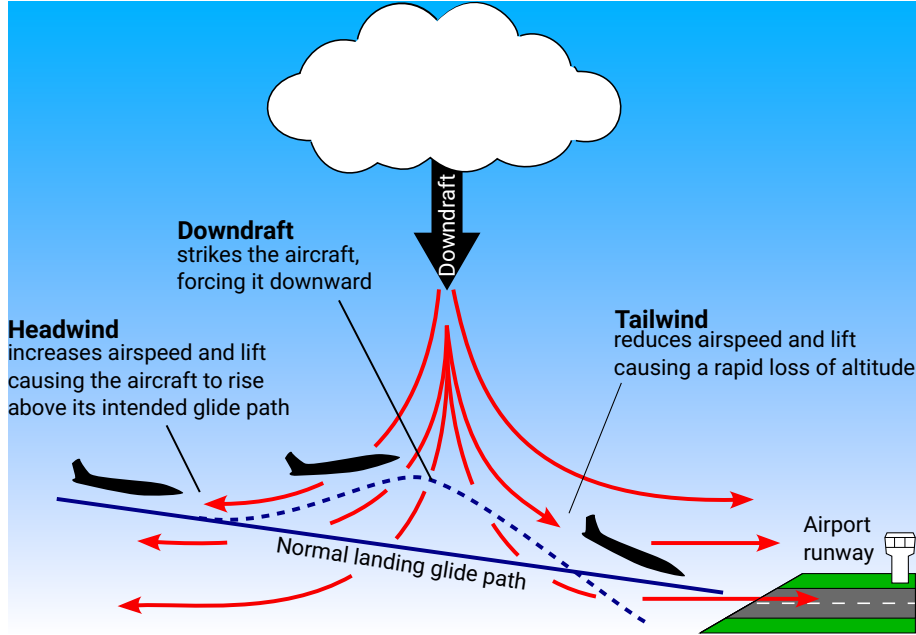


Fig. 1.3. Schematic illustration of an aircraft encountering a microburst-induced wind shear during landing (adapted from [8]). The sequence comprises (i) initial headwind and increased lift, (ii) downdraft and descent, and (iii) tailwind with rapid loss of airspeed and lift.

Extreme wind events impose significant economic burdens worldwide, contributing substantially to global disaster losses [11, 12]. In Europe alone, weather- and climate-related disasters caused cumulative economic losses of approximately 822 billion euros between 1980 and 2024, with meteorological events such as storms, including intense wind gusts, accounting for roughly 27 % of these losses [13]. The incidents mentioned above, together with the associated economic losses, underline how short-duration, high-intensity gusts can critically affect structural stability and operational safety. They highlight the need for more accurate modeling of transient wind loads and for continued improvement of design standards across engineering disciplines.

Beyond isolated case studies and individual research efforts, international standards require gusts to be explicitly considered in engineering design. Across engineering sectors, these standards continue to evolve with the purpose to improve wind-load prediction and safety margins. Nevertheless, they rely on idealized assumptions that do not fully

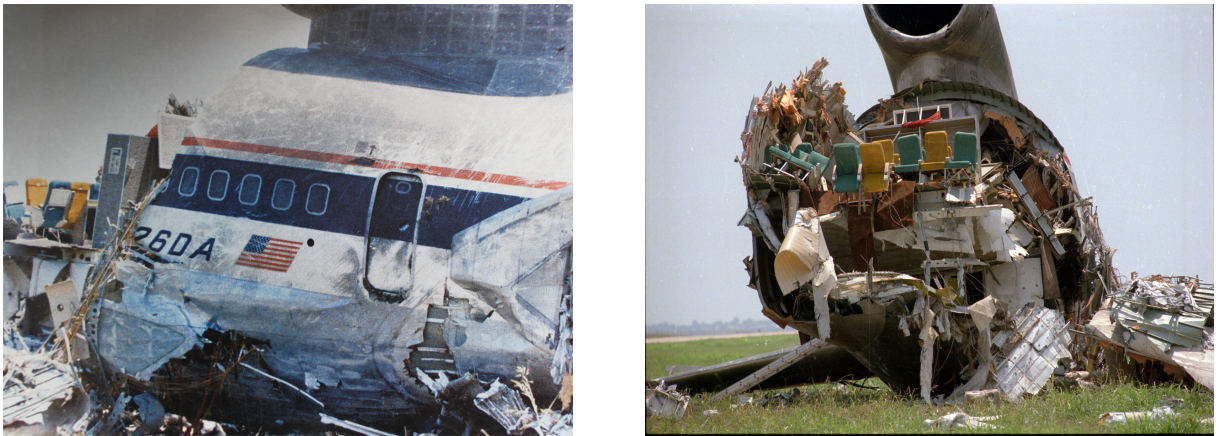


Fig. 1.4. Wreckage of Delta Air Lines Flight 191, which crashed in front of the runway at Dallas-Fort Worth Airport in 1985 due to microburst-induced wind shear during landing [9]. The photographs show severe structural breakup and exposed cabin interior following ground impact.

capture short-duration, high-intensity gusts. In the wind energy industry, the IEC 61400-1 standard [14] offers a consistent framework for turbine certification and defines a set of Design Load Cases (DLCs) to ensure structural integrity under prescribed wind conditions. Among these, gust-related load cases, such as the Extreme Operating Gust (EOG) and the Extreme Direction Change (EDC), are formulated as simplified, one-dimensional variations of wind speed that are assumed to act uniformly across the rotor plane. Such idealized inflow conditions neglect the inherently three-dimensional, intermittent, and direction-changing nature of real atmospheric gusts. As modern turbines continue to grow in size and flexibility, these simplifications become increasingly inadequate for predicting extreme load responses.

Similarly, in civil and structural engineering, design codes such as ASCE 7 [15] (United States) and Eurocode EN 1991-1-4 [16] (Europe) provide methods for calculating wind loads on buildings and structures based on statistically derived gust factors and mean wind profiles. These standards aim to ensure safety under typical storm conditions by prescribing design pressures derived from meteorological records. However, these formulations largely emphasize stationary wind conditions and rely on empirical gust factors that approximate transient effects through simplified multipliers. As a result, they cannot fully resolve the dynamic load evolution or aeroelastic coupling that occur during non-stationary gust events.

In aeronautics, certification requirements such as FAR 25.341 [17] (United States) and CS-25.341 [18] (Europe) mandate that aircraft structures have to be evaluated under discrete gust encounters to ensure safe operation. These regulations prescribe idealized “1-cosine” vertical gust profiles and require that the resulting loads be determined through dynamic analysis accounting for unsteady aerodynamic characteristics and structural degrees of freedom [17, 18]. For preliminary rapid gust-response calculations, the unsteady strip theory is often used in conjunction with Wagner’s and Küssner’s functions. However, its usefulness is restricted by its low fidelity and its limitation to simple geometries [19].

In practice, panel methods (such as vortex-lattice or doublet-lattice methods) offer higher accuracy than the unsteady strip theory and form the basis of certification-level gust-load analyses for transport aircrafts [20]. Nevertheless, they remain limited to flow regimes where the inviscid potential-flow assumption is valid [21]. While effective for certification and early-stage design, these linear aerodynamic models lose accuracy in the presence of flow separation or large disturbances, which motivates the use of high-fidelity CFD and FSI approaches.

To this end, it has been shown that wind gusts can exert drastic effects across different engineering disciplines. Understanding the dynamic interaction between wind gusts and structures is therefore essential for predicting and mitigating transient aerodynamic loads that can compromise safety and performance. Accordingly, the development and application of numerical methods capable of capturing the inherently unsteady and three-dimensional nature of wind gusts provide the physical foundation for understanding these complex phenomena, advancing structural design, and enhancing the reliability of increasingly flexible and sophisticated systems. The work presented in this dissertation, together with the publications on which its chapters are based, builds upon this motivation by developing, implementing, validating, and applying high-fidelity numerical frameworks for the simulation of gust–flow–structure interaction.

1.2 Physical Background

Before exploring what wind gusts are, it is useful to recall why wind exists at all. Winds form because the atmosphere continuously seeks to balance unequal heating across the planet. The equator receives more solar energy than the poles, and Earth’s rotation further disturbs this balance, giving rise to large-scale gradients in temperature and pressure. To restore equilibrium, heat, momentum, and moisture are transported by the wind over the Earth’s surface, guided by Earth’s rotation and the high- and low-pressure systems. While these large-scale processes govern the global circulation of air, the winds that affect structures and everyday life occur much closer to the ground. It is within this near-surface region, known as the *atmospheric boundary layer*.

1.2.1 Atmospheric Boundary Layer (ABL)

The atmospheric boundary layer (ABL), also known as the planetary boundary layer (PBL), is the lowest part of the troposphere and its depth typically ranges from about 100 m to 3000 m, depending on atmospheric conditions [22]. It is the region most strongly influenced by the Earth’s surface through friction, heat exchange, and topographic forcing. Within this layer, turbulence driven by shear and buoyancy creates the highly unsteady flow field in which buildings, wind turbines, vehicles, and most human activities exist.

The ABL undergoes a characteristic diurnal cycle governed by the imbalance between daytime surface heating and nighttime radiative cooling. The principal stages of this evolution are shown in Fig. 1.5, following the canonical description of the boundary layer over land in high-pressure regions given in Stull [22]. During daytime, solar heating

generates buoyant thermals that deepen the *convective boundary layer*, often reaching one to two kilometers in height. At its top lies an *entrainment zone*, a thin transition layer where turbulent thermals mix with the stably stratified free atmosphere and gradually erode the overlying *capping inversion*. After sunset, radiative surface cooling suppresses buoyant turbulence, giving rise to a shallow *stable boundary layer* whose strongly stratified structure limits vertical mixing. Above it, the well-mixed air from the previous day persists as a *residual layer*, largely decoupled from the surface and characterized by weak turbulence. Following sunrise, renewed heating erodes the nocturnal layer from below, establishing a new convective boundary layer. This continuous sequence of growth, decay, and re-formation governs the intensity and vertical distribution of turbulence in the lower atmosphere, thereby shaping the environment in which wind gusts originate and influencing how they interact with structures.

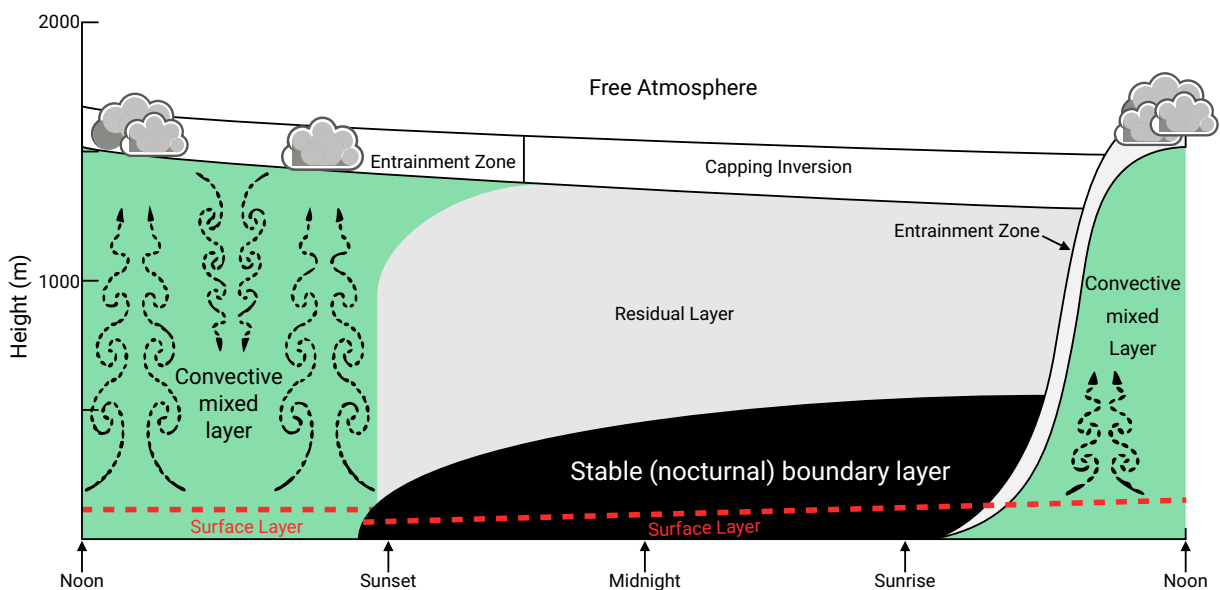


Fig. 1.5. Schematic illustration of the diurnal evolution of the atmospheric boundary layer over land in high-pressure regions. Adapted from the canonical description of Stull [22].

1.2.2 Wind Gusts

Although wind gusts are widely experienced and commonly referenced in weather reports, there is no single universally accepted definition of what a gust is. Different organizations and applications use distinct criteria to distinguish a gust from the mean or sustained wind, and these criteria primarily differ in the averaging period, measurement interval, and magnitude threshold used to identify a short-term wind speed peak.

The World Meteorological Organization (WMO) describes a gust as “*the maximum value of a 3-second running average wind speed*” [23], measured within the observing cycle, which is typically a 10-minute period as defined in the WMO Manual on Codes [24]. The U.S. National Weather Service (NWS) maintains multiple definitions depending on the application. For general meteorological observations, the NWS glossary defines a gust as

“rapid fluctuations in the wind speed with a variation of 10 knots or more between peaks and lulls” [25]. For operational digital forecast products, the NWS adopts a more specific definition, namely “the maximum 3-second wind speed (in knots) forecast to occur within a 2-minute interval at a height of 10 meters above sea level. Wind gust forecasts are valid at the top of the indicated hour” [26, 27]. According to the German Weather Service (Deutscher Wetterdienst, DWD), a gust is “an event in which the 10-minute mean wind speed is exceeded by at least 5.0 m/s (10 knots) for a duration lasting between 3 and 20 seconds” [28]. More broadly, general meteorological literature characterizes a gust as a sudden, brief increase in wind speed, followed by a lull or slackening in the wind speed [29]. Despite differences in formal definitions, these perspectives share a common theme: A gust is a short-duration peak in wind velocity that significantly departs from the mean flow.

The definitions given above are limited to what wind speed sensors typically applied, such as cup or sonic anemometers, are able to capture. These devices record only the temporal fluctuations passing through a single point and therefore cannot resolve the spatial structure of the underlying flow. To determine the spatial scales associated with wind gusts, it is therefore useful to consider the meteorological scales of motion. Figure 1.6 shows a diagram commonly used in meteorology to illustrate the scales of motion within the Earth’s atmosphere and highlights that the time scale of a flow structure is closely linked to its characteristic length as it moves past an observer. Gust events, typically lasting from several seconds to tens of seconds, therefore lie firmly within the *microscale* range and are part of the turbulent motions of the atmospheric boundary layer [30]. Events that originate outside the regular turbulence-generating processes of the atmospheric boundary layer are therefore not considered here. A prominent example is the downdraft associated with thunderstorms, including downbursts and microbursts (see, e.g., Fig. 1.3). These events are driven by precipitation loading and evaporative cooling, which accelerate a column of air downward. After reaching the surface, the descending air spreads out radially from the impact point, generating high wind speeds and elevated turbulence levels. Owing to their kilometer-scale horizontal extent and lifetimes of several minutes to tens of minutes, downbursts and microbursts are classified as mesoscale phenomena [31]. These events arise from the dynamics of convective storms, of which thunderstorms are a primary example, rather than from atmospheric boundary-layer turbulence, and therefore fall outside the scope of the microscale gusts discussed in this work.

1.2.3 Microscale Gust Generation Mechanisms in the ABL

Microscale gusts are a direct manifestation of the turbulence within the atmospheric boundary layer. Within this near-surface region, the flow is continuously disturbed by shear, buoyancy, and surface inhomogeneities (including obstacles), which generate a broad spectrum of eddies spanning the spatial and temporal scales associated with microscale gust events. As these eddies advect past an observer, they produce short-term fluctuations in wind speed that appear as gusts. These gusts typically occur over spatial scales of a few meters to several hundred meters and evolve on time scales from fractions of a second to several seconds. The principal mechanisms responsible for their formation are

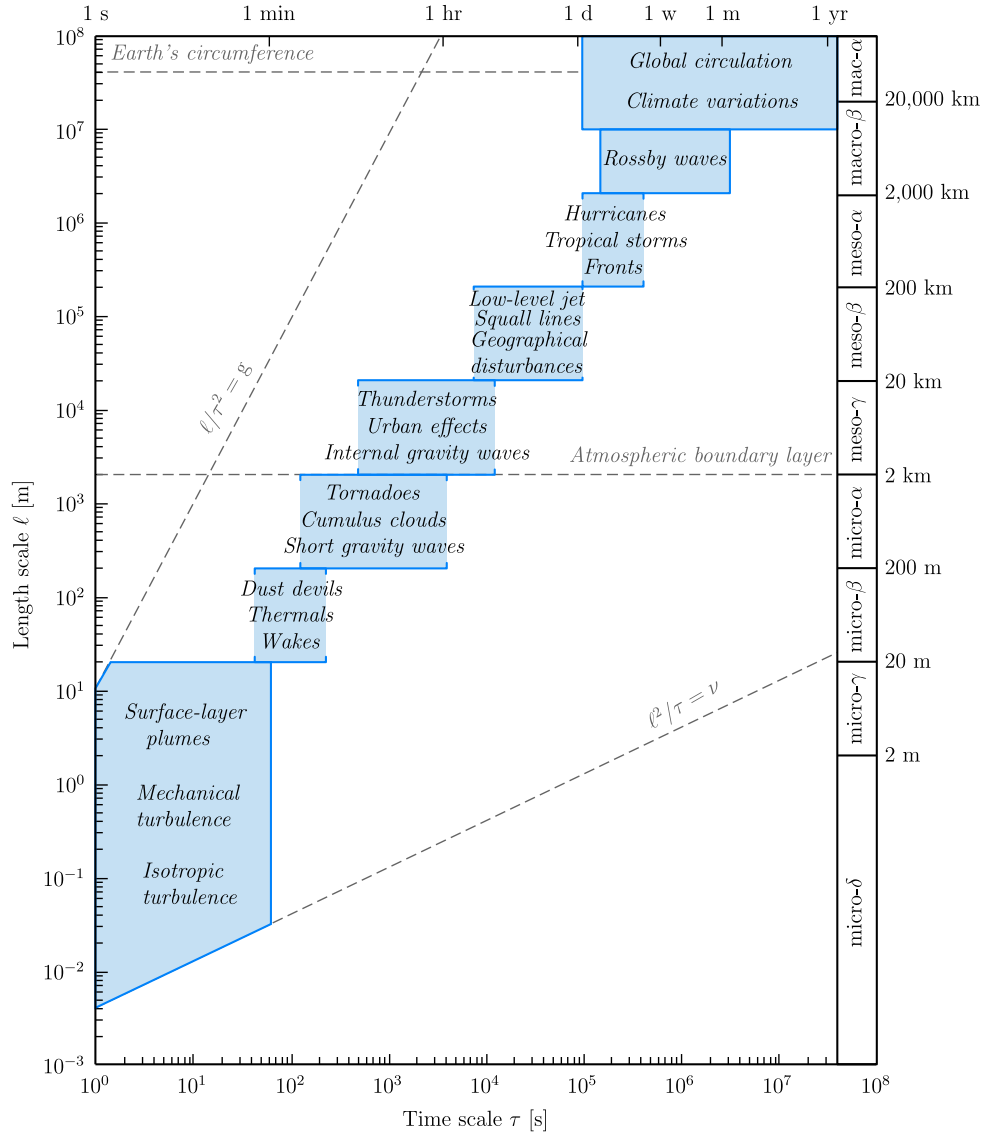


Fig. 1.6. Meteorological scales of motion. Adapted from Bos [30], who compiled them from Stull [22], Orlanski [32] and Steyn et al. [33].

mechanical (shear-driven) turbulence and *thermal (convective) turbulence*, which dominate under neutral and unstable conditions, respectively. These conditions arise naturally from the diurnal evolution of the atmospheric boundary layer described in Section 1.2.1.

Under *neutral* conditions, where buoyancy effects are negligible and turbulence is maintained primarily by mechanical shear, the vertical gradient of wind speed near the ground becomes the main source of turbulent kinetic energy. As air layers move at different velocities, slower close to the surface and faster aloft, the resulting velocity shear can become dynamically unstable and give rise to shear instabilities, including Kelvin-Helmholtz-type motions in sufficiently sharp shear layers. These instabilities generate wave-like perturbations that break down into coherent vortical structures.

Another and most prominent phenomenon are *streamwise vortices* and associated streaks,

which dominate the structure of the neutral atmospheric boundary layer. As these elongated vortical structures rotate, they intermittently transport high-momentum air from aloft downward and lift slower-moving air upward, thereby enhancing vertical momentum exchange. The downward transport events, known as *sweeps*, momentarily accelerate the near-surface flow and manifest as short, intense *shear-driven gusts*. Conversely, the upward ejections of lower-momentum air temporarily reduce the local surface wind, producing the characteristic intermittency of turbulence observed under neutral conditions.

In addition to shear generated by the mean velocity gradient, mechanical turbulence also arises from the flow interacting with obstacles such as buildings, vegetation, and topographic features (see Fig. 1.7). As the wind encounters these structures, flow separation and rapid acceleration around edges create strong local shear layers that roll up into vortices and form turbulent wakes. These small-scale eddies, produced predominantly by mechanical forcing, when advected past an observer, generate brief accelerations in the near-surface flow that appear as obstacle-induced microscale gusts.

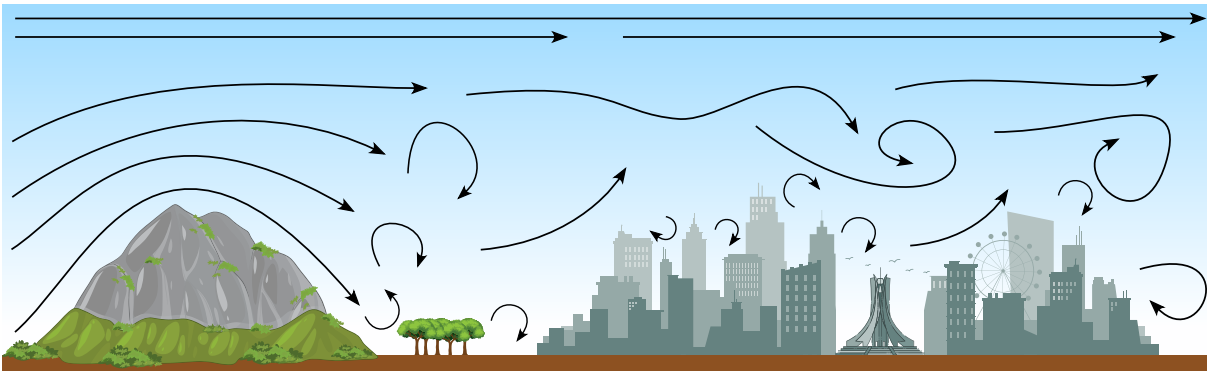


Fig. 1.7. Schematic of mechanical turbulence. Flow over topography, vegetation, and urban structures.

During daytime, intense surface heating creates a steep vertical temperature gradient and a near-surface superadiabatic layer that make the lower atmosphere buoyantly unstable. This causes warm air parcels near the surface to rise as buoyant updrafts (thermals). Meanwhile, slightly cooler mixed-layer air in the surrounding regions subsides to replace the rising air, establishing compensating downdrafts. The resulting pattern of rising thermals surrounded by broader regions of slow subsidence organizes into turbulent plumes that dominate the structure of the convective boundary layer, as depicted in Fig. 1.8. Along the edges of these plumes, vigorous turbulent mixing redistributes heat and momentum throughout the layer. As parts of these circulations descend toward the surface, they transport higher-momentum air from aloft downward and generate smaller-scale turbulent eddies in the surface layer. When these eddies advect past a fixed point near the ground, they produce sharp, transient fluctuations in wind speed that are experienced as microscale convective gusts.

Microscale gusts therefore represent the direct manifestation of the intermittent redistribution of turbulent kinetic energy within the ABL. Their intensity, frequency, and spatial structure depend strongly on surface roughness, atmospheric stability, and mean wind

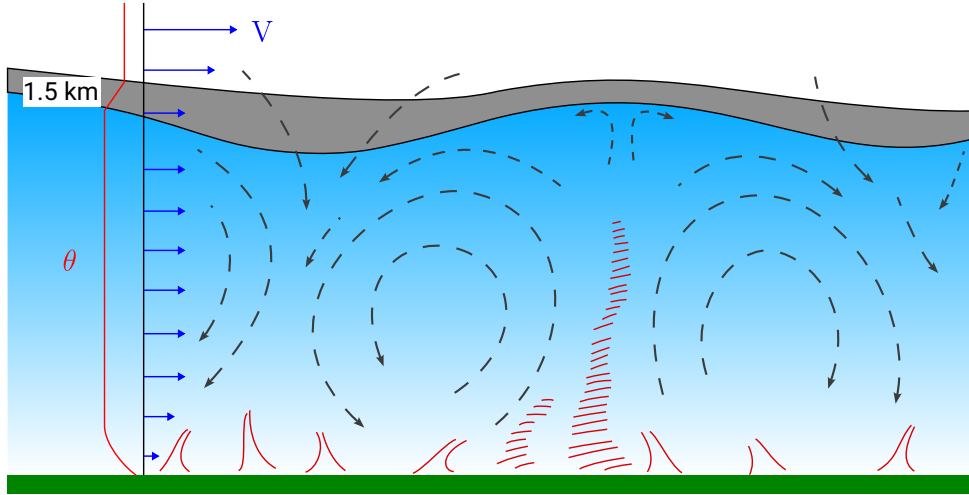


Fig. 1.8. Schematic of thermal turbulence in the convective boundary layer. Adapted from Wyngaard [34], showing boundary layer circulation and entrainment across the entrainment zone. Here, V denotes the mean horizontal wind and θ the temperature.

speed. In the numerical investigations presented in this thesis, these naturally occurring thermal and mechanical processes are not modeled explicitly. Instead, *gusts are introduced as isothermal kinematic perturbations* to the incompressible velocity field.

1.3 Numerical Injection of Discrete Wind Gusts

While full-scale measurements under natural conditions offer the most direct representation of gust-induced loading, they are rarely feasible: Strong gusts cannot be triggered or reproduced on demand, simultaneous flow and structure measurements are difficult to obtain, and large-scale experimental setups are prohibitively expensive. Consequently, numerical simulation has become a central approach for studying wind-gust dynamics and their interaction with structures. Numerical methods allow precise specification of gust characteristics, controlled variation of physical parameters, and direct comparison with experimental data, while providing detailed spatial and temporal resolution of both the flow field and the structural response.

From a modeling standpoint, wind gusts can generally be classified into *continuous* and *discrete* types [35, 36]. Continuous gusts are characterized by persistent fluctuations superimposed on a mean flow, mimicking the stochastic nature of atmospheric turbulence. They are well suited for long-term load assessments and corresponding fatigue analyses but are rarely employed for the determination of extreme transient loads. In contrast, discrete gusts represent short-duration, spatially localized flow disturbances that evolve and dissipate within a finite region of space and time. The present thesis concentrates on the discrete type of gust.

The velocity signal of a discrete gust can be defined using either a *deterministic* model or a *stochastic* model. Deterministic definitions prescribe the gust explicitly (at least during its injection phase) through analytical expressions, allowing full control over its amplitude,

spatial extent, timing, and shape. Such signals enable systematic parametric studies and reproducible comparisons between injection strategies. In contrast, stochastic models generate gusts implicitly by constraining a turbulent velocity signal to pass through a prescribed peak while preserving the statistical properties of atmospheric turbulence [37]. These approaches produce physically more realistic gust-like fluctuations but offer less control over the gust shape and introduce inherent randomness into the signal. Given the focus of this work on controlled evaluation of gust-injection methodologies and their effect on fluid–structure interaction, only deterministic discrete gust definitions are considered in the present dissertation. Accordingly, this dissertation focuses exclusively on *microscale discrete wind gusts* defined *deterministically* and introduced into the computational domain using deterministic model-based injection methods.

Several numerical strategies have been developed to inject such gusts within a computational framework. These strategies differ in how and where the disturbance is introduced, how global conservation properties are maintained, and how strongly the gust–flow–structure coupling is captured. The following outlines the main families of methods and highlights their respective advantages and limitations in the context of incompressible flow simulations.

The *far-field boundary condition* (FBC) method represents one of the most conventional approaches for injecting wind gusts into a computational domain. In this method, the gust is superimposed at the inlet boundary and allowed to propagate freely through the domain. To preserve the global mass balance, particularly in incompressible flow simulations, the inlet velocity must be dynamically adjusted outside the gust injection region to maintain a constant total mass flow rate through the inlet [38, 39]. This adjustment is essential to ensure numerical convergence of incompressible solvers. Because inflow boundaries are often located far from the region of interest, a coarser grid resolution is typically employed upstream to reduce computational cost. As a result, numerical dissipation becomes significant in these regions, causing the injected disturbance to weaken before reaching the target area. To alleviate this issue, a uniformly fine grid can be used throughout the domain or a secondary refined moving mesh can be employed to transport the gust, as in overset-grid implementations of the resolved gust approach [40]. Both remedies, however, considerably increase computational cost and simulation runtime, which limits the practical applicability of the FBC method for high-fidelity studies of localized gust–structure interactions. For these reasons, the FBC method is not employed in the present thesis.

The *prescribed velocity methods* form another class of techniques for gust injection, originally developed in aeronautical applications for the evaluation of indicial aerodynamic responses. In contrast to the FBC approach, which injects the gust at the inlet and then allows it to propagate freely through the domain, these methods define the position and shape of the imposed velocity field at each instant in time. Two main variants are recognized: The *field velocity method* (FVM) and the *split velocity method* (SVM).

The *field velocity method* (FVM) was initially proposed by Parameswaran and Baeder [41] and Singh and Baeder [42]. In this approach, the gust effect is introduced by superimposing a prescribed velocity field onto the governing equations through a modification

of the grid velocity or, equivalently, the convective fluxes of the mass and momentum equations. Although the grid velocity is apparently altered, no actual deformation of the computational mesh occurs. Hence, the FVM employs a grid-velocity term that is formally similar to that appearing in the Arbitrary Lagrangian–Eulerian formulation, but without any actual mesh motion. Historically, FVM has been widely used in inviscid and viscous aerodynamic simulations to compute aerodynamic loads due to step changes in parameters such as the angle of attack or pitch rate, and to model airfoil–gust encounters as well as vortex–blade interactions [41, 43, 44, 45]. While the method provides accurate predictions of indicial responses, it neglects the feedback of the surrounding flow on the imposed gust field, which limits its accuracy for short-wavelength or strongly coupled flow conditions.

The *split velocity method* (SVM), introduced by Wales et al. [19], was developed to address this limitation. In SVM, the total velocity field is decomposed into a prescribed gust component and a remaining background component, as also described by Huntley et al. [46]. The momentum equations are reformulated to solve for the background velocity, while additional source terms are introduced to account for the feedback effect of the flow on the gust. This approach fully captures the interaction between the gust and the object of interest. Initial applications involved two-dimensional Euler simulations of 1-cosine vertical gusts acting on NACA airfoils with different amplitudes and wavelengths [19]. Later studies extended the method to three-dimensional viscous flows and flexible aircraft configurations, demonstrating that SVM provides improved accuracy compared to FVM, particularly for short-wavelength gusts and strong flow–structure interactions [46, 47, 48, 49, 50]. This class of methods is presented in Chapter 2 and employed in Publications I and II.

The *pure source-term* (ST) *formulation* provides an alternative gust-injection strategy that avoids inlet-based gust injection and circumvents the need for fine grid resolution throughout the entire computational domain. In this method, the gust is injected directly within a localized region by introducing an additional source term into the momentum conservation equations [51]. This formulation allows the gust to be generated close to the region of interest rather than at the far-field boundary, which minimizes numerical dissipation and enables the use of coarser grids in regions that are not directly affected by the disturbance. Once injected, the gust propagates freely through the flow field, and the full coupling between the gust and the surrounding fluid is inherently captured. An important advantage of this approach is that it maintains global mass conservation while simultaneously accounting for the feedback of the flow field on the evolving gust. By eliminating the requirement for globally refined meshes, the pure source-term formulation offers a computationally efficient and physically consistent means of simulating localized gust–flow interactions within high-fidelity incompressible solvers. This formulation is presented in detail in Chapter 3 and employed in Publication III.

All discrete gust approaches described so far rely on the assumption that the velocity distribution within the gust, at least during the injection phase, is known a priori and can be explicitly prescribed. In these formulations, deterministic analytical functions

are commonly used to define the spatial and temporal evolution of the gust, such as the “1-cosine” *Extreme Coherent Gust* (ECG) defined in certification regulations [17, 18] or the *Extreme Operating Gust* (EOG) specified in the IEC standard [14].

A conceptually different category of numerical gust-generation techniques is introduced in this dissertation, referred to as the *immersed boundary gust generators* (IBGG). In contrast to the previously discussed methods, this category does not impose a predefined velocity field. Instead, the gust is produced as a physical consequence of the motion of a mechanical device interacting with the surrounding flow. The underlying idea originates from experimental approaches for gust generation that employ *modified boundary layer wind tunnels with active elements*, where mechanical devices such as movable plates or gates are used to produce controlled flow disturbances. The main intention of this category is therefore to provide a complementary numerical counterpart to the laboratory gust-generation mechanisms, referred to as *wind gust generators* (WGG). By directly modeling the kinematics of these devices within the computational domain using the immersed boundary method, the gust field develops naturally from the flow solution. This enables a highly realistic reproduction of experimentally generated gusts and allows for a direct one-to-one comparison between numerical predictions and wind-tunnel measurements. This category, presented in Chapter 4, comprises two immersed boundary gust generators developed in this dissertation, each corresponding to an experimental counterpart, namely the *paddle* [52] and the *adaptive nozzle* [53]. The numerical model of the *paddle* is introduced in Publication IV and employed in Publication V, while the numerical counterpart of the *adaptive nozzle* is presented in Publication VI.

In summary, the numerical strategies reviewed in this section encompass the principal approaches available for simulating discrete wind gusts in incompressible flow solvers. Within the scope of this dissertation, these gust-injection methods (except FBC) are implemented in the finite-volume solver FASTEST-3D [54] and employed to investigate discrete gust–flow–structure interactions under various configurations.

1.4 Objectives of the Thesis

The overarching objective of this thesis is to advance the numerical modeling, simulation, and analysis of discrete wind gusts in incompressible flow solvers, with a particular focus on their interaction with rigid and flexible structures¹. As presented in Section 1.2.3, wind gusts in nature are fundamentally turbulent disturbances that arise from complex turbulence-generating mechanisms. Their short duration, spatial variability, and multi-scale structure make them challenging to reproduce numerically, because a gust must be introduced into an already-evolving flow field without violating conservation laws, disrupting the resolved turbulence, or compromising solver stability. These challenges become more pronounced when the focus shifts from rigid bodies to lightweight or flexible structures. In such configurations, the structural response is highly sensitive to the instantaneous and spatially localized aerodynamic loads imposed by a gust. Errors or

¹This aligns with the objectives of the project that was financially supported by the German Research Foundation (Deutsche Forschungsgemeinschaft, DFG) under grant number BR 1847/17–1.

inconsistencies in how the perturbation is introduced into the computational domain, such as violations of conservation laws or the introduction of discontinuities in the injected gust field, can modify the local pressure distribution and lead to significant deviations in the gust-induced aerodynamic forcing and the corresponding predicted deformation.

As a result, there is a clear need for numerical methodologies that can generate individual gusts or sequences of gusts directly within the computational domain, in close proximity to the structure. Such methods shall be compatible with eddy-resolving simulations (LES) and preserve the full interaction between the injected gust, the background turbulent flow, and the structure. Building on the above considerations, the objectives of this thesis are as follows:

- A.** To develop numerical methodologies for injecting discrete wind gusts relying on an incompressible finite-volume flow solver.
- B.** To extend the application of these gust-injection techniques from rigid configurations to fully coupled fluid–structure interaction problems.
- C.** To evaluate the capabilities and limitations of each gust-injection method.
- D.** To validate the considered gust-injection techniques, where possible, against numerical and especially experimental reference data.

Objective **A** encompasses the three discrete wind gust injection techniques that were either existing or developed within the scope of this thesis. This includes: i) the systematic derivation and implementation of the prescribed velocity methods (FVM, SVM), together with the development of a deep understanding of their underlying physics, ii) revisiting the derivation of the source-term (ST) formulation and addressing its underlying modeling assumptions, iii) the development and implementation of the novel immersed boundary gust generators (IBGG).

Objective **B** involves applying the considered wind-gust injection techniques to a wide range of test cases of increasing complexity. For the prescribed velocity methods, the test cases include fixed and elastically mounted airfoils. For the source-term formulation, the methodology is applied to rigid and flexible (membranous) hemispherical structures. Finally, for the immersed boundary gust generators, the approach is applied to both rigid and flexible T-shaped structures.

Objective **C** is pursued throughout the work associated with the first two objectives and focuses on assessing the strengths and limitations of each technique. The assessment covers the physical accuracy, conservation behavior, the suitability of the methods for different gust orientations, computational efficiency, and the ability of each approach to represent the interaction between the injected gust, the surrounding flow, and the structure (i.e., the feedback effect).

Finally, Objective **D** includes the validation of the wind-gust injection techniques against existing numerical benchmarks (for the prescribed velocity methods) and wind-tunnel measurements (for the immersed boundary gust generators). The source-term formulation is not validated experimentally, since no laboratory setup currently exists that can generate the type of localized gust produced by this approach.

1.5 Outline of the Thesis

The structure of this publication-based dissertation is organized into two parts.

Part I. The first part summarizes the physical context, numerical methodologies for wind gust injection, and the main findings of this work. It begins with Chapter 2, which provides a comprehensive summary of the numerical methodology underlying prescribed velocity gust-injection techniques and gathers the principal findings from the first two publications (Publications I and II). Chapter 3 then introduces the source-term based gust-injection formulation, presenting its derivation and a synthesis of the principal results from Publication III. This is followed by Chapter 4, which provides an overview of artificial discrete gust generation via immersed-boundary gust generators and consolidates the results from Publications IV, V, and VI. The first part concludes with Chapter 5, which provides an outlook toward related future research directions.

Part II. The second part of the dissertation contains the six peer-reviewed publications, on which this work is based. Each publication is presented in full and directly corresponds to the methodologies and findings synthesized in the first part.

2 Discrete Wind Gust Injection via Prescribed Velocity Methods

This chapter presents a comprehensive summary of the numerical methodology underlying prescribed velocity gust injection techniques, along with the main findings from the first two publications of this dissertation (i.e., Publication I¹ and Publication II²). The first publication systematically derives both the Field Velocity Method (FVM) and the Split Velocity Method (SVM), as detailed in Section 2.1. The derivation highlights their connection to the Arbitrary Lagrangian-Eulerian (ALE) formulation and the geometric conservation law (GCL), both of which are commonly employed in the numerical simulation of fluid–structure interaction (FSI) problems. These methods are evaluated using a canonical test case involving vertical and horizontal gusts interacting with a fixed flat plate, as described in Section 2.2. In the second publication, the SVM is applied to a more complex configuration involving an elastically mounted rigid airfoil subjected to vertical gusts, with the objective of investigating the aeroelastic response and determining whether limit-cycle oscillations (LCO) or flutter instabilities are triggered. The main results of this second publication are presented in Section 2.3.

2.1 Numerical Methodology

The following introduces two established methods for injecting gusts into CFD and FSI simulations by prescribing their position, shape, and velocity at each time step within the computational domain. This renders prescribed velocity methods advantageous over other approaches, as numerical dissipation errors are eliminated and a coarser mesh can be used away from the object of interest. As a result, a significant reduction in computational cost can be realized. Within this class of methods, the gust velocity is denoted as $u_{g,i}$ ($i \in \{1, 2, 3\}$) in Cartesian coordinates, while the surrounding flow is represented by the background velocity field $\tilde{u}_i$. The incompressible Navier-Stokes equations are solved for this background field, thereby enforcing the divergence-free constraint $\partial \tilde{u}_i / \partial x_i = 0$. To obtain the total velocity field u_i , needed for instance for visualization, the superposition $u_i = \tilde{u}_i + u_{g,i}$ is required.

As shown in the following sections, regardless of the specific method, the mass fluxes in the governing equations are based on the total velocity. Because the divergence of the prescribed gust velocity field (i.e., $\partial u_{g,i} / \partial x_i$) is generally not zero, but the methods solve the remainder of the flow field, an additional treatment is necessary. To introduce $u_{g,i}$

¹Boulbrachene, De Nayer & Breuer, *J. Wind Eng. Ind. Aerodyn.* **218**, 104790 (2021).

²Boulbrachene & Breuer, *J. Fluids Struct.* **117**, 103828 (2023).

into the Navier-Stokes equations without violating $\partial \tilde{u}_i / \partial x_i = 0$, the ALE formulation is employed in conjunction with the GCL.

2.1.1 Field Velocity Method (FVM)

The field velocity method, also known as the grid velocity approach, was first introduced by Parameswaran and Baeder [41] in the context of aeronautical applications. Its original purpose was to facilitate the computation of indicial aerodynamic responses of airfoils subjected to step changes in the angle of attack, the pitch rate, or the encounter of a sharp-edged gust. The method achieves this by modeling an unsteady external flow field as if it were induced by a virtual (apparent) motion of the computational grid. Physically, the grid velocity corresponds to the velocity assigned to each mesh point during the unsteady displacement of a boundary surface. Therefore, for example, a step change in the angle of attack of an airfoil (see Fig. 2.1a) can be equivalently modeled by prescribing a step change in the vertical grid velocity across the entire flow domain (see Fig. 2.1b). In the FVM, however, the grid remains stationary. Instead, the grid time metrics (i.e., the grid fluxes) are computed to mimic the effect of grid motion, thereby capturing the desired unsteady behavior without physically deforming the mesh (see Fig. 2.1c). Building on this core idea, the FVM models the effects of an external unsteady velocity field (such as a wind gust) by introducing a *pseudo-grid motion* within the ALE framework.

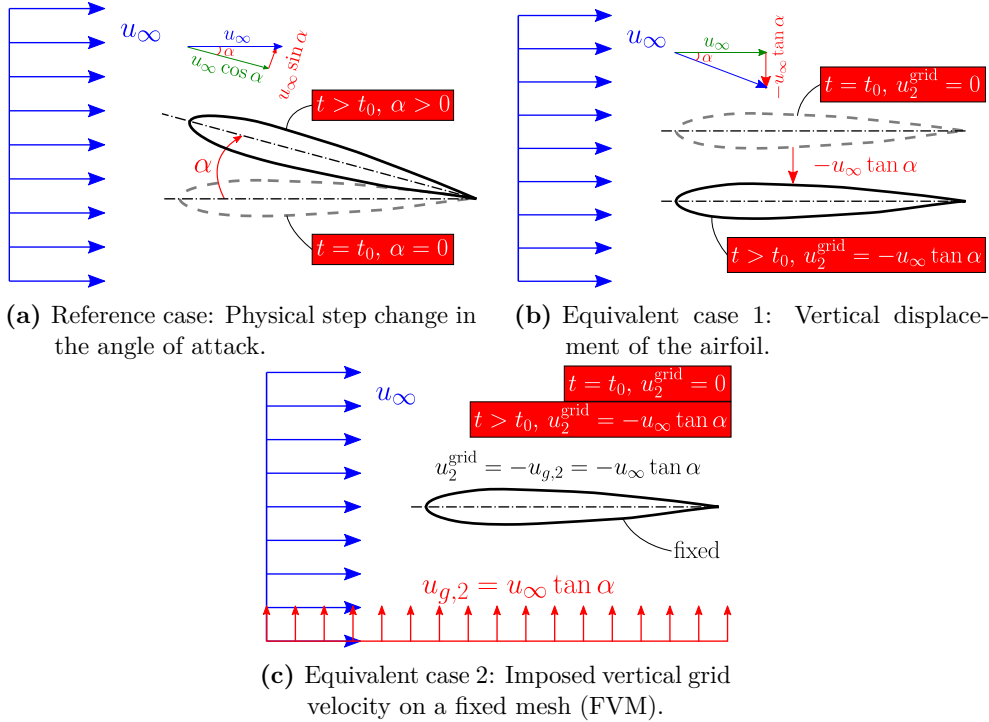


Fig. 2.1. Illustration of the field velocity method: A physical step change in the angle of attack α (a) can be equivalently represented either by a vertical displacement of the airfoil (plunge motion) (b) or by a pseudo-grid motion imposed on a fixed mesh using the FVM (c).

In the following, the ALE formulation is first introduced, followed by the FVM, to establish the underlying connection between both approaches. The ALE formulation provides a general framework for describing fluid flow on moving or deforming computational domains. It is derived by applying the Reynolds transport theorem (RTT) to a control volume (CV) that moves with an arbitrary or a prescribed grid velocity u_i^{grid} . This enables a consistent treatment of the governing equations when the mesh deforms due to structural motion, as is typical in FSI problems. In the context of a temporally varying domain (e.g., see [55]), the Navier-Stokes equations for mass and momentum conservation are expressed in the ALE form. For an incompressible fluid, the integral form of the governing equations reads:

$$\frac{d}{dt} \int_{V(t)} \rho \, dV + \int_{S(t)} \rho (\tilde{u}_i - u_i^{\text{grid}}) n_i \, dS = 0 , \quad (2.1)$$

$$\frac{d}{dt} \int_{V(t)} \rho \tilde{u}_i \, dV + \int_{S(t)} \rho \tilde{u}_i (\tilde{u}_j - u_j^{\text{grid}}) n_j \, dS = - \int_{S(t)} p n_i \, dS - \int_{S(t)} \tilde{\tau}_{ij} n_j \, dS . \quad (2.2)$$

Here, ρ is the fluid density, p is the pressure, and μ is the dynamic viscosity. The stress tensor $\tilde{\tau}_{ij}$ is based on the flow velocity $\tilde{u}_i$. u_i^{grid} denotes the grid velocity with which the surface of a control volume is moving. The control volumes have time-dependent volumes $V(t)$ and surfaces $S(t)$, and n_i is the outward-pointing unit normal vector on each surface of the control volume. In the ALE form, the governing equations describe local changes in a control volume of variable shape and consequently include additional mass and momentum fluxes induced by u_i^{grid} . Evaluating these grid-movement-induced fluxes directly from computed grid velocities generally violates discrete mass and momentum conservation. Instead, the so-called *space conservation law (SCL)* [56, 57], also known as the *geometric conservation law (GCL)* [58] is employed. The integral form of the GCL can be obtained from the mass conservation equation (2.1) by setting $\rho = 1$ and $\tilde{u}_i = 0$:

$$\frac{d}{dt} \int_{V(t)} dV - \int_{S(t)} u_i^{\text{grid}} n_i \, dS = 0 . \quad (2.3)$$

Primarily, the GCL states that the change in volume of a deforming control volume over time must be equal to the sum of the volumes swept by its enclosing surfaces. This additional conservation law ensures that no volume is artificially lost or introduced during the deformation of control volumes. By inserting the GCL (i.e., Eq. (2.3)) into the mass conservation equation (2.1), the grid-movement contribution to the mass fluxes (i.e., $-\int_{S(t)} \rho u_i^{\text{grid}} n_i \, dS$) cancels out the unsteady term (i.e., $\frac{d}{dt} \int_{V(t)} \rho \, dV$), thereby reducing the mass conservation equation to:

$$\int_{S(t)} \rho \tilde{u}_i n_i \, dS = 0 . \quad (2.4)$$

This expression is equivalent to the mass conservation equation on a fixed grid. Consequently, no explicit prediction of additional grid fluxes is required for the mass conservation in the context of moving grids, and the pressure-correction equation used in projection methods remains unchanged. The additional grid fluxes in the momentum equation, however, must still be determined. This is done using the discrete form of the GCL, by which the fluxes are computed from the volumes swept by each face of the control volume, typically using a tetrahedral decomposition [55, 59]. The procedure can be shown by first rearranging the terms in the GCL given in Eq. (2.3) as:

$$\int_{S(t)} u_i^{\text{grid}} n_i dS = \frac{d}{dt} \int_{V(t)} dV . \quad (2.5)$$

In ALE-based solvers, the computational grid is adapted at the beginning of each new time step t^{n+1} . Consequently, all geometric quantities (such as volumes) at t^{n+1} are known, and the Navier-Stokes equations are solved on the updated grid. The temporal derivative of the volume integral appearing in Eq. (2.5) can therefore be expressed, for example, by a first-order accurate approximation as:

$$\frac{d}{dt} \int_{V(t)} dV \approx \frac{V^{n+1} - V^n}{\Delta t} , \quad (2.6)$$

where V^n and V^{n+1} are the volumes of the control volume at the time steps t^n and t^{n+1} , respectively. The volume difference can also be expressed as the sum of the swept volumes associated with each face a of the control volume:

$$\frac{V^{n+1} - V^n}{\Delta t} = \frac{\sum_a \delta V_a^{n+1}}{\Delta t} . \quad (2.7)$$

For the hexahedral control volumes treated by the flow solver used in this thesis (i.e., $a = \{e, w, n, s, t, b\}$), six swept volumes (each associated with a face) are computed. Figure 2.2 illustrates an example of the swept volume associated with the east face of a control volume, resulting from its deformation between two consecutive time steps. Combining the above relations yields the following first-order accurate approximation of the grid fluxes based on swept volumes:

$$\int_{S(t)} \rho \tilde{u}_i u_j^{\text{grid}} n_j dS \approx \sum_{a=\{e,w,n,s,t,b\}} \left(\rho \tilde{u}_{i,a} \frac{\delta V_a^{n+1}}{\Delta t} \right) . \quad (2.8)$$

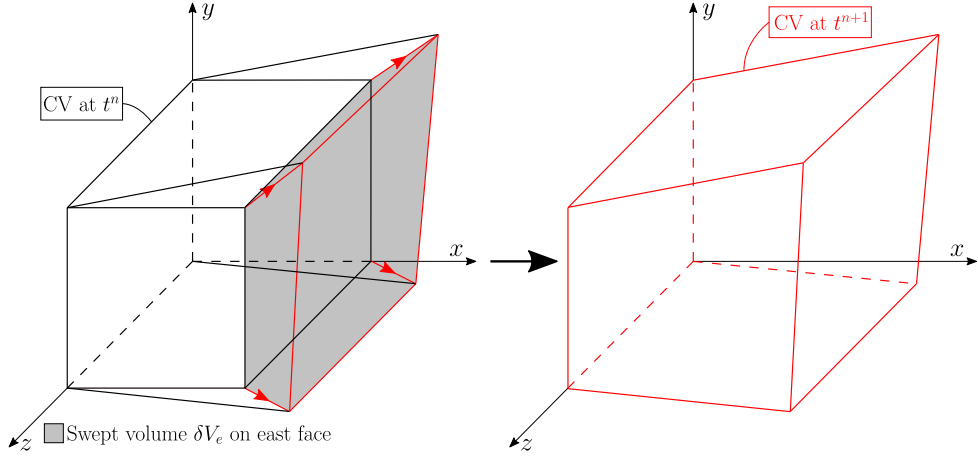


Fig. 2.2. Illustration of the actual swept volume δV_e on the east face of a control volume, resulting from the mesh deformation between two consecutive time steps.

A second-order accurate approximation of the grid flux can be obtained by approximating the temporal derivative in Eq. (2.5) using a second-order backward difference [59]:

$$\begin{aligned} \frac{d}{dt} \int_{V(t)} dV &\approx \frac{3V^{n+1} - 4V^n + V^{n-1}}{2\Delta t} = \frac{3(V^{n+1} - V^n) - (V^n - V^{n-1})}{2\Delta t} \\ &= \frac{\sum_a (3\delta V_a^{n+1} - \delta V_a^n)}{2\Delta t}. \end{aligned} \quad (2.9)$$

The corresponding grid flux is therefore expressed as:

$$\int_{S(t)} \rho \tilde{u}_i u_j^{\text{grid}} n_j dS \approx \sum_{a=\{e,w,n,s,t,b\}} \left(\rho \tilde{u}_{i,a} \frac{3\delta V_a^{n+1} - \delta V_a^n}{2\Delta t} \right). \quad (2.10)$$

It should be noted, however, that this second-order formulation requires information from three consecutive time steps (i.e., t^{n-1} , t^n , and t^{n+1}). Since the time-integration scheme adopted in the present work (three-substeps explicit Runge-Kutta scheme) relies only on two consecutive time steps (t^n and t^{n+1}), this second-order grid-flux evaluation is not applicable, and the first-order approximation given in Eq. (2.8) is used instead.

The field velocity method is constructed by analogy to the previously described ALE formulation. The following focuses on the case of fixed (non-moving) structures, where the computational grid remains undeformed and is therefore fixed. This implies that the control volumes V and surfaces S are no longer time-dependent. A brief explanation concerning the extension to the case of moving structures is provided at the end of this section.

In the FVM, incorporating an external disturbance is achieved by replacing the grid velocity u_i^{grid} appearing in the ALE formulation of the governing equations (i.e., Eqs. (2.1) and (2.2)) with the negative gust velocity $-u_{g,i}$. The motivation for this substitution

has already been discussed in connection with Fig. 2.1. The corresponding governing equations are therefore given by:

$$\frac{d}{dt} \int_V \rho \, dV + \int_S \rho (\tilde{u}_i + u_{g,i}) n_i \, dS = 0 , \quad (2.11)$$

$$\frac{d}{dt} \int_V \rho \tilde{u}_i \, dV + \int_S \rho \tilde{u}_i (\tilde{u}_j + u_{g,j}) n_j \, dS = - \int_S p n_i \, dS - \int_S \tilde{\tau}_{ij} n_j \, dS . \quad (2.12)$$

Since the prescribed gust velocity is not necessarily divergence-free, a GCL must still be enforced, analogous to the ALE formulation, to ensure that the contribution of the pseudo-grid motion to the mass fluxes cancels out the unsteady term in the mass conservation equation (i.e., Eq. (2.11)). In the FVM, strict adherence to a GCL is crucial to prevent the introduction of artificial source terms and to satisfy the postulate of $\partial \tilde{u}_i / \partial x_i = 0$ as found by Sitaraman and Baeder [45]. In view of the fact that the imposed grid movement is purely artificial (i.e., the computational grid remains fixed), the GCL applied in the context of the FVM is referred to as the *pseudo geometric conservation law* in this work. As in the governing equations, the pseudo GCL is obtained by replacing the grid velocity u_i^{grid} with the negative gust velocity $-u_{g,i}$ in Eq. (2.3), resulting in:

$$\frac{d}{dt} \int_V dV + \int_S u_{g,i} n_i \, dS = 0 . \quad (2.13)$$

Following the approach used in the ALE formulation, the pseudo GCL enables the reduction of Eq. (2.11) to Eq. (2.4) and facilitates the computation of the additional grid fluxes in the momentum equation. Figure 2.3 illustrates an arbitrarily prescribed disturbance profile (i.e., $u_{g,1}(z)$) acting on a control volume, along with the resulting swept volume on its east face. Although the actual cell volume remains constant at the time step t^{n+1} , a *pseudo* change in volume must be accounted for. This artificial change represents the apparent grid motion, as described by Sitaraman and Baeder [45]. Incorporating this pseudo-volume change is essential for the temporal integration of the momentum equation.

For problems involving moving structures, the ALE formulation is typically employed, with the computational grid adapted according to displacements prescribed at the fluid–structure interface to maintain conformity. Incorporating an external disturbance via the field velocity method in this context requires the approach to be extended to account for both physical and apparent grid motions [45]. The corresponding governing equations are there-

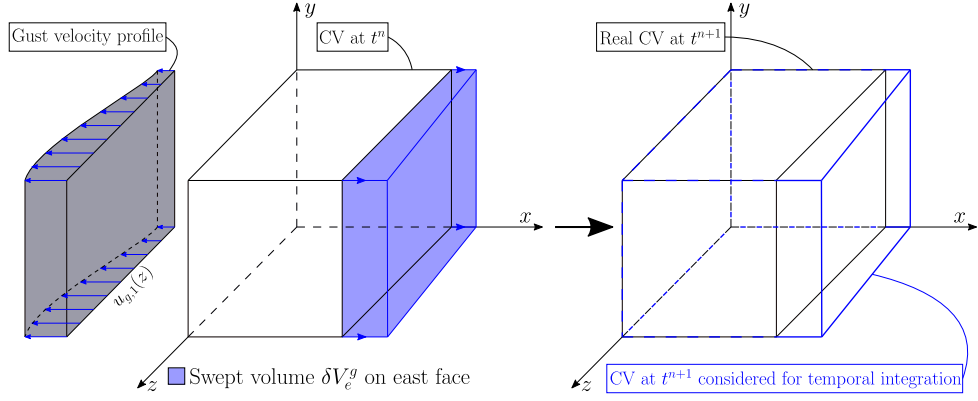


Fig. 2.3. Illustration of apparent swept volume δV_e^g on the east face of a control volume, resulting from the field velocity method.

fore obtained by replacing the grid velocity u_i^{grid} in Eqs. (2.1) and (2.2) with $(u_i^{\text{grid}} - u_{g,i})$, leading to:

$$\frac{d}{dt} \int_{V(t)} \rho dV + \int_{S(t)} \rho (\tilde{u}_i - u_i^{\text{grid}} + u_{g,i}) n_i dS = 0, \quad (2.14)$$

$$\frac{d}{dt} \int_{V(t)} \rho \tilde{u}_i dV + \int_{S(t)} \rho \tilde{u}_i (\tilde{u}_j - u_j^{\text{grid}} + u_{g,j}) n_j dS = - \int_{S(t)} p n_i dS - \int_{S(t)} \tilde{\tau}_{ij} n_j dS. \quad (2.15)$$

Applying the same substitution to the geometric conservation law yields:

$$\frac{d}{dt} \int_{V(t)} dV - \int_{S(t)} (u_i^{\text{grid}} - u_{g,i}) n_i dS = 0. \quad (2.16)$$

In this formulation, grid fluxes are evaluated by accounting for both actual mesh deformation and the imposed gust velocity. Figure 2.4 illustrates the actual swept volume associated with the east face of a control volume (due to grid deformation) together with the apparent swept volume resulting from an arbitrarily prescribed disturbance profile. An extension of the ALE approach to include the prescribed velocity field is therefore realized by computing the grid fluxes based on the accumulated volume swept by each face due to the actual deformation and the apparent volume swept due to the field velocity components [45] (see δV_e and δV_e^g in Fig. 2.4). It is worth mentioning that at time step t^{n+1} the control volume itself is updated using only the volume swept by the actual deformation (i.e., the red CV in the right sketch of Fig. 2.4). All geometric quantities, and hence the fluxes, are evaluated on this updated volume, whereas the volume used for the temporal integration of the momentum equations includes both contributions (i.e., the blue CV in the right sketch of Fig. 2.4).

While the FVM accounts for the influence of the gust on the structure, its major drawback lies in its inability to capture the effect of the surrounding (background) flow

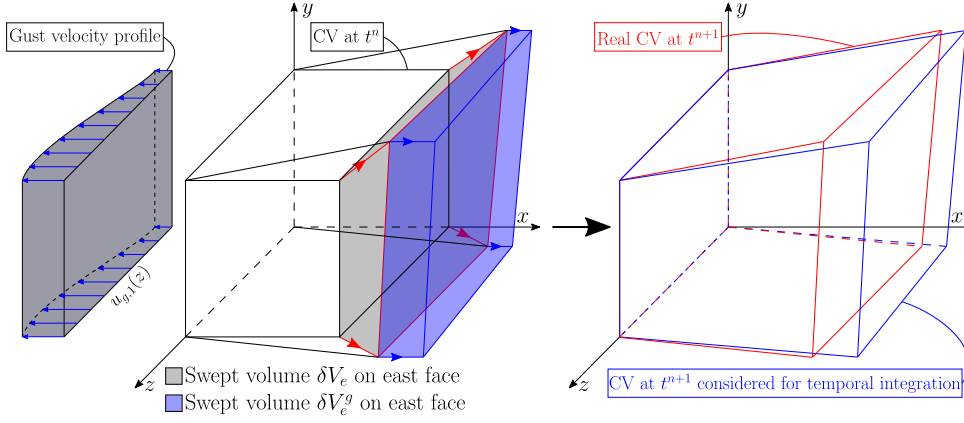


Fig. 2.4. Illustration of the actual swept volume δV_e and the apparent swept volume δV_e^g on the east face of a control volume, resulting from mesh deformation between two consecutive time steps and from the field velocity method, respectively.

on the gust itself (commonly referred to as the *feedback effect*). In addition, unlike in the case of solving the Euler equations, the solution of the Navier-Stokes equations with a non-uniform gust velocity distribution requires a correction of the viscous fluxes. This is because the stress tensor in Eq. (2.12) is based on the background velocity field $\tilde{u}_i$, and the spatial gradients of the prescribed field $u_{g,i}$ are therefore not automatically included. Such a correction is indispensable whenever the gradient of the gust velocity is non-zero [60], and its magnitude depends on how steep the gust velocity profile is. The drawbacks of the original FVM are not present when using the split velocity method discussed next.

2.1.2 Split Velocity Method (SVM)

Similar to the field velocity method, the split velocity method, originally proposed by Wales et al. [19], was also introduced in the field of aeronautics. As discussed in Section 2.1.1, the FVM models the effect of an external disturbance (e.g., a gust) on the structure by considering an equivalent case where the structure moves with the same velocity as the gust but in the opposite direction. This analogy allows the FVM to solve for the gust response by prescribing the grid velocities in a moving-mesh solver to be equal to the negative gust velocity, without physically moving the grid. In contrast, the SVM begins by expressing the governing equations in terms of the total velocity u_i , which is then decomposed into two components: A background velocity $\tilde{u}_i$ and a prescribed gust velocity $u_{g,i}$. The governing equations are then rearranged so that the solution is obtained directly for the background velocity field $\tilde{u}_i$ without requiring simplifying assumptions.

The formulation starts by substituting the decomposed total velocity (i.e., $u_i = \tilde{u}_i + u_{g,i}$) into the integral form of the Navier-Stokes equations on a fixed grid:

$$\int_V \frac{\partial \rho}{\partial t} dV + \int_S \rho (\tilde{u}_i + u_{g,i}) n_i dS = 0, \quad (2.17)$$

$$\begin{aligned} \int_V \frac{\partial (\rho (\tilde{u}_i + u_{g,i}))}{\partial t} dV + \int_S \rho (\tilde{u}_i + u_{g,i}) (\tilde{u}_j + u_{g,j}) n_j dS = & - \int_S p n_i dS \\ & - \int_S \tau_{ij} n_j dS. \end{aligned} \quad (2.18)$$

In contrast to the FVM, τ_{ij} is the stress tensor based on the total velocity u_i . The transient and the convective terms of the momentum equation are expanded using the product rule as follows:

1. Transient term:

$$\begin{aligned} \int_V \frac{\partial (\rho (\tilde{u}_i + u_{g,i}))}{\partial t} dV &= \int_V \frac{\partial (\rho \tilde{u}_i)}{\partial t} dV + \int_V \frac{\partial (\rho u_{g,i})}{\partial t} dV \\ &= \int_V \frac{\partial (\rho \tilde{u}_i)}{\partial t} dV + \int_V \rho \frac{\partial u_{g,i}}{\partial t} dV + \\ &\quad \underbrace{\int_V u_{g,i} \frac{\partial \rho}{\partial t} dV}_* \end{aligned} \quad (2.19)$$

2. Convective term:

$$\begin{aligned} \int_S \rho (\tilde{u}_i + u_{g,i}) (\tilde{u}_j + u_{g,j}) \cdot n_j dS &= \int_V \frac{\partial}{\partial x_j} (\rho (\tilde{u}_i + u_{g,i}) (\tilde{u}_j + u_{g,j})) dV \\ &= \int_V \frac{\partial}{\partial x_j} (\rho \tilde{u}_i (\tilde{u}_j + u_{g,j})) dV + \\ &\quad \int_V u_{g,i} \frac{\partial}{\partial x_j} (\rho (\tilde{u}_j + u_{g,j})) dV + \\ &\quad \int_V \rho (\tilde{u}_j + u_{g,j}) \frac{\partial u_{g,i}}{\partial x_j} dV \\ &= \int_S \rho \tilde{u}_i (\tilde{u}_j + u_{g,j}) \cdot n_j dS + \\ &\quad \underbrace{\int_V u_{g,i} \frac{\partial}{\partial x_j} (\rho (\tilde{u}_j + u_{g,j})) dV}_* \\ &\quad \int_V \rho (\tilde{u}_j + u_{g,j}) \frac{\partial u_{g,i}}{\partial x_j} dV. \end{aligned} \quad (2.20)$$

The summation of the terms marked by * represents the product of the gust velocity $u_{g,i}$ with the mass conservation equation (2.17):

$$\int_V u_{g,i} \frac{\partial \rho}{\partial t} dV + \int_V u_{g,i} \frac{\partial}{\partial x_j} (\rho u_j) dV = \int_V u_{g,i} \left(\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_j)}{\partial x_j} \right) dV = 0 . \quad (2.21)$$

The remaining terms yield the governing equations of the split velocity method, in which the gust-related source terms appear explicitly:

$$\begin{aligned} \int_V \frac{\partial (\rho \tilde{u}_i)}{\partial t} dV + \int_S \rho \tilde{u}_i (\tilde{u}_j + u_{g,j}) n_j dS = & - \int_S p n_i dS - \int_S \tau_{ij} n_j dS \\ & - \underbrace{\int_V \rho \frac{\partial u_{g,i}}{\partial t} dV + \int_V \rho (\tilde{u}_j + u_{g,j}) \frac{\partial u_{g,i}}{\partial x_j} dV}_{-S_{\text{momentum},i}} . \end{aligned} \quad (2.22)$$

The decomposition of the velocity field introduces a source term that captures the convective influence of both the impacted object and the surrounding flow on the gust itself (i.e., the *feedback effect*). The source term can be written in its general form as:

$$S_{\text{momentum},i} = \int_V \rho \left\{ \frac{\partial u_{g,i}}{\partial t} + (\tilde{u}_j + u_{g,j}) \frac{\partial u_{g,i}}{\partial x_j} \right\} dV . \quad (2.23)$$

In the framework of fixed grids, the partial derivative in time within the transient terms of the mass and momentum balances can be replaced by a total time derivative applied outside the integral. This yields the final governing equations solved for the SVM:

$$\frac{d}{dt} \int_V \rho dV + \int_S \rho (\tilde{u}_i + u_{g,i}) n_i dS = 0 , \quad (2.24)$$

$$\begin{aligned} \frac{d}{dt} \int_V \rho \tilde{u}_i dV + \int_S \rho \tilde{u}_i (\tilde{u}_j + u_{g,j}) \cdot n_j dS = & - \int_S p n_i dS - \int_S \tau_{ij} n_j dS \\ & - S_{\text{momentum},i} . \end{aligned} \quad (2.25)$$

Since, in general, the prescribed gust velocity is not necessarily divergence-free, the postulate $\partial \tilde{u}_i / \partial x_i = 0$ may not be satisfied, and special care is required to maintain incompressibility. A direct comparison of the mass conservation equation (2.11) for the FVM with the corresponding equation (2.24) for the SVM reveals that both share the same form. The same holds for the convective term, indicating that both formulations require the computation of grid fluxes arising from the pseudo grid motion. These similarities indicate that, as in the ALE-analog FVM, grid fluxes can be computed and the incompressibility condition for $\tilde{u}_i$ can be enforced by applying the pseudo GCL (i.e., Eq. (2.13)).

Within the momentum equations, the differences between both approaches are confined to the additional source terms in Eq. (2.25) and the viscous flux term. In comparison to the field velocity method, the evaluation of the additional source terms constitutes

an important aspect of the split velocity method, as they enable capturing the effect of both the impacted object and the surrounding flow on the gust itself, i.e., the *feedback effect*. Furthermore, in contrast to the FVM, the SVM uses the total velocity field for the computation of viscous fluxes, making it suitable for problems where viscous effects are not negligible. As a result, the major drawbacks of the FVM are not present in the SVM, although the implementation of the latter requires more code modifications.

2.2 Main Results of Publication I: Gusts on a Fixed Flat Plate with Rounded Edges

The present test case, originally defined in [47, 48, 49] and comprehensively examined in Publication I, evaluates the influence of gusts prescribed via the FVM or SVM on the aerodynamic forces acting on a stationary rigid body. To enable direct comparison with selected results from these earlier studies, the flat plate geometry, Reynolds number, and vertical gust characteristics are retained. However, due to the use of different numerical approaches (structured overset grids versus curvilinear, block-structured body-fitted grids), both the computational domain and the numerical mesh are adapted accordingly, as detailed in Section 5 of Publication I. In the following, a brief overview of the characteristics of each gust configuration is provided, and only the main results for each case are highlighted. A comprehensive presentation of all results and a detailed discussion can be found in Section 6 of Publication I.

2.2.1 Vertical Gusts

The first configuration focuses on the modeling of gusts oriented perpendicular to the streamwise direction of the main flow as shown in Fig. 2.5. For this reason, the term *vertical gust* is employed. While wind gusts are inherently three-dimensional in nature, the methodology outlined in Appendix A allows for the prescription of such complex structures. Nonetheless, for the purpose of fundamental investigations, simplified one-dimensional canonical gusts are more appropriate and are therefore considered. Based on the definition of the gust introduced in Appendix A, gust characteristics of the simulations performed are summarized in Table 2.1. In this configuration, the vertical gust convects in the streamwise direction with $u_{g,\text{conv}} = u_\infty$. This results in $\eta_g = \eta_g^0 + (t - t_0) u_{g,\text{conv}}$, where η denotes the spatial coordinate along the basis vector $\mathbf{g}_2$. At $t = 0$, the gust is centered at the user-defined coordinate η_g^0 . Since various gust wavelengths L_g^η along the $\mathbf{g}_2$ -direction are investigated, η_g^0 is chosen such that the front of the widest gust ($L_g^\eta = 4c$) starts shortly upstream of the plate's leading edge. With the origin placed at the leading edge, this corresponds to $\eta_g^0 = -2.1c$.

2.2.1.1 Temporal Evolution of Flow Fields for $L_g^\eta = 2c$

The analysis begins with a visualization of the flow evolution (captured at four characteristic time instants) during the interaction of an intermediate-length vertical gust of $2c$

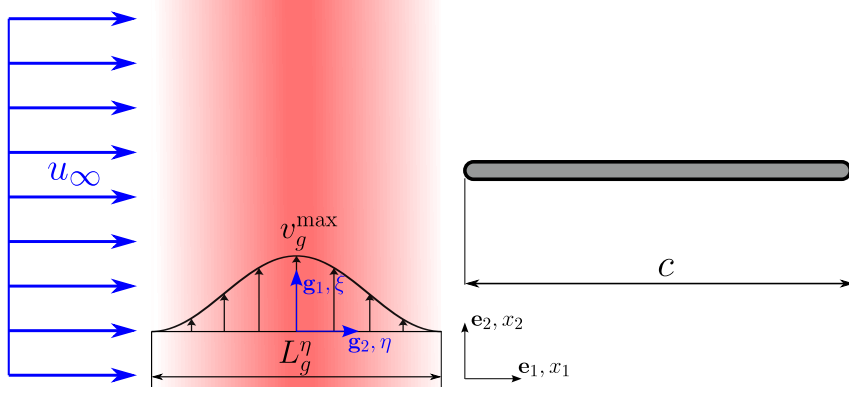


Fig. 2.5. Schematic representation of the vertical gust configuration.

Tab. 2.1. Vertical gust characteristics for the fixed flat plate with rounded edges.

$\mathbf{g}_1$	A_g	$f_t(t) = 1$	$f_1(\xi) = 1$	$f_2(\eta)=\text{ECG}$					$f_3(\zeta) = 1$	
				η_g^0	L_g^η					
$\mathbf{e}_2$	0.8	-	-	$-2.1\,c$	$0.5\,c$	$1\,c$	$1.5\,c$	$2\,c$	$4\,c$	-

with the flat plate. Figure 2.6 presents contours of the total vertical velocity component u_2/u_∞ and the pressure $p/(\rho u_\infty^2)$ fields. As the gust convects over the plate, a series of clockwise-rotating leading-edge vortices (LEVs) form in response to the upward gust impingement. These vortical structures are accompanied by local pressure minima and convect downstream as the gust progresses. While both the FVM and the SVM capture the overall flow behavior consistently, subtle differences emerge. The most pronounced discrepancies appear after the gust has passed over the plate, particularly in its wake. In this region, the shape of the gust becomes less discernible in the total vertical velocity field predicted by SVM. This suggests a stronger interaction between the gust and the wake, leading to a more pronounced attenuation of the gust in the SVM results. Despite this, the overall differences remain modest, particularly with respect to the pressure field. This close agreement is attributed to the nearly vanishing source term in the SVM formulation for the vertical gust configuration as will be shown next.

In the case of a vertical gust, the gust velocity is directed solely along the y -axis. As a result, the source term vector in the momentum equations reduces to $\mathbf{S} = [0, S_2(u_{g,2}(x_1, t)), 0]$. Using Eqs. (2.23) and (A.8), the resulting simplified source term for the y -momentum equation reads (see Appendix A in Publication I for the detailed derivation):

$$S_2(x_1, t) = \rho \left\{ \frac{\partial u_{g,2}}{\partial t} + \tilde{u}_1 \frac{\partial u_{g,2}}{\partial x_1} \right\} = \rho \frac{\pi A_g}{L_g^{x_1}} \sin \left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}} \right) \left\{ \overbrace{u_{g,\text{conv}}}^{u_\infty} - \tilde{u}_1 \right\}. \quad (2.26)$$

It is noteworthy that the source term vanishes when the background velocity $\tilde{u}_1$ matches the free-stream velocity u_∞ . This condition holds in the far-field region, where the influence of the body is negligible and $\tilde{u}_1 \approx u_\infty$. In this region, the source term plays no role and has no effect on the pressure field. This behavior is physically expected as, for instance, one

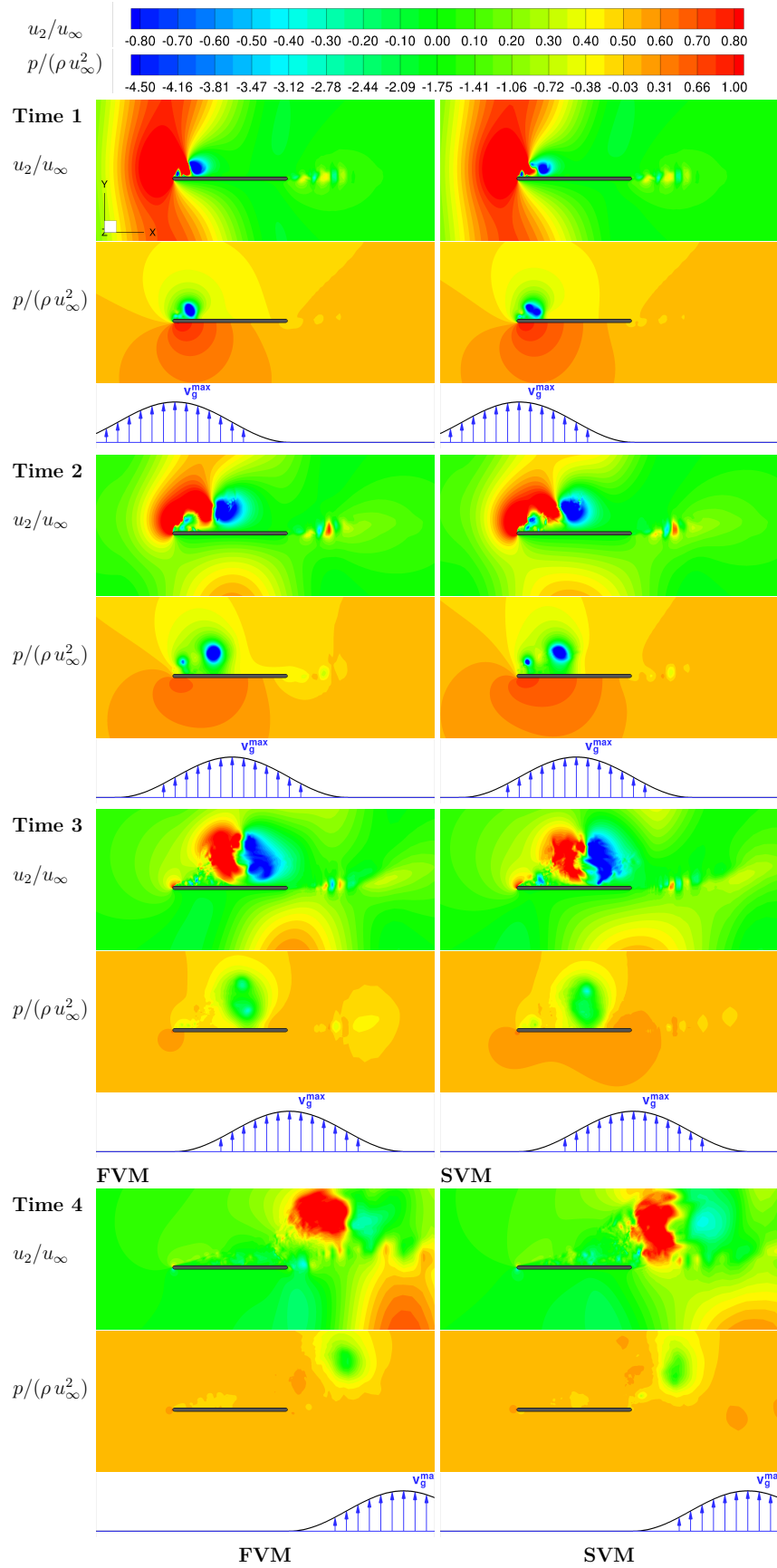


Fig. 2.6. Vertical gust passing around the fixed rounded flat plate: Flow field predictions from the FVM (left) and SVM (right) at time instants 1, 2, 3, and 4. The gust has a length of $L_g^\eta = 2c$.

would anticipate no interaction between a uniform horizontal parallel flow and a vertical jet traveling downstream at the same horizontal velocity. Conversely, in regions influenced by the body, including the area above, below, and within the wake, the background velocity deviates from u_∞ , causing the source term to remain active. This continued contribution, which models the effect of the non-uniform background flow on the gust, is responsible for the discrepancies observed between the FVM and the SVM results during the post-gust interaction phase. As will be shown in Section 2.2.2, the source term remains active in the case of a horizontal gust, including the far-field region. This widespread contribution leads to significant differences between the FVM and the SVM predictions for a horizontal gust.

2.2.1.2 Effect of Gust Length on Force Coefficients

The analysis is extended to the full range of gust lengths, focusing on the resulting time histories of the aerodynamic force coefficients. Figure 2.7 compares the aerodynamic response of the plate to gust encounters as predicted by the FVM and SVM over the convective time t^* ³. At the dimensionless time $t^* = 0$ the front of the gust reaches the leading edge of the plate. Overall, only minor deviations are observed between the two methods. These differences are particularly visible in the lift coefficient for gusts with short wavelengths (e.g., $0.5c$, $1c$, and $1.5c$), where the FVM predicts a secondary local peak occurring after the gust has completely passed over the plate.

The results reveal three key effects of the gust width on the aerodynamic response predicted by the FVM and SVM. First, shorter gusts cause a more rapid increase in aerodynamic forces due to steeper velocity gradients, which lead to higher rates of change in the induced forces. Second, the peak lift coefficient increases nonlinearly with the gust length, as wider gusts sustain a higher effective angle of attack for a longer duration. Third, the peak lift occurs later for wider gusts, as the gust center takes more time to reach the critical region between the leading edge and mid-chord of the plate.

2.2.2 Horizontal Gusts

The second configuration models gusts (also using an ECG shape) aligned with the streamwise flow direction as shown in Fig. 2.8. Hence, the term *horizontal gust* is used. Similar to the vertical gust configuration discussed earlier, a range of gust wavelengths L_g^ξ oriented along the $\mathbf{g}_1$ -direction is investigated. The corresponding parameters and cases are summarized in Table 2.2. As in the vertical gust case, the displacement of the gust in the streamwise direction is governed by the convective velocity $u_{g,\text{conv}} = u_\infty$. This gives $\xi_g = \xi_g^0 + (t - t_0) u_{g,\text{conv}}$, where ξ denotes the spatial coordinate along the basis vector $\mathbf{g}_1$. The initial position ξ_g^0 is also set such that the front of the widest gust ($L_g^\xi = 4c$) is located shortly upstream of the plate's leading edge.

³ $t^* = u_\infty t / c$ is the convective time defined in terms of the number of chord lengths the gust has traveled.

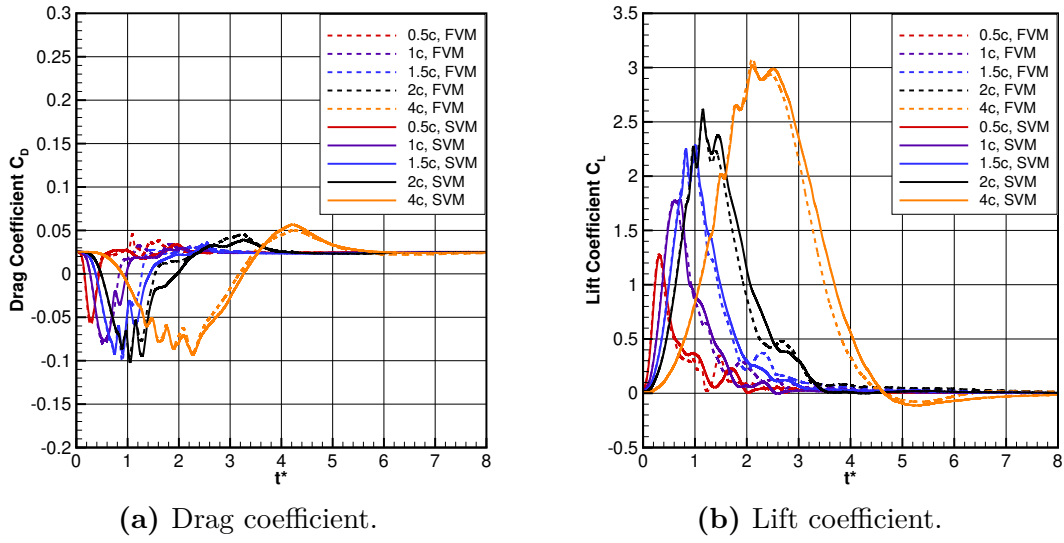


Fig. 2.7. Vertical gust passing around the fixed rounded flat plate: Instantaneous drag and lift coefficients. The gust shape is ECG with a gust amplitude of $A_g = 0.8$: FVM vs. SVM.

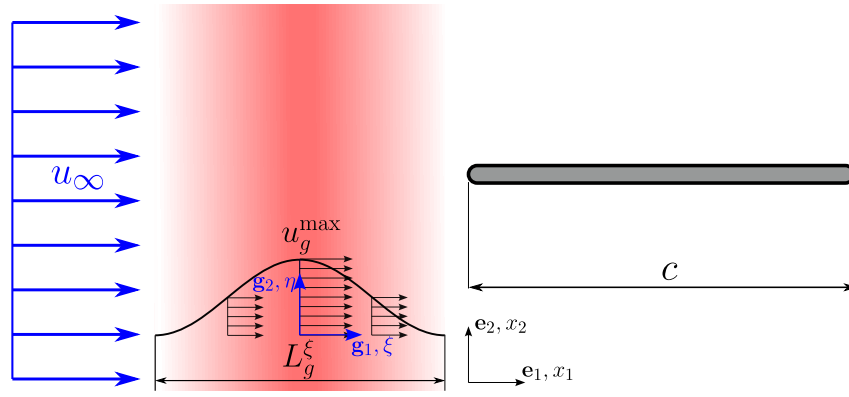


Fig. 2.8. Schematic representation of the horizontal gust configuration.

Tab. 2.2. Horizontal gust characteristics for the fixed flat plate with rounded edges.

$\mathbf{g}_1$	A_g	$f_t(t) = 1$	$f_1(\xi)=\text{ECG}$						$f_2(\eta)=1$	$f_3(\zeta) = 1$
			ξ_g^0	L_g^ξ						
$\mathbf{e}_1$	0.8	-	$-2.1\,c$	$0.5\,c$	$1\,c$	$1.5\,c$	$2\,c$	$4\,c$	-	-

2.2.2.1 Temporal Evolution of Flow Fields for $L_g^\xi = 2c$

Initially, the flow development during a horizontal gust of an intermediate length ($2c$) is examined at the same four characteristic time instants as in the vertical gust case. Figure 2.9 depicts the streamwise background velocity $\tilde{u}_1/u_\infty$, the streamwise total velocity u_1/u_∞ , and the pressure $p/(\rho u_\infty^2)$ during the gust encounter. While both the FVM and SVM initially capture similar streamwise velocities, clear discrepancies arise in the pressure

field. The SVM simulation exhibits a broad low-pressure region that follows the gust's center, attributed to its continuously active source term, whereas the FVM shows only localized pressure drops near the plate. At Time 2, the gust progresses and the background velocity field predicted by SVM begins to show upstream deceleration and downstream acceleration, features that are absent in the FVM result. A comparison with a modified SVM simulation (using the background velocity for the evaluation of diffusive terms like in the FVM) confirms that these deviations stem from the source term rather than from the treatment of diffusive fluxes. In the later stages, vortex shedding becomes more pronounced in the FVM simulation, with small-scale vortices forming at the gust's tail. These structures are absent in SVM, which maintains a smoother flow field due to the stabilizing effect of its source term. After the gust has left the plate, the FVM simulation continues to show pressure-induced unphysical vortical structures near the plate, which are remnants of earlier pressure disturbances. In contrast, the SVM solution returns to the quasi-steady state. These observations highlight the role of the non-vanishing source term in SVM, which consistently governs the interaction between the gust and the background flow across the domain. Unlike in the vertical gust configuration, the SVM formulation yields a widespread active source term in the horizontal gust case, as will be shown next.

For a horizontal gust, the gust velocity is directed solely along the x -axis and the source terms in the momentum equations reduce to $\mathbf{S} = [S_1(u_{g,1}(x_1, t)), 0, 0]$. Using Eqs. (2.23) and (A.8), the simplified source term in the x -momentum equation becomes (see Appendix B in Publication I for detailed derivation):

$$\begin{aligned} S_1(x_1, t) &= \rho \left\{ \frac{\partial u_{g,1}}{\partial t} + (\tilde{u}_1 + u_{g,1}) \frac{\partial u_{g,1}}{\partial x_1} \right\} \\ &= \rho \frac{\pi A_g}{L_g^{x_1}} \sin \left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}} \right) \left\{ \overbrace{u_{g,\text{conv}}}^{u_\infty} - (\tilde{u}_1 + u_{g,1}) \right\}. \end{aligned} \quad (2.27)$$

In contrast to the vertical gust case, the spatial derivative in this configuration is multiplied by the total velocity $u_1 = \tilde{u}_1 + u_{g,1}$, which differs from u_∞ even in the far-field region. Consequently, the role of the source term (2.27) becomes evident already during the early stages of the gust injection through its strong influence on the pressure field. Therefore, significant deviations between the FVM and the SVM results are clearly observed in the horizontal gust configuration.

An additional remark on Eq. (2.27) (and equivalently on Eq. (2.26)) concerns the dependence of the source term on the gust's convective velocity. This dependence is not problematic in aeronautical applications, where $u_{g,\text{conv}}$ is naturally set equal to the free-stream velocity u_∞ . In that case, the problem is equivalent to forcing a moving body (e.g., a wing) through a stationary jet, as demonstrated in the experiments of Biler et al. [47]. In civil engineering applications, however, this dependence raises an important question for the use of the SVM in gust generation: What convective velocity should be specified to obtain a physically consistent total velocity field? In practice, wind gusts in the atmospheric boundary layer are convected naturally and no explicit convective velocity is defined. In view of the FVM's inability to capture the feedback effect, prescribed-velocity

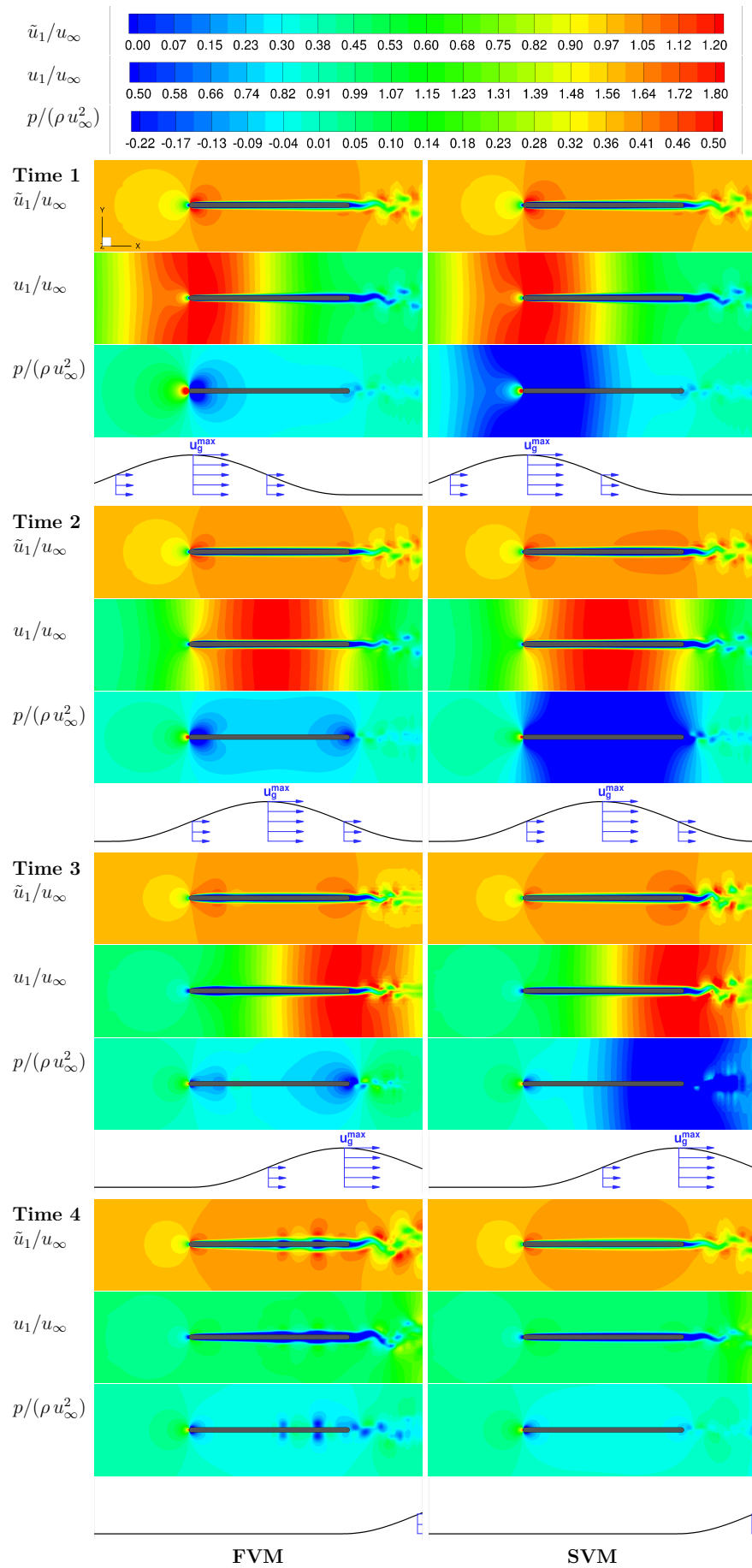


Fig. 2.9. *Horizontal gust passing around the fixed rounded flat plate:* Flow field predictions from FVM (left) and SVM (right) at time instants 1, 2, 3, and 4. The gust has a length of $L_g^\xi = 2c$.

methods appear to be not the most suitable choice for such applications. This limitation motivated the development of the source-term formulation presented in Chapter 3.

2.2.2.2 Time Histories of the Drag Coefficient

Similar to the vertical gust analysis, the investigation is extended to evaluate the aerodynamic force coefficients. Here, results are shown only for the gust length of $2c$, as similar observations can be made for the other gust lengths. The corresponding results for those cases are provided in Section 6.2.2 of Publication I.

Figure 2.10 presents the time histories of the drag coefficient. The pressure and shear contributions (i.e., C_D^p and C_D^s) are also shown to provide a deeper insight into its behavior. In the FVM, the total drag primarily follows the evolution of its shear component. In contrast, SVM results are dominated by the pressure contribution C_D^p , consistent with the pressure field differences induced by the active source term. As shown in Fig. 2.9, the pressure drop shifts from the leading edge to the trailing edge as the gust convects over the plate. This explains the reduction followed by a peak in the SVM drag response. The shear contribution in SVM closely follows the gust shape, whereas in the FVM it shows an unphysical drop below the quasi-steady value for $t^* > 1.5$. The modified SVM simulation (in which the background velocity is used for evaluating the shear stress) confirms that the observed behavior is governed by the source term rather than by the treatment of diffusive fluxes. This finding is further supported by a comparison of magnitudes: the diffusive contribution associated with the gust velocity, $\tau_{ij}(u_{g,i})$, depends on the spatial gradient of the gust profile and the inverse Reynolds number. In the present case, the product of these two factors yields a contribution that is significantly smaller than that of the source term. Although of lesser importance in this case, the lift coefficient (not shown here) remains closely linked to the von Kármán vortex shedding at the trailing edge. Assuming the Strouhal number remains approximately constant during the gust passage, the observed increase in shedding frequency can be attributed to the rise in local streamwise velocity induced by the horizontal gust.

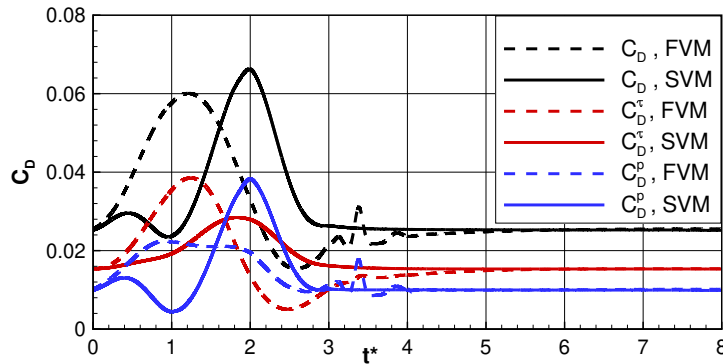


Fig. 2.10. *Horizontal gust passing around the fixed rounded flat plate:* Time history of the drag coefficient C_D , of the pressure part of the drag coefficient C_D^p and of the shear part of the drag coefficient C_D^s for FVM and SVM. The gust has the length of $L_g^x = 2c$.

2.3 Main Results of Publication II: Vertical Gusts on an Elastically Mounted Airfoil

This test case, presented in Publication II, focuses on the numerical investigation of vertical wind gust effects on airfoils within a parameter range relevant to micro-air vehicles. Instead of an elastic wing, a simplified model is used, consisting of a rigid airfoil that is elastically mounted with two degrees of freedom (heave and pitch) as depicted in Fig. 2.11. The fluid–structure interaction problem was previously addressed in De Nayer et al. [61] using a partitioned coupling approach that combines large-eddy simulation (LES) for the flow field with a rigid-body dynamics solver. However, that setup did not incorporate any gust effects. Publication II extends this framework by introducing deterministic 1-cosine vertical gusts of varying lengths and intensities (see Table 2.3). These gusts are modeled using the SVM within an ALE formulation that accommodates temporally deforming control volumes. A detailed description of the FSI case, including the computational domain, numerical grid, and all relevant structural parameters, is provided in Section 4 of Publication II. In the following, the main results (thoroughly presented in Section 5 of the same publication) are summarized, focusing on the aeroelastic response of a NACA 0012 airfoil at a transitional Reynolds number when subjected to gusts of varying lengths and intensities.

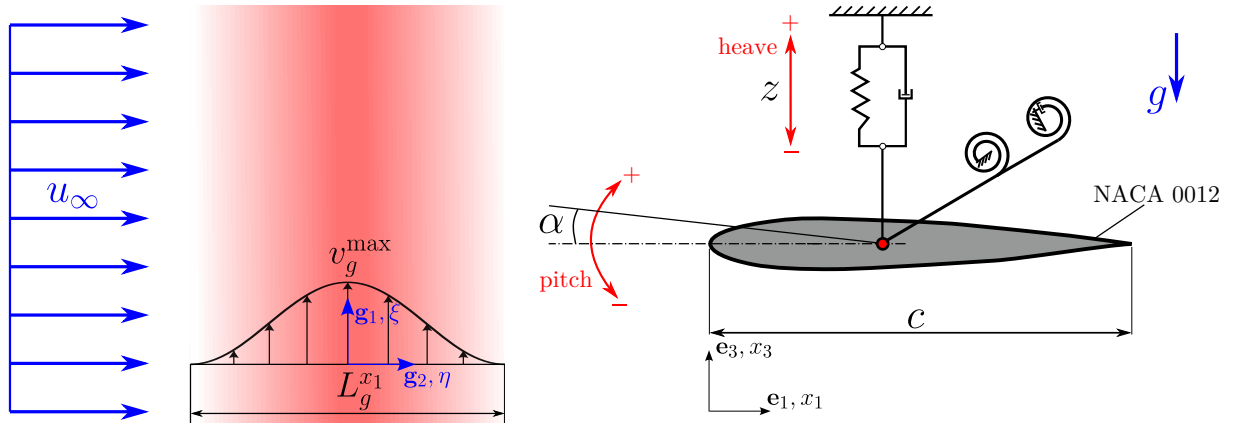


Fig. 2.11. Schematic representation of the investigated FSI case and the introduced vertical gust velocity profile.

Tab. 2.3. Vertical gust characteristics for the elastically mounted airfoil.

$\mathbf{g}_1$	$f_t(t)$	$f_1(\xi)$	$f_2(\eta)$	$f_3(\zeta)$	$L_g^{x_1}/c$	$x_{g,1}^0/c$	A_g/u_∞
$\mathbf{e}_3$	1	1	ECG	1	2	-1	1 2 3
$\mathbf{e}_3$	1	1	ECG	1	4	-2	1 2 3
$\mathbf{e}_3$	1	1	ECG	1	6	-3	1 2 3

2.3.1 Comparison of CFD and FSI Cases

As an initial step, the aerodynamic forces acting on the airfoil are compared for two cases, namely, a fixed and an elastically mounted rigid airfoil. Figure 2.12 depicts the instantaneous lift and pitching moment coefficients for both cases as the airfoil encounters a 1-cosine vertical gust characterized by $L_g^{x_1}/c = 2$ and $A_g/u_\infty = 2$. During the gust interaction phase (i.e., until $t^* = t u_\infty/c = 3$), the force histories are identical for both cases. However, beyond this time instant, the FSI case exhibits sustained oscillations around zero, whereas the rigid CFD counterpart returns to steady-state values. These oscillatory aerodynamic forces play a dual role, contributing to both damping and amplification of the airfoil's motion, as further explored in subsequent analyses.

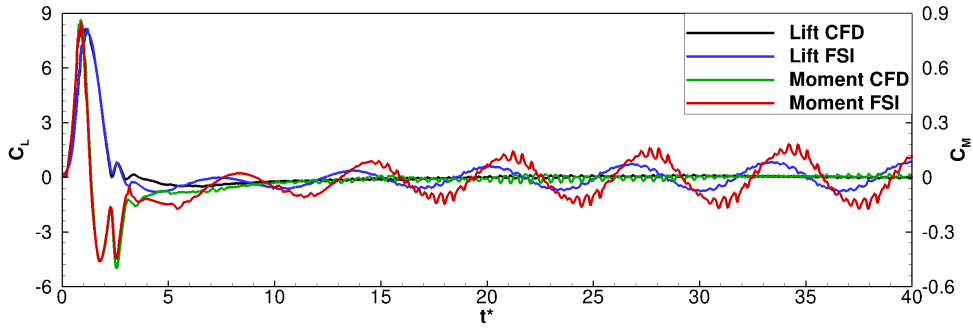


Fig. 2.12. Instantaneous lift and pitching moment coefficients of the fixed and elastically mounted airfoils subjected to a vertical gust with $L_g^{x_1}/c = 2$ and $A_g/u_\infty = 2$.

Figure 2.13 depicts four instants in time capturing the development and convection of flow structures during the airfoil-gust encounter in the elastically mounted case. Leading-edge vortices are formed and transported downstream as the airfoil undergoes heave and pitch motions in response to the imposed aerodynamic loads. It was also observed that, throughout the gust interaction phase, the flow fields of the fixed configuration (shown in Fig. 5 of Publication II) and its elastic counterpart remain nearly indistinguishable, consistent with the close agreement observed in their aerodynamic force histories.

2.3.2 Gust-Induced Aeroelastic Behavior

The aeroelastic response is found to be strongly dependent on the parameters of the imposed gust. Figure 2.14 shows the time histories of the heave and pitch motions of an airfoil subjected to a vertical gust with a wavelength of $L_g^{x_1}/c = 4$. For gust strengths of $A_g/u_\infty = 1$ and 2, both degrees of freedom exhibit an initial increase in amplitude immediately following the gust encounter. This increase is followed by a gradual decay, attributed to the overall positive damping effect of the aerodynamic forces. Eventually, the heave and pitch motions converge to a state characterized by sustained, bounded oscillations, indicative of a limit-cycle oscillation (LCO) regime. A similar dynamic behavior is also observed for gusts with shorter and longer wavelengths, specifically for $(L_g^{x_1}/c = 2, A_g/u_\infty = \{1, 2, 3\})$ and $(L_g^{x_1}/c = 6, A_g/u_\infty = \{1, 2\})$. In contrast, gusts with

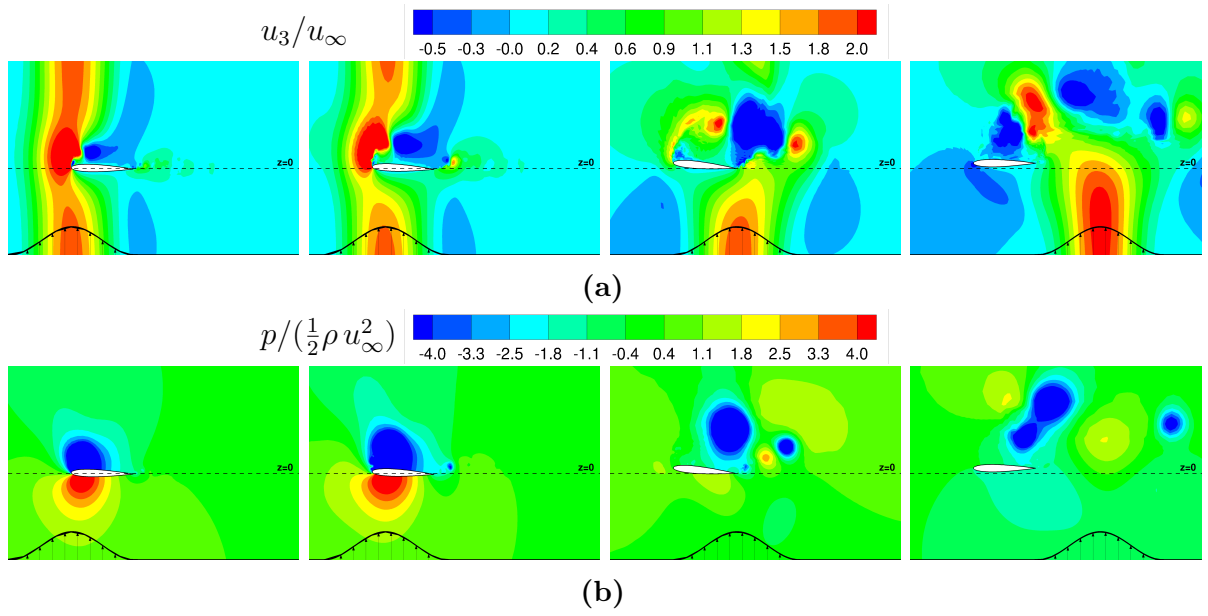


Fig. 2.13. Snapshots of (a) the total vertical velocity and (b) the pressure fields during the gust encounter ($L_g^{x_1}/c = 2$, $A_g/u_\infty = 2$) on the elastically mounted airfoil. From left to right: Gust center at leading edge, gust center when lift is maximum, gust center at trailing edge, gust completely left the airfoil.

$A_g/u_\infty = 3$ and a wavelength of $L_g^{x_1}/c \geq 4$ induce significantly larger excitations, which trigger a pitch-dominated diverging motion of the airfoil, commonly referred to as stall flutter (see blue curve in Fig. 2.14).

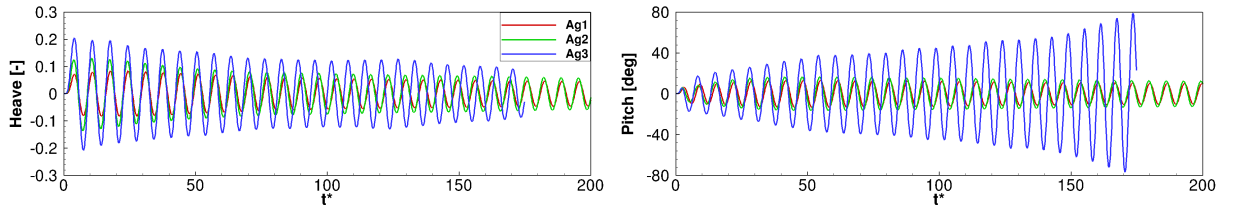


Fig. 2.14. Instantaneous heave and pitch DOFs when imposing a vertical gust with $L_g^{x_1}/c = 4$ and three different gust strength $A_g/u_\infty = 1, 2$ and 3.

2.3.3 Vorticity and Separation Dynamics during an LCO

To gain deeper insight into the flow physics governing a single cycle of a limit-cycle oscillation, snapshots of the spanwise vorticity ω_y , the skin-friction coefficient C_f , and the corresponding flow separation locations are depicted in Fig. 2.15. As the airfoil starts to pitch upward and approaches its maximum angle of attack (see Figs. 2.15a to 2.15d), flow separation, initially located near the trailing edge, progressively moves upstream along the suction side of the airfoil. In the subsequent phase, as the angle of attack starts to decrease, distinct vortices are shed from the suction surface (see Figs. 2.15f and 2.15g).

These vortices dissipate as the angle of attack approaches the opposite extremum (see Fig. 2.15h), leading to the development of a favorable pressure gradient on the upper surface. At this instant, a similar sequence of events begins on the lower surface of the airfoil. The observed laminar separation remains predominantly two-dimensional, and the transition to turbulence is identified further downstream. This transition is indicated by the spanwise variation of the separation line visible in Fig. 2.15h.

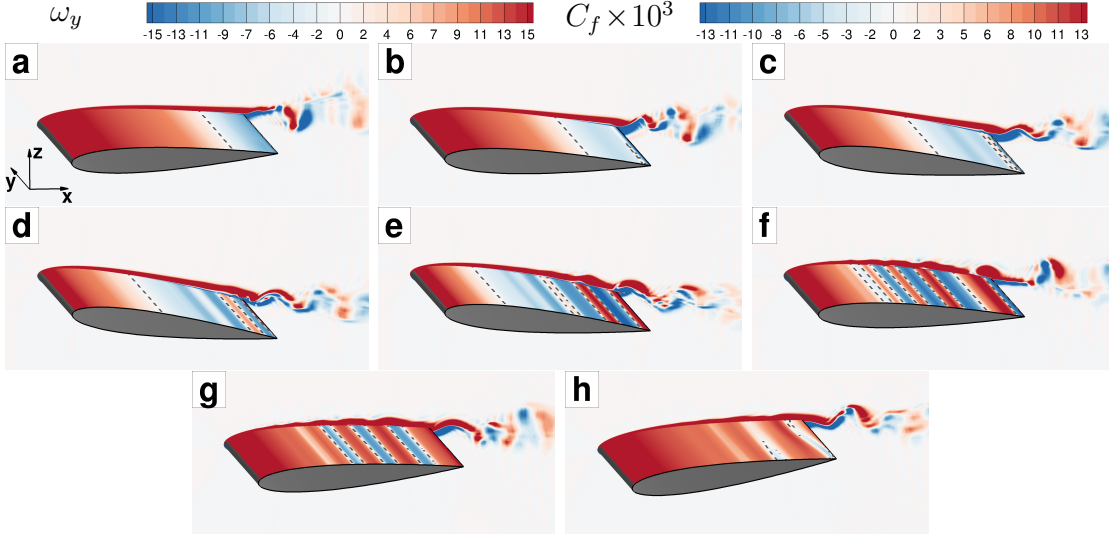


Fig. 2.15. Normalized y -vorticity ω_y (on the mid-span slice), skin-friction coefficient C_f (on the upper surface of the airfoil), and separation location (dashed line) for eight time instants (a-h).

2.3.4 Aerodynamic Hysteresis and Energy Analysis in LCO and Flutter Cases

Figure 2.16 depicts the aerodynamic hysteresis curves of the heave and pitch DOFs after the interaction with two gusts, characterized by $(L_g^{x_1}/c = 2; A_g/u_\infty = 2)$ and $(L_g^{x_1}/c = 4; A_g/u_\infty = 3)$, respectively. The former corresponds to a limit-cycle oscillation, while the latter represents a flutter case. Arrows indicate the direction of advancing time, whereas black and red circles mark the phases, in which energy is extracted from or delivered to the flow. The direction of the loop traversal indicates the sign of the aerodynamic work: clockwise for positive and counterclockwise for negative, following the convention in [62]. The aerodynamic hysteresis loops vary significantly between the two cases. For the LCO case, the clockwise rotation in Fig. 2.16a indicates positive work by the heave motion, while Fig. 2.16b shows that the pitch motion has an overall damping effect. The total energies shown in Fig. 2.17a confirm that the system undergoes a limit-cycle oscillation, as the energy contributions from the heave and pitch degrees of freedom cancel out, resulting in a vanishing total system energy. It is evident that the heave motion serves as the primary contributor to energy extraction in the LCO case, consistent with the behavior observed in the hysteresis curves. In contrast, the diverging motion case

exhibits a continuous increase in the total energy, as shown in Fig. 2.17b. The majority of the energy is extracted by the pitch motion, which continues until the numerical solver can no longer accommodate the severe grid deformation and eventually crashes. The hysteresis curves in Figs. 2.16c and 2.16d do not follow a single turning direction but instead feature a clockwise central loop and counterclockwise outer loops. This behavior reflects the alternating role of the flow in supplying and dissipating energy within a single cycle.

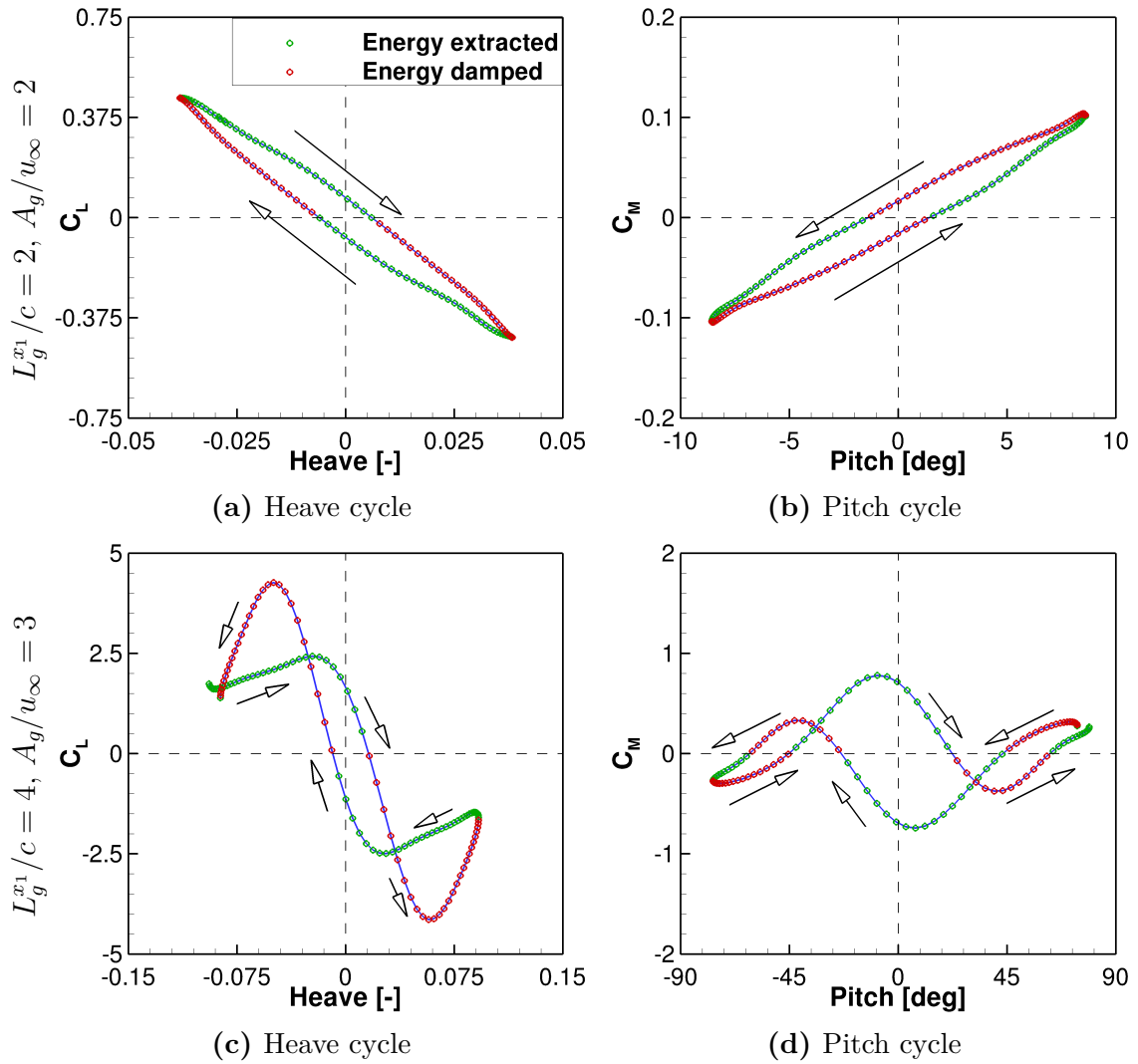


Fig. 2.16. Aerodynamic hysteresis curves within a single cycle for a LCO case (a, b) and a flutter case (c, d).

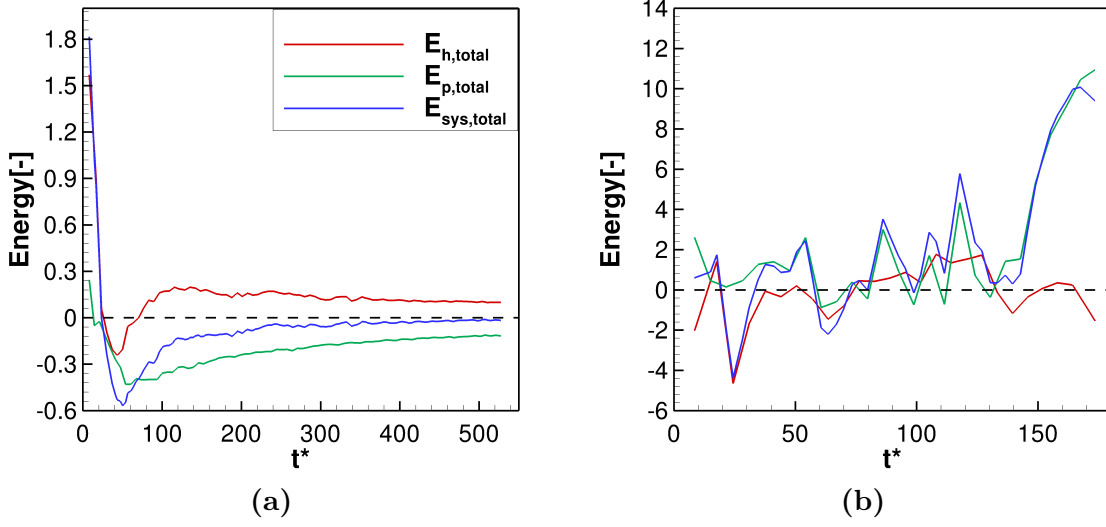


Fig. 2.17. Dimensionless time histories of the instantaneous total energies of both DOFs and the total energy of the system for a LCO case (a) and a flutter case (b).

2.4 Conclusions

This chapter presents a comprehensive summary of Publication I and Publication II, which investigate prescribed velocity wind gust injection methods and assess their aerodynamic and aeroelastic effects through high-fidelity numerical simulations.

In Publication I, the field velocity method and the split velocity method are systematically derived within a finite-volume framework, with particular emphasis on their connection to the Arbitrary Lagrangian Eulerian formulation and the geometric conservation law. The physical behavior of both methods is assessed using vertical and horizontal gusts impinging on a flat plate. In the vertical gust configuration, both the FVM and SVM produce similar results in terms of flow fields and aerodynamic force coefficients, as the source term in SVM becomes negligible in regions where the dimensionless background velocity approaches unity, such as in the far-field. However, in the horizontal gust scenario, pronounced differences emerge. The SVM's explicit treatment of the gust-flow interaction through its additional source term, along with its distinct formulation of the viscous stress tensor, plays a critical role in preventing the formation of unphysical flow structures on the plate surface. In contrast, the FVM lacks this corrective mechanism, leading to unrealistic predictions. These findings highlight the limitations of the FVM when modeling gusts with a significant streamwise component and establish SVM as the more robust and physically consistent method. As such, SVM is recommended as the preferred approach for realistic gust modeling in practical aerodynamic applications.

Building on this foundation, Publication II extends the application of SVM to a fluid-structure interaction setting involving a 2-DOFs elastically mounted NACA 0012 airfoil subjected to vertical gusts at a transitional Reynolds number. Large-eddy simulation and an ALE-based moving mesh strategy are employed to accurately resolve the coupled unsteady flow and structural response. The results show that, although the initial

gust–airfoil interaction produces nearly identical flow fields and aerodynamic forces in both the fixed and the elastically mounted configurations, the post-gust evolution differs significantly. Limit-cycle oscillation cases ultimately converge to a common oscillation amplitude that is independent of the gust parameters, whereas flutter cases exhibit rapid divergence driven by the pitch degree of freedom. An energy-based analysis reveals a clear shift in the dominant energy-contributing mode depending on the dynamic regime. In limit-cycle oscillation cases, the heave motion serves as the primary source of energy, while the pitch motion acts as an energy sink, dissipating energy from the system. In contrast, during flutter, this relationship reverses: the pitch motion becomes the main energy source driving the instability, whereas the heave motion takes on a dissipative role.

Taken together, these two studies demonstrate the necessity of accurate gust modeling for reliable aerodynamic and aeroelastic predictions. The consistent use of the SVM enables the simulation of a broad range of gust configurations and coupling scenarios, making it particularly well suited for future investigations of gust–structure interactions, particularly in aeronautical applications involving flexible wings or micro-air vehicles. Nevertheless, the dependence of the SVM’s source term on a prescribed gust convective velocity limits its applicability in civil-engineering contexts, where no explicit convective velocity is known. Identifying this limitation motivated the source-term formulation presented in the next chapter, which overcomes this constraint and extends the methodology to freely convected wind gusts.

3 Discrete Wind Gust Injection via a Source-term based Formulation

This chapter presents a comprehensive summary of the numerical methodology underlying the source-term based gust injection technique proposed by De Nayer and Breuer [51], along with the main findings from the third publication of this dissertation (Publication III¹). Section 3.1 begins with a detailed derivation of the source-term formulation, highlighting all underlying assumptions. As will be shown in this section, the derived source term includes a spatial derivative of the gust’s prescribed shape function. To ensure a smooth source-term distribution, a modified C^2 “1-cosine” shape based on the original shape introduced in Appendix A is derived and presented in Section 3.2. With these components established, the method is applied in Section 3.3, which presents the main results of Publication III. The injection of a freely traveling gust is demonstrated in a turbulent fluid–structure interaction scenario involving a hemispherical structure immersed in a turbulent boundary layer and subjected to highly transient horizontal wind gusts. Compared to the elastically mounted rigid airfoil case in Publication II (summarized in Section 2.3), the FSI problem now requires a finite-element (FE) solver and highly advanced mapping techniques. The analysis focuses on the flow characteristics prior to the gust impact, the interaction of horizontal gusts of varying intensities with both rigid and flexible hemispheres, and the resulting structural deformations and material response.

3.1 Source-term formulation

The source-term (ST) formulation, as the name suggests, deterministically models discrete wind gusts by introducing a source term on the right-hand side of the momentum conservation equation. In comparison with the prescribed velocity methods discussed in Chapter 2, the idea behind the ST method stems from a physical interpretation, where the gust is perceived as a local, additional volumetric convection of fluid particles. The derivation begins with the integral form of the momentum conservation equation for an incompressible fluid, expressed in vector notation relative to the Cartesian basis $\mathcal{B}_0 = (\mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3)$:

$$\underbrace{\int_V \frac{\partial(\rho \tilde{\mathbf{u}})}{\partial t} dV}_{\text{Variation in time}} + \underbrace{\int_V (\tilde{\mathbf{u}} \cdot \nabla) (\rho \tilde{\mathbf{u}}) dV}_{\text{Convection}} + \underbrace{\int_V \nabla \times [\mu (\nabla \times \tilde{\mathbf{u}})] dV}_{\text{Diffusion}} + \underbrace{\int_V \nabla p dV}_{\text{Pressure force}} = \mathbf{S}_{\text{gust}}^{\text{ST}}, \quad (3.1)$$

where $\tilde{\mathbf{u}}$ denotes the background flow velocity, ρ is the fluid density, p represents the pressure, μ is the dynamic viscosity, and V describes the volume of the computational cell.

¹De Nayer, Breuer & Boulbrachene, *J. Fluids Struct.* **109**, 103462 (2022).

The added *momentum* source term $\mathbf{S}_{\text{gust}}^{\text{ST}}$, whose exact formulation will follow, enables the generation of gusts within the computational domain *without introducing additional mass*. As a result, the gust is purely momentum-driven and the continuity equation remains unchanged, requiring no additional source term.

A local increase in velocity can be physically interpreted as introducing additional convective transport into the flow. Based on this idea, the gust momentum source term is defined to be consistent with the form of the convection term in the momentum conservation equation and is initially formulated in the local basis $\mathcal{B}_1 = (\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$ (described in Appendix A) as:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1} = \int_V (\mathbf{u}|_{\mathcal{B}_1} \cdot \nabla) (\rho \mathbf{u}|_{\mathcal{B}_1}) dV, \quad (3.2)$$

where the total velocity vector $\mathbf{u}$ reads:

$$\mathbf{u} = \tilde{\mathbf{u}} + \mathbf{u}_g.$$

Here $\tilde{\mathbf{u}}$ is the background flow velocity (surrounding the gust) and the primary variable solved for during the simulation. $\mathbf{u}_g$ is the vector containing the velocity profile of the gust without the surrounding fluid. This gust velocity vector is deterministically defined in the local basis $\mathcal{B}_1$ (see Appendix A) based on the analytical functions employed in Eq. (A.4) as:

$$\mathbf{u}_g|_{\mathcal{B}_1}(t, \xi, \eta, \zeta) = u_{g,1}|_{\mathcal{B}_1} \mathbf{g}_1 = A_g f_t(t) f_1(\xi) f_2(\eta) f_3(\zeta) \mathbf{g}_1.$$

The total velocity in the local basis then reads:

$$\mathbf{u}|_{\mathcal{B}_1}(t, \xi, \eta, \zeta) = (\tilde{u}_1|_{\mathcal{B}_1} + u_{g,1}|_{\mathcal{B}_1}) \mathbf{g}_1 + \left(\tilde{u}_2|_{\mathcal{B}_1} + \cancel{u_{g,2}|_{\mathcal{B}_1}} \right)^0 \mathbf{g}_2 + \left(\tilde{u}_3|_{\mathcal{B}_1} + \cancel{u_{g,3}|_{\mathcal{B}_1}} \right)^0 \mathbf{g}_3. \quad (3.3)$$

The ST method relies on two main assumptions:

The first assumption considers the gust to be aligned with the primary flow direction, such that the second and third components of the background velocity (i.e., $\tilde{u}_2|_{\mathcal{B}_1}$ and $\tilde{u}_3|_{\mathcal{B}_1}$ in Eq. (3.3)) are negligible compared to the first component, which comprises both background and gust contributions (i.e., $\tilde{u}_1|_{\mathcal{B}_1} + u_{g,1}|_{\mathcal{B}_1}$). As a result, the total velocity can be expressed as:

$$\mathbf{u}|_{\mathcal{B}_1}(t, \xi, \eta, \zeta) = (\tilde{u}_1|_{\mathcal{B}_1} + u_{g,1}|_{\mathcal{B}_1}) \mathbf{g}_1. \quad (3.4)$$

This reduces Eq. (3.2) to:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1} \approx \int_V u_1|_{\mathcal{B}_1} \frac{\partial}{\partial \xi} (\rho \mathbf{u}|_{\mathcal{B}_1}) dV. \quad (3.5)$$

The second assumption considers the gradient of the background flow velocity along the injection direction (i.e., $\partial \tilde{u}_1|_{\mathcal{B}_1} / \partial \xi$) to be negligible in comparison with that of the prescribed gust velocity (i.e., $\partial u_{g,1}|_{\mathcal{B}_1} / \partial \xi$). This further simplifies Eq. (3.5) to:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1} \approx \int_V u_1|_{\mathcal{B}_1} \frac{\partial}{\partial \xi} (\rho u_{g,1}|_{\mathcal{B}_1} \mathbf{g}_1) dV. \quad (3.6)$$

In the finite-volume discretization employed in this thesis, the midpoint rule is applied to approximate the volume integral:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1} \approx u_1|_{\mathcal{B}_1,cc} \left. \frac{\partial}{\partial \xi} \left(\rho u_{g,1}|_{\mathcal{B}_1,cc} \mathbf{g}_1 \right) \right|_{cc} V . \quad (3.7)$$

Here the subscript cc denotes the cell center. The basis vector $\mathbf{g}_1$ is constant and can be moved out of the derivative. Combining $\mathbf{g}_1$ with $u_1|_{\mathcal{B}_1,cc}$, the velocity $\mathbf{u}|_{\mathcal{B}_1,cc}$ can be reintroduced according to Eq. (3.4):

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1} \approx \underbrace{\left. \frac{\partial}{\partial \xi} \left(\rho u_{g,1}|_{\mathcal{B}_1,cc} \right) \right|_{cc}}_{\text{analog to mass flux}} V \mathbf{u}|_{\mathcal{B}_1,cc} . \quad (3.8)$$

The deduced expression can be interpreted as the product of the mass flow rate based on the gust velocity and the total velocity. It is worth noting that the aforementioned source-term formulation and assumptions follow the derivation carried out in [51]. The same final expression would have been obtained solely from the first assumption if the source term is formulated with the total velocity as the convecting velocity and the gust velocity as the convected velocity:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1} = \int_V \left(\mathbf{u}|_{\mathcal{B}_1} \cdot \nabla \right) \left(\rho \mathbf{u}_g|_{\mathcal{B}_1} \right) dV . \quad (3.9)$$

Upon solely applying the first assumption, the expression simplifies to Eq. (3.6):

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1} \approx \int_V u_1|_{\mathcal{B}_1} \frac{\partial}{\partial \xi} \left(\rho \mathbf{u}_g|_{\mathcal{B}_1} \right) dV = \int_V u_1|_{\mathcal{B}_1} \frac{\partial}{\partial \xi} \left(\rho u_{g,1}|_{\mathcal{B}_1} \mathbf{g}_1 \right) dV . \quad (3.10)$$

By approximating the volume integral and taking the basis vector $\mathbf{g}_1$ outside the derivative, Eq. (3.8) is recovered:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1} \approx \left. \frac{\partial}{\partial \xi} \left(\rho u_{g,1}|_{\mathcal{B}_1,cc} \right) \right|_{cc} V \mathbf{u}|_{\mathcal{B}_1,cc} . \quad (3.11)$$

It is important to note that the first component of the background velocity (i.e., $\tilde{u}_1|_{\mathcal{B}_1}$ contained within $\mathbf{u}|_{\mathcal{B}_1}$) is not directly known with respect to the gust's local basis $\mathcal{B}_1$. In contrast, all background velocity components are available in the Cartesian basis $\mathcal{B}_0$, since $\tilde{\mathbf{u}}$ is the primary variable solved for in Eq. (3.1). Consequently, $\tilde{u}_1|_{\mathcal{B}_1}$ must be evaluated using the transformation matrix $\mathbf{T}_{\mathcal{B}_1 \rightarrow \mathcal{B}_0}$ defined in Eq. (A.2) as:

$$\tilde{u}_1|_{\mathcal{B}_1} = \beta_{11} \tilde{u}_1|_{\mathcal{B}_0} + \beta_{12} \tilde{u}_2|_{\mathcal{B}_0} + \beta_{13} \tilde{u}_3|_{\mathcal{B}_0} . \quad (3.12)$$

The gust source term $\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1}$ can now be expressed in terms of the Cartesian basis $\mathcal{B}_0$. By applying the same transformation procedure as described in Appendix A to the gust velocity component $u_{g,1}|_{\mathcal{B}_1}$, the source term in the Cartesian basis $\mathcal{B}_0$ is obtained as:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_0} \approx \rho A_g f_t(t) f_2(t_\eta(x_{cc}, y_{cc}, z_{cc})) f_3(t_\zeta(x_{cc}, y_{cc}, z_{cc})) \left. \frac{\partial f_1}{\partial \xi} \right|_{cc} V \mathbf{u}|_{\mathcal{B}_{0,cc}} , \quad (3.13)$$

where

$$\mathbf{u}|_{\mathcal{B}_0} = (\beta_{11} \tilde{u}_1|_{\mathcal{B}_0} + \beta_{12} \tilde{u}_2|_{\mathcal{B}_0} + \beta_{13} \tilde{u}_3|_{\mathcal{B}_0}) \mathbf{g}_1 + \mathbf{u}_g|_{\mathcal{B}_0} . \quad (3.14)$$

Here, $\mathbf{u}_g|_{\mathcal{B}_0}$ is given by the formulation in Eq. (A.6).

To remain consistent with the terminology introduced by Pasquetti and Peres [63], who studied the injection of a micro-jet into a fluid domain, it is necessary to distinguish between an expulsion phase and an ingestion phase during gust injection. The gust is generated during the expulsion phase, which corresponds to the period of increasing velocity. In contrast, the ingestion phase disturbs and to a large extent hinders the gust formation. Considering the shape functions commonly used to prescribe gusts (e.g., the ECG), the spatial derivative in Eq. (3.13) inherently exhibits both positive and negative values. Consequently, to isolate the expulsion phase in the present gust source term, this concept must be translated from the temporal to the spatial domain, as described by De Nayer and Breuer [51]. The applied spatial restriction retains only the portion where the derivative $\partial f_1/\partial \xi$ is positive (corresponding to the expulsion phase). Based on the gust shapes employed in this work, this corresponds to the left portion of the $f_1(\xi)$ function (i.e., for $\xi < \xi_g$), ensuring a favorable positive momentum contribution. Hence, the final source-term formulation reads:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_0} \approx \begin{cases} \rho A_g f_t(t) f_2(t_\eta(x_{cc}, y_{cc}, z_{cc})) f_3(t_\zeta(x_{cc}, y_{cc}, z_{cc})) \left. \frac{\partial f_1}{\partial \xi} \right|_{cc} V \mathbf{u}|_{\mathcal{B}_{0,cc}} & \text{for } \xi < \xi_g \\ 0 & \text{else .} \end{cases} \quad (3.15)$$

The key characteristics of the present source-term formulation are summarized as follows. The gust may be introduced at any position within the computational domain. Following injection, it convects naturally, with no further intervention applied, as the governing Navier-Stokes equations alone dictate the subsequent evolution. In contrast to the micro-jet approach by Pasquetti and Peres [63], which employs a local point source-term, the current gust model employs a volumetric distribution of the momentum source. This distinction arises from the fundamental physical differences between micro-jets and gusts. Micro-jets involve both momentum and mass source terms to account for mass injection, whereas gusts are generated solely through momentum injection without the addition of mass. While laboratory experiments often introduce gusts via extra mass flux [see, e.g., 64, 65, 66], this does not reflect natural conditions, where gusts are generated by the local acceleration of the existing fluid. Consequently, the present momentum-only source-term formulation offers a physically more consistent representation for gust modeling. Furthermore, since no additional mass flux is introduced, the divergence-free constraint

remains satisfied within the injection region, in contrast to the micro-jet formulation, which requires relaxation of this constraint [63].

3.2 Smoothing the Gust Source Term Using the ECG- C^2 Function

In the definition of $\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_0}$ provided in Eq. (3.15), the expulsion phase is governed by the positive part of the derivative $\partial f_1/\partial \xi$, while the source term is set to zero during the ingestion phase. Applying this cut-off to a function whose derivative is not continuous at $\xi = \xi_g$ (e.g., the ECG function) introduces a kink in the derivative (see the zoomed view in Fig. 3.1b). In CFD simulations, such discontinuities lead to spurious numerical oscillations during gust injection. To mitigate this issue, the function selected for f_1 is designed such that its derivative remains continuous at $\phi = \phi_g$ when the cut-off is applied, ensuring a smooth transition. For this purpose, a function with a continuous second derivative, referred to as ECG- C^2 and illustrated in Fig. 3.1a, is derived from the classic ECG function in [51]. The expulsion part of the resulting ECG- C^2 function reads:

$$f_i(\phi) = \begin{cases} \frac{2}{L_g^\phi} \left(\phi - \left(\phi_g - \frac{L_g^\phi}{2} \right) + \frac{L_g^\phi}{4\pi} \sin \left(\frac{4\pi(\phi - \phi_g)}{L_g^\phi} + \pi \right) \right) & \text{for } \left(\phi - \left(\phi_g - \frac{L_g^\phi}{4} \right) \right) \in \left[-\frac{L_g^\phi}{4}, \frac{L_g^\phi}{4} \right] \\ 0 & \text{else .} \end{cases} \quad (3.16)$$

It is assumed that the ξ -direction and the time are related through the Taylor hypothesis. Consequently, the gust duration L_g^t is linked to the spatial length scale of the gust in the streamwise direction L_g^ξ by the expression:

$$L_g^\xi = L_g^t u_g^{\text{conv}} .$$

Here, u_g^{conv} represents the convective velocity of the wind gust in the surrounding flow field. In present work, this is approximated by the free-stream velocity, i.e., $u_g^{\text{conv}} = u_\infty$. Based on this assumption, the ECG- C^2 function is applied consistently to both the spatial distribution along ξ and the temporal distribution.

3.3 Main Results of Publication III: Horizontal Gusts Impacting a Hemispherical Structure

The present test case builds upon two successive experimental and numerical studies. In Wood et al. [67], the flow around a wall-mounted rigid hemisphere ($Re = 50,000$) placed within a turbulent boundary layer was investigated through both experimental measurements and large-eddy simulations. This configuration was subsequently extended in Wood et al. [68], where the rigid hemisphere was replaced by a flexible membrane of

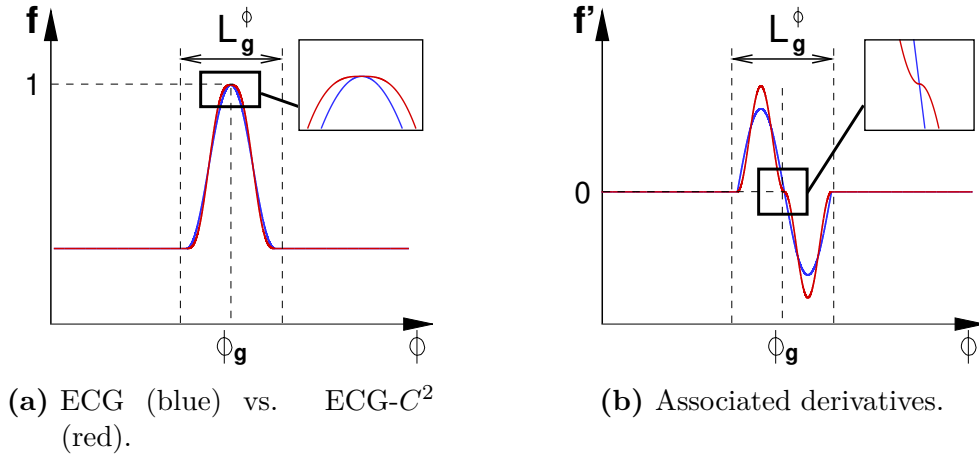


Fig. 3.1. Adapted “1-cosine” vs. modified C^2 -“1-cosine” shapes and their derivatives for a given scale L_g^ϕ and central value ϕ_g .

nominally identical shape and experimentally studied in the same wind tunnel facility ($Re = 50,000, 75,000$ and $100,000$). The flexible membrane setup was further analyzed using fully coupled fluid–structure interaction simulations relying on LES with ALE in De Nayer et al. [69] and Apostolatos et al. [70]. A schematic representation summarizing these previous investigations is provided in Fig. 3.2a.

The novel contribution presented in Publication III introduces a prescribed wind gust superimposed onto the turbulent inflow at $Re = 100,000$. While the gust is injected in close proximity to the object of interest using the source-term formulation described in Section 3.1, the realistic turbulent inflow is modeled by injecting synthetic turbulent perturbations directly upstream of the target region via the source-term method proposed by Schmidt and Breuer [71], Breuer [72], and De Nayer et al. [73]. These perturbations are generated using the so-called Synthetic Turbulent Inflow Generator (STIG) relying on the digital filter concept of Klein et al. [74]. Employed simultaneously, these source-term methods enable the reproduction of a physically realistic turbulent flow including gusts by leveraging the fine grid resolution near the object of interest. As a result, excessive damping of the turbulence intensity and strong numerical dissipation typically associated with gusts injected farther upstream are minimized. This approach enables the assessment of the transient effects of short-duration, high-intensity gust events on the aerodynamic forces and structural deformations.

The computational setup, including boundary conditions, the Synthetic Turbulent Inflow Generator (STIG) window, and the gust injection volume is depicted in Fig. 3.2b. Moreover, the gust characteristics of the simulations performed for the rigid and the flexible cases are summarized in Table 3.1. The selected gust strength range ($0.5 \leq A_g/u_\infty \leq 1.5$) is motivated by its physical relevance [75]. The lower bound results in a peak velocity of $u_\infty + A_g \approx 15$ m/s (54 km/h), which corresponds to a *near gale* wind category. The upper bound yields a peak velocity of approximately $u_\infty + A_g \approx 25$ m/s (90 km/h), corresponding to a *storm*.

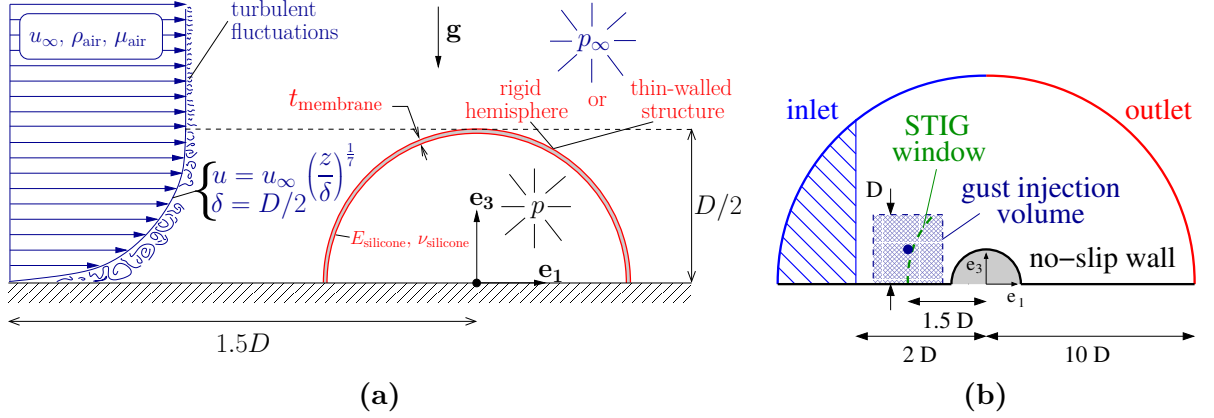


Fig. 3.2. 2D schematic sketch showing (a) the surface-mounted hemispherical geometry (diameter D) within a turbulent boundary layer, and (b) the definition of boundary conditions, the STIG injection window, and the gust injection volume.

Tab. 3.1. Gusts characteristics for ST formulation.

gust	$\mathbf{g}_1$	A_g	$f_t(t) = \text{ECG} - C^2$	$f_1(\xi) = \text{ECG} - C^2$	$f_2(\eta) = \text{ECG}$	$f_3(\zeta) = \text{ECG}$
			L_g^t	L_g^ξ	L_g^η	L_g^ζ
<i>weak</i>	$\mathbf{e}_1$	$0.5 u_\infty$	$L_g^\xi / u_g^{\text{conv}}$	$1 D$	$1 D$	$1 D$
<i>medium</i>	$\mathbf{e}_1$	u_∞	$L_g^\xi / u_g^{\text{conv}}$	$1 D$	$1 D$	$1 D$
<i>strong</i>	$\mathbf{e}_1$	$1.5 u_\infty$	$L_g^\xi / u_g^{\text{conv}}$	$1 D$	$1 D$	$1 D$

3.3.1 Flow Characteristics Prior to Gust Impact

The temporal evolution of the flow is analyzed as a function of the dimensionless time $t^* = t u_\infty / D$, where $t^* = 0$ marks the onset of the gust injection and $t^* = 1$ its completion. General observations concerning the pre-impact phase of the injected gusts listed in Table 3.1 are as follows. Increasing the gust amplitude A_g results in a higher peak of the total velocity, which can be approximately estimated by $u_1^{\text{max}} / u_\infty = 1 + A_g / u_\infty$. Moreover, the temporal evolution of the streamwise velocity at a monitoring point upstream of the hemispherical body reveals that the gusts become increasingly asymmetric with rising A_g . This asymmetry is characterized by a steeper velocity rise and a gradual decay. This behavior is attributed to the fact that fluid elements with higher velocities within the gust structure are convected faster than those with lower velocities, leading to the formation of a fast front that gradually detaches from the trailing portion of the gust. In the following analysis, flow field contours are primarily presented for the *medium gust* case, as the spatial distributions induced by the weak and strong gusts do not differ qualitatively from those observed for the medium gust.

Figure 3.3 illustrates the streamwise velocity u_1 / u_∞ and pressure $p / (\rho_{\text{air}} u_\infty^2)$ fields at $t^* \approx 0.95$, representing the phase shortly before the *medium* gust reaches the rigid hemisphere. For direct comparison, the figure also includes the baseline case without gust, computed under identical turbulent inflow conditions and captured at the same

instant in time. The gust structure is visualized by a red iso-surface of $u_1/u_\infty = 1.4$ in Fig. 3.3a and by a black iso-line of the same velocity magnitude in Fig. 3.3b. At this pre-impact stage, the gust does not alter the flow field in the vicinity of the hemisphere or in its wake, with both velocity and pressure distributions remaining nearly identical to the baseline case. The influence of the gust is confined to the upstream region, where it causes a local acceleration of the streamwise velocity, accompanied by a corresponding pressure drop consistent with Bernoulli's principle. This highlights a key characteristic of the ST method. Despite employing an incompressible flow solver, the effect of the momentum-based injected gust remains spatially localized during the initial phase and does not propagate throughout the computational domain.

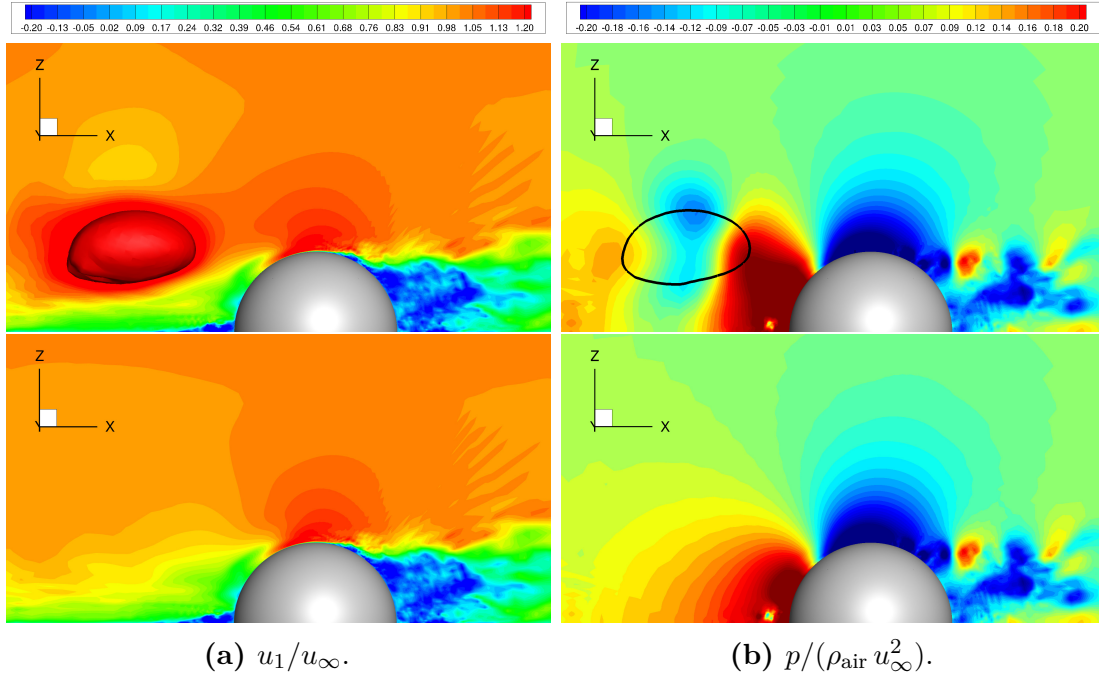


Fig. 3.3. Instantaneous snapshots in the symmetry plane ($y = 0$) showing the *medium* gust approaching the rigid hemisphere at $t^* \approx 0.95$. The top row depicts the case with gust injection, while the bottom row shows the corresponding flow without gust. The gust structure is indicated by a red iso-surface or a black iso-line of the streamwise velocity $u_1/u_\infty = 1.4$.

3.3.2 Horizontal Gusts Impacting Rigid and Flexible Hemispheres

The following analysis investigates the impact of varying wind gust loads on both the rigid and flexible hemispheres. As in the previous section, contour plots are shown exclusively for the *medium* gust, as this case sufficiently captures the characteristic flow features. Figure 3.4 depicts the streamwise velocity within a slice near the bottom wall and in the symmetry plane, highlighting three characteristic time instants that capture the phases of

gust impact, interaction, and decay. To quantitatively assess the impact of the different gusts on the structure, the force coefficients in the streamwise and vertical directions are plotted as a function of the dimensionless time for both the rigid and flexible hemisphere cases in Fig. 3.5. After injection, the gust convects downstream and impacts the tip of the hemisphere, as shown in Fig. 3.4a. For both rigid and flexible cases, this time instant coincides with the maximum of the vertical force coefficient c_z , driven by a pronounced low-pressure region above the hemisphere. However, in contrast to the rigid case, the flexible membrane undergoes significant deformation, forming a hollow indentation at the front. Within this indentation, the local flow velocity decreases, resulting in a localized high-pressure zone. Then, the gust proceeds downstream and moves past the body, as shown in Fig. 3.4b. During this phase, the indentation forming at the front continues to grow larger and deeper. Consequently, a distinct delay is observed between the maxima of the total forces and the maxima of the local front deflections. At the next time instant, shown in Fig. 3.4c, the gust convects further downstream into the wake and decays, while the membrane reaches its peak deflection. The maximum deflection occurs along the symmetry plane on the front side, approximately at the mid-height of the hemisphere. Beyond this point (not shown here), the front indentation begins to oscillate before eventually returning to its original deflected state, driven by the internal gauge pressure of the flexible hemisphere.

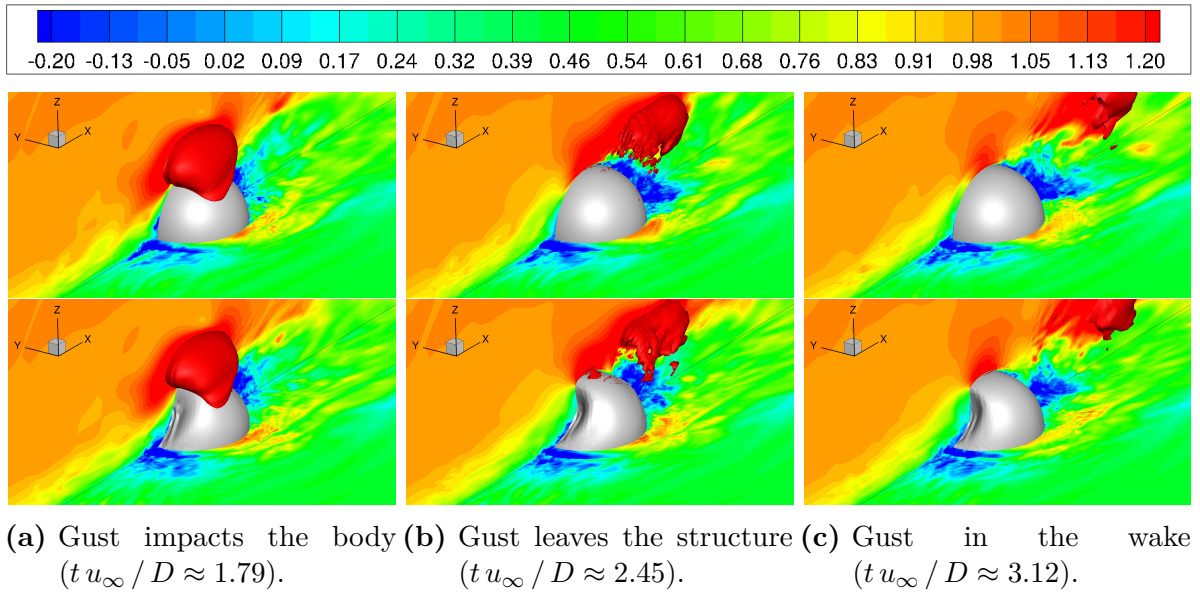


Fig. 3.4. Injected *medium* wind gust convecting downstream and interacting with a rigid hemisphere (top row) and an air-inflated flexible hemisphere (bottom row). The streamwise velocity u_1/u_∞ is visualized within a slice near the bottom wall and in the symmetry plane ($y = 0$). The gust structure is highlighted by a red iso-surface of the streamwise velocity $u_1/u_\infty = 1.4$.

The force coefficients predicted for different gust strengths in the flexible case closely resemble those obtained for the rigid body, as shown in Fig. 3.5. In both cases, increasing

the gust strength leads to higher amplitudes of the two local maxima of the streamwise force coefficient c_x . Moreover, the time interval between these two maxima decreases with stronger gusts, attributed to their higher convection velocity. A similar trend is observed for the vertical force coefficient c_z , whose peak occurs earlier for stronger gusts and exhibits a higher amplitude due to the more pronounced low-pressure region near the hemisphere's tip. A further analysis indicates that pressure forces dominate the total force contribution in both the streamwise and vertical directions across all gust strengths. This dominance is especially evident in the vertical direction, where shear forces contribute only marginally. Consequently, the peak force coefficients exhibit a quadratic dependence on the effective oncoming velocity, which arises from the superposition of the incoming boundary layer flow and the velocity increment introduced by the gust. A key distinction emerges between the rigid and flexible cases when examining the amplitudes of the c_x extrema. The flexible membrane tends to reduce the difference between the two local maxima and the intermediate minimum, resulting in a smoother evolution of the streamwise force coefficient. Moreover, both global force coefficients exhibit oscillations for the flexible membrane after the gust has passed the hemisphere. This behavior is attributed to the structural oscillations of the membrane as it gradually returns to its original deflected state.

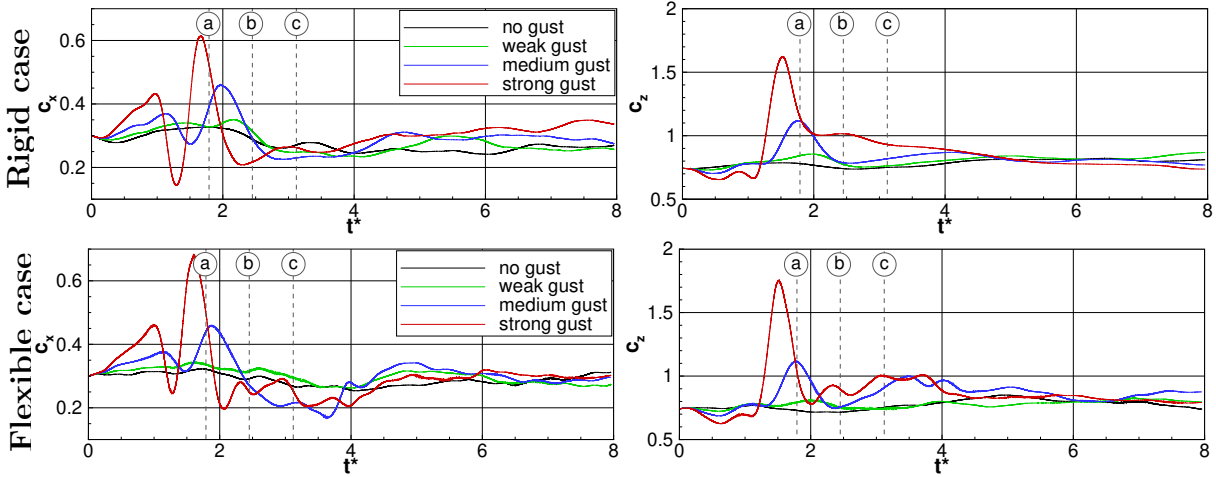


Fig. 3.5. Predicted streamwise and vertical force coefficients for the rigid and flexible hemisphere cases subjected to different gust strengths. The time-axis annotation (a), (b), and (c) correspond to the snapshots shown in Fig. 3.4.

3.3.3 Structural Deformations and Material Response

A comparison of the maximum membrane deflections is presented in Fig. 3.6. Owing to the time lag between force response and structural deformation, the peak membrane deflections occur at later time instants than the corresponding force maxima. As expected, stronger gusts lead to deeper front indentations. This front indentation also affects the overall membrane shape, as increasing gust strength causes the tip of the structure to

rise and the sides to expand outward. These geometric changes result from the combined effect of the external pressure distribution and the constant internal gauge pressure. Once the gust has moved downstream, the front indentation diminishes, and both the tip and sides gradually return to their original positions.

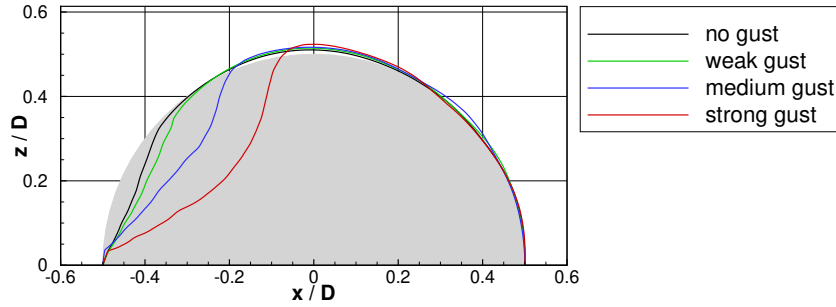


Fig. 3.6. Flexible FEM case: Maximum structural deflections in the symmetry plane for different gust strengths.

To further assess the structural response, a classical criterion for assessing failure in ductile materials is applied by predicting the von Mises stress and comparing this equivalent uniaxial tensile stress to the material's yield stress. Figure 3.7 depicts the von Mises stress distribution at the instant of maximum membrane deflection. The highest stresses consistently occur at the apex of the hemispherical membrane. As expected, increasing gust strength results in higher von Mises stresses, with a factor of approximately 1.73 between the peak values for the weak and strong gust cases. It is important to emphasize that in this generic test case, the von Mises stress remains well below (one order of magnitude less) the yield stress of the silicone membrane. Nevertheless, these results demonstrate the capability of the simulation framework to capture local stress responses under highly transient load conditions.

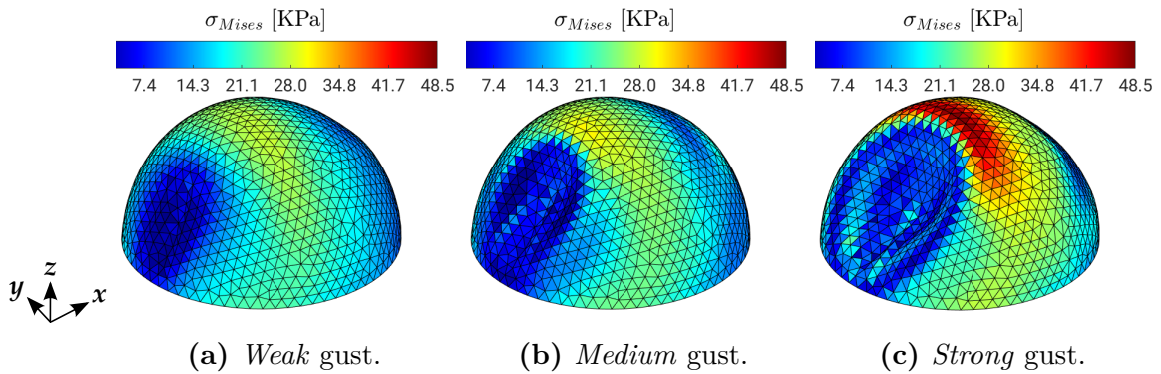


Fig. 3.7. Flexible FEM case: Von Mises stress distributions for the maximally deflected structure at different gust strengths.

3.4 Conclusions

This chapter provides a comprehensive summary of Publication III, which examines the interaction of horizontal wind gusts with a flexible hemispherical membrane exposed to a turbulent boundary layer using a high-fidelity simulation framework.

The study employs a recently developed gust injection technique within a partitioned FSI solution approach, coupling large-eddy simulation for the turbulent flow field relying on an ALE finite-volume formulation with the finite-element method for the structural response. The source-term based gust injection method extends previous applications from laminar and turbulent background flows on Cartesian grids to a more complex turbulent fluid–structure interaction scenario on block-structured curvilinear grids. This configuration allows for the injection area to be positioned directly near the FSI region of interest, significantly improving computational efficiency by reducing numerical dissipation and eliminating the need for long-distance gust propagation from the inlet. Moreover, since the source term is applied exclusively to the momentum equation, the continuity equation remains unaffected, preserving the divergence-free condition of incompressible flows and ensuring stable convergence behavior even under highly transient gust events. Compared to the prescribed velocity methods discussed in Chapter 2, where the gust velocity is imposed at each point in space and convected explicitly at each instant in time, the ST method is less CPU-time intensive, as the source term is applied only during the injection phase and is then allowed to propagate freely through the computational domain.

The analysis reveals several key physical insights. First, the application of horizontal wind gusts significantly amplifies the global force coefficients in both streamwise and vertical directions, with strong gusts doubling the forces compared to the undisturbed boundary layer case. Second, the membrane flexibility slightly attenuates the peak global forces relative to the rigid configuration. However, the local fluid loads, particularly pressure forces, dominate the response and drive considerable structural deformations. These local deformations exhibit a distinct time delay relative to the peak force response. Third, through the fully coupled FSI approach, detailed stress distributions, including the von Mises stress, are evaluated, offering a robust framework to assess structural integrity under transient load conditions. Although the von Mises stress remains well below the material yield limit in the present configuration, the methodology demonstrates a strong capability to predict local stresses in complex FSI environments.

Finally, comparisons between FEM and IGA discretizations (detailed in Publication III) show consistent aerodynamic and structural behavior, with IGA yielding smoother deformations at the cost of significantly higher computational expense. Overall, this study highlights the potential of the ST-based gust injection method combined with high-fidelity FSI modeling to reliably capture gust–structure interactions in turbulent flows. This makes it a powerful and efficient tool for future investigations of flexible structures subjected to transient wind events.

4 Artificial Discrete Wind Gust Injection via Immersed Boundary Gust Generators

Artificial generation of wind gusts using so-called wind gust generators (WGG) enables the controlled and repeatable creation of discrete gusts under laboratory conditions. Unlike natural atmospheric gusts, which are stochastic and difficult to reproduce, an artificial generator produces a flow disturbance with prescribed amplitude, duration, and direction. This allows systematic and comparable investigations of gust–structure interactions. Furthermore, experimental gust generators provide a valuable basis for validating numerical methods, since the same inflow disturbance can be imposed in both experiments and simulations.

In this context, two wind-tunnel gust generators recently developed at HSU Hamburg, namely the *paddle* [52] and the *adaptive nozzle* [53], depicted in Fig. 4.1, are considered. The operating principle of these devices relies on the rapid partial blockage of the wind-tunnel nozzle’s outlet by a rigid body that undergoes large displacements, thereby generating a sudden change in the free-stream velocity. Despite their importance in advancing the understanding of the gust-generation process, wind-tunnel experiments remain constrained by the limited spatial and temporal resolution of measurement techniques, which hampers the detailed characterization of three-dimensional and transient flow phenomena across the entire test section. To address these limitations and to complement the experimental findings, the objective of the work published in this category of numerical gust generation is to replicate the experimental wind-gust generators. This approach enables a detailed analysis of the three-dimensional, time-resolved flow field throughout the entire computational domain and establishes a foundation for fluid–structure interaction simulations under controlled gust conditions. High-fidelity numerical simulations further allow the exploration of flow regions that are difficult to access experimentally, such as the interior of the nozzle, and enable precise quantification of velocity fluctuations, pressure distributions, and turbulence characteristics.

The main modeling challenge in this category of gust generation is to accurately represent the transient blockage effect caused by the fast rigid-body motion of the paddle or the adaptive nozzle’s upper contour. To capture this effect in a computationally efficient manner, the *continuous forcing immersed boundary method* (also referred to as the diffuse-interface IBM) with the *direct-forcing approach* proposed by Uhlmann [76] is employed. Within the scope of the present modeling objective, the IB method is preferred over body-fitted approaches with adaptive meshing, since the latter are not only costly and prone to grid-quality deterioration but also infeasible in this context. This is due to the large vertical displacements of the blocking structure (especially in the paddle case),

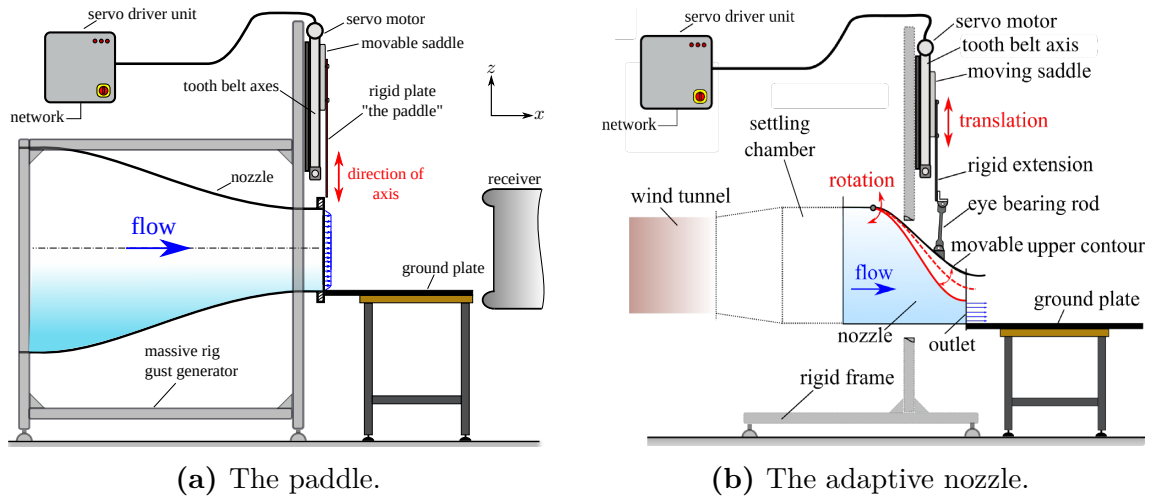


Fig. 4.1. Side view sketches of the experimental wind gust generators: (a) the paddle retrofitted to a Göttingen-type wind tunnel, and (b) the adaptive nozzle contour installed in an Eiffel-type wind tunnel.

which would cause severe grid distortion and numerical instability, ultimately leading to simulation failure. In contrast, the IB method allows the structure to move freely within a fixed Eulerian grid, thereby preserving the grid quality. This provides a robust and flexible framework for modeling complex, time-dependent fluid–structure interactions, particularly when the rigid body undergoes large displacements, as in the gust generator configurations considered here. In this framework, the fluid and structural domains are represented on overlapping discretizations, where Eulerian and Lagrangian frames of reference are used to describe physical quantities associated with the fluid and the structure, respectively. The coupling between both domains is achieved by introducing an appropriate boundary forcing term that enforces the no-slip condition on the immersed interface.

This chapter provides a comprehensive overview of the numerical methodology for artificial discrete wind gust injection via immersed boundary gust generators (IBGG), along with a summary of the corresponding published results. Section 4.1 introduces the immersed boundary method with a direct forcing approach, employed within a predictor-corrector scheme to model moving rigid bodies in a viscous incompressible flow on a uniform Cartesian grid. In practical CFD simulations, it is often desirable to stretch the computational grid and cluster points near solid boundaries to accurately resolve the boundary layers. Such selective refinement avoids unnecessary grid density in regions of lesser importance and yields significant computational savings. The IB method is therefore extended in two main directions, both capable of handling general non-uniform computational grids. Depending on the geometric configuration of the immersed boundary within the computational domain, the suitable formulation is adopted for each gust generator case. Section 4.2 details the extension based on the Reproducing Kernel Particle Method (RKPM), employed to simulate the paddle gust generator presented in Publication IV¹.

¹Boulbrachene & Breuer, *Phys. Fluids* **36**, 015146 (2024).

In this numerical model, the paddle is immersed in a non-uniform Cartesian grid, away from the boundaries of the computational domain. This approach is further combined with a partitioned ALE fluid–structure interaction methodology to simulate the response of a thin, flexible structure deforming under dynamic wind-gust loading, as discussed in Publication V². Section 4.3 introduces the moving least-squares (MLS) version of the direct-forcing IB method, used to simulate the adaptive nozzle gust generator presented in Publication VI³. Here, the nozzle’s upper contour traverses between two side walls, which restricts the applicability of symmetric kernels in near-wall regions. The MLS-based construction is therefore leveraged to generate one-sided IB kernels by modifying the support of a standard symmetric weight function. The main results published from the three publications are summarized in Sections 4.4, 4.5, and 4.6.

4.1 Immersed Boundary Method

The immersed boundary method (IBM) provides a numerical framework for simulating fluid flows involving rigid or flexible bodies using non-conforming grids. Originally introduced by Peskin [77] for flexible structures and later extended to viscous flows around rigid bodies by Uhlmann [76], the continuous forcing formulation of the IBM represents the immersed boundary Γ , which is embedded within the Eulerian fluid domain Ω , by a set of Lagrangian markers as depicted in Fig. 4.2.

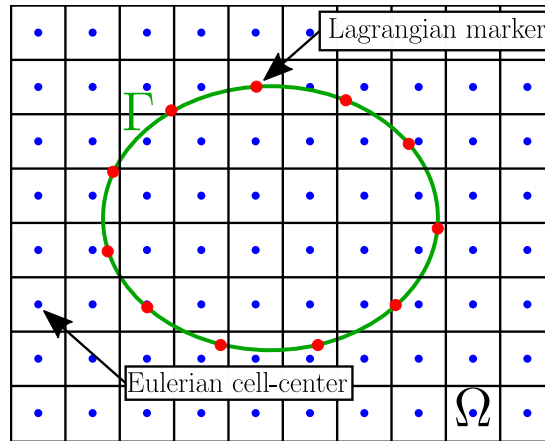


Fig. 4.2. Two-dimensional schematic illustrating the overlapping Lagrangian and Eulerian discretizations.

At the continuous level, the influence of the immersed boundary on the surrounding fluid is incorporated into the governing equations through a forcing term $\mathbf{f}$ added to their

²Breuer, Boulbrachene & De Nayer, *Comput. Fluids* **294**, 106621 (2025).

³Boulbrachene & Breuer, *Comput. Fluids* **301**, 106775 (2025).

right-hand side. Consequently, the Navier–Stokes equations for an incompressible fluid in the domain read:

$$\left. \begin{aligned} \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \nabla \cdot (\mathbf{u}\mathbf{u}) &= \mu \Delta \mathbf{u} - \nabla p + \mathbf{f} \\ \nabla \cdot \mathbf{u} &= 0 \end{aligned} \right\} \quad \mathbf{x} \in \Omega, \quad (4.1)$$

where $\mathbf{f}$ is a volume force arising from the presence of the immersed boundary. At the discrete level, variables are distinguished based on their spatial location. Lower-case letters denote Eulerian quantities defined on the fluid grid, whereas upper-case letters indicate Lagrangian quantities associated with the solid boundary. The in-house finite-volume solver used in this dissertation (i.e., FASTEST-3D [54]) employs a collocated grid arrangement, with Eulerian locations corresponding to the cell centers $\mathbf{x}_e$ (i.e., $\mathbf{x}_e \in \Omega \forall 1 \leq e \leq N_E$, where N_E denotes the total number of cell centers). The Lagrangian locations $\mathbf{X}_l$ correspond to the distributed force points along the immersed boundary Γ (i.e., $\mathbf{X}_l \in \Gamma \forall 1 \leq l \leq N_L$, where N_L denotes the total number of Lagrangian markers).

Within the finite-volume solver employed in this work, the implementation of the continuous-forcing IBM for moving rigid boundaries involves two main components. The first concerns the tracking and the advancement of Lagrangian markers relative to the fixed structured Eulerian grid, as described in Section 4.1.1. The second enforces the no-slip condition at the Lagrangian markers by coupling the Lagrangian and Eulerian frames through the direct forcing approach, detailed in Section 4.1.2.

4.1.1 Lagrangian Marker Tracking

Lagrangian markers are tracked in the computational space (c-space), which is obtained by mapping the structured grid connecting the cell centers (i.e., the $\mathbf{z}$ -grid shown in Fig. 4.3a) to an orthonormal grid [78]. This approach circumvents the need for the time-intensive search in the physical space (p-space) that would otherwise be required to locate the new computational cell containing each Lagrangian marker. In c-space, the position of each Lagrangian marker, denoted as $\boldsymbol{\xi}_l$ (see Fig. 4.3b), is expressed as the sum of the cell's index triple and a fractional offset:

$$\boldsymbol{\xi}_l = [\xi_l, \eta_l, \zeta_l] = [i_l, j_l, k_l] + [\Delta\xi_l^+, \Delta\eta_l^+, \Delta\zeta_l^+]. \quad (4.2)$$

Tracking the Lagrangian markers in this manner also serves in efficiently identifying the nearest cell center to each Lagrangian marker (defined here as $\mathbf{x}_{i,\hat{j},\hat{k}}$), necessary for setting the bounds of the regularized delta function used in the IB method. The index triples $[i_l, j_l, k_l]$ and $[\hat{i}_l, \hat{j}_l, \hat{k}_l]$ can be directly obtained by applying "int" and "nint" operators (available in programming languages) to the computational coordinates $[\xi_l, \eta_l, \zeta_l]$, respectively. The fractional offset can then be evaluated using Eq. (4.2). Its complement then reads:

$$\Delta\xi_l^- = 1 - \Delta\xi_l^+, \quad \Delta\eta_l^- = 1 - \Delta\eta_l^+, \quad \Delta\zeta_l^- = 1 - \Delta\zeta_l^+. \quad (4.3)$$

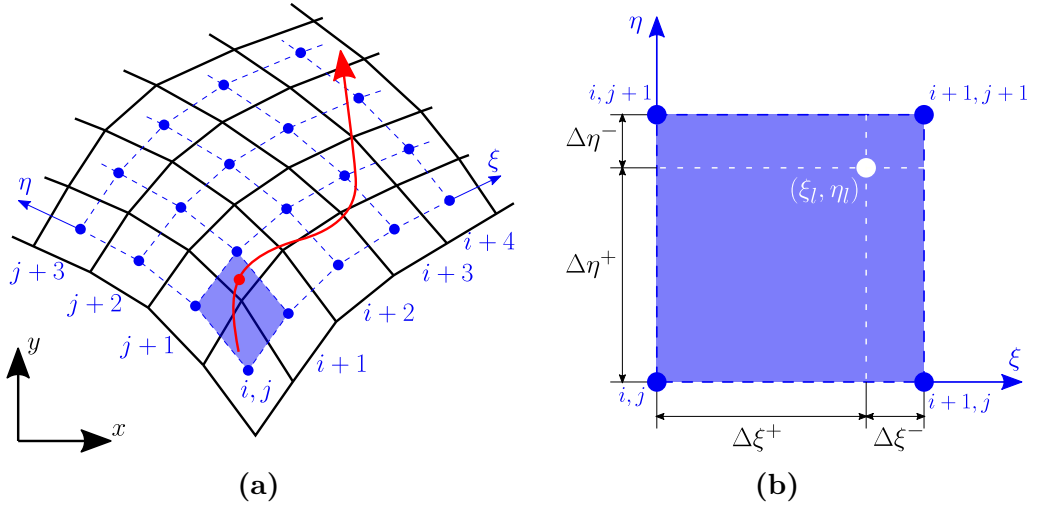


Fig. 4.3. (a) Two-dimensional sketch of the p-space showing a computational grid (black lines), cell centers (blue dots), $\mathbf{z}$ -grid (blue dashed lines), and a marker's trajectory (red line). (b) Corresponding c-space representation showing trilinear interpolation coefficients.

The fractional offset and its complement can be employed to write the transformation T of the marker's position $\boldsymbol{\xi}_l$ to the p-space by means of a trilinear interpolation using the cell centered values $\mathbf{x}_{i,j,k}$ [78]:

$$\begin{aligned} \mathbf{X}_l = T(\boldsymbol{\xi}_l) = & \mathbf{x}_{i,j,k} \left[\Delta\xi_l^- \Delta\eta_l^- \Delta\zeta_l^- \right] + \mathbf{x}_{i+1,j,k} \left[\Delta\xi_l^+ \Delta\eta_l^- \Delta\zeta_l^- \right] + \\ & \mathbf{x}_{i,j+1,k} \left[\Delta\xi_l^- \Delta\eta_l^+ \Delta\zeta_l^- \right] + \mathbf{x}_{i,j,k+1} \left[\Delta\xi_l^- \Delta\eta_l^- \Delta\zeta_l^+ \right] + \\ & \mathbf{x}_{i+1,j+1,k} \left[\Delta\xi_l^+ \Delta\eta_l^+ \Delta\zeta_l^- \right] + \mathbf{x}_{i+1,j,k+1} \left[\Delta\xi_l^+ \Delta\eta_l^- \Delta\zeta_l^+ \right] + \\ & \mathbf{x}_{i,j+1,k+1} \left[\Delta\xi_l^- \Delta\eta_l^+ \Delta\zeta_l^+ \right] + \mathbf{x}_{i+1,j+1,k+1} \left[\Delta\xi_l^+ \Delta\eta_l^+ \Delta\zeta_l^+ \right]. \end{aligned} \quad (4.4)$$

This interpolation is used to determine the physical coordinates of the Lagrangian markers, which are required by the immersed boundary solver (as will be discussed in Section 4.1.2) and to facilitate visualization of the immersed boundary during the post-processing stage.

The immersed boundaries modeled in this work (i.e., the paddle and the nozzle's upper contour) are set into motion by prescribing the desired velocity $\mathbf{U}_l^d(t) = [U_l^d, V_l^d, W_l^d]$ for each Lagrangian marker in the physical space. To update each marker's position within the c-space, the prescribed velocity components must first be transformed into the c-space as follows:

$$\frac{d\xi_l}{dt} = \left(\frac{\partial \xi}{\partial x} \right)_l U_l^d + \left(\frac{\partial \xi}{\partial y} \right)_l V_l^d + \left(\frac{\partial \xi}{\partial z} \right)_l W_l^d, \quad (4.5)$$

$$\frac{d\eta_l}{dt} = \left(\frac{\partial \eta}{\partial x} \right)_l U_l^d + \left(\frac{\partial \eta}{\partial y} \right)_l V_l^d + \left(\frac{\partial \eta}{\partial z} \right)_l W_l^d, \quad (4.6)$$

$$\frac{d\zeta_l}{dt} = \left(\frac{\partial \zeta}{\partial x} \right)_l U_l^d + \left(\frac{\partial \zeta}{\partial y} \right)_l V_l^d + \left(\frac{\partial \zeta}{\partial z} \right)_l W_l^d. \quad (4.7)$$

Hence, the location of a Lagrangian marker is updated by carrying out the temporal integration:

$$\xi_l^{n+1} = \xi_l^n + \left(\frac{\partial \xi}{\partial x} \right)_l \int_{t^n}^{t^{n+1}} U_l^d dt + \left(\frac{\partial \xi}{\partial y} \right)_l \int_{t^n}^{t^{n+1}} V_l^d dt + \left(\frac{\partial \xi}{\partial z} \right)_l \int_{t^n}^{t^{n+1}} W_l^d dt, \quad (4.8)$$

$$\eta_l^{n+1} = \eta_l^n + \left(\frac{\partial \eta}{\partial x} \right)_l \int_{t^n}^{t^{n+1}} U_l^d dt + \left(\frac{\partial \eta}{\partial y} \right)_l \int_{t^n}^{t^{n+1}} V_l^d dt + \left(\frac{\partial \eta}{\partial z} \right)_l \int_{t^n}^{t^{n+1}} W_l^d dt, \quad (4.9)$$

$$\zeta_l^{n+1} = \zeta_l^n + \left(\frac{\partial \zeta}{\partial x} \right)_l \int_{t^n}^{t^{n+1}} U_l^d dt + \left(\frac{\partial \zeta}{\partial y} \right)_l \int_{t^n}^{t^{n+1}} V_l^d dt + \left(\frac{\partial \zeta}{\partial z} \right)_l \int_{t^n}^{t^{n+1}} W_l^d dt. \quad (4.10)$$

For a fixed grid as used in the present case, the spatial metric coefficients appearing in Eqs. (4.8) to (4.10) are time independent. They represent the entries of the Jacobian of the p-space to the c-space coordinate transformation $\mathbf{J}$ and are evaluated at the Lagrangian location. Employing the *CZ+* tracking scheme proposed by Schäfer and Breuer [78], the Jacobian of the inverse coordinate transformation $\mathbf{J}^{-1}$ (i.e., c-space to p-space) is analytically evaluated by differentiating Eq. (4.4). The first column of $\mathbf{J}^{-1}$ at a position ξ_l reads:

$$\begin{aligned} \mathbf{J}_1^{-1}(\xi_l) = \frac{\partial \mathbf{X}_l}{\partial \xi_1} = \frac{\partial T(\xi_l)}{\partial \xi_1} = & \Delta \eta_l^- \Delta \zeta_l^- (\mathbf{x}_{i+1,j,k} - \mathbf{x}_{i,j,k}) + \\ & \Delta \eta_l^+ \Delta \zeta_l^- (\mathbf{x}_{i+1,j+1,k} - \mathbf{x}_{i,j+1,k}) + \\ & \Delta \eta_l^- \Delta \zeta_l^+ (\mathbf{x}_{i+1,j,k+1} - \mathbf{x}_{i,j,k+1}) + \\ & \Delta \eta_l^+ \Delta \zeta_l^+ (\mathbf{x}_{i+1,j+1,k+1} - \mathbf{x}_{i,j+1,k+1}). \end{aligned} \quad (4.11)$$

Following the evaluation of $\mathbf{J}_m^{-1}(\xi_l)$ where $m = 1, 2, 3$ and inverting it, the spatial metrics in Eqs. (4.8) to (4.10) are obtained.

4.1.2 IBM with a Direct Forcing Approach

The direct forcing method computes the force exerted by the solid on the fluid at the fluid–solid interface within the Lagrangian frame of reference. The force is formulated in such a manner that it enforces (to a certain extent) the no-slip boundary condition at each Lagrangian marker. In the context of the predictor-corrector method, the Lagrangian volume force at a time step t^n is given by [76]:

$$\mathbf{F}_l^n = \rho \frac{\mathbf{U}_l^d - \mathbf{U}_l^*}{\Delta t}. \quad (4.12)$$

Here, $\mathbf{U}_l^d$ and $\mathbf{U}_l^*$ denote the desired and the intermediate velocity after the predictor step at a Lagrangian location $\mathbf{X}_l$, respectively. The desired velocity $\mathbf{U}_l^d$ is explicitly prescribed by the rigid-body motion of the immersed boundary. For dynamic immersed boundaries, Lagrangian markers are generally not coinciding with the Eulerian cell centers. Since the fluid solver computes the intermediate velocity only at the Eulerian locations $\mathbf{x}_e$, $\mathbf{U}_l^*$ has to be evaluated by interpolating the Eulerian flow variables to the corresponding Lagrangian

locations $\mathbf{X}_l$. This process of transferring information between the two discrete frames of reference is referred to as the interpolation operation. On a uniform one-dimensional Eulerian grid with grid spacing h , this operation is performed by employing a regularized Dirac delta function δ_d [79]:

$$\delta_d(x_e - X_l) = \frac{1}{d} \varphi(r) , \quad (4.13)$$

where d is the dilation parameter ($d = h$ for uniform grids) and φ is a continuous and symmetric kernel function of compact support. The dilation parameter defines the extent of the kernel's support Ω_l , while r represents the normalized distance between an Eulerian and a Lagrangian coordinate ($r = (x - X)/d$). An example of a three-point kernel defined by Roma et al. [80] is depicted in Fig. 4.4a. On a three-dimensional *uniform Cartesian* grid g_h with grid spacing h , the regularized delta function is constructed as the tensor product of three one-dimensional functions (one for each coordinate direction) and is used to perform the interpolation operation $\mathcal{I}(\mathbf{u}_e^*)$:

$$\mathbf{U}_l^* = \sum_{e \in N_{E,l}} \mathbf{u}_e^* \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e , \quad (4.14)$$

where $N_{E,l}$ denotes the set of Eulerian cells whose centroids $\mathbf{x}_e$ are located within the support of the kernel associated with the l th Lagrangian marker, and Δv_e is the volume of the e th Eulerian cell. An illustration of the interpolation operation, which evaluates a discrete Lagrangian quantity as a weighted average of neighboring discrete Eulerian quantities, is depicted in Fig. 4.4b.

After the interpolation step, the Lagrangian boundary force $\mathbf{F}_l$ is directly computed at each Lagrangian force point using Eq. (4.12). The effect of the immersed boundary is subsequently incorporated into the momentum equations by spreading the Lagrangian volume forces onto the Eulerian grid, where they appear as a volumetric source term on the right-hand side. The spreading operation $\mathcal{S}(\mathbf{F}_l)$, depicted in Fig. 4.4c, employs the

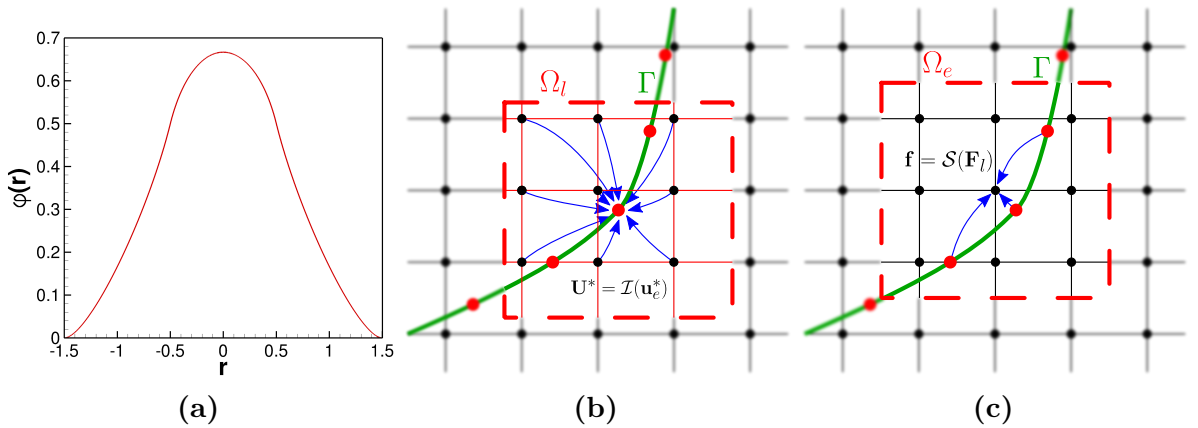


Fig. 4.4. (a) Three-point kernel by Roma et al. [80]; (b) interpolation of Eulerian velocities to a Lagrangian location; and (c) spreading of Lagrangian forces to an Eulerian location.

same regularized delta function as the interpolation operation to evaluate the Eulerian volume force:

$$\mathbf{f}_e = \sum_{l \in N_{L,e}} \mathbf{F}_l \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta V_l, \quad (4.15)$$

where $N_{L,e}$ denotes the set of Lagrangian markers whose kernel support contains the e th Eulerian cell, and ΔV_l is the volume associated with the l th Lagrangian marker. The Lagrangian volume can be expressed as $\Delta V_l = \Delta S_l \varepsilon_l$, where ΔS_l represents the surface area associated with the l th marker and ε_l denotes its characteristic thickness. For uniform Eulerian grids, this volume is typically assumed to equal the fluid cell volume, provided that the Lagrangian markers are uniformly distributed such that $\Delta S_l \approx \Delta v_e^{2/3}$ (i.e., $\varepsilon_l \approx h$ [76]). An alternative approach for determining the characteristic thickness involves setting it such that, when the force is spread from the Lagrangian marker and interpolated back to it, the exact force is recovered [81]. This method requires solving a linear system of equations that enforces the duality of the interpolation and the spreading operators. Zhou and Balachandar [82], however, have shown that the Lagrangian volume serves as a relaxation factor governing how fast the no-slip condition is satisfied. The authors have also found that the optimal choice of the Lagrangian volume corresponds to the largest value permitted by a stability condition. Eventually, the choice of the Lagrangian volume is somewhat arbitrary, as it primarily influences the magnitude of the feedback forcing [83]. In this work, the actual physical thickness of the modeled geometries (namely, the paddle and the nozzle's upper contour) is explicitly used to define the third dimension of the Lagrangian grid.

The steps of the predictor-corrector method, modified to account for the immersed boundary, can now be summarized by the following equations:

$$1. \quad \mathbf{u}_e^* = \mathbf{u}_e^n + \frac{\Delta t}{\rho} \mathbf{rhs}^n \quad \forall \mathbf{x}_e \in g_h, \quad (4.16)$$

$$2. \quad \mathbf{U}_l^* = \mathcal{I}(\mathbf{u}_e^*) \quad \forall \mathbf{X}_l \in \Gamma, \quad (4.17)$$

$$3. \quad \mathbf{F}_l = \rho \frac{\mathbf{U}_l^d - \mathbf{U}_l^*}{\Delta t} \quad \forall \mathbf{X}_l \in \Gamma, \quad (4.18)$$

$$4. \quad \mathbf{f}_e = \mathcal{S}(\mathbf{F}_l) \quad \forall \mathbf{x}_e \in g_h, \quad (4.19)$$

$$5. \quad \mathbf{u}_e^{**} = \mathbf{u}_e^* + \frac{\Delta t}{\rho} \mathbf{f}_e \quad \forall \mathbf{x}_e \in g_h, \quad (4.20)$$

$$6. \quad \Delta p_e' = \frac{\rho}{\Delta t} \nabla \cdot \mathbf{u}_e^{**} \quad \forall \mathbf{x}_e \in g_h, \quad (4.21)$$

$$7. \quad \mathbf{u}_e^{n+1} = \mathbf{u}_e^n - \frac{\Delta t}{\rho} \nabla p_e' \quad \forall \mathbf{x}_e \in g_h. \quad (4.22)$$

In comparison to the standard predictor-corrector scheme, steps 2 to 5 are additionally introduced. These steps lead to a second intermediate velocity $\mathbf{u}_e^{**}$, which satisfies (with a decreasing error at each time step) the no-slip boundary condition at the fluid–solid interface. This velocity is then used to construct the right-hand side of the Poisson equation (4.21) for computing the pressure correction p_e' (step 6). Finally, the pressure correction

is used to determine the updated velocity $\mathbf{u}_e^{n+1}$ (step 7).

Several regularized delta function kernels, effectively smearing the boundary forces over Eulerian grid cells within their compact support, have been derived in the literature [see, e.g., 77, 79, 80, 84]. The choice of a specific kernel typically depends on the required level of accuracy, the smoothing properties, and the desired numerical efficiency. Nevertheless, all smoothing kernels are designed to satisfy, at a minimum, the zeroth- and the first-order discrete moment conditions on uniform grids:

$$\sum_{e \in N_{E,l}} \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e = 1 \quad \forall l, \quad (4.23)$$

$$\sum_{e \in N_{E,l}} (\mathbf{x}_e - \mathbf{X}_l) \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e = 0 \quad \forall l. \quad (4.24)$$

These discrete analogues of the kernel's fundamental properties ensure the integral conservation of the force and torque in the spreading operation:

$$\begin{aligned} \sum_{e \in N_E} \mathbf{f}_e \Delta v_e &= \sum_{e \in N_E} \sum_{l \in N_{L,e}} \mathbf{F}_l \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta V_l \Delta v_e \\ &= \sum_{l \in N_L} \mathbf{F}_l \left(\sum_{e \in N_{E,l}} \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e \right) \Delta V_l = \sum_{l \in N_L} \mathbf{F}_l \Delta V_l, \end{aligned} \quad (4.25)$$

$$\begin{aligned} \sum_{e \in N_E} \mathbf{x}_e \times \mathbf{f}_e \Delta v_e &= \sum_{e \in N_E} \mathbf{x}_e \times \left(\sum_{l \in N_{L,e}} \mathbf{F}_l \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta V_l \right) \Delta v_e \\ &= \sum_{l \in N_L} \left(\sum_{e \in N_{E,l}} \mathbf{x}_e \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e \right) \times \mathbf{F}_l \Delta V_l = \sum_{l \in N_L} \mathbf{X}_l \times \mathbf{F}_l \Delta V_l. \end{aligned} \quad (4.26)$$

A key limitation of directly using analytically derived IB kernels is their reliance on a uniform Cartesian Eulerian grid. In fluid–structure interaction problems, however, it is often advantageous to employ stretched or curvilinear Eulerian grids to improve computational efficiency and avoid unnecessary refinement in regions far from the object of interest. In this dissertation, two types of non-uniform Eulerian grids are employed. For the paddle gust generator, a non-uniform Cartesian grid is used to cluster grid points near the walls and accurately resolve the boundary layer. For the adaptive nozzle gust generator, a curvilinear grid is adopted to concentrate grid points around the nozzle's upper contour and within its range of motion. Consequently, the Lagrangian markers must be embedded within non-uniform structured Eulerian grids, either Cartesian in the paddle case or curvilinear in the adaptive nozzle case. In such cases, the direct use of IB kernels formulated for uniform grids becomes unsuitable, since they fail to preserve the

integral conservation of force and moment. To address these issues, modified formulations are employed, as presented in Secs. 4.2 and 4.3.

4.2 Reproducing Kernel Particle Method based IBM

The wind-tunnel geometry simulated in Publications IV and V requires a stretched grid that clusters grid points close to the walls in order to accurately capture the boundary layer. The grid is therefore *non-uniform Cartesian* as shown in Fig. 4.5. To account for grid non-uniformity, the Reproducing Kernel Particle Method is used to define the interpolation and spreading operators that satisfy the conservation properties in Eqs. (4.25) and (4.26), following the technique of Pinelli et al. [81]. The RKPM was originally proposed by Liu et al. [85] as a mesh-free approximation technique based on corrected kernel functions that enforce polynomial reproduction conditions. This guarantees that constants and polynomials up to a chosen order are exactly reproduced, which ensures both accuracy and consistency even on non-uniform point distributions. Owing to these characteristics, the RKPM offers a suitable framework for extending interpolation and spreading operators of the immersed boundary method to non-uniform Cartesian grids, as required for the paddle gust generator case. For the sake of writing the equations in a compact manner,

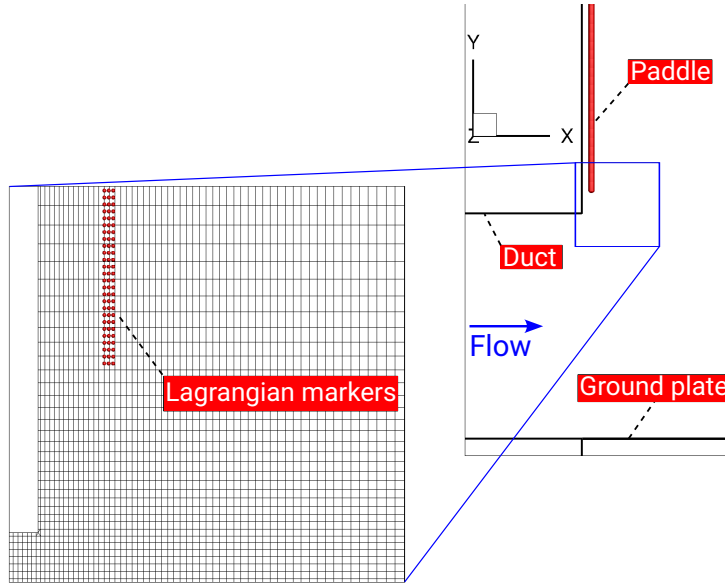


Fig. 4.5. Schematic showing the distribution of Lagrangian markers in the z -plane of the non-uniform Cartesian grid used for simulating the paddle gust generator.

in the following the shift between an Eulerian and a Lagrangian coordinate (e.g., $x - X$) shall be referred to by the subscript "s" (i.e., $x_s = x - X$, $y_s = y - Y$, and $z_s = z - Z$). In 1D, the non-uniformity of the grid is accounted for by defining a modified window function according to Pinelli et al. [81]:

$$\tilde{\delta}_d(x_s) = \sum_{c=1}^n b_c(X, d) x_s^{(c-1)} \delta_d(x_s) , \quad (4.27)$$

where $b_c(X, d)$ are the unknown polynomial coefficients and their number n is to be defined later. The unknown polynomial coefficients are determined by imposing a set of modified moment conditions:

$$\tilde{m}_{(i-1)}(X) = \int_{\Omega} x_s^{(i-1)} \tilde{\delta}_d(x_s) dx = \delta_{i1} , \quad (4.28)$$

where δ_{ij} is the Kronecker delta. It is worth noting that it can be shown, by induction, that the modified moment conditions are equivalent to the polynomial reproduction conditions:

$$\int_{\Omega} x^{(i-1)} \tilde{\delta}_d(x_s) dx = X^{(i-1)} . \quad (4.29)$$

By substituting Eq. (4.27) in Eq. (4.28) one can write the modified moment conditions in terms of the original moments:

$$\begin{aligned} \tilde{m}_{(i-1)}(X) &= \int_{\Omega} x_s^{(i-1)} \sum_{c=1}^n b_c(X, d) x_s^{(c-1)} \delta_d(x_s) dx \\ &= \sum_{c=1}^n b_c(X, d) \int_{\Omega} x_s^{(i+c-2)} \delta_d(x_s) dx = \sum_{c=1}^n b_c(X, d) m_{(i+c-2)}(X) = \delta_{i1} . \end{aligned} \quad (4.30)$$

The number of polynomial coefficients required in Eq. (4.27) (i.e., n) primarily depends on the number of reproducing moment conditions to be satisfied, the dimensionality of the problem, and the order of the underlying Eulerian spatial discretization scheme [81]. In 3D, the linear combinations of $\{1, x, y, z\}$ guarantee the integral conservation of the force and moment (i.e., Eq. (4.25) and Eq. (4.26)). Because the governing equations are discretized in space by a second-order accurate scheme (i.e., surface and volume integrals are approximated by the mid-point rule), and to achieve a sharp definition of the boundary, the modified window function employed reads [81]:

$$\begin{aligned} \tilde{\delta}_{\xi, \eta, \zeta}(x_s, y_s, z_s) &= \left[b_1 + b_2 x_s + b_3 y_s + b_4 z_s + \right. \\ &\quad b_5 (x_s y_s) + b_6 (y_s z_s) + b_7 (z_s x_s) + \\ &\quad \left. b_8 x_s^2 + b_9 y_s^2 + b_{10} z_s^2 \right] \delta_{\xi, \eta, \zeta}(x_s, y_s, z_s) , \end{aligned} \quad (4.31)$$

where the dependency of the coefficients can be expressed by $b_c = b_c(X, Y, Z, \xi, \eta, \zeta)$, $c = 1, \dots, 10$ and the original window function is defined as the product:

$$\delta_{\xi, \eta, \zeta}(x_s, y_s, z_s) = \delta_{\xi}(x_s, y_s, z_s) \delta_{\eta}(x_s, y_s, z_s) \delta_{\zeta}(x_s, y_s, z_s) , \quad (4.32)$$

with the subscripts ξ , η and ζ being the dilatation parameters in each Cartesian coordinate direction. Hence, 10 modified moment conditions need to be satisfied to evaluate the unknown polynomial coefficients. The continuous form of these conditions read:

$$\tilde{m}_{\mathbf{i}(r),\mathbf{j}(r),\mathbf{k}(r)}(X, Y, Z) = \int_{\Omega} x_s^{\mathbf{i}(r)} y_s^{\mathbf{j}(r)} z_s^{\mathbf{k}(r)} \sum_{c=1}^{n=10} b_c x_s^{\mathbf{i}(c)} y_s^{\mathbf{j}(c)} z_s^{\mathbf{k}(c)} \delta_{\xi,\eta,\zeta}(x_s, y_s, z_s) dx dy dz = \delta_{r1}, \quad (4.33)$$

where the indices r and c range from 1 to 10, and the vectors $\mathbf{i}$, $\mathbf{j}$, and $\mathbf{k}$ are defined such that the coordinate shifts (x_s , y_s and z_s) within the summation are raised to their respective powers according to Eq. (4.31):

$$\mathbf{i} = [0, 1, 0, 0, 1, 0, 1, 2, 0, 0], \quad (4.34)$$

$$\mathbf{j} = [0, 0, 1, 0, 1, 1, 0, 0, 2, 0], \quad (4.35)$$

$$\mathbf{k} = [0, 0, 0, 1, 0, 1, 1, 0, 0, 2]. \quad (4.36)$$

Further simplification yields:

$$\begin{aligned} \tilde{m}_{\mathbf{i}(r),\mathbf{j}(r),\mathbf{k}(r)}(X, Y, Z) &= \sum_{c=1}^{n=10} b_c \int_{\Omega} x_s^{\mathbf{i}(r)+\mathbf{i}(c)} y_s^{\mathbf{j}(r)+\mathbf{j}(c)} z_s^{\mathbf{k}(r)+\mathbf{k}(c)} \delta_{\xi,\eta,\zeta}(x_s, y_s, z_s) dx dy dz \\ &= \sum_{c=1}^{n=10} b_c m_{\mathbf{i}(r)+\mathbf{i}(c),\mathbf{j}(r)+\mathbf{j}(c),\mathbf{k}(r)+\mathbf{k}(c)}(X, Y, Z) = \delta_{r1}. \end{aligned} \quad (4.37)$$

Letting r and c refer to the row and column number, respectively, the resulting symmetric linear system of equations that needs to be solved for each Lagrangian location reads:

$$\begin{bmatrix} \tilde{m}_{0,0,0} \\ \tilde{m}_{1,0,0} \\ \tilde{m}_{0,1,0} \\ \tilde{m}_{0,0,1} \\ \tilde{m}_{1,1,0} \\ \tilde{m}_{0,1,1} \\ \tilde{m}_{1,0,1} \\ \tilde{m}_{2,0,0} \\ \tilde{m}_{0,2,0} \\ \tilde{m}_{0,0,2} \end{bmatrix} = \begin{bmatrix} m_{0,0,0} & m_{1,0,0} & m_{0,1,0} & m_{0,0,1} & m_{1,1,0} & m_{0,1,1} & m_{1,0,1} & m_{2,0,0} & m_{0,2,0} & m_{0,0,2} \\ & m_{2,0,0} & m_{1,1,0} & m_{1,0,1} & m_{2,1,0} & m_{1,1,1} & m_{2,0,1} & m_{3,0,0} & m_{1,2,0} & m_{1,0,2} \\ & & m_{0,2,0} & m_{0,1,1} & m_{1,2,0} & m_{0,2,1} & m_{1,1,1} & m_{2,1,0} & m_{0,3,0} & m_{0,1,2} \\ & & & m_{0,0,2} & m_{1,1,1} & m_{0,1,2} & m_{1,0,2} & m_{2,0,1} & m_{0,2,1} & m_{0,0,3} \\ & & & & m_{2,2,0} & m_{1,2,1} & m_{2,1,1} & m_{3,1,0} & m_{1,3,0} & m_{1,1,2} \\ & & & & & m_{0,2,2} & m_{1,1,2} & m_{2,1,1} & m_{0,3,1} & m_{0,1,3} \\ & & & & & & m_{2,0,2} & m_{3,0,1} & m_{1,2,1} & m_{1,0,3} \\ & & & & & & & m_{4,0,0} & m_{2,2,0} & m_{2,0,2} \\ & & & & & & & & m_{0,4,0} & m_{0,2,2} \\ & & & & & & & & & m_{0,0,4} \end{bmatrix} \begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ b_4 \\ b_5 \\ b_6 \\ b_7 \\ b_8 \\ b_9 \\ b_{10} \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}. \quad (4.38)$$

The entries of the symmetric matrix are numerically evaluated for each Lagrangian marker $\mathbf{X}_l = (X_l, Y_l, Z_l)$ by transforming the integral in Eq. (4.37) to a summation over the Eulerian cells falling within the cage $\Omega_l(\xi_l, \eta_l, \zeta_l)$. First, the closest cell center $\mathbf{x}_{\hat{e}} = \mathbf{x}_{\hat{i},\hat{j},\hat{k}} = (x_{\hat{i},\hat{j},\hat{k}}, y_{\hat{i},\hat{j},\hat{k}}, z_{\hat{i},\hat{j},\hat{k}})$ is sought and two of its direct neighbors (in a structured grid) in each direction are identified (i.e., $N_{E,l} = \{\mathbf{x}_{\hat{i}+ne,\hat{j}+ne,\hat{k}+ne}\}$ for $ne = -1, 0, 1$). Maximum and

minimum local grid spacings in each direction are then evaluated according to Pinelli et al. [81]:

$$\begin{aligned}
h_x^+(\mathbf{X}_l) &= \max\{|x_{i,j,k} - x_{i-1,j,k}|\}, x_{i,j,k}, x_{i-1,j,k} \in N_{E,l}, \\
h_x^-(\mathbf{X}_l) &= \min\{|x_{i,j,k} - x_{i-1,j,k}|\}, x_{i,j,k}, x_{i-1,j,k} \in N_{E,l}, \\
h_y^+(\mathbf{X}_l) &= \max\{|y_{i,j,k} - y_{i,j-1,k}|\}, y_{i,j,k}, y_{i,j-1,k} \in N_{E,l}, \\
h_y^-(\mathbf{X}_l) &= \min\{|y_{i,j,k} - y_{i,j-1,k}|\}, y_{i,j,k}, y_{i,j-1,k} \in N_{E,l}, \\
h_z^+(\mathbf{X}_l) &= \max\{|z_{i,j,k} - z_{i,j,k-1}|\}, z_{i,j,k}, z_{i,j,k-1} \in N_{E,l}, \\
h_z^-(\mathbf{X}_l) &= \min\{|z_{i,j,k} - z_{i,j,k-1}|\}, z_{i,j,k}, z_{i,j,k-1} \in N_{E,l},
\end{aligned} \tag{4.39}$$

and the dilation parameters depending on the local grid spacing are defined as:

$$\xi_l = \frac{5}{6} h_x^+(\mathbf{X}_l) + \frac{1}{6} h_x^-(\mathbf{X}_l), \tag{4.40}$$

$$\eta_l = \frac{5}{6} h_y^+(\mathbf{X}_l) + \frac{1}{6} h_y^-(\mathbf{X}_l), \tag{4.41}$$

$$\zeta_l = \frac{5}{6} h_z^+(\mathbf{X}_l) + \frac{1}{6} h_z^-(\mathbf{X}_l). \tag{4.42}$$

The discrete counterpart of the matrix entries can now be written as:

$$\begin{aligned}
m_{i,j,k}(X_l, Y_l, Z_l) &= \sum_{e \in N_{E,l}} (x_e - X_l)^i (y_e - Y_l)^j (z_e - Z_l)^k \\
&\quad \delta_{\xi_l, \eta_l, \zeta_l}(x_e - X_l, y_e - Y_l, z_e - Z_l) \Delta v_e.
\end{aligned} \tag{4.43}$$

In summary, the RKPM-based IBM modifies the kernel used in the interpolation and spreading steps of the standard direct-forcing scheme. Instead of employing a regularized delta function designed for uniform Cartesian grids, locally corrected kernels satisfying polynomial reproduction conditions are used, ensuring force and torque conservation on non-uniform meshes. This extension maintains the simplicity of the original method while preserving the accuracy on stretched grids.

4.3 Moving Least Squares based IBM

The numerical model developed to simulate the adaptive nozzle gust generator employs a *curvilinear stretched grid* to discretize the nozzle region (as shown in Fig. 4.6a and detailed in Section 4 of Publication VI). This grid configuration clusters grid points near the walls, enabling an accurate resolution of the boundary layer, while the curvilinear layout prevents unnecessary refinement in regions of lesser importance, resulting in substantial computational savings. Since the nozzle's upper contour moves between two side walls (see Fig. 4.6b), Lagrangian points must inevitably be positioned near the boundaries of the computational domain to ensure a perfect seal at the corners. This proximity limits the applicability of kernels with symmetric support for these specific Lagrangian

markers, necessitating the use of *one-sided kernels* that account for the boundaries of the computational domain.

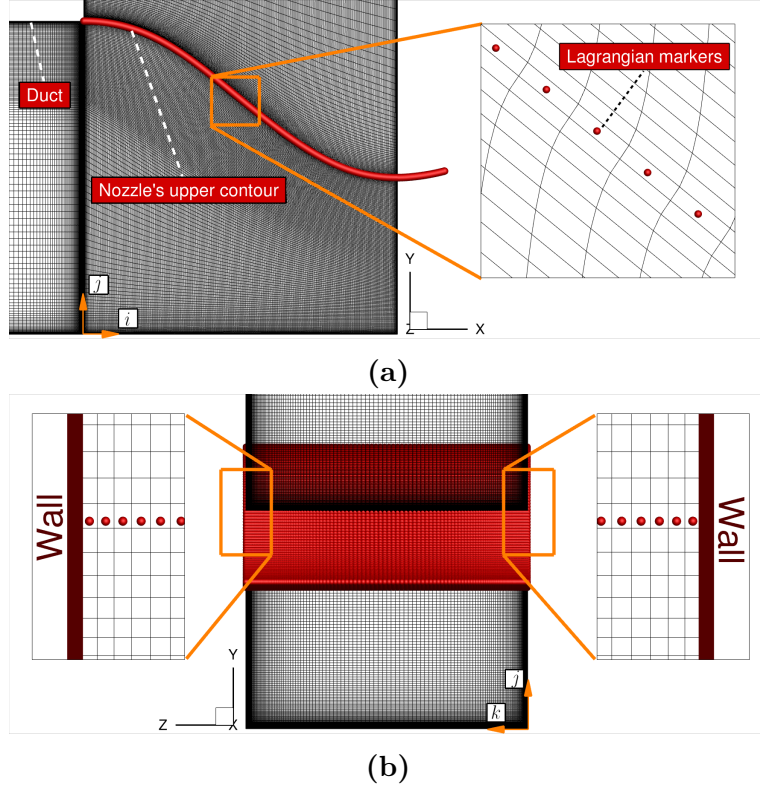


Fig. 4.6. Schematic showing the distribution of Lagrangian markers (red spheres) (a) in the z -plane of the curvilinear grid used for simulating the adaptive nozzle gust generator, and (b) near the wall boundaries in an i -plane of the grid within the nozzle region.

Remedies to the two aforementioned deviations from the classical IBM (i.e., markers on curvilinear stretched grids and markers with one-sided kernels) have been previously proposed in the literature. This work incorporates the moving least-squares version of the direct forcing immersed boundary method (MLS-IBM) introduced by Vanella and Balaras [86], beside its recent one-sided extension proposed by Bale et al. [87]. A key advantage of the MLS method is that it does not impose a specific arrangement of Eulerian interpolation points around a Lagrangian marker (e.g., uniform distribution). This characteristic makes the MLS approach effective for handling immersed boundaries on curvilinear grids. The work of Bale et al. [87] extends the MLS technique to construct one-sided (non-symmetric) IB kernels from standard two-sided (symmetric) ones. This advancement enables velocity interpolation and force spreading exclusively on one side of the fluid–solid interface. This specific ability is particularly beneficial for avoiding spurious flows within enclosed areas and for facilitating the handling of immersed structures near the boundaries of the computational domain.

The following provides a summary of the MLS method, along with a new proposition for its application to curvilinear block-structured grids. A detailed explanation of how the general framework is applied to different Lagrangian markers is also presented.

4.3.1 Standard MLS Formulation

The basic idea of the moving least-squares version of the direct forcing immersed boundary method is to compute interpolation weights that transfer a quantity to a Lagrangian marker from arbitrarily distributed Eulerian cell centers in its vicinity by solving a weighted least-squares optimization problem. The MLS method approximates the value of the Eulerian variable g at the discrete location of the l th Lagrangian marker by:

$$g_l = \sum_{i=1}^m p_i(\mathbf{X}_l) c_i^g(\mathbf{X}_l) = \mathbf{p}(\mathbf{X}_l)^\top \mathbf{c}^g(\mathbf{X}_l) . \quad (4.44)$$

Here, $\mathbf{p}(\mathbf{X}_l) \in \mathbb{R}^m$ is the basis function vector evaluated at the Lagrangian location $\mathbf{X}_l$ and $\mathbf{c}^g(\mathbf{X}_l) \in \mathbb{R}^m$ is the vector of unknown coefficients depending on the Eulerian quantity g_e . For a second-order accurate three-dimensional spatial discretization scheme, it is sufficient to define $\mathbf{p}$ as a linear basis function vector (i.e., $m = 4$ and $\mathbf{p}(\mathbf{x}) = [1 \ x \ y \ z]$). The unknown coefficient vector $\mathbf{c}^g(\mathbf{X}_l)$ is found by minimizing the weighted L^2 -norm of the error function:

$$\min_{\mathbf{c}^g(\mathbf{X}_l)} J_l = \frac{1}{2} \sum_{e \in N_{E,l}} \phi_{e,l} (\mathbf{p}(\mathbf{x}_e)^\top \mathbf{c}^g(\mathbf{X}_l) - g_e)^2 , \quad (4.45)$$

where ϕ denotes the MLS weighting function and $\phi_{e,l} = \phi(\mathbf{x}_e - \mathbf{X}_l)$ represents the discrete interpolation weight for the e th Eulerian cell. This discrete weight is equivalent to the product of the kernel function normalized by the dilation coefficient and the volume of the fluid cell Δv_e (i.e., $\delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e$ appearing in Eq. (4.14)). Equation (4.45) can also be written in a matrix form as:

$$\min_{\mathbf{c}^g(\mathbf{X}_l)} J_l = \frac{1}{2} (\mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_l) - \mathbf{g})^\top \text{diag}(\boldsymbol{\phi}_l) (\mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_l) - \mathbf{g}) , \quad (4.46)$$

where the entries of the polynomial matrix $\mathcal{A} \in \mathbb{R}^{m \times N_{E,l}}$ are the values of the basis functions at the Eulerian interpolation stencil (i.e., $\mathcal{A}_{ij} = p_i(\mathbf{x}_j)$, with $i = 1, \dots, m$, $j = 1, \dots, N_{E,l}$). The minimization of the error J_l with respect to the vector of unknown coefficients (i.e., $\frac{\partial J_l}{\partial \mathbf{c}^g(\mathbf{X}_l)} = 0$) yields (see Appendix A.1 in Publication VI for details):

$$\mathbf{c}^g(\mathbf{X}_l) = \mathcal{G}^{-1} \mathcal{B} \mathbf{g} , \quad (4.47)$$

where $\mathcal{G} = \mathcal{A} \text{diag}(\boldsymbol{\phi}_l) \mathcal{A}^\top$ is the positive-definite Gram matrix and $\mathcal{B} = \mathcal{A} \text{diag}(\boldsymbol{\phi}_l)$. Substituting the solution of $\mathbf{c}^g(\mathbf{X}_l)$ in Eq. (4.44) yields:

$$g_l = \mathbf{p}(\mathbf{X}_l)^\top \mathcal{G}^{-1} \mathcal{B} \mathbf{g} = \tilde{\boldsymbol{\phi}}_l^\top \mathbf{g} = \sum_{e \in N_{E,l}} \tilde{\phi}_{e,l} g_e , \quad (4.48)$$

where $\tilde{\phi}_l \in \mathbb{R}^{N_{E,l}}$ denotes the MLS kernel (or generating) function vector given by:

$$\tilde{\phi}_l = \mathcal{B}^\top \mathcal{G}^{-1} \mathbf{p}(\mathbf{X}_l) . \quad (4.49)$$

Unlike the expression given in Eq. (4.44), where the coefficient vector (i.e., $\mathbf{c}^g(\mathbf{X}_l)$) is dependent on both the Lagrangian marker's position and the Eulerian variable being interpolated, the form in Eq. (4.48) depends only on the coordinates of the Lagrangian marker. In the context of the immersed boundary method, it is more practical to directly use the latter form as this enables the use of a single kernel function for the interpolation of velocity components and the spreading of the forces.

It is worth mentioning that, although not explicitly imposed, the generating functions evaluated by the MLS method satisfy the discrete moment conditions (i.e., the polynomial reproduction conditions):

$$\sum_{e \in N_{E,l}} p_i(\mathbf{x}_e) \tilde{\phi}_{e,l} = p_i(\mathbf{X}_l), \quad \forall i = 1, \dots, m, \quad (4.50)$$

with the matrix form written as:

$$\mathcal{A} \tilde{\phi}_l = \mathbf{p}(\mathbf{X}_l) . \quad (4.51)$$

A proof based on the Backus-Gilbert formulation of the moving least-squares method can be found in Appendix A.2 of Publication VI.

4.3.2 New Proposition for Curvilinear Block-Structured Grids

In diffuse-interface IB methods, the dilation parameter (appearing in Eq. (4.13)) defines the interpolation window Ω_l and, consequently, the number of Eulerian cells included in the stencil of the Lagrangian marker ($N_{E,l}$). In addition, this parameter plays a role in computing the initial interpolation weights ($\phi_{e,l}$) required for the MLS procedure.

In most classical IBM applications, the structure of interest is immersed in a Cartesian uniform grid. In such cases, the dilation parameter for each coordinate direction is straightforward, and it is equal to the uniform grid spacing along that direction, as described in Section 4.1.2. In non-uniform grids, however, local dilation parameters are estimated in the literature for each Lagrangian marker using a criterion that considers the nearest Eulerian cell ($\mathbf{x}_e$) and its immediate neighbors (see, e.g., [81, 88, 89]). Then, based on the evaluated parameters, the set of Eulerian cell centers within the interpolation window is sought. However, some Eulerian centroids may lie at the boundary of the interpolation window, resulting in near-zero weight values, which can render the moment matrix ($\mathcal{G}$ in Eq. (4.47)) singular. In addition, this approach introduces an extra computational overhead due to the need for a larger search radius around the Lagrangian marker. That would also result in systems of varying sizes for different Lagrangian markers, requiring dynamic memory allocation at each time step (for moving immersed boundaries) for the vectors and matrices needed for the MLS method. The issue becomes more pronounced when solving the problem in parallel, particularly when the interpolation window of a

Lagrangian marker, located on a connectivity patch between two parallel blocks of a block-structured grid, extends to cover multiple ghost cells. In second-order accurate finite-volume codes, however, only one ghost cell is typically employed.

The proposition made in Publication VI makes the determination of the interpolation weights independent of the dilation parameter. Consequently, there is no need to estimate the dilation parameter, and the risk of obtaining zero interpolation weights is eliminated. The interpolation window is also fixed to encompass the exact number of Eulerian cells that the kernel φ is intended to cover. In other words, a n -point kernel incorporates n Eulerian cell centers. This is achieved by evaluating the interpolation weights with respect to the computational space (explained in Section 4.1.1) instead of the physical space as:

$$\phi_{e,l}^c = \frac{1}{\Delta v_{\hat{e}}} \varphi(\xi_e - \xi_l) \varphi(\eta_e - \eta_l) \varphi(\zeta_e - \zeta_l) \Delta v_e, \quad (4.52)$$

where the superscript c indicates that the value is computed with respect to the computational space and $\Delta v_{\hat{e}}$ is the volume of the closest Eulerian cell. Note that the normalized parameter r has been replaced by the difference between the Eulerian and the Lagrangian computational coordinates, which are already normalized. For a uniform grid, $\phi_{e,l}^c$ will immediately satisfy the moment conditions in Eqs. (4.23) and (4.24) provided that the full support of the kernel φ , which satisfies these conditions, is used. For a non-uniform distribution of Eulerian cells around a Lagrangian marker, the kernel's weights are evaluated without accounting for the non-uniformity of the grid. However, they are scaled by the ratio $\frac{\Delta v_e}{\Delta v_{\hat{e}}}$ for each cell center. This scaling causes $\phi_{e,l}^c$ to deviate from the moment conditions as $\frac{\Delta v_e}{\Delta v_{\hat{e}}} \neq 1$ for $e \neq \hat{e}$.

A thorough validation study of the proposed immersed boundary approach on a block-structured curvilinear grid is performed in Appendix B of Publication VI based on the flow past a stationary and an oscillating cylinder.

4.3.3 General Framework

The MLS procedure detailed in Section 4.3.1 is used in conjunction with the approach proposed in Section 4.3.2. If a Lagrangian marker is far from the boundaries of the computational domain, the 3-point kernel proposed by Roma et al. [80] is applied in all local directions leading in total to 27 points. Contrarily, when a Lagrangian marker is near the domain boundary, the 5-point kernel by Bao et al. [84] is employed only along the constrained directions. Figure 4.7 depicts the selection of kernels based on the marker's proximity to the boundaries of the computational domain. This selection is motivated by the findings of Bale et al. [87], which demonstrate that using a wider kernel helps to prevent the Gram matrix in the one-sided MLS from becoming ill-conditioned due to insufficient support.

The general procedure for each Lagrangian marker can be summarized as follows:

1. The nearest Eulerian cell center in the computational space and its direct neighbors in each local block direction (i.e., i , j and k) are identified.

2. A check is carried out whether any of the Eulerian neighbors belong to the boundaries of the computational domain and the affected local directions are constrained accordingly to exclude those neighbors (e.g., wall). If a local direction is constrained, it is extended from its other side to include one extra Eulerian interpolation point.
3. Based on the absolute difference between the Lagrangian computational coordinates and its Eulerian counterparts along each local block direction, a kernel function is assigned. For instance, if $\max(|\xi_e - \xi_l|) \leq 1.5 \ \forall e \in N_{E,l}$, the 3-point kernel is employed. This will be the case (in all local block directions) for all Lagrangian markers far from the boundaries of the computational domain. However, if $\max(|\xi_e - \xi_l|) > 1.5$, this indicates that the interpolation window was extended in step 2 and the 5-point kernel is assigned to the i th direction.
4. The MLS weights are evaluated based on Eq. (4.52). These weights do not satisfy the discrete moment conditions due to the curvilinear Eulerian grid, the reduced (one-sided) kernel support, or both.
5. The MLS kernel function vector is found using Eq. (4.49). These modified weights satisfy the polynomial reproduction conditions needed for the conservation of force and torque.

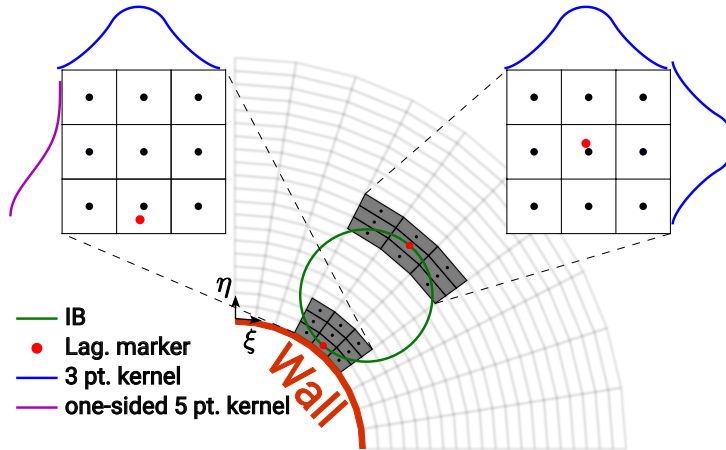


Fig. 4.7. Schematic sketch showing two Lagrangian markers within a curvilinear block-structured grid. The sketch highlights the interpolation kernels employed along each local coordinate direction for markers near and far from the boundary of the computational domain.

4.4 Main Results of Publication IV: High-Fidelity Numerical Model of the Wind-Tunnel Gust Generator – The Paddle

The objective of the test case, presented in Publication IV, is to create a numerical counterpart of the experimental wind gust generator of Wood et al. [52], denoted the paddle. The numerical model is designed to accurately reproduce the wind-gust generation observed in the wind tunnel with high fidelity. By resolving all flow-field variables across the computational domain, it enables a deeper analysis of the complex paddle-induced flow and establishes a foundation for coupled FSI studies aimed at investigating the effect of wind-gust loading on flexible structures. Figure 4.8 shows a schematic of the wind-tunnel test section with the eleven monitoring points located in the symmetry plane, where streamwise and wall-normal velocities were recorded using laser-Doppler anemometry (LDA) in Wood et al. [52]. The numerical-experimental comparison is carried out at seven streamwise positions in the range $0 \leq x/w \leq 0.6$ (width of the nozzle's outlet $w = 0.5$ m), which corresponds to the region where a wind-tunnel model is typically placed.

To reproduce the experimental conditions numerically, the computational setup is designed to mimic the essential geometric and flow characteristics of the wind-tunnel environment while maintaining computational efficiency. The following summarizes the main modeling choices and boundary conditions adopted in the numerical reconstruction of the paddle gust generator. The turbulent flow is resolved using the LES technique on a block-structured Cartesian grid refined near the walls to capture the thin boundary layers. The complex wind-tunnel geometry depicted in Fig. 4.1a is simplified, where the nozzle is replaced by a straight duct and its contraction ratio is neglected. The paddle geometry is simplified to a rectangular parallelepiped, and its Lagrangian markers are distributed according to the minimum local Eulerian grid spacing to ensure adequate coupling between the two grids. At the duct inlet, a uniform velocity distribution is imposed so that the center velocity at the duct outlet matches that of the experimental setup. No-slip and impermeability conditions are applied to all solid boundaries (duct, nozzle, ground plate, and far-field), and a convective boundary condition is specified at the outlet.

The numerical model includes two prescribed velocity boundary conditions that govern the flow field during the gust event. The first is applied to the Lagrangian markers defining the paddle. Unlike the prescribed velocity or source-term formulations, in which the gust shape inside the computational domain is imposed by an analytical function (e.g., ECG), here the paddle's kinematics dictate the shape of the generated gust. Among the five motion patterns investigated by Wood et al. [52], the symmetric bell-shaped motion shown in Fig. 4.9 is adopted, as it closely resembles the standard 1-cosine gust shape. The second boundary condition is applied at the inlet. In the experiments by Wood et al. [52], a temporary reduction of the streamwise velocity below the free-stream level was observed in the wake of the gust. This phenomenon is attributed to pressure losses caused by the sudden partial blockage of the nozzle's outlet. This effect, combined with the fact that the blower is not operated in a mode that compensates for these increased pressure losses, results in a reduced mass flow rate through the wind tunnel and, consequently, in an

undershoot of the free-stream velocity. To reproduce this undershoot in the simulation, a transient inlet velocity is prescribed, capturing the temporal deficit in the incoming flow. Details about the procedure to determine the shape of this velocity deficit and its final form can be found in Publication IV. Furthermore, a detailed description of the computational setup and a comprehensive presentation of the results are provided in Secs. 5 and 6 of Publication IV, respectively. In the following, the main results are highlighted, with emphasis on the instantaneous flow field induced by the paddle and the comparison between ensemble-averaged experimental measurements and numerical predictions.

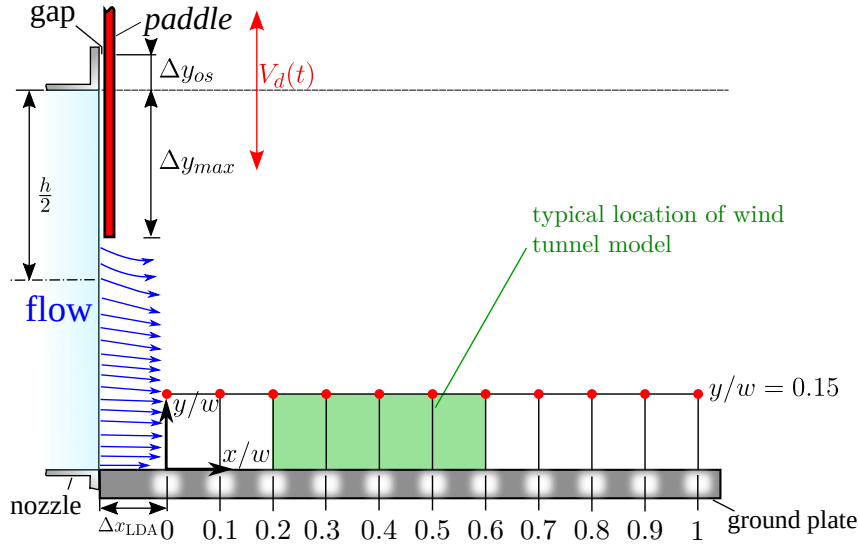


Fig. 4.8. Schematic of the paddle gust generator showing the monitoring points (in red) above the ground plate used for validation.

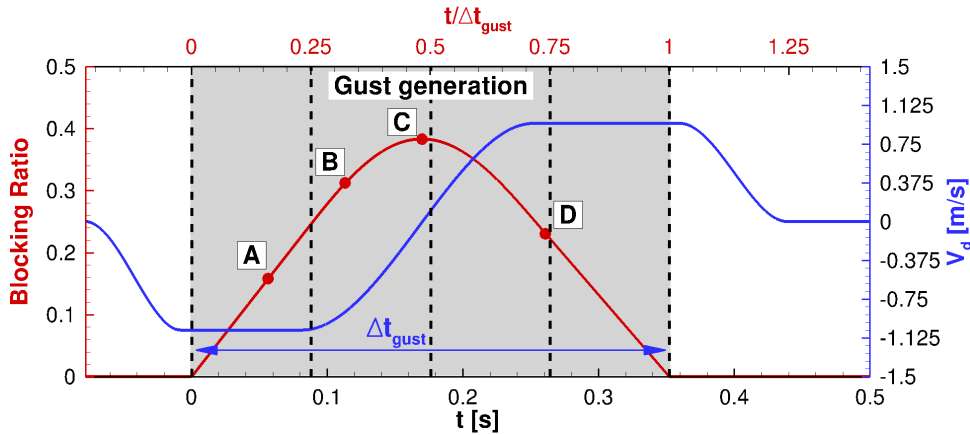


Fig. 4.9. Kinematic characteristics of the gust *type 1* paddle movement pattern. Nozzle's blocking ratio (red line) annotated with visualization time instants and paddle's vertical velocity (blue line) during the downward and upward strokes.

4.4.1 Flow Field Induced by the Paddle Wind Gust Generator

The initial results focus on the flow characteristics of the gust generated by the moving paddle. Figure 4.10 shows the streamwise velocity component and the pressure field at the x - y symmetry plane at four characteristic instants in time. The monitoring points labeled 1 through 7 indicate the positions selected for comparison with the experimental measurements of Wood et al. [52]. At instant A ($t/\Delta t_{\text{gust}} = 0.16$), the paddle has moved downward from its idle position, blocking 15.8 % of the nozzle's outlet. The flow is deflected from its original path, leading to an acceleration of the streamwise velocity and the development of negative vertical velocities below the paddle. Flow separation at the paddle's lower edge produces a strong vortex with a pressure minimum at its center, located at the paddle's downstream side. At instant B ($t/\Delta t_{\text{gust}} = 0.32$), the downstroke continues leading to a higher streamwise velocity that extends to the ground plate and elevated pressure values propagating upstream into the duct. As the primary vortex grows in size and is convected downstream, a new shear layer forms behind the paddle's edge and follows its trajectory. At instant C ($t/\Delta t_{\text{gust}} = 0.48$), the paddle reaches its maximum blocking ratio of 38.3 %. In the reduced outlet region of the nozzle above the ground plate, high streamwise velocities are observed, characterizing the strong horizontal gust. At instant D ($t/\Delta t_{\text{gust}} = 0.74$), the paddle starts traveling back to its idle position. The large vortical structure impinges on the ground plate, causing the boundary layer to separate and lift off the surface. At this stage, the shear layer breaks up into smaller vortices, and the gust weakens, resulting in highly transient velocity fields. As the paddle clears the nozzle's outlet, the flow begins to recover toward its initial state.

The spanwise characteristics of the generated gust are evaluated based on the evolution of the streamwise velocity field at the x -plane $x/w = 0.2$ (see Fig. 4.11), which marks the beginning of the region typically used for FSI models. The paddle edges are projected onto this plane solely for visualization and as a spatial reference, since they are actually located upstream. Excluding about 20 % of the total width ($w = 0.5$ m) from both lateral boundaries, the flow field is nearly homogeneous in the spanwise direction. This agrees with the experimental findings of Wood et al. [52], who reported an almost constant streamwise velocity component of the generated gust across this spanwise range.

4.4.2 Comparison of Numerical and Experimental Results

The predicted data are compared with the laser Doppler anemometer (LDA) measurements of Wood et al. [52], focusing on the streamwise and wall-normal velocities at the aforementioned seven points (see Fig. 4.8). For clarity, the time histories are shifted by adding the product of a freely chosen time increment $\Delta t_{\text{inc}} = 0.15$ s and a spacing factor $a \in (0, \dots, 6)$. Because of the shear layer at the top surface of the jet exiting the nozzle, small vortical structures are irregularly formed and convected downstream. Consequently, the flow induced by the paddle depends on the instant at which it begins blocking the nozzle's outlet. To obtain statistically representative results, ensemble averaging is performed on the numerical results based on 25 different initial conditions and on the experimental measurements based on 25 realizations.

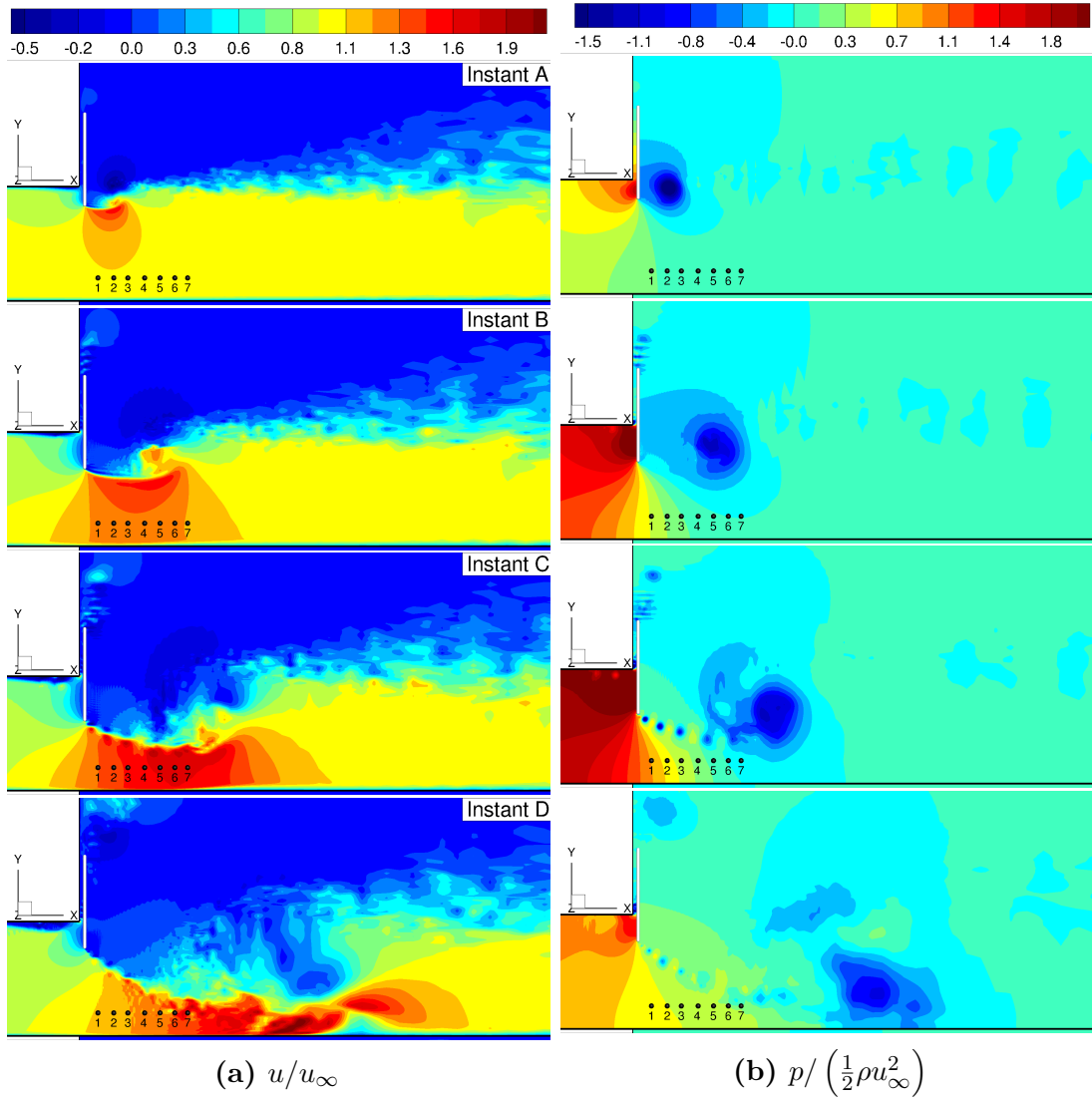


Fig. 4.10. Snapshots of the (a) streamwise velocity and the (b) pressure during the paddle's downward and upward strokes. The corresponding blocking ratios are: instant A: 15.8 %, instant B: 31.2 %, instant C: 38.3 %, instant D: 22 %.

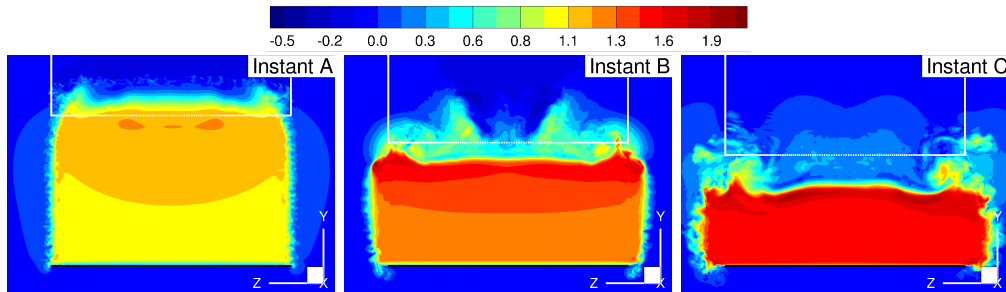


Fig. 4.11. Snapshots of the streamwise velocity u/u_∞ at the x -plane $x/w = 0.2$.

Focusing on the streamwise velocity component (see Fig. 4.12a), the time history at a single point exhibits a bell-shaped profile consistent with the paddle displacement curve. According to mass conservation, the flow beneath the paddle accelerates as soon as it begins blocking the nozzle outlet and continues until the paddle reaches its maximum displacement. When the paddle starts traveling back to its idle position, the streamwise velocity decelerates. The drop below the free-stream velocity arises from the prescribed transient inlet velocity, which models the reduced discharge of the wind-tunnel blower. The numerical predictions are in close agreement with the experimental data. Both results demonstrate an increase of the maximum streamwise velocity along the downstream direction. At the last three locations, the velocity signal exhibits a chaotic and turbulent character along the descending slope, caused by the free shear layer and vortices shedding from the paddle's lower edge. Increasing the number of realizations in the ensemble averaging is found to mitigate this phenomenon.

An analogous agreement between the experimental and the numerical results is observed for the wall-normal velocity component (see Fig. 4.12b). The vertical velocity is predominantly negative, indicating that the flow is directed toward the ground plate, consistent with the paddle's displacement curve. During the downstroke, the paddle deflects the jet downward, while during the upstroke this steering effect gradually weakens, allowing the jet to recover its original trajectory. Beyond the range $x/w \geq 0.3$, increasing positive vertical velocities are initially observed, caused by the counter-clockwise vortex forming at the paddle's lower edge. As in the streamwise direction, this region is characterized by strong fluctuations induced by vortices shed from the paddle's lower edge.

Compared to the streamwise velocity signals, the pressure time histories exhibit a contrasting trend, with peak values decreasing along the downstream direction (see Fig. 4.13). This behavior results from the partial blockage of the nozzle's outlet by the paddle, which causes a significant pressure rise in the upstream region. Further downstream, however, the pressure decreases with increasing distance from the nozzle (see the pressure distribution at instant C in Fig. 4.10b). The resulting negative pressure gradient drives the acceleration of the streamwise velocity in the downstream direction.

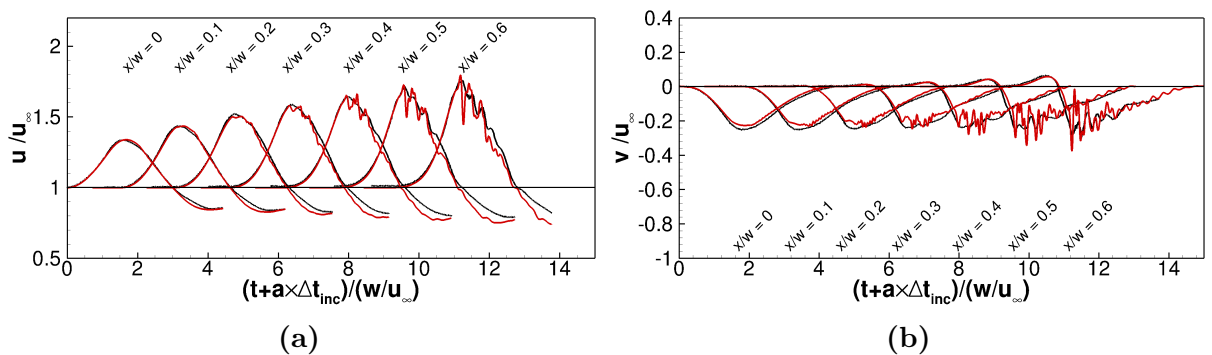


Fig. 4.12. Comparison between the ensemble-averaged experimental streamwise velocity u/u_∞ (black) by Wood et al. [52] and its numerical counterpart (red) of the gust generated by the paddle at the first seven points in Fig. 4.8.

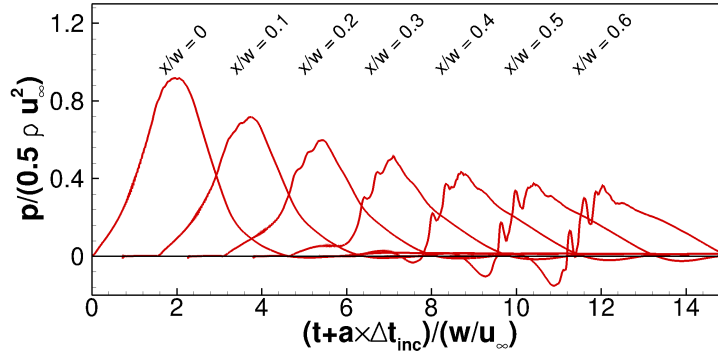


Fig. 4.13. Predicted ensemble-averaged pressure at the first seven points in Fig. 4.8.

4.5 Main Results of Publication V: Artificial Wind Gusts Impacting a Flexible T-Structure

The novel wind-tunnel gust generator proposed by Wood et al. [52] and numerically reproduced in Publication IV was successfully applied in a follow-up experimental study by Breuer and Neumann [90], in which synchronous high-speed measurements were conducted on a flexible T-structure subjected to wind-gust loading. The objective of Publication V is to establish a numerical counterpart of this experimental investigation on the T-structure. To this end, two numerical approaches for simulating fluid–structure interaction problems are combined, allowing the respective strengths of each method to be exploited. The first approach is the IB methodology introduced in Section 4.3 and employed in Publication IV to generate artificial wind gusts by means of a rigid paddle undergoing large prescribed displacements, which partially obstruct the wind-tunnel outlet. The second approach is a classical partitioned FSI methodology coupling a finite-volume fluid solver, formulated in an Arbitrary Lagrangian-Eulerian framework, with a finite-element solver for structural analysis, enabling the simulation of a thin, flexible T-shaped structure deforming under the dynamic loading of a wind gust.

The present computational setup, illustrated in Fig. 4.14, builds directly on the validated paddle gust generator described in Section 4.4. The same geometric configuration and gust-generation approach are employed, and the model is extended to include a flexible T-structure interacting with the generated gust. The T-structure is mounted in the symmetry plane of the test section at a distance of $d_s = 0.25$ m from the nozzle’s outlet. It is made of a thin brass sheet and consists of a rectangular rod supporting a shield-shaped part, forming a simplified aeroelastic benchmark geometry. Its geometric dimensions correspond to those of the experimental setup by Breuer and Neumann [90], while the properties of the material model, including the Young’s modulus and Rayleigh damping coefficients, are determined from the preliminary study detailed in Appendix A of Publication V. The computational grid of the fluid domain, initially Cartesian before structural deformation, is locally refined around the flexible structure and near solid boundaries to capture the development of thin boundary layers and accurately resolve the fluid–structure interaction. The structural domain (i.e., the T-structure) is discretized

using quadrilateral four-node shell elements and solved with the Carat++ finite-element solver [91]. A detailed description of the computational setup is provided in Section 3 of Publication V. For the subsequent visualization of the results, the Cartesian coordinate system shown in Fig. 4.14 is employed. Its origin lies in the symmetry plane at the clamping point of the T-structure on the ground plate, and its coordinates are normalized by the T-structure height h_s . The time t is nondimensionalized with the gust-generation period Δt_{gust} (see Fig. 4.9), and the reference point $t = 0$ corresponds to the instant at which the paddle begins to obstruct the nozzle's outlet. In the following, the main results of the publication are summarized, focusing on the flow field and the structural displacements with and without gust loading.

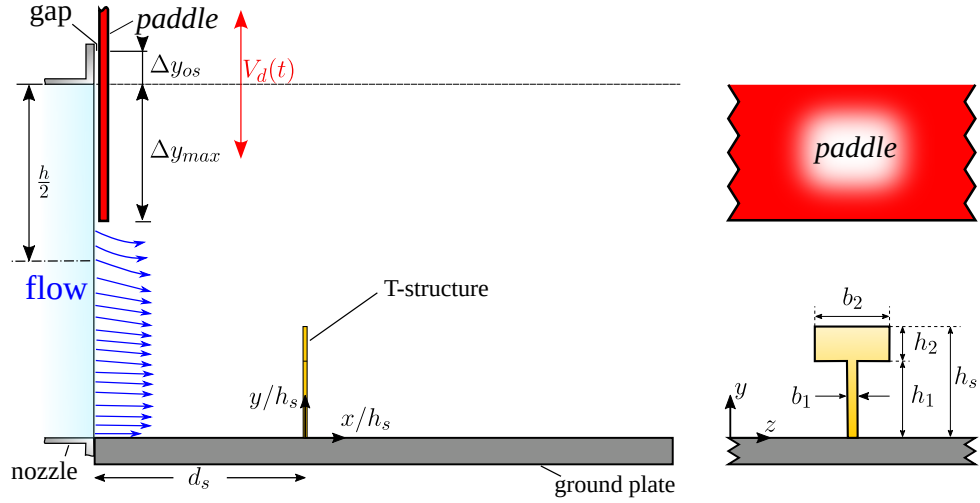


Fig. 4.14. Schematic of the paddle gust generator and the flexible T-structure mounted on the ground plate.

4.5.1 Flow Field and Structural Displacements without Gust

The time-resolved measurements in Breuer and Neumann [90] were repeated 104 times under identical wind-tunnel and gust-generator conditions, enabling phase-averaging of both flow and structural data. Data recorded before gust injection ($t/\Delta t_{\text{gust}} \leq 0$) capture the undisturbed flow, where T-structure deflections are minimal. This permits time-averaging of the numerical flow field for comparison with the phase-averaged experimental data. The field was averaged over 10^5 samples, corresponding to a time interval of 10 s and about 68 oscillations at the first bending eigenfrequency (determined in Appendix A of Publication V as $f_1^{\text{num}} = 6.826$ Hz), ensuring statistical convergence.

Figure 4.15 shows the averaged streamwise velocity contours and the corresponding streamlines in the geometric symmetry plane ($z = 0$) for both the measured and the predicted data. The flow separates at the sharp corners of the T-structure and forms a large recirculation zone. This zone, highlighted by the $\bar{u} = 0$ contour, is most pronounced behind the shield-shaped part and extends to about 1.25 times the total height h_s . By comparing the numerical and the experimental data, an overall very good agreement is found. However,

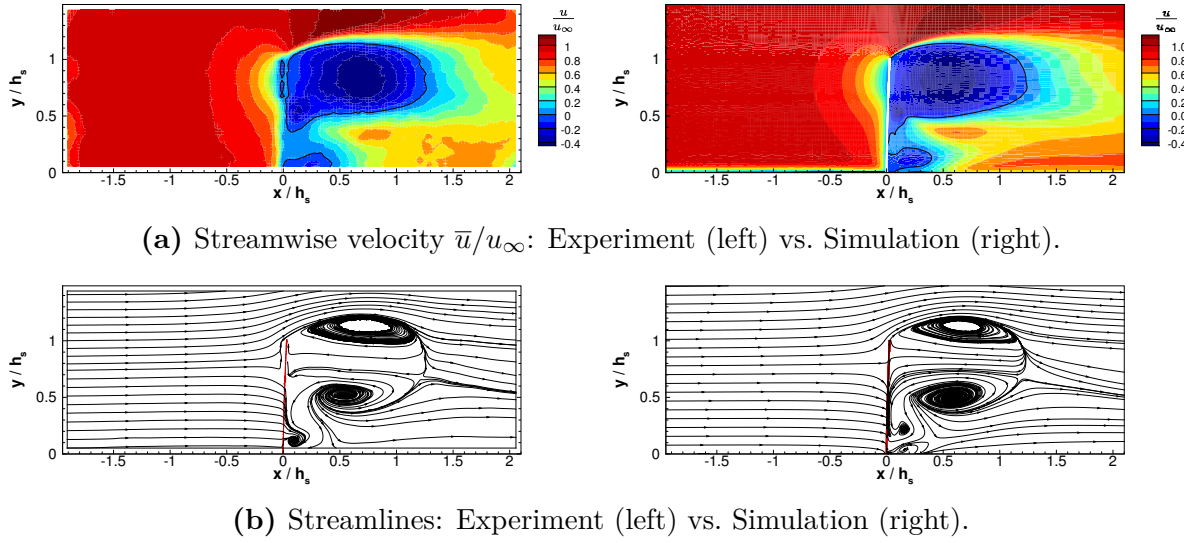


Fig. 4.15. Comparison between measured and predicted data of the averaged flow field in the symmetry plane. Experimental data taken from Breuer and Neumann [90].

two aspects merit further attention. First, reflections of the laser light used for particle-image velocimetry (PIV) prevent velocity measurements in the immediate vicinity of the ground plate, resulting in the white gap visible in all presented PIV figures. Second, the simulation statistics are better converged since the flow field was continuously averaged over 10 s with about 10^5 samples, whereas the experimental means are based on only 104 independent measurements. The relatively lower number of realizations in the latter case is attributed to the fact that the experiment was designed to capture instantaneous and statistically unsteady gust processes rather than time-averaged quantities. Consequently, small differences in the streamlines based on the averaged data are observed.

A quantitative comparison between the experimental and the numerical data is shown in Fig. 4.16. In Fig. 4.16a, the averaged streamwise velocity profiles are extracted in the symmetry plane at four streamwise positions ($x/h_s = -1/6, 1/6, 2/3$, and $4/3$) indicated by the dotted lines. Each profile is plotted at its actual position and the deformed T-structure is indicated by a red line for reference. Despite the minor deviations, the experimental and the numerical profiles agree closely. Figure 4.16b shows the averaged deformation of the T-structure in the symmetry plane, revealing bending caused by fluid loading and demonstrating good agreement between predictions and measurements. Because of the structure's small thickness, shear forces have only a marginal contribution to the resulting streamwise fluid force. In contrast, pressure forces dominate, driven by near-stagnation pressure ($c_p \approx 1$) over most of the front face and by the low-pressure recirculation zone with negative c_p at the back (not shown here). Nevertheless, the deflections remain small, with the center of the shield-shaped part of the T-structure deflecting by about $2.45\% h_s$ and the upper edge by about $3.2\% h_s$.

Figure 4.17a shows the time histories of the total displacement of the center point of the shield-shaped part in the flow without gust. The T-structure exhibits two oscillation modes, alternating irregularly and sometimes overlapping between a bending-dominated

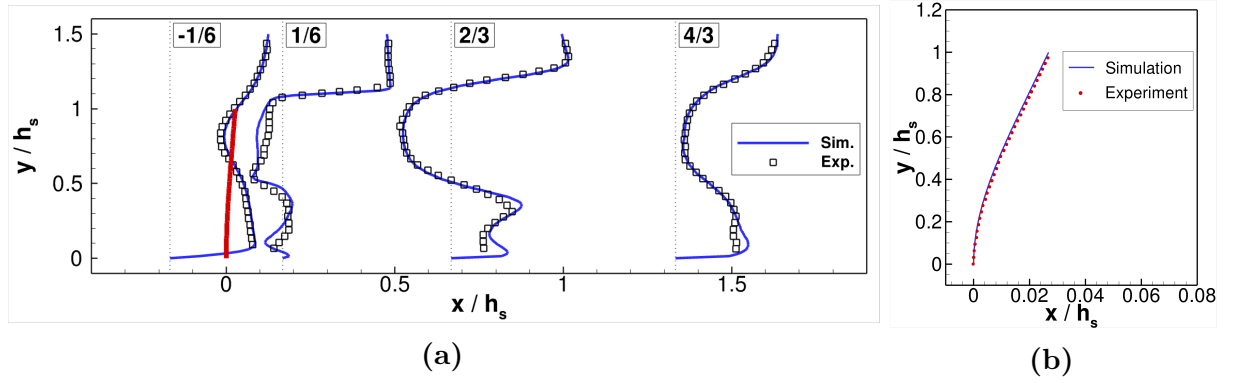


Fig. 4.16. Comparison of predicted and measured time-averaged: (a) Streamwise velocity at four positions in the flow field ($0.3 \bar{u}/u_\infty + x/h_s$) (T-structure in red) and (b) deformation of the T-structure in the symmetry plane. Experimental data by Breuer and Neumann [90].

mode with large amplitudes and a twisting mode with smaller amplitudes. Fast Fourier transforms of these deflections (Fig. 4.17b) reveal two dominant frequencies, $f = 6.68$ Hz and $f = 6.34$ Hz, both close to the natural bending frequency of 6.82 Hz. A detailed analysis of the predicted data shows that the higher-frequency peak corresponds to the bending-dominated mode, whereas the lower one indicates increased twisting influence. As a final observation, the measured oscillation amplitudes exceed those in the simulations, likely because inflow turbulence, which could not be prescribed without detailed turbulence data, is absent in the numerical model.

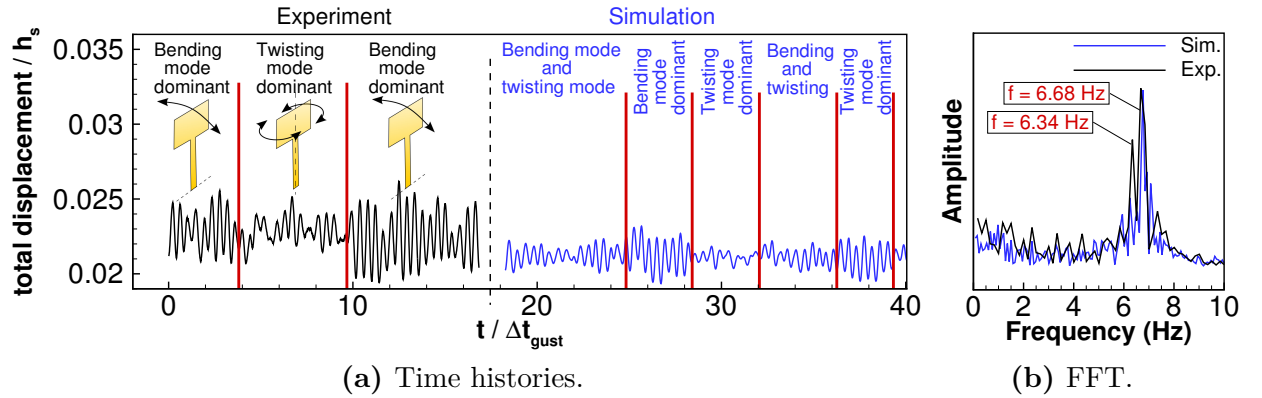


Fig. 4.17. Time histories and fast Fourier transformation of the total displacements of the center of the upper rectangular part of the T-structure. Predicted data and experimental data by Breuer and Neumann [90] for the case without gust.

4.5.2 Flow Field and Structural Displacements with Gust

The subsequent analysis examines the more complex case of gust impact. Unlike the quasi-steady incoming flow considered earlier, the gust produces a statistically unsteady

flow. Therefore, time averaging is no longer valid when an active gust generator is used and phase averaging is applied instead. In both the experiments and the numerical simulations, phase averaging is performed using the paddle position as the temporal reference. The experimental phase averaging of Breuer and Neumann [90] relied on 104 realizations, achieved by synchronizing the measurement devices and the gust generator with a single trigger signal. Due to the high computational cost of the coupled simulations, the numerical phase averaging is based on 32 predictions, starting from randomly chosen instants of the no-gust simulation.

The simulation results and measurements are compared at the time instants $t/\Delta t_{\text{gust}} = 0.25, 0.5, 0.75, 1.0$, and 2.5 (see Fig. 4.9 for the definition of instants), using the phase-averaged streamwise velocity shown in Fig. 4.18. At $t/\Delta t_{\text{gust}} = 0.25$, the developing gust slightly perturbs the upstream flow and the region above the T-structure, while the T-structure deflection remains minimal. Around $t/\Delta t_{\text{gust}} = 0.5$ the paddle arrives at its reversal point and the gust reaches its peak intensity, leading to a pronounced enlargement of the recirculation region behind the shield-shaped part and to the maximum structural deformation of about $0.1 h_s$ (reached shortly after at $t/\Delta t_{\text{gust}} = 0.6$). As the paddle withdraws (i.e., at $t/\Delta t_{\text{gust}} = 0.75$ and 1.0), the recirculation region contracts and the structure swings back toward its original shape. By $t/\Delta t_{\text{gust}} = 2.5$, the flow field and structural deflections have essentially returned to their pre-gust state.

Focusing on the structural deformations, Fig. 4.19 shows the phase-averaged time histories of the center-point total displacement from both experiments and simulations. Before the paddle enters the fluid stream ($t < 0$), minor flow-induced oscillations are visible, with slightly larger amplitudes in the measurements than in the predictions, consistent with the absence of inflow turbulence in the simulations (see Figs. 4.19a and 4.19b). After gust initiation, a delay of about $0.2 \Delta t_{\text{gust}}$ precedes the structural response, reflecting the distance between the nozzle's outlet and the T-structure representing the gust development time. The deflection then rises rapidly to a maximum of about $0.07 h_s$ near $t/\Delta t_{\text{gust}} \approx 0.6$, with cycle-to-cycle variations affecting the peak value. Experiments show slightly higher maxima, resulting in a marginally greater phase-averaged peak. During the upstroke, the structure swings back reaching its pre-gust state at $t/\Delta t_{\text{gust}} = 0.95$ as the gust effect diminishes. In the interval $0.9 \leq t/\Delta t_{\text{gust}} \leq 1.3$, the largest differences appear between the simulated and the measured phase-averaged displacements, where the simulation predicts smaller initial rebounds. This underprediction is mainly due to the absence of inflow turbulence in the computational framework, as confirmed by preliminary tests showing that background turbulence increases the response amplitude of the T-structure. Following the initial rebound, and owing to the structure's elasticity and inertia, it oscillates beyond the pre-gust position, reaching a minimum deformation of about $0.01 h_s$ around $t/\Delta t_{\text{gust}} \approx 1.3$ before gradually settling back to its equilibrium state. Overall, the experimental and numerical results agree well and both exhibit significant cycle-to-cycle variations, mainly linked to the structure's pre-deflected position at gust onset.

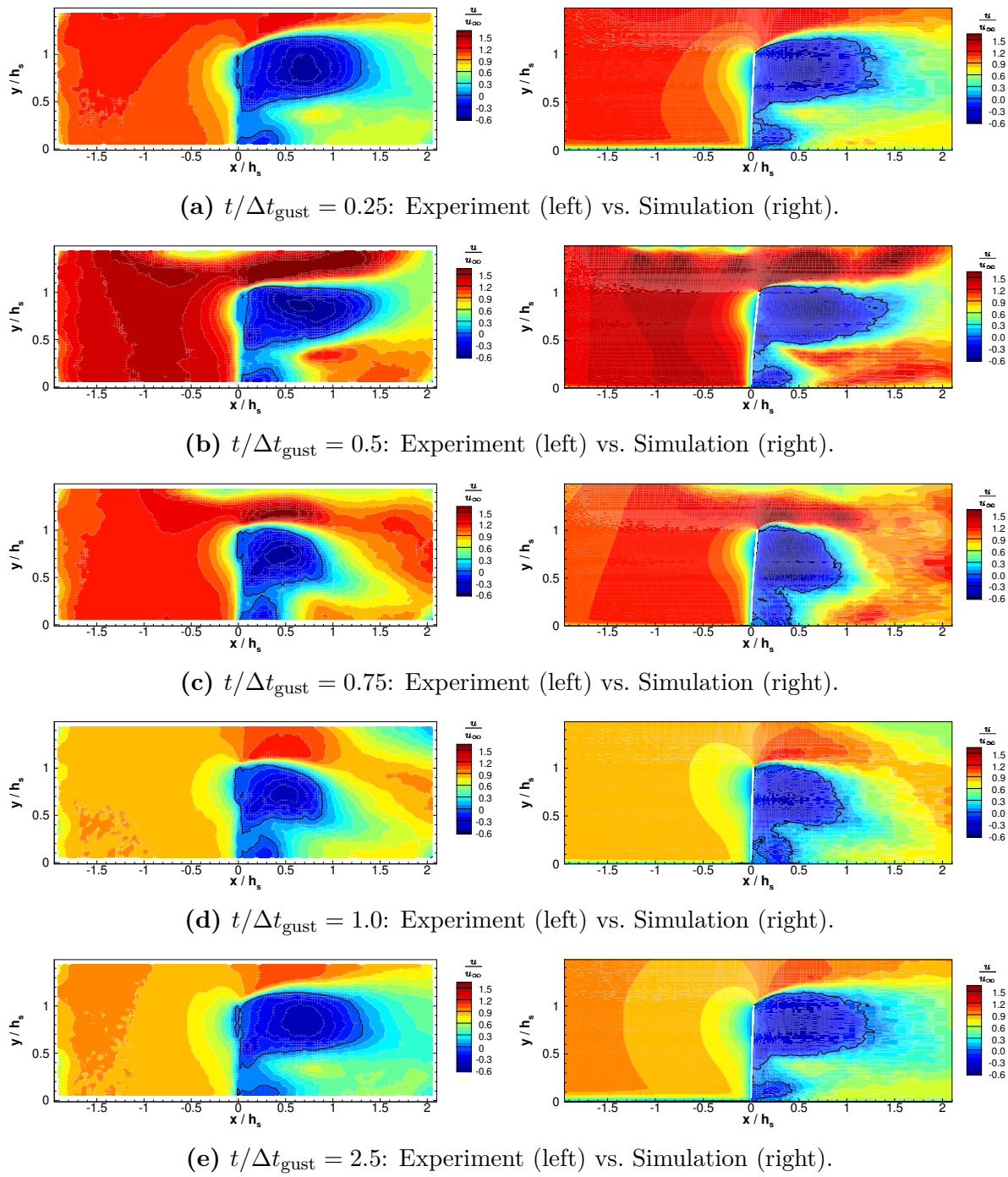


Fig. 4.18. Comparison between measured and predicted data of the phase-averaged streamwise velocity in the symmetry plane at different time instants of the gust encounter. Experimental data taken from Breuer and Neumann [90].

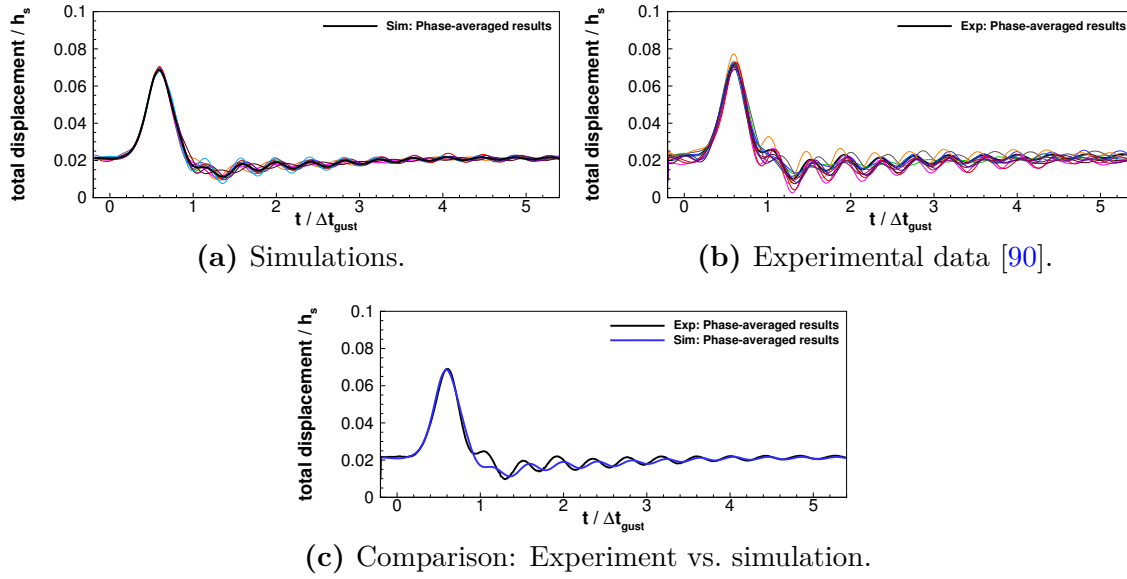


Fig. 4.19. Time histories of the total displacements of the center of the upper rectangular part of the T-structure: Phase-averaged data (black thick line) and 10 singular gusts (colored lines).

4.6 Main Results of Publication VI: High-Fidelity Numerical Model of the Wind-Tunnel Gust Generator – The Adaptive Nozzle

Building upon the gust-generation principle of the wind-tunnel gust generator, the *paddle* [52], Wood and Breuer [53] introduced a successor design referred to as the *adaptive nozzle*. This configuration features a fully rotatable upper contour, enabling a rapid reduction of the nozzle’s outlet area within a short time interval. The primary objective of the new concept was to reduce pressure losses caused by flow blockage while preserving the original horizontal trajectory of the flow along the streamwise direction. The experiments of Wood and Breuer [53] validated the improved design, demonstrating a smoother transition of the gusty flow into the test section and avoiding the abrupt cut-off observed with the paddle. The objective of Publication VI is to numerically reproduce the highly unsteady turbulent flow field generated by the adaptive nozzle. The numerical model aims to resolve the three-dimensional flow throughout the entire computational domain, thereby providing a detailed insight into the instantaneous gust characteristics. Beyond flow characterization, this study lays the foundation for future investigations on interactions between wind gusts generated by the adaptive nozzle and flexible structures. This stems from its ability to quantify the transient aerodynamic loads acting on wind-tunnel specimens, which remain difficult to obtain experimentally.

Figure 4.20 shows a schematic of the adaptive nozzle with its rotatable upper contour and the open test section of the wind tunnel. In the experimental work of Wood and Breuer [53], several measurement techniques were employed in the x - y symmetry plane

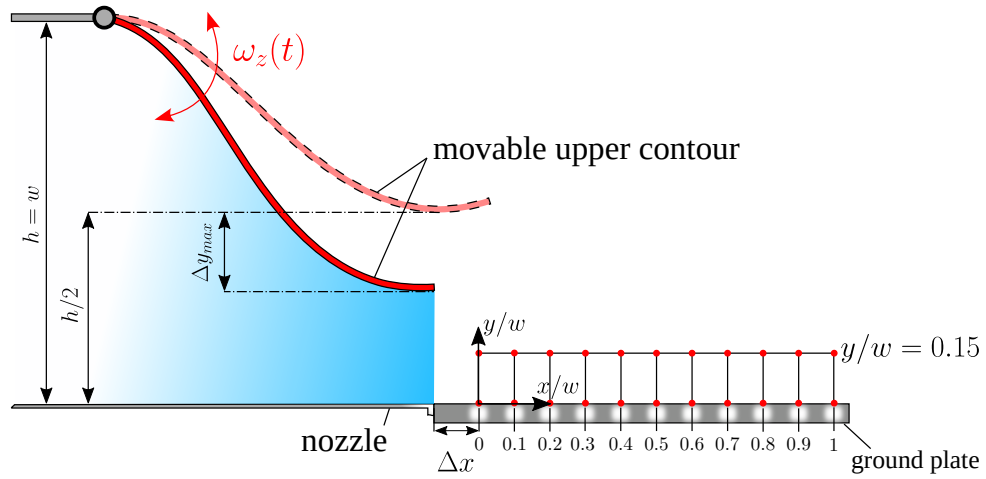


Fig. 4.20. Schematic of the adaptive nozzle gust generator showing the monitoring points (in red) above and on the ground plate used for validation.

of the test section to analyze the flow field during the gust event (see Section 2 of Publication VI). These included particle-image velocimetry (PIV), constant-temperature anemometry (CTA) for time-resolved velocity data at 11 monitoring points (at $y/w = 0.15$ in Fig. 4.20), and a pressure transducer for recording time histories of surface pressure signals (at $y/w = 0$ in Fig. 4.20). Together, these measurements formed the basis for validating the numerical simulations presented in Publication VI.

To mimic the experimental conditions, the computational setup is tailored to preserve the essential geometry and flow characteristics of the adaptive nozzle while maintaining computational efficiency. Unlike the paddle gust generator, where the nozzle's contraction ratio was neglected, the adaptive nozzle's geometry, featuring a contraction ratio of 2:1, is exactly replicated. However, the upstream wind-tunnel section, including the flow straightener and diffuser, is simplified by modeling only the square duct directly ahead of the nozzle. The cubic spline used to define the fabricated upper contour of the nozzle in the experiments of Wood and Breuer [53] serves as the basis for generating the block-structured curvilinear grid. This grid is refined near solid boundaries to capture thin boundary layers, and the turbulent flow is resolved using LES. The immersed boundary representing the upper contour is discretized by a set of Lagrangian markers distributed along the spline. The spacing between markers in each local direction is defined based on the minimum Eulerian grid spacing encountered by the IB during its closing and opening motion. This ensures sufficient overlap between Lagrangian and Eulerian grids, thereby preventing any fluid penetration through the immersed surface. At the inlet of the upstream duct, a uniform horizontal velocity is imposed to reproduce the free-stream conditions measured at the nozzle's outlet in the experiment. No-slip and impermeability conditions are imposed on all solid walls (duct, nozzle, ground plate, and far-field), and a convective condition is applied at the outlet.

As in the paddle gust generator, the model includes two prescribed velocity boundary conditions governing the gust event: One applied to the Lagrangian markers and another at the inlet. In the paddle gust generator of Publication IV, it was sufficient to fit the

translational velocity of the saddle to obtain the kinematics of the moving paddle. In the adaptive nozzle, however, the nozzle's upper contour undergoes a rotational motion driven by an eye-bearing that converts the saddle's translational motion (see Fig. 4.1b). Accordingly, the translational velocity is converted into a rotational velocity, which is then prescribed to the Lagrangian markers defining the nozzle's upper contour. Figure 4.21 illustrates the resulting rotational velocity together with the transient blocking ratio during gust generation. The second prescribed velocity boundary condition concerns the inlet flow. A transient inlet velocity is applied to capture the temporary undershoot observed experimentally, caused by pressure losses resulting from the sudden partial blockage of the nozzle's outlet. Wood and Breuer [53] confirmed this effect by placing a Prandtl probe inside the main tunnel section during gust generation and calculating the dynamic pressure as the difference between the measured total and static pressures, revealing a temporary reduction in inlet velocity. In the simulation, this velocity drop is determined numerically from the continuity equation applied between the inlet patch and the time-dependent area at the first monitoring point, ensuring that the simulated inflow deficit reproduces the experimentally observed mass-flow reduction. A detailed description of the computational setup is provided in Section 4 of Publication VI. The main results, discussed in detail in Section 5 of the same publication, are summarized in the following, highlighting the instantaneous flow field induced by the adaptive nozzle and comparing the ensemble-averaged numerical predictions with the corresponding experimental measurements.

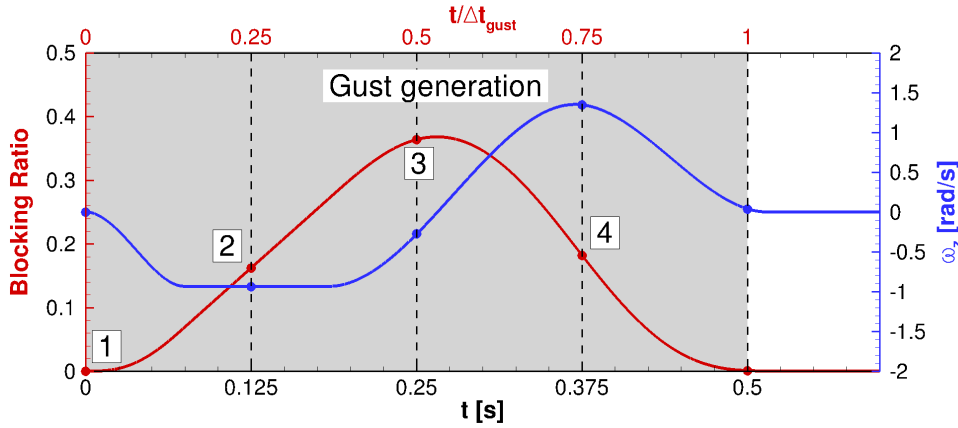


Fig. 4.21. Nozzle's blocking ratio annotated with visualization time instants; rotational velocity of the upper contour during the downward and upward strokes.

4.6.1 Flow Field Induced by the Adaptive Nozzle Wind Gust Generator

The initial analysis focuses on demonstrating the time-dependent flow characteristics of the gust induced by the rotational motion of the nozzle's upper contour. Figure 4.22 depicts snapshots of the instantaneous streamwise velocity component and the pressure fields in the x - y symmetry plane at four distinct time instants annotated in Fig. 4.21.

The monitoring points, labeled 1 to 11, are those defined in Fig. 4.20 at the ground plate and at the wall-normal distance $y/w = 0.15$. They denote specific positions within the wind-tunnel test section selected for comparison with the experimental data by Wood and Breuer [53, 92]. At instant 1 (i.e., at $t/\Delta t_{\text{gust}} = 0$), the nozzle's upper contour is in its idle position and the flow fields define the initial conditions for the numerical gust generator. The flow is fully developed, with the immersed boundary stationary to establish a stable reference. Boundary layers form along the nozzle walls and ground plate, while a free shear layer separates the core jet from the surrounding slower fluid. At instant 2 (i.e., at $t/\Delta t_{\text{gust}} = 0.25$), the upper contour moves downward, blocking about 16 % of the outlet area. This accelerates the streamwise velocity above the ground plate while preserving the main flow direction, which is a key advantage of the new design over the paddle gust generator. The contour's clockwise rotation leads to a pressure buildup upstream, increasing the static pressure within the nozzle. Simultaneously, local separation near the contour's tip triggers a small counter-clockwise vortex. At instant 3 (i.e., at $t/\Delta t_{\text{gust}} = 0.5$), the downstroke is nearly complete, reaching a blocking ratio of 36 %. High streamwise velocities along the ground plate mark the strong horizontal gust generated. The initially thick free shear layer evolves into a thin layer, dynamically adjusting its elevation to follow the contour's downward motion. Higher internal pressures develop compared to instant 2, while the vortex near the contour's tip grows and convects downstream. Instant 4 (i.e., $t/\Delta t_{\text{gust}} = 0.75$) captures the contour's upward return, where the gust weakens as the flow expands through the enlarging outlet. The free shear layer tilts as it lags behind the contour's rapid upward motion. Consequently, small vortices are shed from the upper contour's tip, destabilizing the shear layer and leading to strong velocity fluctuations. The lowest pressure values of all instants are recorded inside the nozzle, as the counter-clockwise rotation reduces both flow redirection and pressure buildup.

4.6.2 Comparison of Numerical and Experimental Results

The numerical model is validated through a direct comparison with the experimental measurements of Wood and Breuer [53]. Following the approach adopted for the paddle gust generator, phase-averaged quantities capturing the characteristic development of the gust, are used for comparison. Phase-averaging is essential due to the free shear layer at the nozzle's outlet, where small vortical structures are irregularly generated and convected downstream, rendering the flow response sensitive to the initial state at the onset of motion. Accordingly, 46 gust realizations were phase-averaged in the experiments and compared against numerical predictions based on 46 distinct initial condition states. The comparison is carried out using the time histories of the phase-averaged velocity magnitude and the pressure coefficient at their respective monitoring points (in the flow or on the ground plate) depicted in Fig. 4.20. For clarity, the time histories are shifted by adding the product of a freely chosen dimensionless time increment $\Delta t_{\text{inc}}/\Delta t_{\text{gust}} = 2$ and a spacing factor $a \in (0, \dots, 10)$.

The comparison between the phase-averaged CTA measurements of the velocity magnitude and the corresponding numerical predictions is depicted in Fig. 4.23. The time histories at the monitoring points exhibit a bell-shaped evolution consistent with the

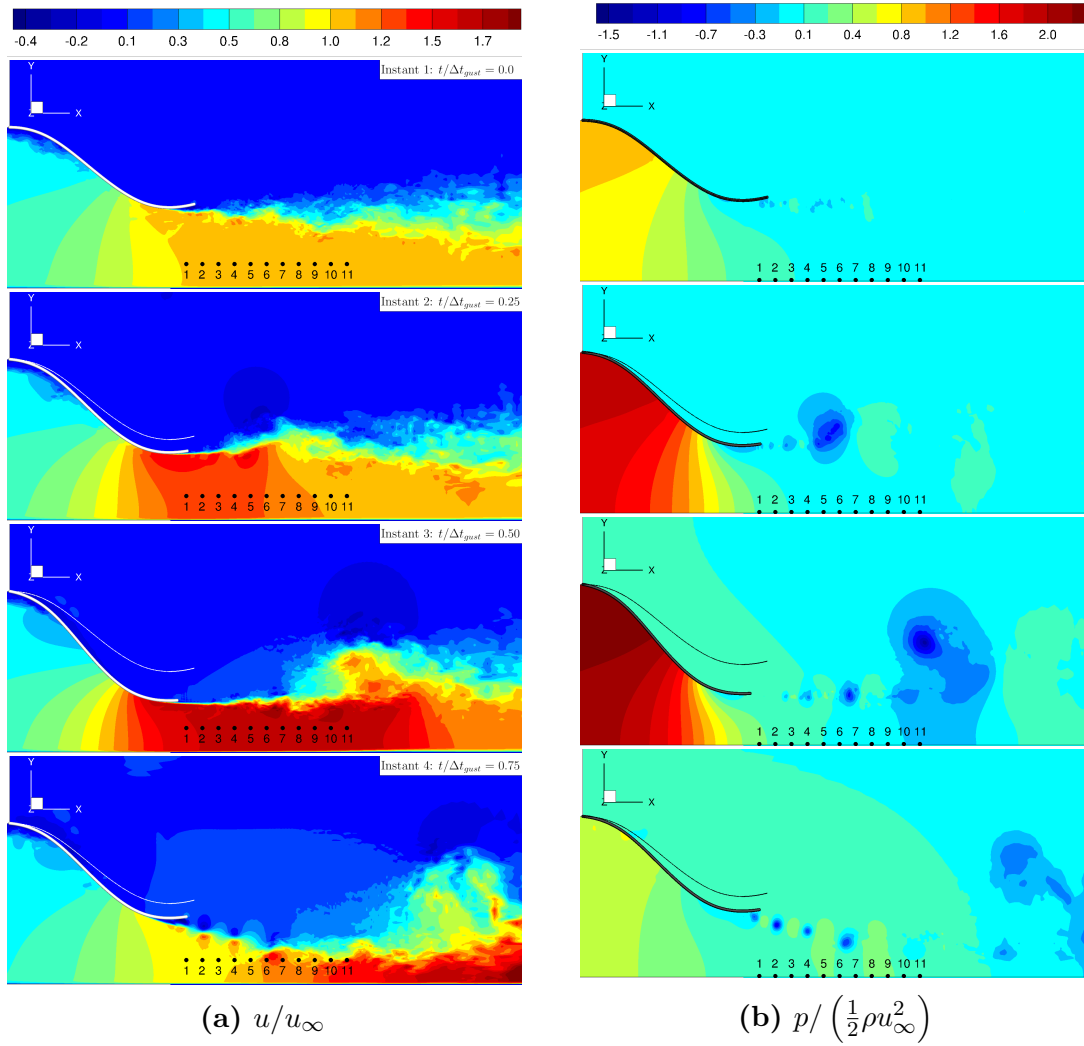


Fig. 4.22. Snapshots of the streamwise velocity (a) and the pressure (b) fields during the nozzle's downward and upward strokes. Instants correspond to those depicted in Fig. 4.21. For the instants 2 to 4, a line indicating the upper contour's idle position is depicted for reference.

nozzle's blocking ratio. As expected from mass conservation, the streamwise velocity beneath the upper contour increases with the outlet contraction, reaching its maximum at the peak blocking ratio and decreasing as the contour returns to its idle position. The subsequent undershoot below the free-stream velocity is linked to the prescribed transient inlet velocity, which reflects the reduced airflow through the wind-tunnel blower caused by flow blockage. This effect becomes more pronounced as the gust propagates downstream. As shown in Fig. 4.23, the predicted velocity signals at the near-nozzle positions, located within the range $(0 \leq x/w \leq 0.2)$, closely follow the measurements. Further downstream $(0.3 \leq x/w \leq 1)$, however, an additional velocity peak emerges in the descending slope of these curves (see zoom-in in Fig. 4.23). This fluctuation originates from the flow separation near the tip of the nozzle's upper contour, triggered by its rapid upward motion, which

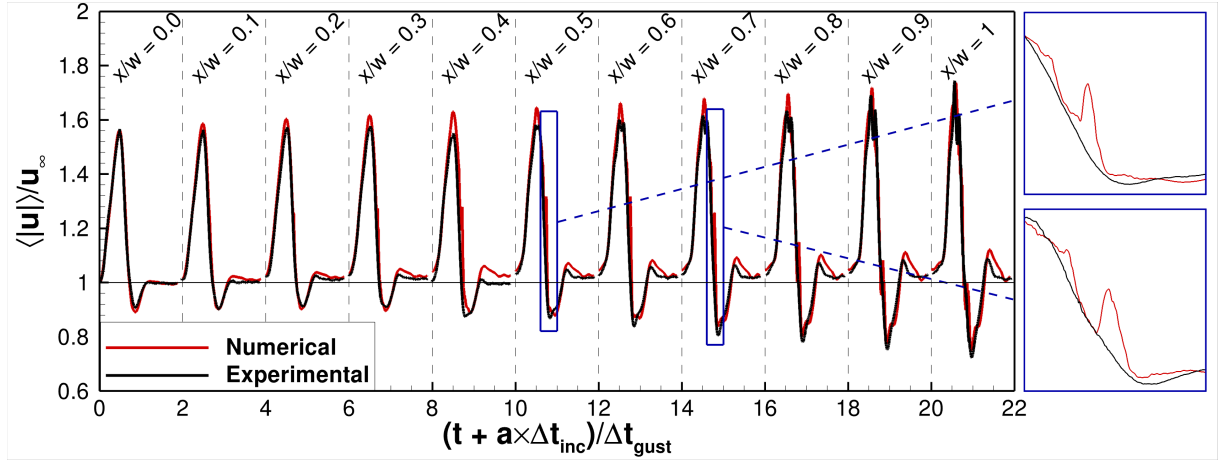


Fig. 4.23. Time histories of the phase-averaged experimental velocity magnitude [53, 92] and its numerical counterpart at the 11 monitoring points depicted in Fig. 4.22a.

generates a series of small vortices in the detached, inclined shear layer. These vortices destabilize the layer and ultimately disrupt the flow at the downstream monitoring points (see instant 4 in Fig. 4.22a). While this peak persists in the numerical phase-averaged data, it is absent in the experimental averages. Here it is assumed that inflow turbulence desynchronizes the vortex shedding process across the 46 realizations, smoothing out the effect. This is an issue, which should be investigated in the future.

Another noteworthy observation regarding the velocity peaks occurs after the gust is generated particularly at the monitoring points $0.4 \leq x/w \leq 1$. A second local velocity maximum is observed in the far wake of the generated gust (i.e., at $t/\Delta t_{gust} > 1$). This overshoot, together with the preceding undershoot, becomes more pronounced downstream. Overall, the comparisons demonstrate that the proposed numerical model effectively reproduces the gust generation process and agrees well with measurements, apart from the discrepancies in the shear layer.

The time history of the predicted phase-averaged pressure coefficient $\langle c_p \rangle = 2 \langle p \rangle / (\rho u_\infty^2)$ and its measured counterpart at the 11 streamwise surface monitoring points is depicted in Fig. 4.24. The temporal evolution of $\langle c_p \rangle$ follows a characteristic four-stage behavior across all monitoring points. Initially, during the downward stroke of the nozzle, the pressure rises due to outlet contraction, with the magnitude of this rise decreasing downstream. It then undergoes a sharp drop, reaching negative values for $x/l > 0.4$, caused by the large vortex formed at the contour's tip. While this vortex only weakly influences the near-nozzle monitoring points, it grows in strength downstream, leading to a pronounced pressure reduction. Following this drop, the pressure recovers as the vortex convects away and the continued buildup inside the nozzle increases the surface pressure again. Near-outlet points (i.e., $0 \leq x/l \leq 0.1$) reach their peak at the end of the downstroke, while downstream points ($0.2 \leq x/l \leq 1$) reach their maxima mainly during the upstroke phase, consistent with the simultaneous velocity decrease. Unlike the nearly constant velocity peaks across the monitoring points, the pressure peaks increase in magnitude downstream and eventually level off for $x/l > 0.7$. Finally, a sharp pressure drop occurs during the

upstroke, coinciding with the recovery of the velocity from its undershoot.

Overall, the comparison between numerical and experimental surface pressures demonstrates reasonable agreement in both trend and magnitude. Minor discrepancies at the first two monitoring points are attributed to geometric inaccuracies of the fabricated nozzle. This surface-pressure analysis confirms that the numerical model not only reproduces the velocity field but also accurately captures the unsteady surface-pressure distribution in the test section, thereby reinforcing the validity of the proposed framework for gust generation.

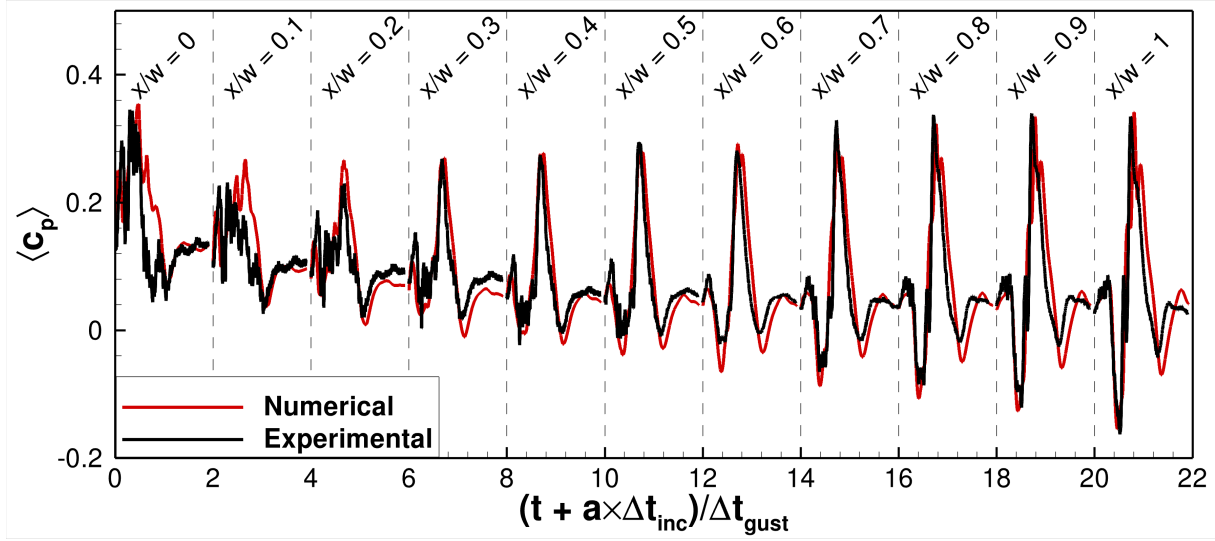


Fig. 4.24. Time histories of the phase-averaged experimental surface pressure coefficient [53] and its numerical counterpart at the 11 monitoring points depicted in Fig. 4.22b.

Additional comparisons between the predicted and measured flow fields based on phase-averaged PIV data are provided in Section 5 of Publication VI. These are not shown here for brevity. Nevertheless, the PIV comparisons confirm good agreement between the numerical predictions and the experimental measurements, further validating the accuracy and reliability of the developed numerical model.

4.7 Conclusions

This chapter provides a comprehensive summary of Publications IV, V, and VI, which present the numerical counterparts of the experimental investigations performed on the *paddle* [52, 90] and the *adaptive nozzle* [53] wind-tunnel gust generators. The publications address the challenge of replicating experimentally generated discrete wind gusts within a numerical framework, thereby establishing a robust coupled fluid–structure interaction formulation capable of analyzing the dynamic response of flexible structures to controlled gusts.

Since both gust generators share the same operating principle, namely the rapid partial blockage of the nozzle outlet by a rigid body undergoing large displacements, this

feature must be accurately modeled in the numerical simulations. Among the available modeling approaches, the immersed boundary method with a continuous direct-forcing formulation is adopted, as it efficiently handles large rigid-body motions without requiring costly remeshing. Its main advantage lies in the use of overlapping Eulerian (fluid) and Lagrangian (structure) discretizations, coupled through a regularized delta-function kernel that smoothly spreads the boundary volume force over the surrounding fluid cells. This allows the nozzle’s transient blockage effect to be captured accurately while maintaining stable and physically consistent pressure and velocity fields throughout the simulation.

To enable grid stretching and clustering near regions of interest while reducing the overall computational cost, the grids used in the present simulations deviate from the uniform Cartesian layouts typically adopted in classical IBM applications. Consequently, alternative interpolation schemes are employed to construct IB kernels that rigorously enforce force and moment conservation on arbitrary underlying Eulerian grids. For the paddle gust generator, corrected kernel functions are constructed based on the RKPM, whereas the adaptive nozzle employs the MLS approach to construct one-sided kernels in near-wall regions.

In Publication IV, a high-fidelity numerical model of the wind-tunnel gust generator, the *paddle*, is developed to replicate the experimentally generated discrete gust with high accuracy. The IBM formulation employed ensures smooth pressure and velocity fields without numerical oscillations and conserves force and moment coefficients, confirming the robustness of the numerical approach. The simulations reproduce the key flow dynamics during both the downward and upward strokes and capture the associated acceleration and pressure variations within the test section. Ensemble-averaged predictions show close agreement with LDA measurements at all seven monitoring points, including the characteristic bell-shaped temporal evolution of the streamwise velocity and the spanwise uniformity of the generated gust. The undisturbed gust region confined between the ground plate and the free shear layer is found to be relatively narrow, in agreement with experimental observations. Minor deviations in the velocity signals during the upstroke are attributed to the vortical unsteadiness in the shear layer, which diminishes with increased ensemble averaging. The validated numerical model thus provides a reliable reference for paddle-induced gusts and a solid foundation for subsequent coupled fluid–structure interaction studies.

In Publication V, the fluid–structure interaction problem of a flexible T-structure subjected to the artificial wind gust generated by the paddle is numerically reproduced by combining two complementary methods: The immersed boundary approach for modeling the moving paddle and an ALE-based partitioned approach for the deformable structure. The hybrid framework successfully captures both the gust generation process in the wind tunnel and the complex dynamic response of the thin structure. Without gust excitation, the structure exhibits only minor deflections characterized by a dominant bending mode and a weaker torsional component, and the predicted time-averaged flow and structural deformations closely match the experimental phase-averaged measurements. Under gust loading, the phase-averaged simulations reproduce the transient acceleration of the incoming flow, the enlargement of the recirculation region behind the shield-shaped section, and the short-lived structural deformation that gradually decays as the gust diminishes.

The results also reveal alternating oscillation modes and notable cycle-to-cycle variations consistent with the measurements, while the absence of inflow turbulence in the numerical simulations explains the slightly smaller oscillation amplitudes. Overall, the study demonstrates the effectiveness of the combined IBM–ALE methodology in replicating the wind-tunnel experiments and provides a validated numerical foundation for future investigations of flexible or membrane-type structures subjected to controlled wind-gust excitations.

In Publication VI, a dynamic immersed boundary method is developed for the application on curvilinear block-structured grids to reproduce the highly unsteady turbulent flow generated by the adaptive nozzle wind-tunnel gust generator. The numerical model accurately captures the time-resolved flow field throughout the entire computational domain, including the interior of the nozzle, and reproduces the ensemble-averaged experimental data with good agreement in both velocity and pressure distributions. Small discrepancies in the detached shear layer are again attributed to the absence of inflow turbulence in the simulations, which desynchronizes vortex formation in the experiments. Overall, the study establishes an efficient and robust IB methodology for curvilinear grids, validated against experimental data, and provides a versatile foundation for future investigations of wind-gust interactions with flexible structures.

5 Outlook

The developments presented in this thesis provide several opportunities for extending gust-injection techniques and for broadening their use in future fluid–structure interaction studies. The three approaches explored in this work (the prescribed velocity methods, the source-term formulation, and the immersed boundary gust generators) represent distinct strategies for introducing discrete gusts, each offering different directions for future developments and applications.

The *prescribed velocity methods* employed in this work idealize a discrete gust as a coherent structure convected through the domain with a constant, user-defined velocity, effectively treating the gust as a frozen disturbance. Although this assumption becomes questionable in atmospheric settings, where gusts evolve on comparable time scales to their convection, it offers a high degree of control when the goal is to impose well-defined transient disturbances rather than to reproduce realistic wind fields. In such idealized studies, prescribed-velocity gusts serve as synthetic coherent disturbances that enable systematic probing of structural sensitivity to specific transient velocity excitations. Because these methods rely only on prescribing the disturbance and its convection velocity, they are not inherently restricted to aeronautical applications. They can equally support idealized sensitivity studies in other engineering disciplines, for example by examining how wind turbines, buildings, or long-span bridges respond to prescribed transient gust-like disturbances. Thus, the advantages of these methods could be explored in other fields of application. In addition, while most published applications of these methods have focused almost exclusively on idealized analytical gust profiles (e.g., 1-cosine, sharp-edge, ramps), the methods themselves place no restriction on the functional form of the imposed disturbance. In principle, they can prescribe arbitrary space–time gust fields, including tabulated or numerically generated profiles, thereby enabling controlled studies with more complex or non-analytic transient disturbances when such inputs are available.

The *source-term formulation* presented in this work was developed to inject gusts that align with the main flow direction. Its derivation relies on a key simplifying assumption that suppresses the transverse components of the background velocity, which yields a compact and easily implementable source-term expression but also limits its applicability to more complex gust orientations (e.g., inclined or vertical gusts). A further limitation is that the injected momentum scales with the instantaneous local total flow velocity, meaning that the gust disturbance depends not only on its prescribed shape but also on the evolving flow field within the injection region, including any background turbulence. As a result, the injected disturbance effectively “rides” on the instantaneous flow field, implying that the effective momentum input is not uniquely determined by the prescribed gust shape alone and may be unintentionally amplified or weakened. Addressing these two aspects may involve adopting a more general and controlled definition of the convecting

velocity in the source term defined by Eq. (3.9). Allowing all directional components in the convecting velocity would remove the restriction to main-flow-aligned gusts. In addition, formulating a user-defined convecting velocity would prevent the gust strength from depending on the evolving flow field. These modifications could form the basis of a second-generation source-term formulation with broader applicability and more controlled gust characteristics.

The *immersed boundary gust generators*, used within the hybrid IBM–ALE framework, provide a foundation for extending the method to more demanding fluid–structure interaction problems. A natural next step is the application to membrane-type structures, where larger deformations are expected and a strongly coupled FSI treatment is required. These cases would provide more challenging benchmarks for assessing the robustness of the framework. With both gust generators available numerically and experimentally, membrane-type structures could be investigated in each setting and directly compared for systematic validation. Such complementary experimental-numerical studies are planned for the near future.

Finally, machine-learning techniques, such as artificial neural networks, offer a promising way to rapidly predict structural responses during gust encounters. Such models can be trained for different structures and across a broad parameter space that includes various gust shapes, widths, and amplitudes. The predicted quantities may include maximum aerodynamic forces, structural deformations, and other key structural response metrics. The simulation frameworks and test cases developed in this thesis provide a suitable basis for generating the training data needed to construct such predictive models, enabling fast assessments of gust–structure interactions at a fraction of the computational cost associated with full fluid–structure interaction simulations.

A Definition of the Gust Shape

The gust shape in Chapters 2 and 3 is defined with respect to the local orthonormal basis $\mathcal{B}_1 = (\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$, as illustrated in Fig. A.1. It is assumed that the gust has a non-zero velocity component only in the direction of the first basis vector of the local coordinate system (i.e., $\mathbf{g}_1$).

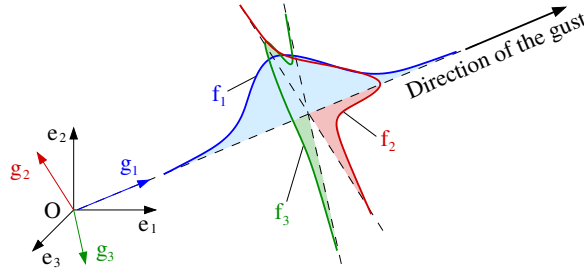


Fig. A.1. Local basis definition $\mathcal{B}_1 = (\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$ of the gust.

Letting $u_{g,j}|_{\mathcal{B}_0}$ and $u_{g,j}|_{\mathcal{B}_1}$ represent the components of the gust velocity vector expressed in the Cartesian basis $\mathcal{B}_0 = (\mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3)$ and in the local orthonormal basis $\mathcal{B}_1 = (\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$, respectively, the gust velocity can then be written as:

$$\mathbf{u}_g = u_{g,1}|_{\mathcal{B}_0} \mathbf{e}_1 + u_{g,2}|_{\mathcal{B}_0} \mathbf{e}_2 + u_{g,3}|_{\mathcal{B}_0} \mathbf{e}_3 = u_{g,1}|_{\mathcal{B}_1} \mathbf{g}_1 + \cancel{u_{g,2}|_{\mathcal{B}_1} \mathbf{g}_2} + \cancel{u_{g,3}|_{\mathcal{B}_1} \mathbf{g}_3} = u_{g,1}|_{\mathcal{B}_1} \mathbf{g}_1. \quad (\text{A.1})$$

The first basis vector $\mathbf{g}_1$ of the local coordinate system is user-defined. The remaining vectors (i.e., $\mathbf{g}_2$ and $\mathbf{g}_3$) may either be specified manually or computed such that the triplet $(\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$ forms an orthonormal basis. To express the gust velocity in the global Cartesian system, the transformation matrix $\mathbf{T}_{\mathcal{B}_1 \rightarrow \mathcal{B}_0}$, mapping vectors from the local basis $\mathcal{B}_1$ to the Cartesian basis $\mathcal{B}_0$, is required. Introducing the shorthand notation $\beta_{ij} = \partial u_{g,j}|_{\mathcal{B}_0} / \partial u_{g,i}|_{\mathcal{B}_1}$, the transformation can be written in matrix form as:

$$\begin{aligned} [\mathbf{g}_i] &= [\beta_{ij}] [\mathbf{e}_j], \\ \underbrace{\begin{bmatrix} \mathbf{g}_1 \\ \mathbf{g}_2 \\ \mathbf{g}_3 \end{bmatrix}}_{\text{old basis}} &= \underbrace{\begin{bmatrix} \frac{\partial u_{g,1}|_{\mathcal{B}_0}}{\partial u_{g,1}|_{\mathcal{B}_1}} & \frac{\partial u_{g,2}|_{\mathcal{B}_0}}{\partial u_{g,1}|_{\mathcal{B}_1}} & \frac{\partial u_{g,3}|_{\mathcal{B}_0}}{\partial u_{g,1}|_{\mathcal{B}_1}} \\ \frac{\partial u_{g,1}|_{\mathcal{B}_0}}{\partial u_{g,2}|_{\mathcal{B}_1}} & \frac{\partial u_{g,2}|_{\mathcal{B}_0}}{\partial u_{g,2}|_{\mathcal{B}_1}} & \frac{\partial u_{g,3}|_{\mathcal{B}_0}}{\partial u_{g,2}|_{\mathcal{B}_1}} \\ \frac{\partial u_{g,1}|_{\mathcal{B}_0}}{\partial u_{g,3}|_{\mathcal{B}_1}} & \frac{\partial u_{g,2}|_{\mathcal{B}_0}}{\partial u_{g,3}|_{\mathcal{B}_1}} & \frac{\partial u_{g,3}|_{\mathcal{B}_0}}{\partial u_{g,3}|_{\mathcal{B}_1}} \end{bmatrix}}_{\mathbf{T}_{\mathcal{B}_1 \rightarrow \mathcal{B}_0}} \underbrace{\begin{bmatrix} \mathbf{e}_1 \\ \mathbf{e}_2 \\ \mathbf{e}_3 \end{bmatrix}}_{\text{new basis}} = \begin{bmatrix} \beta_{11} & \beta_{12} & \beta_{13} \\ \beta_{21} & \beta_{22} & \beta_{23} \\ \beta_{31} & \beta_{32} & \beta_{33} \end{bmatrix} \begin{bmatrix} \mathbf{e}_1 \\ \mathbf{e}_2 \\ \mathbf{e}_3 \end{bmatrix}. \quad (\text{A.2}) \end{aligned}$$

Here, β_{11} , β_{12} , and β_{13} are the x , y , and z components, respectively, of the user-defined unit vector which defines the direction of the gust in the Cartesian basis. Based on the relation derived in Eq. (A.2), the expression of the gust velocity in the Cartesian basis can be written as:

$$\mathbf{u}_g = u_{g,1}|_{\mathcal{B}_1} \mathbf{g}_1 = u_{g,1}|_{\mathcal{B}_1} (\beta_{11} \mathbf{e}_1 + \beta_{12} \mathbf{e}_2 + \beta_{13} \mathbf{e}_3) . \quad (\text{A.3})$$

In deterministic gust models, the component $u_{g,1}|_{\mathcal{B}_1}$ characterizes the spatial and temporal variation of the gust through analytical functions:

$$u_{g,1}|_{\mathcal{B}_1}(t, \xi, \eta, \zeta) = A_g f_t(t) f_1(\xi) f_2(\eta) f_3(\zeta) . \quad (\text{A.4})$$

A_g denotes the user-defined amplitude of the gust. The functions f_t , f_1 , f_2 , and f_3 are analytic expressions specified by the user, representing the temporal shape of the gust and its spatial distribution along the $\mathbf{g}_1$ -, $\mathbf{g}_2$ -, and $\mathbf{g}_3$ -directions, respectively. The variables ξ , η , and ζ correspond to the spatial coordinates in the local orthonormal basis ($\mathbf{g}_1$, $\mathbf{g}_2$, $\mathbf{g}_3$). These local coordinates can be expressed as functions of the Cartesian coordinates by (see [51] for further details):

$$\begin{bmatrix} \xi \\ \eta \\ \zeta \end{bmatrix} = \begin{bmatrix} t_\xi(x, y, z) \\ t_\eta(x, y, z) \\ t_\zeta(x, y, z) \end{bmatrix} = \begin{bmatrix} \beta_{11} & \beta_{12} & \beta_{13} \\ \beta_{21} & \beta_{22} & \beta_{23} \\ \beta_{31} & \beta_{32} & \beta_{33} \end{bmatrix} \begin{bmatrix} x \\ y \\ z \end{bmatrix} . \quad (\text{A.5})$$

This leads to the following expression for the gust velocity in the Cartesian basis $\mathcal{B}_0$:

$$\mathbf{u}_g|_{\mathcal{B}_0}(t, x, y, z) = A_g f_t(t) f_1(t_\xi(x, y, z)) f_2(t_\eta(x, y, z)) f_3(t_\zeta(x, y, z)) \mathbf{g}_1 . \quad (\text{A.6})$$

Commonly employed deterministic functions used in Eq. (A.4) to describe the spatial and temporal distribution of a gust include the “1-cosine” profile, referred to here as the *Extreme Coherent Gust* (ECG). This gust definition is specified in FAR 25.341 [17] and CS-25.341 [18]. Another deterministic gust model is the *Extreme Operating Gust* (EOG), which is defined in the IEC standard [14]. Alternative formulations include Gaussian-shaped profiles or the model introduced by Knigge and Raasch [93], which was derived from large-eddy simulations (LES) of turbulent flows and is considered to represent gust behaviour more realistically. In the present work, only the ECG model is employed. The original formulation of the 1-cosine gust shape, as specified in the certification regulations [17, 18], is expressed as:

$$f_i(\phi) = \begin{cases} \frac{1}{2} \left(1 - \cos \left(\frac{\pi \phi}{L_g^\phi} \right) \right) & \text{for } \phi \in [0, L_g^\phi] \\ 0 & \text{else .} \end{cases} \quad (\text{A.7})$$

In order to achieve more control, the original shape is adapted introducing the central value ϕ_g [39]:

$$f_i(\phi) = \begin{cases} \frac{1}{2} \left(1 + \cos \left(\frac{2\pi (\phi - \phi_g)}{L_g^\phi} \right) \right) & \text{for } (\phi - \phi_g) \in \left[-\frac{L_g^\phi}{2}, \frac{L_g^\phi}{2} \right] \\ 0 & \text{else .} \end{cases} \quad (\text{A.8})$$

The subscript i is equal to 1, 2, 3 or t. The variable ϕ corresponds to the coordinate ξ, η, ζ or to the time t for $i = 1, 2, 3$ or t, respectively. The constant ϕ_g is the user-defined central value of the gust distribution for the corresponding ϕ and L_g^ϕ is its user-defined length or time scale. Note that the present methodology allows to prescribe the wind gust as a three-dimensional instantaneous phenomenon as will be shown in Chapter 3.

Bibliography

- [1] Intergovernmental Panel on Climate Change (IPCC). *Climate change 2021 – The physical science basis: Working group I contribution to the sixth assessment report of the intergovernmental panel on climate change*. Cambridge University Press. Cambridge, 2023.
- [2] R. R. Rogers and M. K. Yau. *A short course in cloud physics*. 3rd. Pergamon Press. Oxford and New York, 1989.
- [3] WMO. *State of the global climate 2022*. World Meteorological Organization. Geneva, Switzerland, 2023.
- [4] P. Hawbecker, S. Basu and L. Manuel. Realistic simulations of the July 1, 2011 severe wind event over the Buffalo Ridge Wind Farm. *Wind Energy* 20 (11) (2017), 1803–1822.
- [5] X. Chen and J. Z. Xu. Structural failure analysis of wind turbines impacted by super typhoon Usagi. *Engineering Failure Analysis* 60 (2016), 391–404.
- [6] J. L. Gross, J. A. Main, L. T. Phan, F. H. Sadek, S. A. Cauffman and D. P. Jorgensen. *Final report on the collapse of the Dallas Cowboys indoor practice facility, May 2, 2009 (NIST IR 7661)*. 2010.
- [7] The Courier. *Storm Ali: Watch the moment a huge marquee at St Andrews Old Course is lifted into the air by strong wind*. Sept. 19, 2018. URL: <https://www.thecourier.co.uk/fp/news/fife/728519/storm-ali-watch-the-moment-a-huge-marquee-at-st-andrews-old-course-is-lifted-into-the-air-by-strong-wind/> (visited on 11/18/2025).
- [8] J. McCarthy, R. Serafin, J. Wilson, J. Evans, C. Kessinger and W. P. Mahoney III. Addressing the microburst threat to aviation: Research-to-operations success story. *Bulletin of the American Meteorological Society* 103 (12) (2022).
- [9] National Transportation Safety Board. *Aircraft accident report: Delta Air Lines Flight 191, Lockheed L-1011-385-1, N726DA, Dallas/Fort Worth international airport, Texas, August 2, 1985*. Tech. rep. NTSB-AAR-86-05. National Transportation Safety Board. Washington, D.C., 1986.
- [10] K. Barnstorff. *Wind shear accident was catalyst for technology*. NASA Langley Research Center. National Aeronautics and Space Administration (NASA). July 28, 2010. URL: <https://www.nasa.gov/history/wind-shear-accident-was-catalyst-for-technology/> (visited on 11/06/2025).

- [11] World Meteorological Organization. *Atlas of mortality and economic losses from weather, climate and water-related hazards (1970–2021)*. World Meteorological Organization. Geneva, May 22, 2023.
- [12] Munich Reinsurance Company. *Hail, tornadoes and flash floods: Losses from thunderstorms on the rise*. June 2025. URL: <https://www.munichre.com/en/risks/natural-disasters/thunderstorms-hail-tornados.html> (visited on 11/06/2025).
- [13] European Environment Agency. *Economic losses from weather- and climate-related extremes in Europe*. Oct. 14, 2025. URL: <https://www.eea.europa.eu/en/analysis/indicators/economic-losses-from-climate-related> (visited on 11/06/2025).
- [14] IEC 61400-1: Wind energy generation systems – Part 1: Design requirements. International Electrotechnical Commission, 2019.
- [15] ASCE/SEI 7-22: Minimum design loads and associated criteria for buildings and other structures. American Society of Civil Engineers, 2022.
- [16] EN 1991-1-4: Eurocode 1: Actions on structures – Part 1-4: General actions – Wind actions. European Committee for Standardization, 2005.
- [17] Federal Aviation Administration. *14 CFR § 25.341: Gust and turbulence loads*. 2025.
- [18] European Union Aviation Safety Agency. *CS-25.341: Gust and turbulence loads*. 2023.
- [19] C. Wales, D. Jones and A. Gaitonde. Prescribed velocity method for simulation of aerofoil gust responses. *Journal of Aircraft* 52 (1) (2014), 64–76.
- [20] Federal Aviation Administration. *Advisory circular 25.341-1: Dynamic gust loads*. Tech. rep. AC 25.341-1. 2014.
- [21] C. Badrya, J. Baeder and A. Jones. Application of prescribed velocity methods to a large-amplitude flat-plate gust encounter. *AIAA Journal* 57 (8) (2019), 1–13.
- [22] R. B. Stull. *An Introduction to Boundary Layer Meteorology*. Atmospheric Sciences Library. Springer. Dordrecht, 1988.
- [23] World Meteorological Organization. *OSCAR variable: Wind gust*. URL: https://space.oscar.wmo.int/variables/view/wind_gust (visited on 11/13/2025).
- [24] World Meteorological Organization. *Manual on codes (WMO-No. 306), Volume I.3*. Geneva, 2023.
- [25] National Weather Service. *NWS glossary: Wind gust*. URL: <https://forecast.weather.gov/glossary.php?word=wind+gust> (visited on 11/13/2025).
- [26] National Weather Service. *Implementation of operational wind gust grids*. Tech. rep. National Oceanic and Atmospheric Administration. 2007.

- [27] National Weather Service. *National digital forecast database definitions*. URL: <https://graphical.weather.gov/supplementalpages/definitions.php> (visited on 11/13/2025).
- [28] Deutscher Wetterdienst. *Glossar: Bö*. URL: <https://www.dwd.de/DE/service/lexikon/Functions/glossar.html?lv2=100310&lv3=100488> (visited on 11/13/2025).
- [29] American Meteorological Society. *Glossary of meteorology: Gust*. URL: <https://glossary.ametsoc.org/wiki/Gust> (visited on 11/13/2025).
- [30] R. Bos. Extreme gusts and their role in wind turbine design. PhD thesis. Delft University of Technology, The Netherlands, 2017.
- [31] T. T. Fujita. *The downburst: microburst and macroburst*. Tech. rep. SMRP Research Paper 210. Department of the Geophysical Sciences, University of Chicago. 1985.
- [32] I. Orlanski. A rational subdivision of scales for atmospheric processes. *Bulletin of the American Meteorological Society* 56 (5) (1975), 527–530.
- [33] D. G. Steyn, T. R. Oke, J. E. Hay and J. L. Knox. On scales in meteorology and climatology. *McGill Climatological Bulletin* 30 (1981), 1–8.
- [34] J. C. Wyngaard. Scalar fluxes in the planetary boundary layer — Theory, modeling, and measurement. *Boundary-Layer Meteorology* 50 (1990), 49–75.
- [35] F. M. Hoblit. *Gust loads on aircraft: Concepts and applications*. AIAA Education Series, Washington D.C. 1988.
- [36] Z. Wu, Y. Cao and M. Ismail. Gust loads on aircraft. *The Aeronautical Journal* 123 (1266) (2019), 1216–1274.
- [37] W. A. A. M. Bierbooms. Constrained Stochastic Simulation of Wind Gusts for Wind Turbine Design. PhD thesis. Delft University of Technology, Delft, The Netherlands, 2009.
- [38] S. E. Norris, J. E. Cater, K. A. Stol and C. P. Unsworth. Wind turbine wake modelling using large-eddy simulation. In: *Proceedings of the 17th Australian Fluid Mechanics Conference*. Curran Associates, Inc. University of Auckland, Australia, 2010.
- [39] G. De Nayer, M. Breuer, P. Perali and K. Grollmann. Modeling of wind gusts for large-eddy simulations related to fluid–structure interactions. In: *ERCRAFT Series, Direct and Large-Eddy Simulation XI, 11th Int. ERCRAFT Workshop on Direct and Large-Eddy Simulation: DLES-11, Pisa, Italy, May 29–31, 2017*. Ed. by M. V. Salvetti, V. Armenio, J. Fröhlich, B. J. Geurts and H. Kuerten. Vol. 25. Springer Nature Switzerland AG. 2019, 453–459.
- [40] R. Heinrich. Simulation of interaction of aircraft and gust using the TAU-code. In: *New Results in Numerical and Experimental Fluid Mechanics IX: Contributions to the 18th STAB/DGLR Symposium, Stuttgart, Germany, 2012*. Ed. by A. Dillmann,

G. Heller, E. Krämer, H.-P. Kreplin, W. Nitsche and U. Rist. Springer International Publishing, Cham, 2014, 503–511.

- [41] V. Parameswaran and J. D. Baeder. Indicial aerodynamics in compressible flow—Direct computational fluid dynamic calculations. *Journal of Aircraft* 34 (1) (1997), 131–133.
- [42] R. Singh and J. D. Baeder. Direct calculation of three-dimensional indicial lift response using computational fluid dynamics. *Journal of Aircraft* 34 (4) (1997), 465–471.
- [43] J. Sitaraman and J. D. Baeder. Enhanced unsteady airload models using CFD. In: *Fluids 2000 Conference and Exhibit*. Denver, Colorado, June 2000.
- [44] J. Sitaraman. CFD based unsteady aerodynamic modeling for rotor aeroelastic analysis. PhD thesis. University of Maryland, College Park, USA, Jan. 2003.
- [45] J. Sitaraman and J. D. Baeder. Field velocity approach and geometric conservation law for unsteady flow simulations. *AIAA Journal* 44 (9) (2006), 2084–2094.
- [46] S. J. Huntley, D. Jones and A. Gaitonde. Aeroelastic gust response of an aircraft using a prescribed velocity method in viscous flows. In: *23rd AIAA Computational Fluid Dynamics Conference*. American Institute of Aeronautics and Astronautics. Denver, Colorado, 2017, 1–18.
- [47] H. Biler, C. Badrya and A. R. Jones. Experimental and computational investigation of transverse gust encounters. *AIAA Journal* 57 (11) (2019), 4608–4622.
- [48] C. Badrya and J. D. Baeder. Numerical study of a flat plate wing response to large transverse gusts at low Reynolds number. In: *AIAA Scitech 2019 Forum*. San Diego, California, 2019, p. 0638.
- [49] C. Badrya, H. Biler, A. R. Jones and J. D. Baeder. Effect of gust width on flat-plate response in large transverse gust. *AIAA Journal* 59 (1) (2021), 49–64.
- [50] K. Boulbrachene, G. De Nayer and M. Breuer. Assessment of two wind gust injection methods: Field velocity vs. split velocity method. *Journal of Wind Engineering and Industrial Aerodynamics* 218 (2021), 104790.
- [51] G. De Nayer and M. Breuer. A source-term formulation for injecting wind gusts in CFD simulations. *Journal of Wind Engineering and Industrial Aerodynamics* 207 (2020), 104405.
- [52] J. N. Wood, M. Breuer and T. Neumann. A novel approach for artificially generating horizontal wind gusts based on a movable plate: The Paddle. *Journal of Wind Engineering and Industrial Aerodynamics* 230 (2022), 105170.
- [53] J. N. Wood and M. Breuer. Experimental study on a novel wind gust generator based on an adaptive nozzle design. *Journal of Wind Engineering and Industrial Aerodynamics* 261 (2025), 106080.

- [54] F. Durst and M. Schäfer. A parallel block-structured multigrid method for the prediction of incompressible flows. *International Journal for Numerical Methods in Fluids* 22 (6) (1996), 549–565.
- [55] M. Breuer, G. De Nayer, M. Münsch, T. Gallinger and R. Wüchner. Fluid–structure interaction using a partitioned semi–implicit predictor–corrector coupling scheme for the application of large–eddy simulation. *Journal of Fluids and Structures* 29 (2012), 107–130.
- [56] I. Demirdžić and M. Perić. Space conservation law in finite-volume calculations of fluid flow. *International Journal for Numerical Methods in Fluids* 8 (9) (1988), 1037–1050.
- [57] I. Demirdžić and M. Perić. Finite-volume method for prediction of fluid flow in arbitrarily shaped domains with moving boundaries. *International Journal for Numerical Methods in Fluids* 10 (7) (1990), 771–790.
- [58] M. Lesoinne and C. Farhat. Geometric conservation laws for flow problems with moving boundaries and deformable meshes, and their impact on aeroelastic computations. *Computer Methods in Applied Mechanics and Engineering* 134 (1-2) (1996), 71–90.
- [59] M. Münsch. Entwicklung und Anwendung eines semi-impliziten Kopplungsverfahrens zur numerischen Berechnung der Fluid-Struktur-Wechselwirkung in turbulenten Strömungen mittels Large-Eddy Simulationen. PhD thesis. University of Erlangen-Nuremberg, Germany, 2015.
- [60] N. Gu, Z. Lu and T. Guo. Simulation of viscous flows around a moving airfoil by field velocity method with viscous flux correction. *Advances in Applied Mathematics and Mechanics* 4 (3) (2012), 294–310.
- [61] G. De Nayer, M. Breuer and J. N. Wood. Numerical investigations on the dynamic behavior of a 2-DOF airfoil in the transitional Re number regime based on fully coupled simulations relying on an eddy-resolving technique. *International Journal of Heat and Fluid Flow* 85 (2020), 108631.
- [62] D. Poirel, Y. Harris and A. Benaissa. Self-sustained aeroelastic oscillations of a NACA 0012 airfoil at low-to-moderate Reynolds numbers. *Journal of Fluids and Structures* 24 (2008), 700–719.
- [63] R. Pasquetti and N. Peres. A penalty model of synthetic micro-jet actuator with application to the control of wake flows. *Computers and Fluids* 114 (2015), 203–217.
- [64] R. Volpe, A. Da Silva, V. Ferrand and L. Le Moyne. Experimental and numerical validation of a wind gust facility. *Journal of Fluids Engineering* 135 (1) (2013), 011106.

- [65] M. Tischmacher and B. Ruck. Interaction of gusts and forest edges – An experimental wind-tunnel study. *Forestry: An International Journal of Forest Research* 86 (5) (2013), 523–532.
- [66] C. Gromke and B. Ruck. On wind forces in the forest-edge region during extreme-gust passages and their implications for damage patterns. *Boundary-Layer Meteorology* 168 (2018), 269–288.
- [67] J. N. Wood, G. De Nayer, S. Schmidt and M. Breuer. Experimental investigation and large-eddy simulation of the turbulent flow past a smooth and rigid hemisphere. *Flow, Turbulence and Combustion* 97 (1) (2016), 79–119.
- [68] J. N. Wood, M. Breuer and G. De Nayer. Experimental studies on the instantaneous fluid–structure interaction of an air-inflated flexible membrane in turbulent flow. *Journal of Fluids and Structures* 80 (2018), 405–440.
- [69] G. De Nayer, A. Apostolatos, J. N. Wood, K. U. Bletzinger, R. Wüchner and M. Breuer. Numerical studies on the instantaneous fluid–structure interaction of an air-inflated flexible membrane in turbulent flow. *Journal of Fluids and Structures* 82 (2018), 577–609.
- [70] A. Apostolatos, G. De Nayer, K.-U. Bletzinger, M. Breuer and R. Wüchner. Systematic evaluation of the interface description for fluid–structure interaction simulations using the isogeometric mortar-based mapping. *Journal of Fluids and Structures* 86 (2019), 368–399.
- [71] S. Schmidt and M. Breuer. Source term based synthetic turbulence inflow generator for eddy-resolving predictions of an airfoil flow including a laminar separation bubble. *Computers and Fluids* 146 (2017), 1–22.
- [72] M. Breuer. Effect of inflow turbulence on an airfoil flow with laminar separation bubble: An LES study. *Flow, Turbulence and Combustion* 101 (2) (2018), 433–456.
- [73] G. De Nayer, S. Schmidt, J. N. Wood and M. Breuer. Enhanced injection method for synthetically generated turbulence within the flow domain of eddy-resolving simulations. *Computers and Mathematics with Applications* 75 (7) (2018), 2338–2355.
- [74] M. Klein, A. Sadiki and J. Janicka. A digital filter based generation of inflow data for spatially-developing direct numerical or large-eddy simulations. *Journal of Computational Physics* 186 (2003), 652–665.
- [75] P. Friederichs, M. Göber, S. Bentzien, A. Lenz and R. Krampitz. A probabilistic analysis of wind gusts using extreme value statistics. *Meteorologische Zeitschrift* 18 (6) (2009), 615–629.
- [76] M. Uhlmann. An immersed boundary method with direct forcing for the simulation of particulate flows. *Journal of Computational Physics* 209 (2) (2005), 448–476.

- [77] C. Peskin. Flow patterns around heart valves: A digital computer method for solving the equations of motion. PhD thesis. Albert Einstein College of Medicine of Yeshiva University, NY, USA, 1972.
- [78] F. Schäfer and M. Breuer. Comparison of c-space and p-space particle tracing schemes on high-performance computers: Accuracy and performance. *International Journal for Numerical Methods in Fluids* 39 (4) (2002), 277–299.
- [79] C. S. Peskin. The immersed boundary method. *Acta Numerica* 11 (2002), 479–517.
- [80] A. M. Roma, C. S. Peskin and M. J. Berger. An adaptive version of the immersed boundary method. *Journal of Computational Physics* 153 (2) (1999), 509–534.
- [81] A. Pinelli, I. Naqavi, U. Piomelli and J. Favier. Immersed-boundary methods for general finite-difference and finite-volume Navier-Stokes solvers. *Journal of Computational Physics* 229 (24) (2010), 9073–9091.
- [82] K. Zhou and S. Balachandar. An analysis of the spatio-temporal resolution of the immersed boundary method with direct forcing. *Journal of Computational Physics* 424 (2021), 109862.
- [83] K. Zhou, Z. Ding and K. Sun. Is Lagrangian weight crucial in the direct forcing immersed boundary method? *Journal of Physics: Conference Series* 1324 (2019), 012081.
- [84] Y. Bao, J. Kaye and C. S. Peskin. A Gaussian-like immersed-boundary kernel with three continuous derivatives and improved translational invariance. *Journal of Computational Physics* 316 (2016), 139–144.
- [85] W. K. Liu, S. Jun and Y. F. Zhang. Reproducing kernel particle methods. *International Journal for Numerical Methods in Fluids* 20 (8) (1995), 1081–1106.
- [86] M. Vanella and E. Balaras. A moving-least-squares reconstruction for embedded-boundary formulations. *Journal of Computational Physics* 228 (18) (2009), 6617–6628.
- [87] R. Bale, A. P. S. Bhalla, B. E. Griffith and M. Tsubokura. A one-sided direct forcing immersed boundary method using moving least squares. *Journal of Computational Physics* 440 (2021), 110359.
- [88] S. Krishnan, E. S. Shaqfeh and G. Iaccarino. Fully resolved viscoelastic particulate simulations using unstructured grids. *Journal of Computational Physics* 338 (2017), 313–338.
- [89] M. H. Abdol Azis, F. Evrard and B. van Wachem. An immersed boundary method for incompressible flows in complex domains. *Journal of Computational Physics* 378 (2019), 770–795.
- [90] M. Breuer and T. Neumann. Synchronous high-speed measurements of a flexible structure under wind gust load. *Physics of Fluids* 36 (7) (2024), 075147.

- [91] K.-U. Bletzinger, R. Wüchner and A. Kupzok. Algorithmic treatment of shells and free form-membranes in FSI. In: *Fluid-Structure Interaction*. Ed. by H.-J. Bungartz and M. Schäfer. Vol. 53. Lecture Notes in Computational Science and Engineering, LNCSE. Springer. Heidelberg, 2006, 336–355.
- [92] J. N. Wood and M. Breuer. Artificial wind gust generation based on an adaptive nozzle design. In: *Proc. der 31. Gala-Fachtagung 'Experimentelle Strömungsmechanik', Sept. 3–5, 2024*. Ed. by C. Rauh, B. Ruck and A. Leder. Berlin, 2024, 9-1–9-8.
- [93] C. Knigge and S. Raasch. Improvement and development of one- and two-dimensional discrete gust models using a large-eddy simulation model. *Journal of Wind Engineering and Industrial Aerodynamics* 153 (2016), 46–59.

Part II

Publications

Publication I: Assessment of two wind gust injection methods: Field velocity vs. split velocity method

Reference

K. Boulbrachene, G. De Nayer and M. Breuer. Assessment of two wind gust injection methods: Field velocity vs. split velocity method. *Journal of Wind Engineering and Industrial Aerodynamics* 218 (2021), 104790. DOI: [10.1016/j.jweia.2021.104790](https://doi.org/10.1016/j.jweia.2021.104790)

Copyright

© 2021 The Authors. Published by Elsevier Ltd. This is an open access article distributed under the terms of the Creative Commons CC-BY license (<http://creativecommons.org/licenses/by/4.0/>), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.



Assessment of two wind gust injection methods: Field velocity vs. split velocity method

Khaled Boulbrachene, Guillaume De Nayer, Michael Breuer*

Professur für Strömungsmechanik, Helmut-Schmidt-Universität Hamburg, D-22043 Hamburg, Germany

ARTICLE INFO

Keywords:

Field velocity method
Split velocity method
Vertical and horizontal wind gust
Feedback effect
ECG

ABSTRACT

The objective of the present paper is to revisit two well-known wind gust injection methods in a consistent manner and to assess their performance based on different application cases. These are the field velocity method (FVM) and the split velocity method (SVM). For this purpose, both methods are consistently derived pointing out the link to the Arbitrary Lagrangian Eulerian formulation and the geometric conservation law. Furthermore, the differences between FVM and SVM are worked out and the advantages and disadvantages are compared. Based on a well-known test case considering a vertical gust hitting a plate and a newly developed case taking additionally a horizontal gust into account, the methods are evaluated and the deviations resulting from the disregard of the feedback effect in FVM are assessed. The results show that the deviations between the predictions by FVM and SVM are more pronounced for the horizontal gust justifying the introduction of this new test case. The main reason is that the additional source term in SVM responsible for the feedback effect of the surrounding flow on the gust itself nearly vanishes for the vertical gust, whereas it has a significant impact on the flow field and the resulting drag and lift coefficients for the horizontal gust. Furthermore, the correct formulation of the viscous stress tensor relying on the total velocity as done in case of SVM plays an important role, but is found to be negligible for the chosen Reynolds number of the present test cases. The study reveals that SVM is capable of delivering physical results in contradiction to FVM. It paves the way for investigating further complex gust configurations (e.g., inclined gusts) and practical applications towards coupled fluid–structure interaction simulations of engineering structures impacted by wind gusts.

1. Introduction

The study of extreme events constitutes an important aspect of many engineering applications nowadays. Investigating extreme events was found to enable the design of safe components that will withstand harsh conditions and circumvent any sudden failure which may occur. In the field of fluid mechanics, wind gusts represent a typical example of such extreme phenomena which might lead to devastating consequences on flexible as well as rigid structures. For instance, aircraft design requires the study of different load cases and the evaluation of their aerodynamic responses (Wu et al., 2019). This ensures that the encounter of aircraft by wake vortices of a preceding plane (Sitaraman and Baeder, 2006; Gordnier and Visbal, 2015), a streamwise wind gust (Granlund et al., 2014) or an impulsive change in the angle of attack (Parameswaran and Baeder, 1997; Singh and Baeder, 1997) is controlled properly. In order to guarantee the safety of trains moving over bridges and to evaluate the risk of derailment, Montenegro et al. (2020) simulated the effect of discrete gusts and turbulent wind models on aerodynamic forces. That allows to determine critical wind speeds,

which is especially of interest for high-speed trains. Beside aeronautics and rail transport, the wind energy sector is constantly dealing with the design of structurally reliable wind turbine tower systems (Kwon et al., 2012) and especially wind turbine blades (for example, see Onol and Yesilyurt (2017)). A turbine blade is, like a helicopter blade, exposed to trailing-edge vortices in the wake of other blades as well as to potential gusts which may strip it away. For example, Hawbecker et al. (2017) reported on extreme structural damages to turbines of the Buffalo Ridge Wind Farm in Minnesota in 2011. Blades from multiple turbines broke away and a tower buckled in the intense winds. They attempted to characterize meteorological conditions over the wind farm area during this event and carried out weather research and forecasting model simulations of the event that considered current and anticipated future operational model setups.

More recently, lightweight thin structures such as membrane-covered and tensile-membrane structures have become widely used in civil engineering applications. Due to the unique flexible characteristics of the fabric membranes such structures allow the engineers

* Corresponding author.

E-mail address: breuer@hsu-hh.de (M. Breuer).

<https://doi.org/10.1016/j.jweia.2021.104790>

Received 6 April 2021; Received in revised form 21 September 2021; Accepted 22 September 2021

0167-6105/© 2021 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

to create visually exciting and iconic structures while maintaining a faster installation and an overall lower cost in comparison to traditional constructions. However, these structures are vulnerable to wind events and to determine to what extent the aerodynamic loads influence their integrity is essential (for example, see Gross et al. (2013)). Although design standards are prescribed to provide guidelines in order to take such events into account (for example, see Frost et al. (1978), IEC-Standard (2002), Burton et al. (2001), Kasperski (2007)), these are based on simplifying assumptions to facilitate the design process. Therefore, the development of methods capable of providing a rapid and comprehensive investigation of proposed designs would substantially contribute to the design procedure. Highly resolved numerical simulations of the flow field (and the resulting deformations) are considered to be a reliable and powerful method for this objective. That was recently shown in De Nayer et al. (2014), De Nayer and Breuer (2014), De Nayer et al. (2018) and Apostolatos et al. (2019), so far not for gusts but at least for turbulent flows around flexible membranous structures.

In the following, the description is restricted to wind gusts and methodologies to take these short but often violent phenomenon into account. One of the conventional approaches to inject wind gusts into the computational domain is to superimpose them at the inlet boundary and to allow the gusts to freely propagate through the domain. This technique is known in the literature as the far-field boundary condition (FBC) method. To avoid the violation of the global mass balance especially for incompressible fluid flows, this method requires the dynamic adaptation of the inlet velocity away from the gust to ensure a constant total mass flow through the inlet (Norris et al., 2010; De Nayer et al., 2019). Such a requirement is necessary for the convergence of incompressible flow solvers. Owing to the fact that inflow boundaries are often far away from the region of interest, a relatively coarse grid resolution is used in the upstream region to reduce the computational costs. Consequently, numerical dissipation is prevalent in these regions and the induced disturbances are strongly damped while being convected through the flow domain. To circumvent this problem, one would have to use a highly resolved grid throughout the entire computational domain, or a secondary fine moving mesh transporting the gust within the computational domain (e.g., an overset-grid technique such as the resolved gust approach of Heinrich (2014)). Nevertheless, the aforementioned remedies are computationally highly demanding leading to long simulation times.

Several approaches have been proposed to overcome the requirement of using fine grids throughout the computational domain. A category of these methods is referred to as the pure source-term formulation method. Recently, De Nayer and Breuer (2020) proposed an original method for injecting wind gusts in the computational domain based on a source-term formulation. The authors derived a source-term formulation for the momentum conservation equations which allows to place the injection region of the gust close to the region of interest. The gust can then freely propagate through the domain and, consequently, the full coupling (i.e., interaction) between the gust and the surrounding flow is captured.

Contrary to the aforementioned methods which allow gusts to freely travel after the injection phase, another family of methods which defines the position of the imposed field at each instant in time is denoted the prescribed velocity methods. There exist two prescribed velocity methods, namely, the field velocity method (FVM) and the split velocity method (SVM). The field velocity or grid velocity approach (also denoted the disturbance velocity approach by Heinrich and Reimer (2013)) was initially proposed by Parameswaran and Baeder (1997) and Singh and Baeder (1997). FVM works by incorporating gust velocities by suitably modifying the grid velocity and consequently modifying the grid time metrics. However, this modification does not lead to a real grid deformation. Hence, a pseudo grid motion is simulated without an actual grid variation. FVM could be thought of as an extension of the surface transpiration method (Sankar et al., 1986), where instead of applying the velocity correction only to the surface, it is applied to the

entire flow field. Since FVM is similar to moving grid problems, grid time metrics have to be consistently evaluated to satisfy the geometric conservation law (GCL) (Sitaraman and Baeder, 2006).

The history of FVM started in the field of aeronautics, where its main objective was to tackle the problem of evaluating indicial responses (i.e., aerodynamic loads) due to a step change in one of the influencing parameters (e.g., angle of attack or pitch rate) in the context of inviscid three-dimensional compressible flows while having full insight into the flow features (velocity and pressure distributions) (Parameswaran and Baeder, 1997; Singh and Baeder, 1997). The method could successfully solve the challenge of decoupling the step change in the angle of attack from the pitch rate, delivered results that agree with the attainable analytical results, and avoided numerical instabilities associated with this problem. Moreover, in the context of inviscid Euler equations FVM was employed to simulate the response of an airfoil penetrating through a sharp-edged vertical gust (Parameswaran and Baeder, 1997; Sitaraman and Baeder, 2000), and to model the interaction of an isolated vortex with a rotor blade (Singh and Baeder, 1996). Furthermore, FVM was used to aid the prediction of aerodynamic loads on a helicopter blade by incorporating the velocity field caused by trailed tip vortices from all other blades (Sitaraman, 2003; Sitaraman and Baeder, 2006). Nevertheless, FVM is incapable of accounting for the effect of the surrounding fluid flow on the induced field.

In contrast, SVM (originally proposed by Wales et al. (2014)) fully captures the interaction between the gust and the object of interest. SVM is based on decomposing the velocity field to a prescribed gust velocity and the remaining velocity (or background velocity as denoted by Huntley et al. (2017)). The momentum equations are effectively rearranged to be solved for the background velocity. As a result, additional source terms are derived to account for the feedback effect which is missing in FVM. Initial applications of SVM concentrated on applying 1-cosine vertical gusts on several NACA airfoils with different gust lengths and amplitudes (Wales et al., 2014). The simulations, however, were performed in the context of two-dimensional Euler equations. Later, Huntley et al. (2017) extended SVM to investigate the gust response of a flexible aircraft to a series of 1-cosine gusts in the context of three-dimensional viscous flows. Both research groups compared their results with those obtained by FVM and found SVM to be more accurate when short wavelength gusts are imposed. Recently, Biler et al. (2019), Badrya and Baeder (2019) and Badrya et al. (2021) investigated the flow physics resulting from a flat plate-gust encounter. The analysis showed the ability of analytical models (Küssner's solution (Küssner, 1936)) in predicting gust response for a 0° geometric angle of attack while the need for CFD solutions remains in order to formulate empirical formulas predicting responses for high angles of attack.

The steps in this paper are as follows:

1. Provide a comprehensive description of the FVM and SVM methods along with the modifications needed to be made in the governing equations.
2. Deliver best practice guidelines describing the implementation of the methods in a finite-volume code thoroughly.
3. Apply the methods to simulate the encounter of a vertical as well as a horizontal gust (which can be said to be relevant for many engineering applications) on a horizontal flat plate.
4. Highlight the differences among the methods and their limitations.

The main objective is to figure out the most appropriate methodology to simulate the effect of wind gusts impacting on mechanical or civil engineering structures. For the latter especially flexible membranous structures impacted by horizontal gusts are in the focus of medium-term interest, whereby such kind of complex fluid-structure interaction problems are to be simulated by high-fidelity eddy-resolving schemes such as the large-eddy simulation (LES) technique.

2. Prescribed gust methods

In the following, two well-established methods for injecting gusts into a CFD simulation by prescribing the position in the computational domain, the shape and the velocity of the gusts at each time step are introduced. In the present work the gust velocity is denoted $u_{g,i}$ ($i \in \{1, 2, 3\}$) in Cartesian coordinates. The flow field surrounding the gust is denoted as the background velocity field $\tilde{u}_i$. The Navier–Stokes equations for an incompressible fluid are solved for this background velocity. The prescribed velocity methods postulate that the divergence of the background velocity is equal to zero (i.e., $\frac{\partial \tilde{u}_i}{\partial x_i} = 0$). To present the total flow field u_i containing the resolved flow field and the prescribed gust, the sum of both contributions (i.e., $u_i = \tilde{u}_i + u_{g,i}$) has to be considered by superposition. Since the divergence of the prescribed gust velocity on a fixed grid (i.e., $\frac{\partial u_{g,i}}{\partial x_i}$) is not necessarily equal to zero, the divergence of the total velocity will not automatically be zero. In order to introduce the prescribed gust velocity into the Navier–Stokes equations while maintaining the postulate of $\frac{\partial u_i}{\partial x_i} = 0$, the Arbitrary Lagrangian Eulerian (ALE) formulation of the Navier–Stokes equations is required as described below.

2.1. Field Velocity Method (FVM)

The original idea of this method is the incorporation of unsteady flow conditions via a pseudo-grid movement in the computational domain according to the Arbitrary Lagrangian Eulerian (ALE) formulation. In this way, disturbance velocities (e.g., gust velocities) could be prescribed by changing the grid time metrics while the grid is actually not altered or distorted.

In the context of a temporally varying domain (as in fluid–structure interaction (FSI), see Breuer et al. (2012) for example) the Navier–Stokes equations (mass and momentum) are written in the Arbitrary Lagrangian Eulerian (ALE) form. For an incompressible fluid, the integral form of these equations read (here in a Cartesian coordinate system for the sake of simplicity):

$$\frac{d}{dt} \int_{V(t)} \rho dV + \int_{S(t)} \rho (\tilde{u}_j - u_j^{\text{grid}}) \cdot n_j dS = 0, \quad (1)$$

$$\begin{aligned} \frac{d}{dt} \int_{V(t)} \rho \tilde{u}_i dV + \int_{S(t)} \rho \tilde{u}_i (\tilde{u}_j - u_j^{\text{grid}}) \cdot n_j dS \\ = - \int_{S(t)} \tilde{\tau}_{ij} \cdot n_j dS - \int_{S(t)} p \cdot n_i dS. \end{aligned} \quad (2)$$

Here, the control volumes (CV) have time-dependent volumes $V(t)$ and surfaces $S(t)$. Since in this case the grid is deformable, the grid velocity with which the surface of a CV is moving, is taken into account via u_i^{grid} . $\tilde{u}_i$ describes the velocity of the flow field. A grid deformation in the inner domain will not affect $\tilde{u}_i$. However, the flow velocity $\tilde{u}_i$ changes due to movements or deformations of walls. $\tilde{\tau}_{ij}$ is the stress tensor based on the flow velocity $\tilde{u}_i$.

In the context of a temporally varying domain the ALE formulation has to additionally satisfy the so-called *space conservation law* (SCL) (Demirdžić and Perić, 1988, 1990) or *geometric conservation law* (GCL) (Lesoinne and Farhat, 1996):

$$\frac{d}{dt} \int_{V(t)} dV - \int_{S(t)} u_j^{\text{grid}} \cdot n_j dS = 0. \quad (3)$$

This extra conservation law is necessary to assure that no space is lost during the change of CVs. Employing the GCL, the contribution of the grid movement to the mass fluxes will cancel the unsteady term leading to the reduction of the mass conservation equation to:

$$\int_{S(t)} \rho \tilde{u}_j \cdot n_j dS = 0, \quad (4)$$

which is equivalent to the mass conservation equation on a fixed grid. Consequently, the pressure-correction equation in the context of

projection methods does not have to be modified. The discrete GCL is used to evaluate the additional grid fluxes in the momentum equation by evaluating the change of the position or the shape of a CV to compute the unknown grid velocity u_i^{grid} .

The field velocity approach is built per analogy based on the previously written ALE formulation. The grid is now undeformed and thus fixed, i.e., the cell volumes V and surfaces S are no longer time-dependent. Furthermore, the grid velocity u_i^{grid} is replaced by the negative gust velocity $-u_{g,i}$. Thus, the equations read:

$$\frac{d}{dt} \int_V \rho dV + \int_S \rho (\tilde{u}_j + u_{g,j}) \cdot n_j dS = 0, \quad (5)$$

$$\begin{aligned} \frac{d}{dt} \int_V \rho \tilde{u}_i dV + \int_S \rho \tilde{u}_i (\tilde{u}_j + u_{g,j}) \cdot n_j dS \\ = - \int_S \tilde{\tau}_{ij} \cdot n_j dS - \int_S p \cdot n_i dS. \end{aligned} \quad (6)$$

Even though the grid is fixed, it is still necessary to satisfy a GCL in the context of FVM due to the fact that the prescribed gust velocity component is not necessarily divergence-free. This will again enable the reduction of the mass conservation equation (i.e., Eq. (5) to Eq. (4)) and thus the pressure-correction equation still remains unaffected. Hence, it is clear that a rigorous fulfillment of the GCL will suppress any spurious source terms that might be generated. Despite the undeformed grid (V^{grid} is the real volume of the cell and taken as constant) a pseudo change in the cell volumes (denoted $\Delta V = V^* - V^{\text{grid}}$ and referred to as apparent grid movement in Sitaraman and Baeder (2006)) has to be taken into account in order to ensure the pseudo geometric conservation law:

$$\frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} dV + \int_{S^*} u_{g,j} \cdot n_j dS = 0. \quad (7)$$

Since the pseudo updated surface S^* of each CV is not available, the pseudo geometric conservation law is rewritten applying the Gaussian integral theorem:

$$\frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} dV + \int_{V^*} \frac{\partial u_{g,j}}{\partial x_j} dV = 0.$$

In a first approximation, when a second-order approximation of the volume integral such as the mid-point rule is employed in the context of a finite-volume scheme, it follows:

$$\frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} dV + \gamma \int_{V^{\text{grid}}} \frac{\partial u_{g,j}}{\partial x_j} dV = 0,$$

where the scaling factor γ is defined as $\gamma = \frac{V^*}{V^{\text{grid}}}$. A back transformation leads to:

$$\frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} dV + \gamma \int_S u_{g,j} \cdot n_j dS = 0. \quad (8)$$

The derived pseudo geometric conservation law now only depends on the pseudo updated volume V^* . S is the current real surface of the grid and thus denoted S^{grid} in the following.

Coming back to the mass and momentum conservation equations, the contributions due to the pseudo grid movement have to be evaluated on the updated volumes V^* and surfaces S^* , whereas all other terms related to the background velocity are computed on the real grid:

$$\frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} \rho dV + \int_{S^{\text{grid}}} \rho \tilde{u}_j \cdot n_j dS + \int_{S^*} \rho u_{g,j} \cdot n_j dS = 0, \quad (9)$$

$$\begin{aligned} \frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} \rho \tilde{u}_i dV + \int_{S^{\text{grid}}} \rho \tilde{u}_i \tilde{u}_j \cdot n_j dS + \int_{S^*} \rho \tilde{u}_i u_{g,j} \cdot n_j dS \\ = - \int_{S^{\text{grid}}} \tilde{\tau}_{ij} \cdot n_j dS - \int_{S^{\text{grid}}} p \cdot n_i dS. \end{aligned} \quad (10)$$

Applying the methodology mentioned above to the mass and momentum conservation leads to:

$$\frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} \rho dV + \int_{S^{\text{grid}}} \rho \tilde{u}_j \cdot n_j dS + \gamma \int_{S^{\text{grid}}} \rho u_{g,j} \cdot n_j dS = 0, \quad (11)$$

$$\frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} \rho \tilde{u}_i dV + \int_{S^{\text{grid}}} \rho \tilde{u}_i \tilde{u}_j \cdot n_j dS + \gamma \int_{S^{\text{grid}}} \rho \tilde{u}_i u_{g,j} \cdot n_j dS$$

$$= - \int_{S_{\text{grid}}} \tilde{\tau}_{ij} \cdot n_j dS - \int_{S_{\text{grid}}} p \cdot n_i dS. \quad (12)$$

Note that in analogy to the ALE approach, employing the GCL to the mass conservation equation (11) allows to cancel out the contribution of the pseudo-grid movement to the mass fluxes with the unsteady term such that the mass conservation equation again reduces to Eq. (4). Since the gust velocity is a prescribed velocity, the additional mass fluxes in the momentum equation can be directly evaluated.

For a flow solver already able to solve ALE problems, the integration of the FVM is straightforward. Further advantages and disadvantages are as follows. In FVM the gust is prescribed and the remaining field is solved, hence no numerical dissipation of the gust is appearing. Moreover, the gust shape and position can be prescribed at any time in the computational domain. It has the advantage that even when the amplitude of the gust is significantly larger than the background velocity, the simulation will converge. Despite the reasonable results delivered by FVM, it is not supported by a clear description of simplifications used in its derivation. Even though FVM accounts for the influence of the gust on the surrounding flow field, its major drawback lies on its failure to capture the effect of the surrounding on the gust itself (often denoted as the feedback effect). In addition, FVM was found to be accurate when solving the Euler equations. The reason is that the gust velocity is taken into account when the convective fluxes are calculated across cell faces. However, when solving the Navier–Stokes equations with a non-uniform gust velocity distribution, viscous flux correction is required. Gu et al. (2012) demonstrated this issue for accurately simulating practical flow phenomena (such as pitching and plunging motions of an airfoil). Their contribution relies on the principle of relative motion to show that the velocity which needs to be used for the calculation of the viscous fluxes must be a summation of the background and the gust velocity. They show that this correction is indispensable if the gradient of the gust velocity is different from zero. Consequently, the magnitude of the applied correction is depending on how steep the gust velocity profile is. The drawbacks of the original FVM are not existent when using the split velocity method (SVM) explained next.

2.2. Split Velocity Method (SVM)

The split velocity method was proposed by Wales et al. (2014). As before, the total velocity u_i is decomposed into two components, namely, a background $\tilde{u}_i$ and a prescribed gust velocity $u_{g,i}$. However, now the governing equations are written on a fixed grid based on the total velocity u_i and rearranged on that grid without the need for any simplifying assumptions. As explained below, in order to fulfill the postulate of $\frac{\partial \tilde{u}_i}{\partial x_i} = 0$, the ALE formulation of the Navier–Stokes equations is used once more including the necessary application of the pseudo GCL. The SVM has been applied to 2-D Euler equations (Wales et al., 2014) and to simulate 2-D as well as 3-D viscous flows (Huntley et al., 2016).

The formulation starts by writing the integral form of the Navier–Stokes equations on a fixed grid:

$$\int_{V_{\text{grid}}} \frac{\partial \rho}{\partial t} dV + \int_{S_{\text{grid}}} \rho u_i \cdot n_i dS = 0, \quad (13)$$

$$\begin{aligned} & \int_{V_{\text{grid}}} \frac{\partial (\rho u_i)}{\partial t} dV + \int_{S_{\text{grid}}} \rho u_i u_j \cdot n_j dS \\ &= - \int_{S_{\text{grid}}} \tau_{ij} \cdot n_j dS - \int_{S_{\text{grid}}} p \cdot n_i dS. \end{aligned} \quad (14)$$

τ_{ij} is the stress tensor based on the total velocity u_i . Substituting the decomposed total velocity u_i , the conservation equations read:

$$\int_{V_{\text{grid}}} \frac{\partial \rho}{\partial t} dV + \int_{S_{\text{grid}}} \rho (\tilde{u}_j + u_{g,j}) \cdot n_j dS = 0, \quad (15)$$

$$\int_{V_{\text{grid}}} \frac{\partial (\rho (\tilde{u}_i + u_{g,i}))}{\partial t} dV + \int_{S_{\text{grid}}} \rho (\tilde{u}_i + u_{g,i}) (\tilde{u}_j + u_{g,j}) \cdot n_j dS$$

$$= - \int_{S_{\text{grid}}} \tau_{ij} \cdot n_j dS - \int_{S_{\text{grid}}} p \cdot n_i dS. \quad (16)$$

The terms of the momentum equation are rewritten as follows:

1. Transient term (note that in the context of fixed grids, the temporal derivative inside the volume integral can be directly replaced by a temporal derivative outside the integral):

$$\begin{aligned} & \int_{V_{\text{grid}}} \frac{\partial (\rho (\tilde{u}_i + u_{g,i}))}{\partial t} dV \\ &= \int_{V_{\text{grid}}} \frac{\partial (\rho \tilde{u}_i)}{\partial t} dV + \int_{V_{\text{grid}}} \frac{\partial (\rho u_{g,i})}{\partial t} dV \\ &= \frac{\partial}{\partial t} \int_{V_{\text{grid}}} \rho \tilde{u}_i dV + \int_{V_{\text{grid}}} \rho \frac{\partial u_{g,i}}{\partial t} dV + \underbrace{\int_{V_{\text{grid}}} u_{g,i} \frac{\partial \rho}{\partial t} dV}_{\bullet}. \end{aligned} \quad (17)$$

2. Convection term:

$$\begin{aligned} & \int_{S_{\text{grid}}} \rho (\tilde{u}_i + u_{g,i}) (\tilde{u}_j + u_{g,j}) \cdot n_j dS \\ &= \int_{V_{\text{grid}}} \frac{\partial}{\partial x_j} (\rho (\tilde{u}_i + u_{g,i}) (\tilde{u}_j + u_{g,j})) dV \\ &= \int_{V_{\text{grid}}} \frac{\partial}{\partial x_j} (\rho \tilde{u}_i (\tilde{u}_j + u_{g,j})) dV + \int_{V_{\text{grid}}} u_{g,i} \frac{\partial}{\partial x_j} (\rho (\tilde{u}_j + u_{g,j})) dV \\ &+ \int_{V_{\text{grid}}} \rho (\tilde{u}_j + u_{g,j}) \frac{\partial u_{g,i}}{\partial x_j} dV \\ &= \int_{S_{\text{grid}}} \rho \tilde{u}_i (\tilde{u}_j + u_{g,j}) \cdot n_j dS + \underbrace{\int_{V_{\text{grid}}} u_{g,i} \frac{\partial}{\partial x_j} (\rho (\tilde{u}_j + u_{g,j})) dV}_{\bullet} \\ &+ \int_{V_{\text{grid}}} \rho (\tilde{u}_j + u_{g,j}) \frac{\partial u_{g,i}}{\partial x_j} dV. \end{aligned} \quad (18)$$

The summation of the terms marked by $\bullet$ represents the product of the gust velocity $u_{g,i}$ with the mass conservation equation (13) expressed by $u_i = \tilde{u}_i + u_{g,i}$:

$$\begin{aligned} & \int_{V_{\text{grid}}} u_{g,i} \frac{\partial \rho}{\partial t} dV + \int_{V_{\text{grid}}} u_{g,i} \frac{\partial}{\partial x_j} (\rho u_i) dV \\ &= \int_{V_{\text{grid}}} u_{g,i} \left(\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_j} \right) dV = 0. \end{aligned} \quad (19)$$

Hence, the momentum equation can be rewritten in its simplified form as:

$$\begin{aligned} & \frac{\partial}{\partial t} \int_{V_{\text{grid}}} \rho \tilde{u}_i dV + \int_{S_{\text{grid}}} \rho \tilde{u}_i (\tilde{u}_j + u_{g,j}) \cdot n_j dS \\ &= - \underbrace{\int_{V_{\text{grid}}} \rho \frac{\partial u_{g,i}}{\partial t} dV - \int_{V_{\text{grid}}} \rho (\tilde{u}_j + u_{g,j}) \frac{\partial u_{g,i}}{\partial x_j} dV}_{-S_{\text{momentum},i}} \\ &- \int_{S_{\text{grid}}} \tau_{ij} \cdot n_j dS - \int_{S_{\text{grid}}} p \cdot n_i dS. \end{aligned} \quad (20)$$

In this formulation the convective fluxes are described by mass fluxes, which are predicted by the total velocity (i.e., the sum of the background and the gust velocity). Furthermore, the decomposition of the velocity leads to the formation of a source term. This term models the convective effect of the object being hit by the gust on the gust itself. The source term can be written in its general form as:

$$S_{\text{momentum},i} = \int_V \rho \left\{ \frac{\partial u_{g,i}}{\partial t} + (\tilde{u}_j + u_{g,j}) \frac{\partial u_{g,i}}{\partial x_j} \right\} dV. \quad (21)$$

In the framework of fixed grids, the partial derivative in time inside the volume integral can be replaced by a total derivative outside the volume integral. Hence, the mass conservation equation (15) reads:

$$\frac{d}{dt} \int_{V_{\text{grid}}} \rho dV + \int_{S_{\text{grid}}} \rho (\tilde{u}_j + u_{g,j}) \cdot n_j dS = 0. \quad (22)$$

The transient term in Eq. (22) vanishes for an incompressible fluid implying that the divergence of the total velocity field is equal to zero. Since the prescribed velocity is not necessarily divergence-free, the postulate $\frac{\partial \tilde{u}_i}{\partial x_i} = 0$ might not be fulfilled. In order to comply with this condition, the pseudo GCL (i.e., Eq. (7)) is employed on the transient term of Eq. (22):

$$\frac{d}{dt} \int_{V^{\text{grid} \rightarrow V^*}} \rho dV = - \int_{S^*} \rho u_{g,j} \cdot n_j dS. \quad (23)$$

Substituting Eq. (23) into Eq. (22), Eq. (4) is retrieved. In a similar manner, the partial temporal derivative in time in the momentum equation (20) on a fixed grid is substituted by a total derivative and the ALE formulation is employed consistently:

$$\begin{aligned} \frac{d}{dt} \int_{V^{\text{grid} \rightarrow V^*}} \rho \tilde{u}_i dV + \int_{S^{\text{grid}}} \rho \tilde{u}_i \tilde{u}_j \cdot n_j dS \\ + \int_{S^*} \rho \tilde{u}_i u_{g,j} \cdot n_j dS = -S_{\text{momentum},i} \\ - \int_{S^{\text{grid}}} \tau_{ij} \cdot n_j dS - \int_{S^{\text{grid}}} p \cdot n_i dS. \end{aligned} \quad (24)$$

Employing equation (8) deduced in the FVM section, the final governing equations solved for SVM containing the scaling factor γ are:

$$\frac{d}{dt} \int_{V^{\text{grid} \rightarrow V^*}} \rho dV + \int_{S^{\text{grid}}} \rho \tilde{u}_j \cdot n_j dS + \gamma \int_{S^{\text{grid}}} \rho u_{g,j} \cdot n_j dS = 0, \quad (25)$$

$$\begin{aligned} \frac{d}{dt} \int_{V^{\text{grid} \rightarrow V^*}} \rho \tilde{u}_i dV + \int_{S^{\text{grid}}} \rho \tilde{u}_i \tilde{u}_j \cdot n_j dS \\ + \gamma \int_{S^{\text{grid}}} \rho \tilde{u}_i u_{g,j} \cdot n_j dS = -S_{\text{momentum},i} \\ - \int_{S^{\text{grid}}} \tau_{ij} \cdot n_j dS - \int_{S^{\text{grid}}} p \cdot n_i dS. \end{aligned} \quad (26)$$

Comparing the mass conservation equation (11) for FVM with the corresponding equation (25) for SVM, both formulations are identical and both allow a reduction to the original mass conservation equation (4) by the GCL. Obviously, the differences between both approaches arise from the momentum equations. In comparison to the field velocity approach, the evaluation of the additional source terms in Eq. (21) constitutes an important aspect of the split velocity method, as these terms enable to capture the effect of the body on the wind gust. Furthermore, in contrast to FVM, SVM uses the total velocity field for the computation of the viscous fluxes. Hence, it is suitable for the simulation of viscous flow problems. So the major drawbacks of FVM are not present in SVM. However, the implementation of SVM needs more changes in the code than FVM.

Note that even in case of LES predictions the viscous terms cannot always be neglected in comparison with the resolved stresses. For flow problems with separation for example at curved walls or with transition to turbulence, the viscous stresses are decisive for the correct prediction of the location of these phenomena and thus the entire flow development.

2.3. Boundary conditions

At the boundary patches of a computational domain, velocities have to be assigned and convective and diffusive fluxes have to be computed depending on the boundary condition type (e.g., inlet, outlet, symmetry, periodic and wall). The distinction made between the background and gust velocities in the prescribed velocity methods may act as a potential source of misconception. Therefore, special care has to be taken on the application of the boundary conditions in the context of FVM and SVM.

1. **Velocities:** In both methods, the variable for which the momentum equations are solved is the background velocity $\tilde{u}_i$. Therefore, velocities at the faces belonging to a boundary patch are assigned based on this velocity. For example, for a stationary

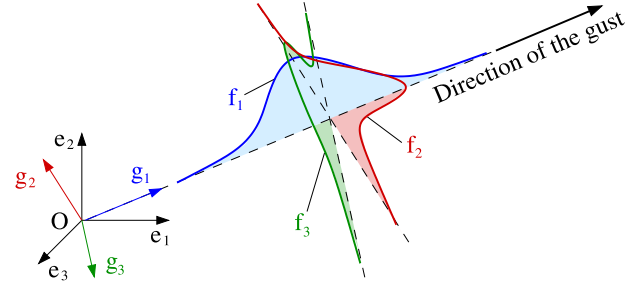


Fig. 1. Local basis definition $B_1 = (\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$ of the gust.

wall, the total velocity is equal to zero fulfilling the no-slip and impermeability conditions. Hence, the background flow velocity at the wall is set to $-\mathbf{u}_{g,i}$. Similarly, the velocities at the inlet, outlet and symmetry faces are assigned based on the primary variable $\tilde{u}_i$.

2. **Convective fluxes:** In both methods, the convective fluxes should eventually include the contribution of both velocities (i.e., the background as well as the gust velocity). For example, at a no-slip wall, mass fluxes are first computed based on the assigned background velocity and then corrected using the gust velocity. Hence, this yields a mass flux fulfilling the impermeability condition of a no-slip wall. The same treatment is applied to all boundary conditions.
3. **Diffusive fluxes:** The only difference between FVM and SVM in the application of boundary conditions lies on the computation of the diffusive fluxes. While the diffusive fluxes on boundary faces (e.g., wall shear stress of a no-slip wall or the wall normal stress of a symmetry patch) are solely computed based on the background velocity in FVM, these are evaluated using the contribution of both velocities in SVM.

3. Definition of the gust shape

The gust shape is defined in the local orthonormal basis $B_1 = (\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$ depicted in Fig. 1 assuming that the only non-zero gust velocity component is that of the first basis vector of the local coordinate system (i.e., of $\mathbf{g}_1$).

Letting $u_{g,j}$ and θ_j denote the components of the gust velocity vector in the Cartesian basis $B_0 = (\mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3)$ and in the local basis $B_1 = (\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$, respectively, the gust velocity reads:

$$\mathbf{u}_g = u_{g,1} \mathbf{e}_1 + u_{g,2} \mathbf{e}_2 + u_{g,3} \mathbf{e}_3 = \theta_1 \mathbf{g}_1 + \theta_2 \mathbf{g}_2 + \theta_3 \mathbf{g}_3 = \theta_1 \mathbf{g}_1. \quad (27)$$

The first basis vector of the local coordinate system, i.e., $\mathbf{g}_1$ is imposed by the user. $\mathbf{g}_2$ and $\mathbf{g}_3$ can be imposed or computed so that $(\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$ forms an orthonormal basis. The transformation matrix $\mathbf{T}_{B_1 \rightarrow B_0}$ from the local basis B_1 to the Cartesian basis B_0 is required.

Using the abbreviation $\beta_{ij} = \frac{\partial u_{g,j}}{\partial \theta_i}$, one could write the transformation in a matrix notation as:

$$\begin{aligned} [\mathbf{g}_i] &= [\beta_{ij}] [\mathbf{e}_j], \\ \underbrace{\begin{bmatrix} \mathbf{g}_1 \\ \mathbf{g}_2 \\ \mathbf{g}_3 \end{bmatrix}}_{\text{old basis}} &= \underbrace{\begin{bmatrix} \frac{\partial u_{g,1}}{\partial \theta_1} & \frac{\partial u_{g,2}}{\partial \theta_1} & \frac{\partial u_{g,3}}{\partial \theta_1} \\ \frac{\partial u_{g,1}}{\partial \theta_2} & \frac{\partial u_{g,2}}{\partial \theta_2} & \frac{\partial u_{g,3}}{\partial \theta_2} \\ \frac{\partial u_{g,1}}{\partial \theta_3} & \frac{\partial u_{g,2}}{\partial \theta_3} & \frac{\partial u_{g,3}}{\partial \theta_3} \end{bmatrix}}_{\mathbf{T}_{B_1 \rightarrow B_0}} \underbrace{\begin{bmatrix} \mathbf{e}_1 \\ \mathbf{e}_2 \\ \mathbf{e}_3 \end{bmatrix}}_{\text{new basis}} = \begin{bmatrix} \beta_{11} & \beta_{12} & \beta_{13} \\ \beta_{21} & \beta_{22} & \beta_{23} \\ \beta_{31} & \beta_{32} & \beta_{33} \end{bmatrix} \begin{bmatrix} \mathbf{e}_1 \\ \mathbf{e}_2 \\ \mathbf{e}_3 \end{bmatrix}. \end{aligned} \quad (28)$$

Using the relation deduced in Eq. (28), the definition of the gust velocity with respect to the physical Cartesian basis reads:

$$\mathbf{u}_g = \theta_1 \mathbf{g}_1 = \theta_1 (\beta_{11} \mathbf{e}_1 + \beta_{12} \mathbf{e}_2 + \beta_{13} \mathbf{e}_3). \quad (29)$$

In the context of deterministic gust models, the gust velocity component θ_1 describes the spatial and temporal distributions of the gust using analytic functions:

$$\theta_1(t, \xi, \eta, \zeta) = A_g f_t(t) f_1(\xi) f_2(\eta) f_3(\zeta). \quad (30)$$

A_g is the user-defined amplitude of the gust. f_t , f_1 , f_2 and f_3 are user-defined analytic functions representing the shape of the gust in time and space (in $\mathbf{g}_1$ -, $\mathbf{g}_2$ - and $\mathbf{g}_3$ -direction), respectively. ξ, η, ζ are the spatial coordinates in the local orthonormal basis ($\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3$). The local coordinates ξ, η and ζ can be expressed depending on the Cartesian coordinates (see De Nayer and Breuer (2020) for more details).

Typically used deterministic functions required by Eq. (30) for the spatial and temporal distributions related to a gust are the *Extreme Coherent Gust* (ECG) and the *Extreme Operating Gust* (EOG) defined in the IEC-Standard (2002). Other options are a Gaussian distribution or the model recently proposed by Knigge and Raasch (2016). The latter was derived from turbulent flow fields computed by LES and is supposed to describe gusts more realistically. Here, solely the ECG is applied which relies on the “1-cosine” shape. The original “1-cosine” shape found in the literature (IEC-Standard, 2002) is expressed as follows:

$$f_i(\phi) = \begin{cases} \frac{1}{2} \left(1 - \cos \left(\frac{\pi \phi}{L_g^\phi} \right) \right) & \text{for } \phi \in [0, L_g^\phi] \\ 0 & \text{else} \end{cases} \quad (31)$$

In order to achieve more control, the original shape is adapted introducing the central value ϕ_g (De Nayer et al., 2019):

$$f_i(\phi) = \begin{cases} \frac{1}{2} \left(1 + \cos \left(\frac{2\pi (\phi - \phi_g)}{L_g^\phi} \right) \right) & \text{for } (\phi - \phi_g) \in \left[-\frac{L_g^\phi}{2}, \frac{L_g^\phi}{2} \right] \\ 0 & \text{else} \end{cases} \quad (32)$$

The subscript i is equal to 1, 2, 3 or t . The variable ϕ corresponds to the coordinate ξ, η, ζ or to the time t for $i = 1, 2, 3$ or t , respectively. The constant ϕ_g is the user-defined central value of the gust distribution for the corresponding ϕ and L_g^ϕ is its user-defined length or time scale. Note that the present methodology allows to prescribe the wind gust as a three-dimensional instantaneous phenomenon.

4. Implementation into a finite-volume NS solver

4.1. Flow solver

The prescribed gust injection techniques mentioned above are incorporated into an incompressible Navier–Stokes solver, i.e., an enhanced and well validated (De Nayer et al., 2014; De Nayer and Breuer, 2014) version of FASTEST-3D (Durst and Schäfer, 1996; Breuer et al., 2012). The Navier–Stokes equations are discretized based on the finite-volume technique on a curvilinear, block-structured body-fitted grid with a collocated variable arrangement. The surface and volume integrals are approximated by the midpoint rule. Most flow variables are linearly interpolated to the cell faces leading to a second-order accurate central scheme. The convective fluxes are approximated by the technique of flux blending (Khosla and Rubin, 1974; Ferziger and Perić, 2002) to stabilize the simulation. For the current case the flux blending includes 3% of a first-order accurate upwind scheme and 97% of a second-order accurate central scheme. In order to avoid unwanted oscillations, the momentum interpolation technique of Rhie and Chow (1983) for non-staggered grids is applied to couple the pressure and the velocity fields.

A semi-implicit predictor–corrector scheme (Breuer et al., 2012) of second-order accuracy is used to solve the pressure–velocity coupling problem. First, the momentum equations are time-marched by a low-storage multi-stage Runge–Kutta method to obtain an intermediate velocity. Then, the corrector step ensures that mass conservation is achieved in form of a divergence-free velocity field. For this purpose, a Poisson equation for the pressure correction is solved by an incomplete LU decomposition method (Stone, 1968). The whole procedure provides second-order accuracy in space and time. The solver can be applied for direct numerical simulations and large-eddy simulations (Breuer et al., 2012; De Nayer et al., 2014; De Nayer and Breuer, 2014, 2020).

4.2. Additional implementations for FVM and SVM

In order to implement the field and the split velocity method into a CFD code, the gust velocity $u_{g,i}$ is prescribed in the domain at the beginning of each time step. The pseudo updated volume V^* is then computed and applied to the time discretization during the solution of the momentum equation. The momentum equation includes convective fluxes relying on $u_{g,i}$ and $\bar{u}_i$. For a three-dimensional finite-volume CFD solver working on curvilinear grids (see Section 4.1) these steps are detailed below:

- The gust shape is prescribed as described in Section 3.
- Based on the definition of the gust velocity, the volumetric flow rate Q through the faces of a finite volume due to the gust velocity reads:

$$Q = \theta_1 (\beta_{11} A_x + \beta_{12} A_y + \beta_{13} A_z). \quad (33)$$

A_x, A_y and A_z are the face area projected on $\mathbf{e}_1, \mathbf{e}_2$ and $\mathbf{e}_3$, respectively.

- In the context of the prescribed gust methods, a volume change due to a pseudo grid movement takes place if a gust velocity component varies along its direction. The pseudo cell volume V^* is computed based on the definition of the pseudo GCL according to Eq. (8):

$$\frac{d}{dt} \int_{V_{\text{grid}} \rightarrow V^*} dV = \gamma \sum_a \int_{S_a} -u_{g,j} \cdot n_j dS, \\ \frac{V^* - V_{\text{grid}}}{\Delta t} = -\frac{V^*}{V_{\text{grid}}} (Q_e + Q_w + Q_n + Q_s + Q_t + Q_b),$$

which leads to:

$$V^* = \frac{V_{\text{grid}}}{1 + \frac{\Delta t}{V_{\text{grid}}} (Q_e + Q_w + Q_n + Q_s + Q_t + Q_b)}. \quad (34)$$

{e,w,n,s,t,b} are the indices of the east, west, north, south, top and bottom faces of a hexahedral control volume, respectively.

- The additional mass fluxes due to the prescribed gust methods can now be evaluated based on the previously mentioned volumetric flow rate and the pseudo updated volume. For example, for the east face the additional mass flux $\dot{m}_e$ reads:

$$\dot{m}_e = \int_{S_e^*} \rho u_{g,j} \cdot n_j dS \approx \rho \underbrace{\frac{V^*}{V_{\text{grid}}}}_{\gamma} \underbrace{\theta_1 (\beta_{11} A_{x,e} + \beta_{12} A_{y,e} + \beta_{13} A_{z,e})}_{Q_e \text{ from Eq. (33)}} \\ = \rho \gamma Q_e. \quad (35)$$

- Similar to moving grids, the pseudo change in volume has to be also included in the integration scheme in time. In FASTEST-3D, a three-substeps second-order explicit Runge–Kutta time integration scheme is employed to march the solution in time. For a conserved quantity the momentum equation is integrated with

respect to time as follows:

$$\begin{aligned} \int_{t^n}^{t^{n+1}} \frac{d}{dt} (\rho \phi V) dt &= \int_{t^n}^{t^{n+1}} \text{RHS} dt, \\ [\rho \phi V]^{n+1} - [\rho \phi V]^n &= \Delta t \text{RHS}(\phi^n), \\ [\rho \phi]^{n+1} V^* - [\rho \phi]^n V^{\text{grid}} &= \Delta t \text{RHS}(\phi^n), \\ \phi^{n+1} &= \left(V^{\text{grid}} \phi^n + \frac{\Delta t}{\rho} \text{RHS}(\phi^n) \right) \frac{1}{V^*}. \end{aligned} \quad (36)$$

Incorporating Runge–Kutta coefficients (α_k , $k = 1, 2, 3$), the conserved quantity is advanced from the time step t^n to t^{n+1} in substeps by:

$$\phi^k = \left(V^{\text{grid}} \phi^n + \alpha_k \frac{\Delta t}{\rho} \text{RHS}(\phi^{k-1}) \right) \frac{1}{V^*}.$$

Following the work of Münsch (2015), a linear variation of the volume of the cell is assumed within the Runge–Kutta substeps:

$$\phi^k = \left(V^{\text{grid}} \phi^n + \alpha_k \frac{\Delta t}{\rho} \text{RHS}(\phi^{k-1}) \right) \frac{1}{V^{\text{grid}} + \alpha_k (V^* - V^{\text{grid}})}.$$

The momentum equation is solved for the background velocity $\tilde{u}_i$. Replacing the quantity ϕ by $\tilde{u}_i$ delivers the final time discretization scheme:

$$\tilde{u}_i^k = \left(V^{\text{grid}} \tilde{u}_i^n + \alpha_k \frac{\Delta t}{\rho} \text{RHS}(\tilde{u}_i^{k-1}) \right) \frac{1}{V^{\text{grid}} + \alpha_k (V^* - V^{\text{grid}})}. \quad (37)$$

- The prescribed gust methods can be coupled with a temporally varying domain in case of FSI. In the case of moving grids, the temporally discretized momentum equation is (Breuer et al., 2012):

$$\tilde{u}_i^k = \left(V^n \tilde{u}_i^n + \alpha_k \frac{\Delta t}{\rho} \text{RHS}(\tilde{u}_i^{k-1}) \right) \frac{1}{V^n + \alpha_k (V^{n+1} - V^n)}.$$

where V^n and V^{n+1} are the geometric cell volumes corresponding to the deformed grid at t^n and t^{n+1} , respectively. The insertion of the prescribed gust methods relies on the addition of the pseudo volume change to the geometric cell volumes of the current deformed grid, i.e., V^{n+1} . Hence, the solution is now marched in time using the updated cell volume (i.e., $(V^{n+1})^*$ evaluated by Eq. (34)) as:

$$\tilde{u}_i^k = \left(V^n \tilde{u}_i^n + \alpha_k \frac{\Delta t}{\rho} \text{RHS}(\tilde{u}_i^{k-1}) \right) \frac{1}{V^n + \alpha_k ((V^{n+1})^* - V^n)}. \quad (38)$$

The implementation of the additional source terms required for SVM is straightforward and thus needs no further explanation.

5. Test case: Flat plate with rounded edges

The aim of the present test cases originally defined in Biler et al. (2019), Badrya and Baeder (2019) and Badrya et al. (2021) is to compare the impact of gusts prescribed by FVM or SVM on the aerodynamic forces of a fixed rigid body. To allow a direct comparison with some of the results of these previous studies, the geometry of the flat plate, the Reynolds number and the properties of the vertical gusts are taken over. However, since different numerical methods (structured overset grid vs. curvilinear, block-structured body-fitted grid) are applied, the computational domain and the numerical grid have to be adjusted accordingly.

5.1. Setup and initial conditions

The geometry of the body is taken as simple as possible: A thin flat plate (chord c) with a small thickness ($h = 3 \times 10^{-2} c \ll c$) and rounded front and rear edges (half-cylinder with a radius of $h/2$) is oriented along the horizontal axis $\mathbf{e}_1 = \mathbf{e}_x$ as depicted in Fig. 2. Rounded edges have the advantage that in comparison to a sharp edge the location of separation is not fixed, which renders the test case more challenging

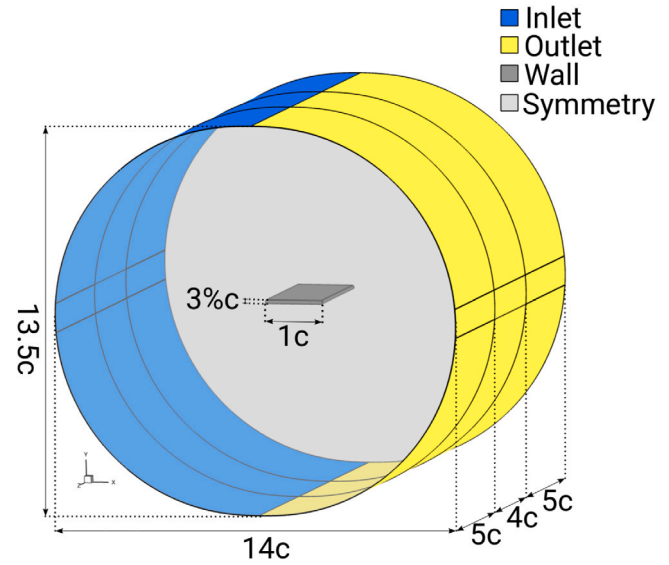


Fig. 2. Geometry of the test case and definition of the boundary conditions.

Table 1
Geometrical setup of the flat plate case.

Plate thickness	$h/c = 3 \times 10^{-2}$
Plate width	$B/c = 4$
Domain length	$D/c = 14$
Domain height	$H/c = 13.5$
Domain width	$W/c = 14$

for the prediction of viscous flows. The separation point depends on the Reynolds number chosen, which is not the case for the sharp counterpart. The plate has a finite width of $B = 4c$. The side edges of the plate are sharp. The x - y -cross-section of the computational domain is ovally shaped with its center at the center of the plate. Its large radius oriented in streamwise direction is equal to $R = 7c$ and its small radius oriented in wall-normal direction is equal to $R = 6.75c$. Thus, the boundaries of the domain are far away from the plate to avoid spurious numerical effects in the far-field when gusts encounter the body. Note that the distances are above the recommendations provided by classical best practice guidelines (Franke and Baklanov, 2007; Franke et al., 2011). In spanwise direction the domain has a width of $14c$. The plate is in the middle and two spanwise side domains with a width of $5c$ are present. Thus, the recommendations provided by classical best practice guidelines are also satisfied in the lateral direction.

In the far-field the flow has a uniform inlet velocity ($u_1^{\text{inlet}} = u_\infty$, $u_2^{\text{inlet}} = 0$, $u_3^{\text{inlet}} = 0$). No-slip and impermeability boundary conditions are applied on the surface of the plate. Symmetry boundary conditions are assumed on the lateral sides of the domain in the spanwise direction. In order to avoid reflections and disturbances at the outlet, a convective boundary condition (Breuer, 2002) is defined on the outlet patch with a convective velocity ($u_1^{\text{conv}} = u_\infty$, $u_2^{\text{conv}} = 0$, $u_3^{\text{conv}} = 0$).

The geometrical and physical characteristics of the test case are summarized in Table 1. Based on the chord length c of the plate, the far-field velocity u_∞ , the density ρ and the dynamic viscosity μ of the fluid, the Reynolds number $\text{Re}_c = \rho c u_\infty / \mu$ is set to 20,000.

One of the important aspects that has to be taken into account when generating a grid for such a test case is to guarantee a reasonably fine grid near the plate to capture the flow details near the boundary. These viscous effects are significant and lead to the formation of thin boundary layers at the wall. In order to quantify the maximum cell size in the vicinity of the wall, the flat-plate boundary layer theory is employed. For a Reynolds number of $\text{Re}_c = 20,000$, the flow is assumed to be laminar. Hence, the skin friction coefficient at the trailing edge

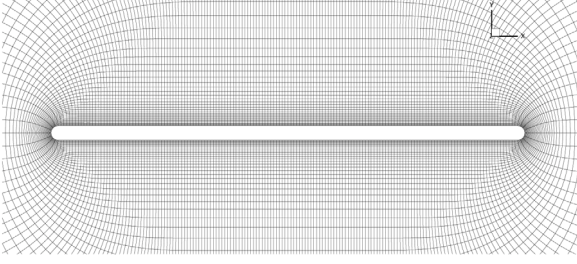


Fig. 3. x-y plane of the computational O-type grid. Only every second grid line in each direction is shown.

of the plate can be evaluated to $c_f = 0.664/\sqrt{\text{Re}_c} = 4.7 \times 10^{-3}$. The wall shear stress at this position is equal to $\tau_w = c_f \rho u_\infty^2/2$ and the friction velocity normalized by the free-stream velocity is given by $u_\tau/u_\infty = \sqrt{\tau_w/(\rho u_\infty^2)} = \sqrt{c_f/2} = 4.85 \times 10^{-2}$. Hence to resolve the near-wall region by a few grid points one could approximate the maximum height of the first grid cell off the wall (Δy) resulting in $y_{cc}^+ = 0.2$ at the cell center by $\Delta y/c = 2 \times 0.2/(\text{Re}_c \sqrt{c_f/2}) \approx 4 \times 10^{-4}$. Therefore, a dimensionless spacing of 4×10^{-4} is used in the wall-normal direction and the grid is stretched according to a geometric series with a mild stretching factor of 1.05. A zoom of the O-type grid in the x-y plane in the vicinity of the flat plate is shown in Fig. 3. Note that at the leading edge the skin friction theoretically rises to infinity. An estimation at a distance of 5% chord length from the leading edge shows that even in this region the current wall-normal resolution satisfies the condition that the first cell center is below $y_{cc}^+ = 1$.

An equidistant grid spacing of $2 \times 10^{-2}c$ is assigned in the spanwise direction to the cells of the middle block (4c) above and below the plate to capture the three-dimensional flow structures during the gust encounter. To guarantee a smooth grid transition on the side edges in the spanwise direction of the other blocks, a first cell size of $2 \times 10^{-2}c$ and a stretching factor of 1.05 are set. A summary of the fine grid on which the simulations are performed is provided in Table 2. In addition to this grid, two coarser grids are applied for a grid convergence study (see Appendix B).

To ensure the stability of the explicit time integration scheme used in the following simulations, a dimensionless time step size of $\Delta t u_\infty/c = 2 \times 10^{-4}$ is applied. That leads to a CFL number of 0.35, where in case of an incompressible fluid CFL is based on the characteristic fluid velocity u_∞ instead of the speed of sound.

The initial flow conditions are taken from the quasi steady-state flow solution in order to suppress the effects of the flow development stage. The drag and lift coefficients are chosen as the parameters indicating whether the simulation reaches its quasi steady-state solution. The dimensionless force coefficients are defined as the force components normalized by the product of the dynamic pressure (i.e., $\frac{1}{2} \rho u_\infty^2$) and the area of the plate (i.e., $A = c B$):

$$C_D = \frac{F_D}{\frac{1}{2} \rho u_\infty^2 A}, \quad C_L = \frac{F_L}{\frac{1}{2} \rho u_\infty^2 A}. \quad (39)$$

The simulation was carried out for a total dimensionless time interval of $t^* = t u_\infty/c = 10$ starting from initial conditions given by a parallel flow with $u = u_\infty$. The force coefficients recorded are shown in Fig. 4.

While the drag coefficient has converged to a value of about 0.025, the lift coefficient is fluctuating around a mean value of zero. This was expected as a result of the flow separation and the resulting vortex shedding taking place at the trailing edge of the plate depicted by the flow fields shown in Fig. 5. Note that the amplitudes of the lift variation are small compared to those expected by the vertical gusts.

5.2. Vertical gusts

The first configuration is devoted to the modeling of gusts having their direction perpendicular to the streamwise direction of the main flow. Hence, the name *vertical gust* is used in the present work. Note that wind gusts are in general three-dimensional structures. As mentioned above, the methodology described in Section 3 allows to prescribe the gust in this manner. However, for fundamental investigations canonical one-dimensional wind gusts are more suitable and are thus considered here.

A generic vertical gust configuration allows the evaluation of the rapid increase in lift forces when a structure is subjected to a vertical wind gust. These gusts are widely used in the literature (Sitaraman, 2003; Zaide and Raveh, 2006; Heinrich and Reimer, 2013; Wales et al., 2014; Ghoreyshi et al., 2018; Biler et al., 2019; Badrya et al., 2021) to investigate the response of an airfoil penetrating through a gust and consequently facilitating the design optimization process. In these investigations, a variety of gust shapes is employed. However, the current work is restricted to the application of gusts with an ECG shape not varying in time (see Heinrich and Reimer (2013), Biler et al. (2019) and Badrya et al. (2021)).

Based on the definition of the gust introduced in Section 3, the basis vector $\mathbf{g}_1$ (defining the gust direction) has to be oriented towards the basis vector $\mathbf{e}_2$. The gust velocity component θ_1 given by Eq. (30) is defined as follows. The amplitude A_g is set to 0.8 as in Badrya and Baeder (2019) and Badrya et al. (2021). Since the gust shape is independent of time, because it is purely convected downstream, the temporal shape function $f_t(t)$ is a uniform value (i.e., $f_t(t) = 1$). Along the $\mathbf{g}_1$ and $\mathbf{g}_3$ directions (i.e., the vertical and spanwise directions, respectively), the gust is uniform. Hence, a uniform value is assigned to $f_1(\xi)$ and $f_3(\zeta)$. Along the $\mathbf{g}_2$ direction (i.e., the streamwise direction in the case of a vertical gust), the ECG shape is set ($f_2(\eta) = ECG$). The center η_g of the ECG shape is convected downstream in time. In this vertical configuration, the displacement of the gust in the streamwise direction is prescribed by the convective velocity $u_{\text{conv}} = u_\infty$ leading to $\eta_g = \eta_g^0 + (t - t_0) u_{\text{conv}}$. At $t = t_0$, the gust is prescribed at the center η_g^0 . Different gust wavelengths L_g^η in $\mathbf{g}_2$ -direction are evaluated. The value of η_g^0 is chosen so that the front of the widest gust, i.e., $L_g^\eta = 4c$, starts just before the leading edge of the plate. Thus, if the origin of the frame is taken at the leading edge, $\eta_g^0 = -2.1c$. The gust characteristics of the simulations performed are summarized in Table 3.

The length scale of the gust is chosen between 0.5 and 4 times the chord length c of the plate. As mentioned in the introduction, in the intermediate run investigation of the effect of wind gusts on flexible (mechanical and civil engineering) structures are intended. Thus, this is the range of length scales which might be most important for the structural integrity and safety of the design. If the scales of the wind gusts are much smaller, the length scales can be considered as a part of the spectrum of the turbulent flow. If the scales are much larger, the corresponding flow fields are seen as long-term variations of the flow which are less critical than short-term highly instantaneous loads. The latter may lead to high tensile stresses, which might exceed the yield stress of the material and thus are critical for the safety. That explains the range of the investigated length scales.

5.3. Horizontal gusts

The second configuration is devoted to the modeling of gusts (also with an ECG shape) having their direction parallel to the streamwise direction of the main flow. Such a gust configuration is often experienced in engineering applications. For example, civil engineers are concerned with the effect of horizontal gusts on the integrity of flexible structures as well as on the stability of bridges and high-rise buildings. Moreover, energy harvesting wind turbines are also vulnerable to such gusts and this has consequently drawn the attention of the wind energy sector. A practical application in the field of aeronautics, where such a test case

Table 2
Grid information for the fine grid.

Grid	Total no. of CVs	CVs along streamwise direction	CVs in wall-normal direction	CVs in spanwise direction
Middle block	27,014,400	1008	134	200
Side blocks	2 × 7,934,400	–	–	50

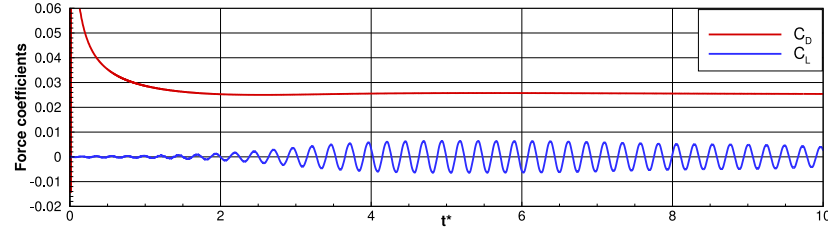


Fig. 4. Time history of the drag and lift coefficients.

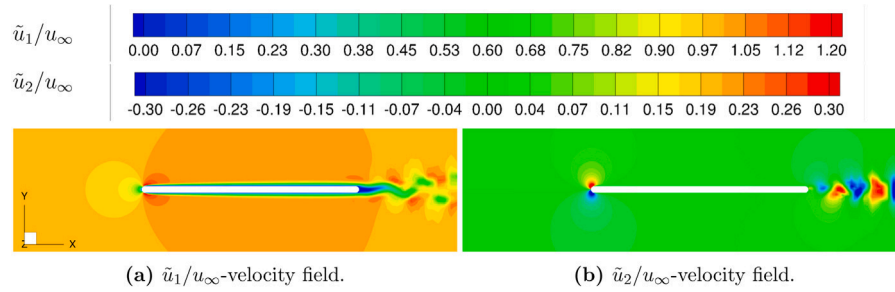


Fig. 5. Snapshot of the quasi-steady flow fields.

Table 3
Vertical gust characteristics.

$\mathbf{g}_1$	A_g	$f_t(t) = 1$	$f_1(\xi) = 1$	$f_2(\eta) = \text{ECG}$					$f_3(\zeta) = 1$
				η_g^0	L_g^η				
$\mathbf{e}_2$	0.8	–	–	–2.1 c	0.5 c	1 c	1.5 c	2 c	4 c

is critical, can be found in high-altitude and long-endurance aircraft employing high aspect-ratio wings. Such wings are susceptible to flow-induced vibrations (i.e., flutter) and associated instabilities. Subjecting such a geometry to a horizontal gust may amplify the streamwise velocity to levels exceeding the critical flutter speed.

Here, the vector $\mathbf{g}_1$ is oriented towards the streamwise direction $\mathbf{e}_1$. The horizontal gust was chosen to vary solely in the streamwise direction and the computation of the gust velocity component θ_1 was as follows. The same amplitude A_g as for the previous configuration is used (i.e., $A_g = 0.8$). Again, no gust shape variation in time is allowed. Therefore, the temporal shape function $f_t(t)$ is set to a uniform value (i.e., $f_t(t) = 1$). Along the $\mathbf{g}_1$ direction (i.e., the streamwise direction), the gust varies with an ECG shape function ($f_1(\xi) = \text{ECG}$). The center coordinate ξ_g of the ECG shape is convected downstream in time. In this configuration, the displacement of the gust in the streamwise direction is prescribed by the convective velocity $u_{\text{conv}} = u_\infty$ following $\xi_g = \xi_g^0 + (t - t_0) u_{\text{conv}}$. At $t = t_0$, the gust is introduced at the gust center coordinate ξ_g^0 . Different gust wavelengths L_g^ξ in $\mathbf{g}_1$ direction are investigated (see Table 4). As for the vertical gust, ξ_g^0 is set so that the front of the widest gust, i.e., $L_g^\xi = 4c$, starts just before the leading edge of the plate. Along the $\mathbf{g}_2$ and $\mathbf{g}_3$ directions (i.e., the vertical and spanwise directions, respectively), uniform values $f_2(\eta) = 1$ and $f_3(\zeta) = 1$ are used. As motivated in Section 5.2, the length scale of the gust is again chosen between 0.5 and 4 times the chord length c of the plate.

6. Results: FVM vs. SVM

In the following, the results of both methods are evaluated based on the vertical as well as the horizontal gust configurations. In a first step, the flow field of one specific setup with an intermediate gust length of $2c$ is visualized at different characteristic instants in time in order to illustrate the flow development when the gust encounters the plate. This also allows to work out deviations observed between the predictions based on FVM and SVM and to illuminate the resulting time histories of the lift and drag coefficients. In a second step, the full range of gust lengths is exploited and the resulting time histories of the force coefficients are evaluated.

6.1. Vertical gusts

6.1.1. Flow development at $L_g^\eta = 2c$

Fig. 7 depicts the flow field based on the vertical background velocity $\tilde{u}_2/u_\infty$, the total vertical velocity u_2/u_∞ and the pressure normalized by ρu_∞^2 . Six instants in time are chosen to present the development of the flow. At the first snapshot (see Fig. 6, time 1) the vertical gust is still in front of the plate and the ECG shape just reaches the leading edge of the plate. As expected in both predictions the velocity field is the same as shown in Fig. 5 for the quasi-steady initial conditions. At the second snapshot (see Fig. 6, time 2) the center of the gust is at the leading edge of the plate as visible in the contours of the total vertical velocity. The gust hits the plate from below leading to a flow separation

Table 4
Horizontal gust characteristics.

g_1	A_g	$f_1(t)=1$	$f_1(\xi)=\text{ECG}$					$f_2(\eta)=1$	$f_3(\zeta)=1$
			ξ_g^0	L_g^ξ					
e_1	0.8	–	$-2.1c$	$0.5c$	$1c$	$1.5c$	$2c$	$4c$	–

on the top and the development of a strong vortical clockwise rotating structure as visible in the pressure minimum in the vicinity of the leading edge. At this stage already some first small deviations between the results achieved by FVM and SVM are obvious. The third instant in time (see Fig. 7, time 3) is the one at which the plate experiences the largest lift force. The center of the ECG shape is at a distance of about $0.2c$ from the leading edge. A second pressure minimum indicating a further strong clockwise rotating vortical structure can be seen close to the leading edge. Again small deviations between the results of FVM and SVM are apparent, but the differences have not grown visibly. At the fourth snapshot (see Fig. 7, time 4) the center of the gust is located at the center of the plate. The two leading-edge vortices (LEV) are convected downstream in comparison to time 3. At this phase the lift force is already decreasing again. At the fifth instant in time (see Fig. 8, time 5) the center of the gust reaches the trailing edge of the plate. Now the first LEV is located further downstream and its strength has significantly decreased, whereas the second vortical structure is hardly visible in the FVM prediction but still apparent in the SVM results. At the last snapshot (see Fig. 8, time 6) the gust already detaches from the plate completely. Now the deviations between the FVM and SVM predictions are the largest. Nevertheless, the deviations observed between FVM and SVM are rather small. This is especially true for the pressure field. The reason for this behavior is the nearly vanishing source term in the SVM prediction in case of the vertical gust (see Appendix A.1). As will be shown below, the source term is not disappearing in the horizontal gust case and thus significant differences between FVM and SVM results occur. Furthermore, in case of the vertical gust no unphysical flow phenomena are observed either for FVM nor for SVM.

6.1.2. Force coefficients and effect of gust length

Beside the focus of this configuration to illustrate the effect of the gust width on the forces of the flat plate, it was also used to validate the implementation of SVM in FASTEST-3D based on the results in the literature (Badrya and Baeder, 2019; Badrya et al., 2021). Fig. 9 shows the agreement achieved between the aerodynamic responses predicted by SVM using FASTEST-3D and those from the literature. Here, the aerodynamic responses of the plate are plotted as a function of the dimensionless time t^* .¹ Small deviations (see Table 5) are observed among the responses and this could be related to the use of different discretization schemes of the underlying equations. For instance, Badrya and Baeder (2019) and Badrya et al. (2021) employed the Monotonic Upstream-centered Scheme for Conservation Laws (MUSCL) for the computation of the inviscid terms. The use of flux limiters makes the solution total variation diminishing and, therefore, spurious oscillation free. On the contrary, FASTEST-3D relies on a second-order accurate central scheme with 3% flux blending of a first-order accurate upwind scheme. Such a low-dissipative scheme is favored since it does not damp out small-scale structures typically occurring in large-eddy simulations which are the final objective of this study. Obviously, such small-scale structures develop in the present simulations leading to local maxima for example in the time history of the lift coefficient. Despite the small deviations between the present results and the literature data (see Table 5), a good arrangement is achieved for the purpose of SVM validation.

¹ $t^* = u_\infty t/c$ is the convective time defined in terms of the number of chords traveled by the gust.

At $t^* = 0$, the plate-gust interaction commence at the leading edge of the plate yielding an increase in the lift and a drop in the drag coefficients. While the rise in the lift coefficients is a consequence of an increase in the effective angle of attack of the plate, the drop in the drag coefficients is due to the formation of a clockwise rotating leading-edge vortex (LEV). The elevation in the lift coefficients continues until a peak value is reached. Then, these coefficients start decaying to eventually recover the steady-state values when the gusts have completely left the plate. On the other hand, drag coefficients keep on decreasing until the leading edge vortex has reached its largest size. Finally, the steady-state values of the drag coefficient are retrieved following the convection of the leading-edge vortex downstream along the chord length of the plate.

In Fig. 10 the aerodynamic responses of the plate due to the gust encounter predicted by FVM are plotted against those delivered by SVM. One could notice that only some minor deviations are present between the results of the two methods (see Table 5). These deviations can be detected in the lift coefficients of gusts with short wavelengths (e.g., $0.5c$, $1c$ and $1.5c$), where FVM predicts a second local peak when the gusts have entirely left the plate.

Based on the results found, one could deduce three main effects of the gust width on the aerodynamic responses appearing in FVM and SVM:

1. The rate at which the forces change: It could be observed that the shorter the gust is, the higher is the rate at which the forces increase. This could be related to the fact that a gust with a shorter wavelength has larger gradients along its profile, and therefore, leads to higher force rates.
2. The peak forces recorded: The instantaneous lift coefficient steadily increases to reach a maximum value which is depending on the gust width. In fact, the maximum lift coefficient non-linearly increases with the gust width as found in (Biler et al., 2019) and shown in Fig. 11(a), where the slope decreases for larger gust lengths. This behavior is expected since for a wider gust, the plate experiences a higher effective angle of attack along the chord for a longer period of time. Fig. 11(a) also shows that the current results of both SVM and FVM are in close agreement with the predicted data by Badrya et al. (2021) (see Table 5).
3. The instant in time at which the peaks are recorded: It was found that the peak lift occurs when the center of the gust is located between the leading edge and the mid-chord of the plate. Since wider gusts require more time to reach this region, their resulting maximum peak values occur at later times. It was also found in Biler et al. (2019) that the convective time associated with the maximum lift linearly increases with the gust width as shown in Fig. 11(b). Again the present results are found to be in good agreement with the predictions in Badrya et al. (2021) (see Table 5).

6.2. Horizontal gusts

6.2.1. Flow development at $L_g^\xi = 2c$

The flow fields induced by a prescribed horizontal gust of $L_g^\xi = 2c$ are depicted in Figs. 12 and 13 at six instants in time using the streamwise background velocity $\bar{u}_1/u_\infty$, the streamwise total velocity u_1/u_∞ and the pressure normalized by ρu_∞^2 . Fig. 14 highlights the differences in the results between both methods at the same time

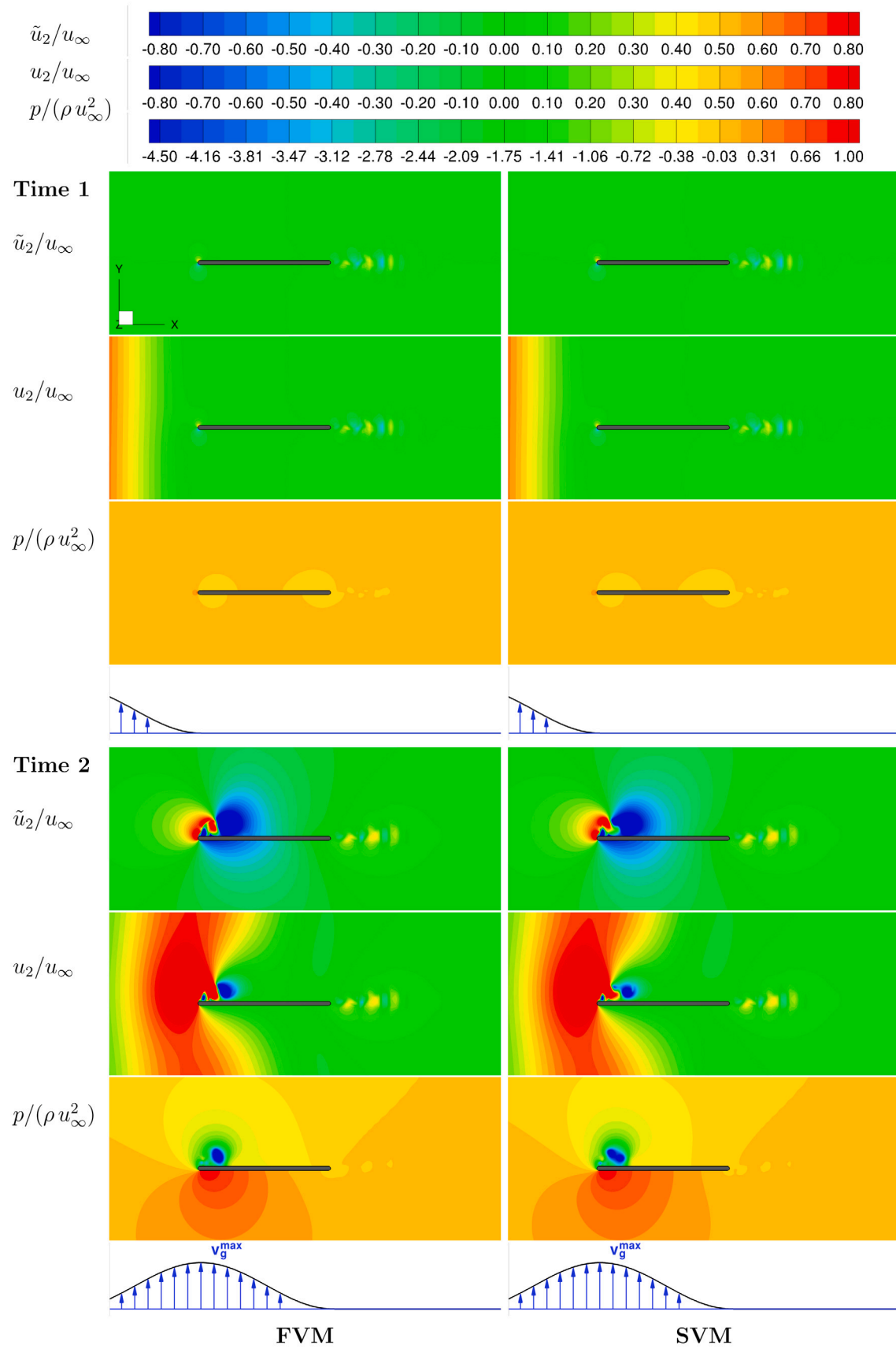


Fig. 6. Vertical gust passing around the rounded flat plate: Predictions based on FVM (left) and SVM (right) at time 1 and 2.

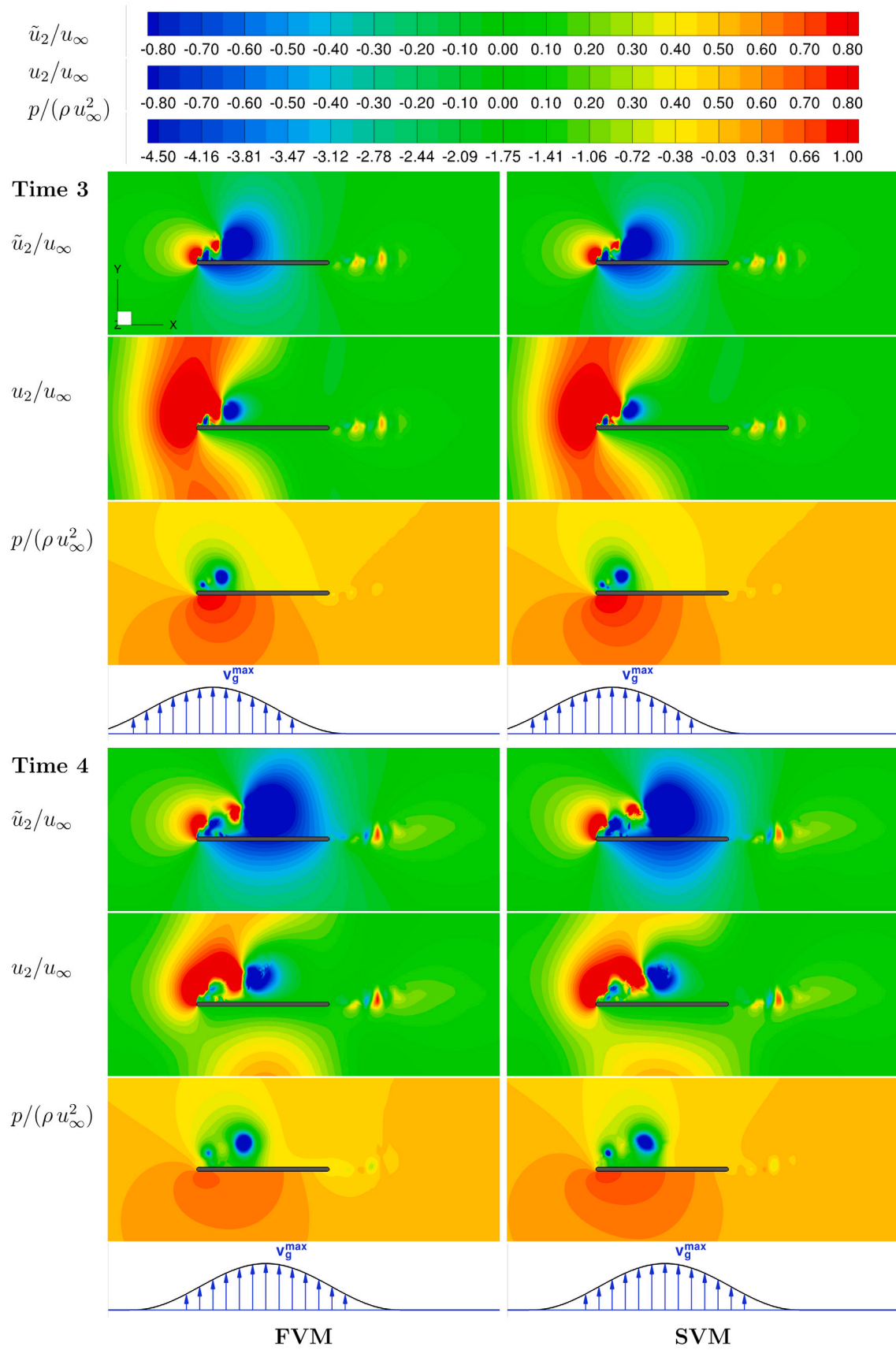


Fig. 7. Vertical gust passing around the rounded flat plate: Predictions based on FVM (left) and SVM (right) at time 3 and 4.

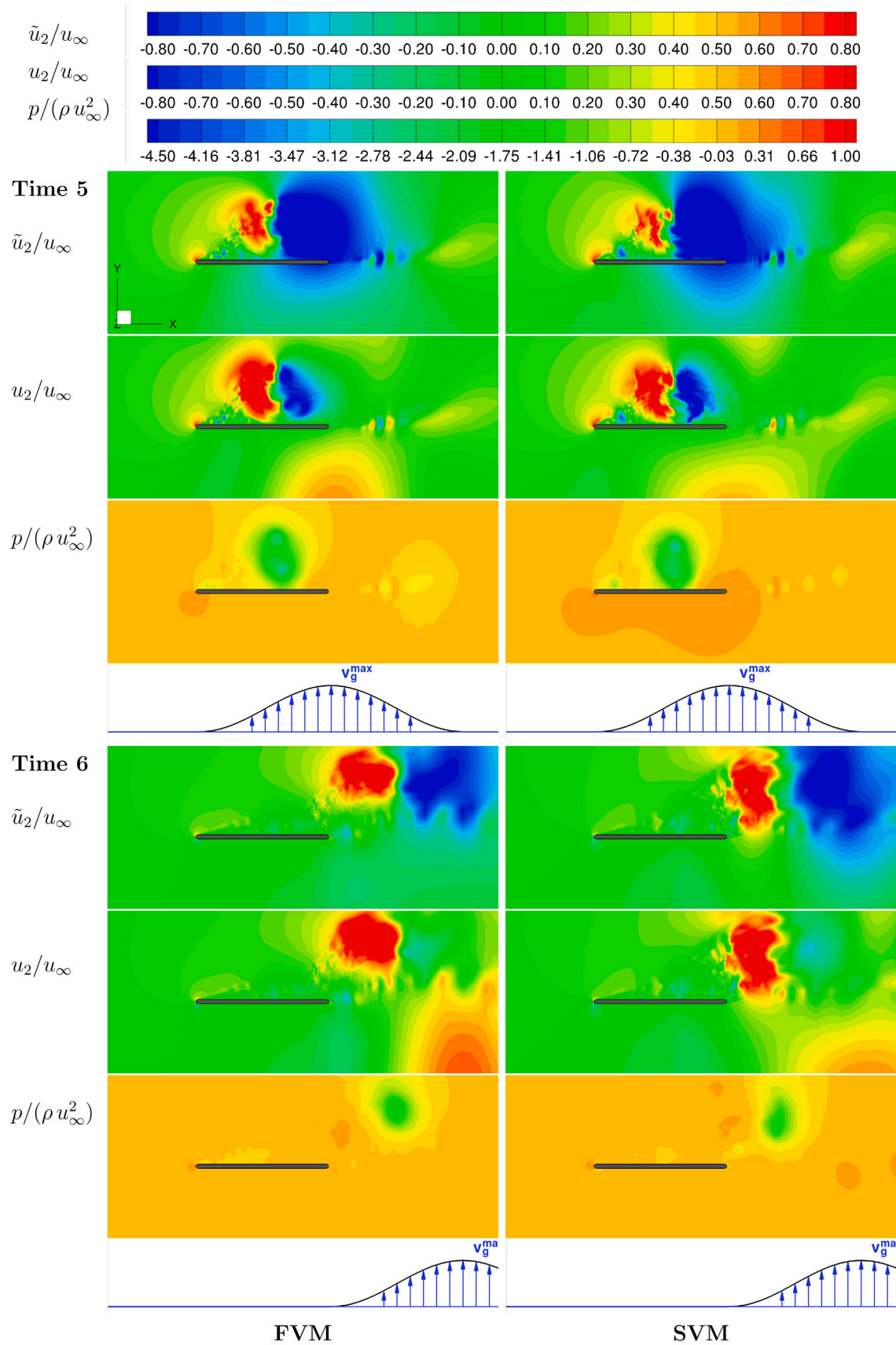


Fig. 8. Vertical gust passing around the rounded flat plate: Predictions based on FVM (left) and SVM (right) at time 5 and 6.

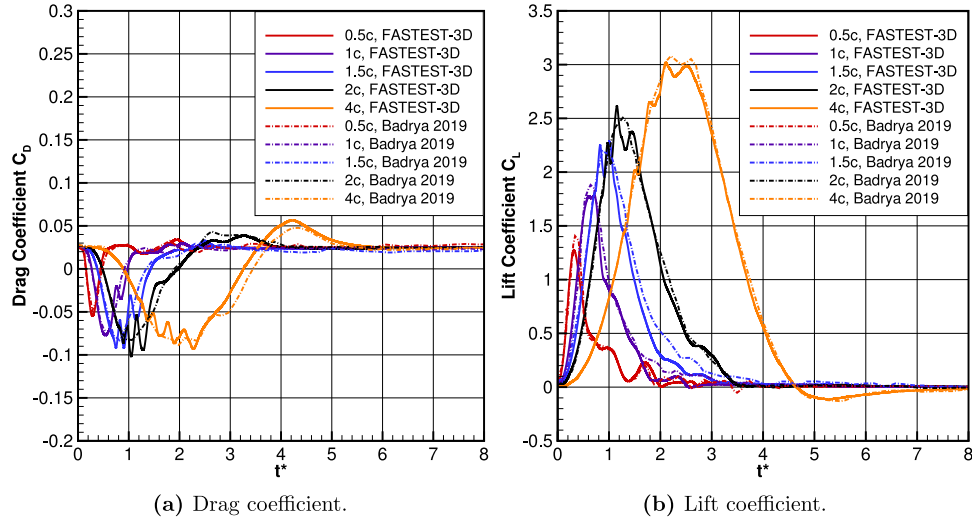


Fig. 9. Vertical gust passing around the rounded flat plate: Instantaneous drag and lift coefficients. The gust shape is ECG with a gust amplitude of $A_g = 0.8$: FASTEST-3D vs. Badrya and Baeder (2019) SVM results.

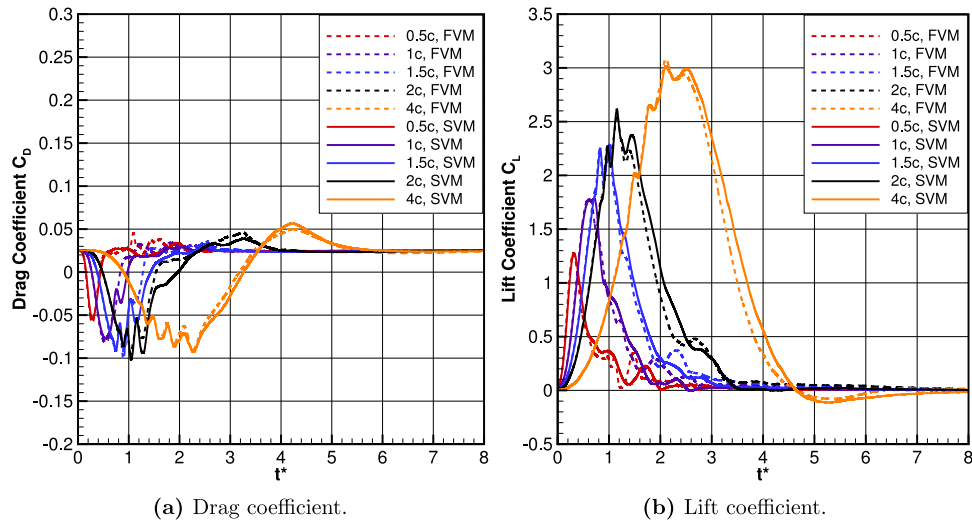


Fig. 10. Vertical gust passing around the rounded flat plate: Instantaneous drag and lift coefficients. The gust shape is ECG with a gust amplitude of $A_g = 0.8$: FVM vs. SVM.

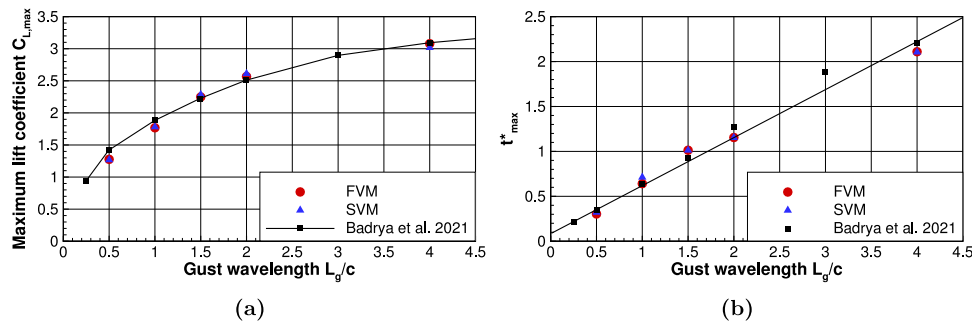


Fig. 11. Vertical gust passing around the rounded flat plate: Lift coefficient analysis: (a) Maximum lift coefficient as a function of the gust wavelength and (b) Convective time corresponding to the maximum lift coefficient as a function of the gust wavelength.²

instants. Please note that the current six time instants are partially different from those used for the vertical gust configuration.

² For FVM and a gust wavelength of $1.5c$ the second local maximum is used.

At the first snapshot (time 1) the horizontal gust has reached the front of the plate and the first quarter of the gust has already penetrated the leading edge of the plate. The predictions (FVM and SVM) show that the streamwise background velocity field and the streamwise total velocity field (see Fig. 12, time 1) are very similar to each other

Table 5

Normalized root-mean-square error of the lift and drag coefficients found in Fig. 9 (present simulation vs. Badrya and Baeder (2019)) and Fig. 10 (FVM vs. SVM). The relative errors are normalized by the corresponding ranges (maximum–minimum).

Case	Normalized RMSE [%]	0.5 c	1.0 c	1.5 c	2.0 c	4.0 c
Present sim. vs. B&B	ΔC_D	5.0	4.9	6.3	4.9	4.7
	ΔC_L	5.0	4.6	4.6	2.8	1.3
FVM vs. SVM	ΔC_D	3.5	4.6	4.4	4.3	2.4
	ΔC_L	3.5	3.3	2.2	3.3	3.5

and also very similar to the quasi-steady flow shown in Fig. 5. However, strong differences are visible in the pressure distribution (see Fig. 14(a)). In case of SVM an area of negative pressure forms at the prescribed gust location³. Obviously, the observed pressure distribution is due to the contribution of the SVM source term (see Appendix A.2 for further details), since a similar observation cannot be made for the FVM prediction, which does not use such a source term. A region with a strong pressure drop at the location of the gust is reasonable, since an increase of the total streamwise velocity occurs. Therefore, the pressure sinks. At the second snapshot (Fig. 12, time 2) the center of the gust is exactly at the leading edge of the plate. A minor augmentation of the background velocity is observed at the leading edge. This acceleration of the flow is more pronounced in the SVM results. The previously mentioned area of the pressure drop follows the gust prescribed by SVM. For FVM a locally occurring pressure reduction is observed near the leading edge, where the flow is locally accelerated during the deflection around the leading edge. That is also the case in the SVM prediction. However, it is not visible due to the presence of the large region of negative pressure. Contrary to the vertical gust configuration the gust hits a much smaller area of the plate. Therefore, no strong vortical structure is shed in the vicinity of the front of the plate. At the third instant in time (Fig. 12, time 3) the center of the gust is located at the center of the plate. The largest deviations are again found in the pressure. Assuming that the wide area of negative pressure following the gust observed in SVM is physical (velocity increases, thus the pressure decreases), the pressure field connected with FVM is more doubtful. The gust velocity prescribed by FVM does not directly influence the pressure. Only two negative pressure areas are observed at both plate ends, where the flow is accelerated due to the deflection induced by the plate. In the middle of the structure, where the gust is the strongest, only a mild decrease of the pressure is visible. Contrary to time 1 and 2, the streamwise background velocity field predicted by FVM and SVM starts to differ in the vicinity of the plate and in the wake (see Fig. 14(c)). The additional source term and the evaluation of the shear stress tensor based on the total velocity are the two decisive differences characterizing SVM.

In order to track which of these two issues is responsible for the difference observed in the background velocity, an altered SVM simulation, where the shear stress tensor is evaluated based on the background velocity (as for FVM) instead of the total velocity, is carried out. Since the streamwise background velocity field predicted by SVM and the altered SVM are in close agreement, it can be concluded that the SVM source term is responsible for the change in the streamwise background velocity field near the plate. The SVM source term generates the previously mentioned pressure drop area, which implies an acceleration of the flow in the tail of the gust and a deceleration in front of the gust (particularly visible near the plate in Fig. 14(c)).

The gust reaches the trailing edge of the plate in the fourth snapshot (Fig. 13, time 4). Looking at the pressure field, where the changes are most obvious, it can be noticed that the vortex shedding at the trailing edge becomes stronger in both simulations. This can be further

observed at the fifth instant in time (Fig. 13, time 5), when the plate is exposed to the last quarter of the gust. In this snapshot the background velocity predicted by FVM strongly differs from the SVM results (see Fig. 14(e)). Several short local areas of increased streamwise velocity connected with negative pressure areas are present in the vicinity of the plate in case of FVM (see Fig. 13, time 5). These small vortices are located near the plate in the tail of the gust. These strong unphysical vortical structures are neither present in SVM nor in the altered SVM. The physical explanation is that the acceleration of the flow field generated by the SVM pressure drop area induced by the source term in the tail of the gust avoids the formation of these vortices. At the last snapshot (Fig. 13, time 6) the gust detaches from the plate completely. In case of SVM the flow looks very similar to the quasi-steady case (see Fig. 5). However, the unphysical vortices are further present in the FVM simulation (see Fig. 13, time 6 and Fig. 14(f)).

In summary, the test case of the horizontal gust reveals two findings, which are not so clearly visible in the vertical gust case. These are directly related to the differences in the formulation of the FVM and SVM methods. First, the additional source term in SVM is responsible for a meaningful forecast of the pressure field and consequently of the background velocity field near the structure (feedback effect). Second, the difference in the formulation of the viscous shear stresses between FVM and SVM does not have a major effect on the predictions, at least for the present configuration. This could be related to the fact that the diffusive contribution of the gust velocity (i.e., $\tau_{ij}(u_{g,i})$) is depending on two main parameters, namely, the gust profile and the reciprocal of the Reynolds number. In the present case, the product of the spatial derivative of the gust profile with the reciprocal of the Reynolds number has an order of magnitude which is significantly smaller than that of the SVM source term.

6.2.2. Force coefficients and effect of gust length

Similar to the vertical gust case Fig. 15 depicts the time histories of the drag and lift coefficients for the case of the horizontal gust. To ensure a better visibility of the results, separate subfigures are provided for each gust length. At the dimensionless time instant $t^* = 0$ the branch of the gust reaches the leading edge of the plate. As visible from the drag coefficient depicted in Fig. 15(a), the duration of the gust-plate interaction lasts slightly longer than about $1 + L_g^*/c$ convective time units, which is the time the gusts needs to travel completely over the plate. Obviously, the deviations observed between the drag coefficients predicted by FVM and SVM are significantly larger than for the vertical gust case. Furthermore, the lift coefficient has obviously less importance. However, it is directly connected to the van Kármán vortex shedding occurring at the trailing edge. The frequency of the vortex shedding rises during the passage of the gust over the plate. Assuming a constant Strouhal number of the shedding process the rise of the frequency can be explained by the increase of the velocity due to the horizontal gust.

In order to better understand the time history of the drag coefficient, the pressure part of the drag coefficient C_D^p and the shear part of the drag coefficient C_D^r are separately plotted in Fig. 16 for FVM and SVM for the gust length of $L_g^* = 2c$. It is obvious that the temporal evolution of the drag coefficient C_D predicted by FVM mainly corresponds to the temporal evolution of its shear contribution. On the contrary, the curve of C_D obtained by SVM is directly connected to the pressure part. This

³ Note that the reference point of the pressure is located just after the inlet patch at the height of the plate. Thus, a negative pressure has to be interpreted as the difference to the pressure at the reference point.

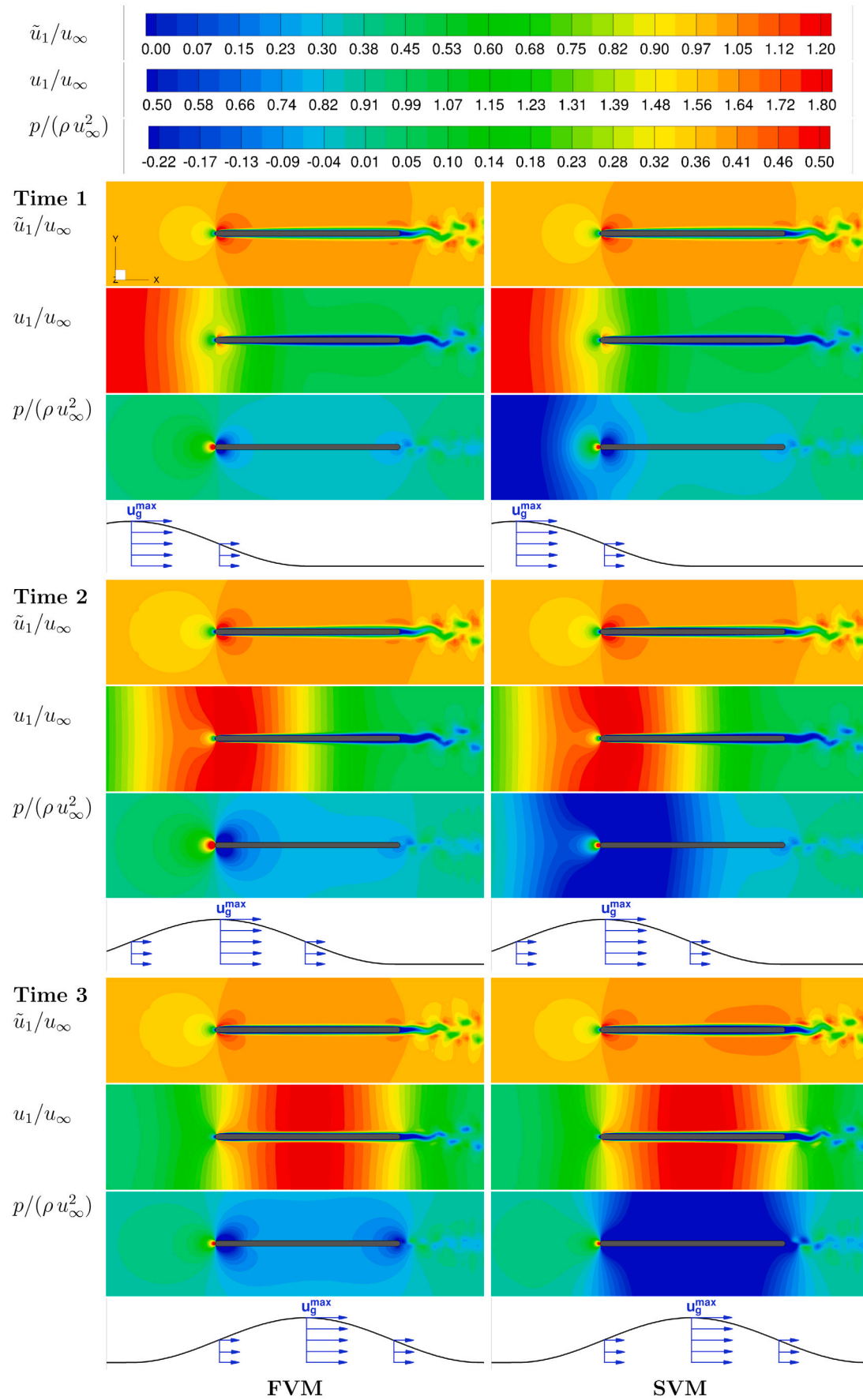


Fig. 12. Horizontal gust passing around the rounded flat plate: Predictions based on FVM (left) and SVM (right) at time 1, 2 and 3.

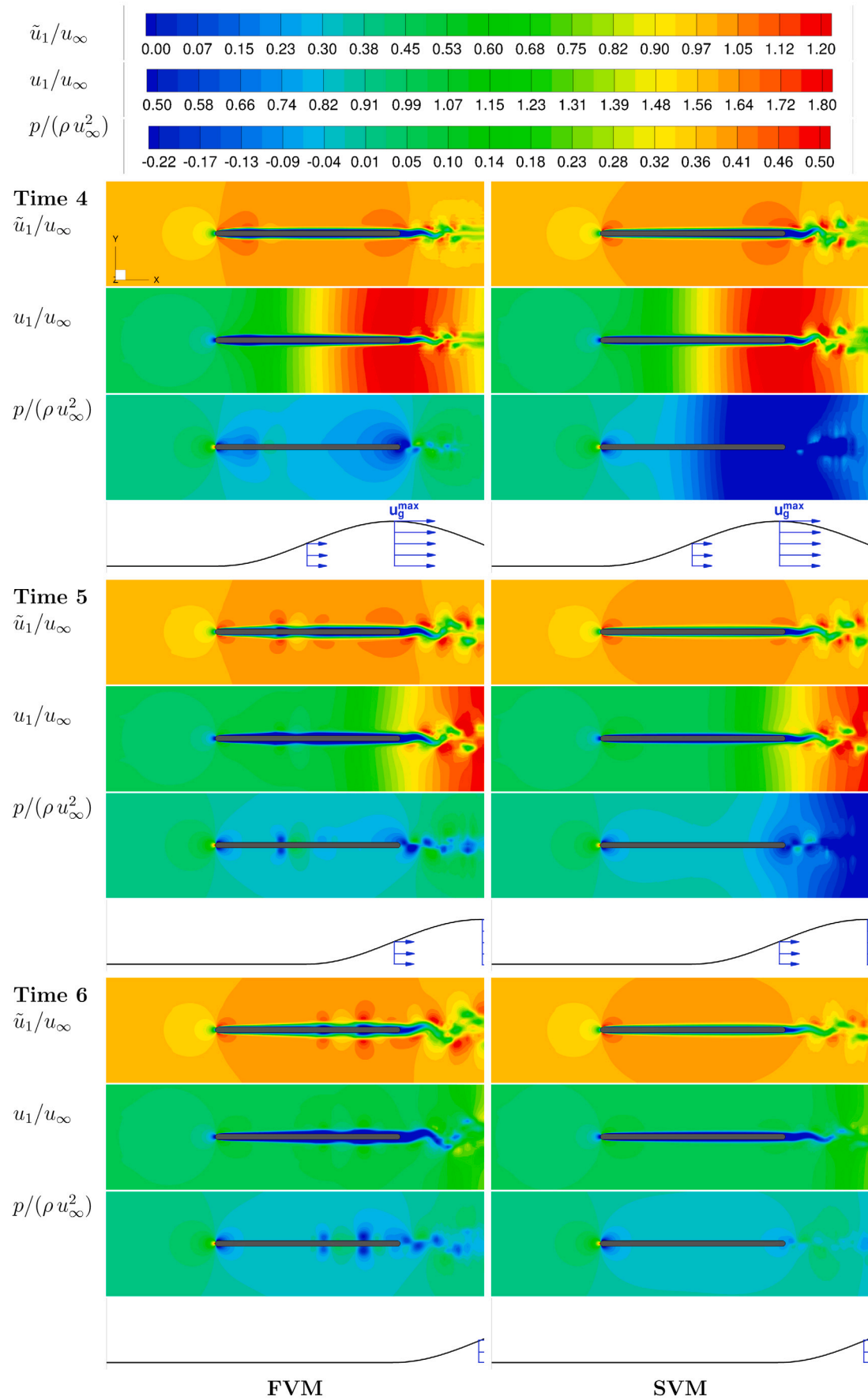


Fig. 13. Horizontal gust passing around the rounded flat plate: Predictions based on FVM (left) and SVM (right) at time 4, 5 and 6.

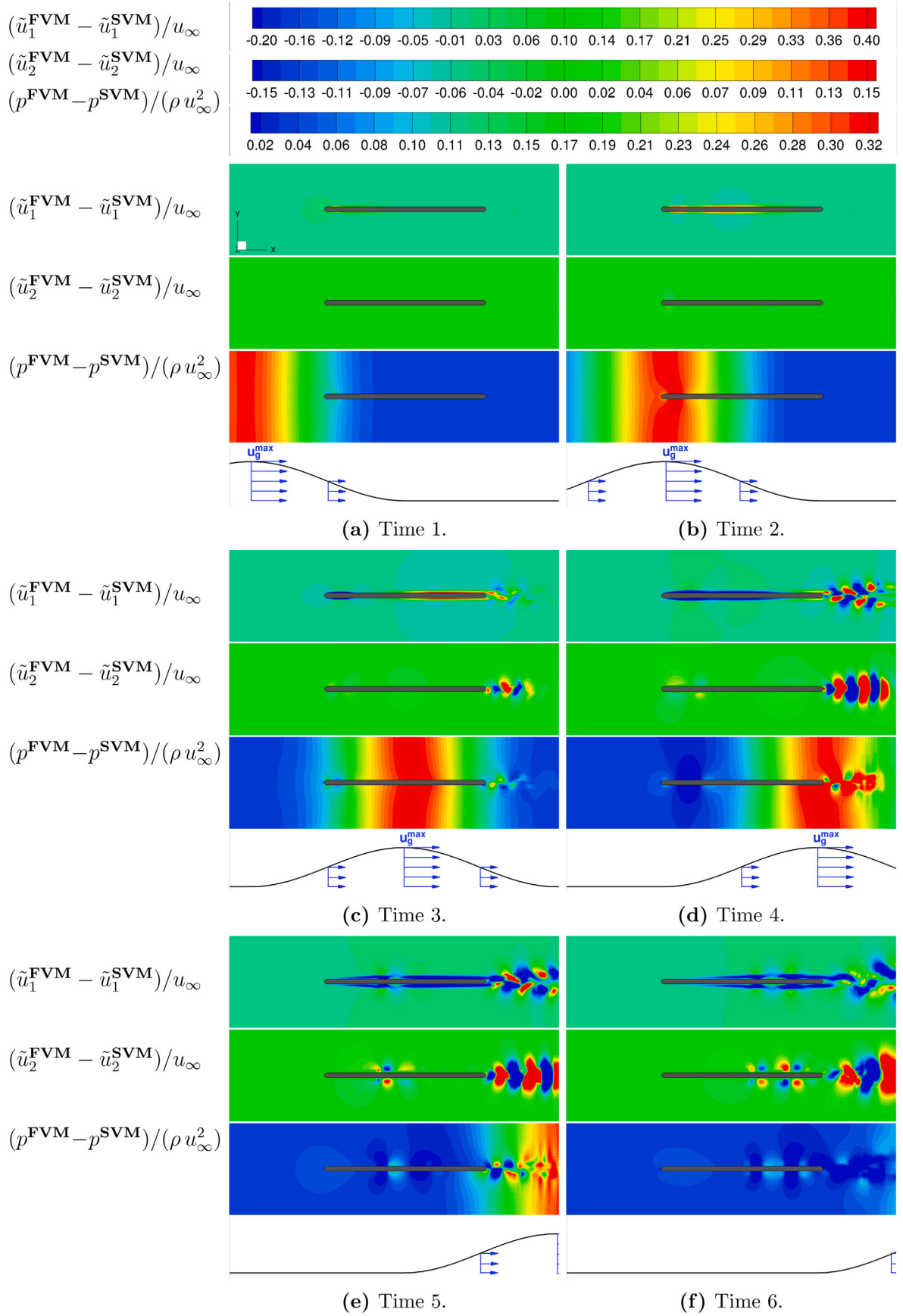


Fig. 14. Horizontal gust passing around the rounded flat plate: Flow differences between FVM and SVM at six instants in time.

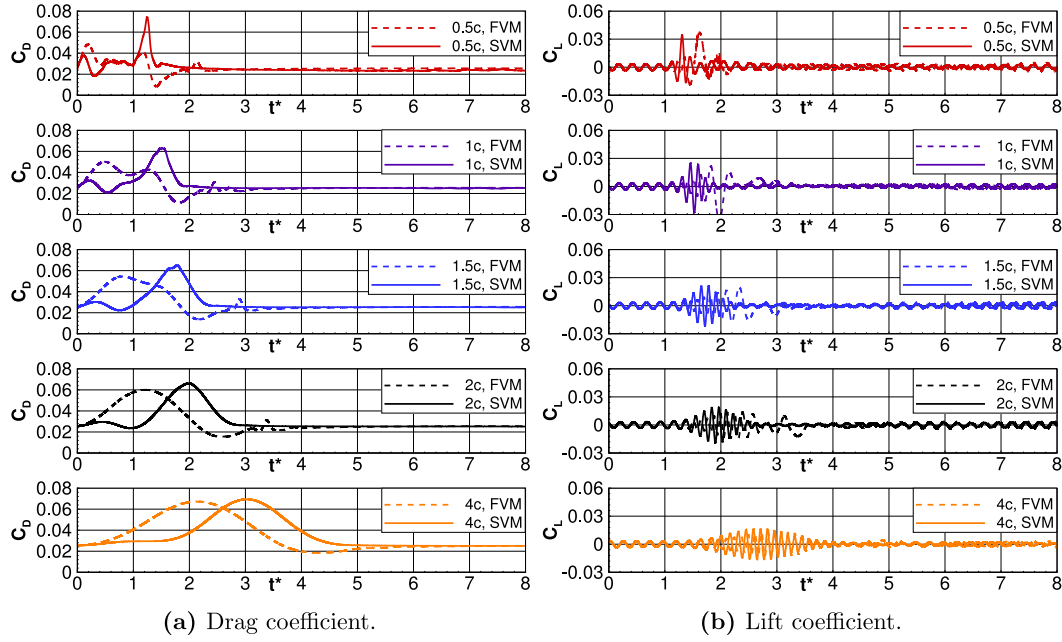


Fig. 15. Horizontal gust passing around the rounded flat plate: Instantaneous drag and lift coefficients for different gust lengths.

fits to the observations made on the flow fields in the previous section. The SVM source term significantly influences the pressure distribution at the gust location and this has a strong impact on the temporal evolution of the pressure part of the drag coefficient.

At $t^* = 1$, the center of the gust reaches the leading edge. Consequently, the maximum pressure drop is also located at the leading edge. Therefore, a decrease in the drag coefficient is observed at this time instant. At $t^* = 2$, the center of the gust reaches the trailing edge and the pressure drop is located at this end of the plate. Thus, a maximum of the drag coefficient occurs at this instant in time. Of further interest is also the temporal evolution of the shear stresses. In case of SVM the shear stress part of the drag coefficient looks like the shape of the gust. On the contrary, the time history of C_D^r obtained by FVM reveals that the shear part of the drag coefficient temporarily drops below the quasi-steady value. This observation is made for the second half of the period of the gust-plate interaction and can hardly be explained by the physics of the event. The additional source term and the evaluation of the shear stress tensor based on the total velocity are the two main differences between FVM and SVM.

As in the previous section, the altered SVM simulation, where the shear stress tensor is evaluated based on the background velocity (as for FVM) instead of the total velocity is applied to clarify the source of the changes. The temporal evolution of the force coefficients predicted by SVM and the altered SVM are quasi identical. This means that the SVM source term is responsible for the change in the C_D^r shape. As explained in the previous section, this outcome is expected as the SVM source term contribution is more pronounced.

Please note that the evaluation of the shear stress tensor based on the total velocity does not have a major impact in the present case owing to the currently selected gust configuration. Indeed, the gust shape only changes in $\mathbf{g}_1$ -direction. If the gust would also vary in $\mathbf{g}_2$ - and/or $\mathbf{g}_3$ -direction, a shear stress tensor based on the total velocity would have a stronger effect on the results.

7. Conclusions

The field velocity method and the split velocity method are wind gust injection methods which are often used in the literature. Nevertheless, a comprehensive and comprehensible derivation in the context of a finite-volume scheme applying the Arbitrary Lagrangian Eulerian

formulation and the geometric conservation law was missing. Such a consistent derivation is provided here. This works out the similarities between the two approaches but also their differences. Their effects on the results are assessed based on vertical and horizontal gusts hitting a plate. The following results can be recorded:

- The derivation has highlighted that both methods are strongly linked to the Arbitrary Lagrangian Eulerian formulation and the geometric conservation law. The difference between both approaches are on the one hand the additional source terms in SVM describing the feedback effect of the surrounding flow field on the gust itself and the formulation of the viscous stress tensor by the background and the total velocity, respectively.
- From the numerical point of view and the required computational effort both methods are more or less equivalent. Therefore, using FVM does not really make sense anymore. This statement is supported by the computational results achieved by both methods.
- For the classical case of a vertical gust often used in the context of aircraft, the deviations in the results (flow field and force coefficients) are rather small, i.e., the normalized root-mean-square errors are in the range between 2.4% and 4.6%. An analytical derivation revealed that the source term vanishes in the regions where the dimensionless background velocity $\bar{u}_1$ is equal to unity (e.g., the far-field region), which explains its minor impact on the predictions in the vertical gust case.
- For the newly introduced horizontal gust hitting a plate the observed differences both in the flow field and in the resulting force coefficients are significantly more pronounced. The non-vanishing source term as well as the modified formulation of the viscous shear stresses can be made responsible for this observation. However, an additional study clearly showed that for the chosen Reynolds number the consideration of the feedback effect due to the source term is the decisive difference between FVM and SVM, which prevents unphysical flow structures from being observed on the plate.

The significance of the outcome of this study lies on the assessment of the capability of the prescribed velocity methods to introduce wind gusts of different configurations. Especially the horizontal gust case reveals the shortcomings of the underlying assumptions used in FVM and

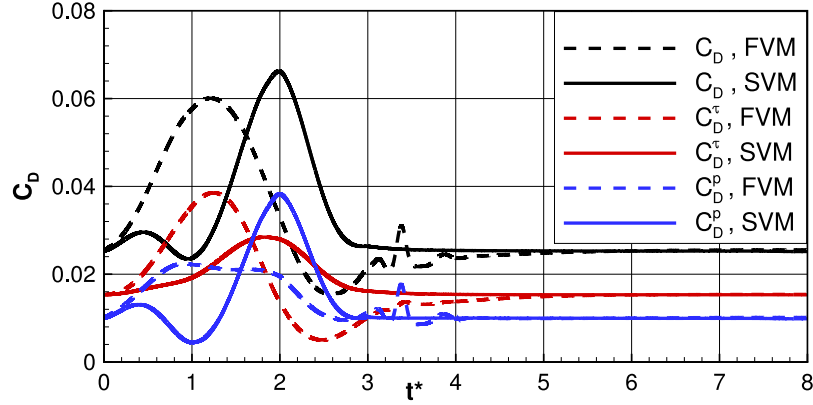


Fig. 16. Horizontal gust passing around the rounded flat plate: Time history of the drag coefficient C_D , of the pressure part of the drag coefficient C_D^p and of the shear part of the drag coefficient C_D^s for FVM and SVM. The gust has the length of $L_g^x = 2c$.

suggests the use of SVM for the injection of gusts having a streamwise velocity component. That will allow to tackle practically relevant flow problems in the near future, e.g., to predict the impact of strong wind gusts on flexible membranous structures and to answer the question of their structural integrity.

CRediT authorship contribution statement

Khaled Boulbrachene: Conceptualization, Methodology, Software, Validation, Investigation, Visualization, Writing – original draft, Writing – review & editing. **Guillaume De Nayer:** Conceptualization, Methodology, Software, Validation, Investigation, Visualization, Writing – original draft. **Michael Breuer:** Conceptualization, Methodology, Resources, Writing – original draft, Writing – review & editing, Supervision, Project administration, Funding acquisition.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgments

The project is financially supported by the *German Research Foundation (Deutsche Forschungsgemeinschaft)* under the contract number BR 1847/17-1.

Appendix A. Evaluation and effect of the svm source term

A.1. Vertical gusts

For a vertical gust, the gust velocity points only in the y -direction and is given by $\mathbf{u}_g = [0, u_{g,2}(x, t), 0]$. Hence, the source terms of the momentum equations simplify to $\mathbf{S} = [0, S_2(u_{g,2}(x, t)), 0]$. Employing Eq. (21), the simplified source term of the y -momentum equation reads:

$$S_2 = \rho \left\{ \frac{\partial u_{g,2}}{\partial t} + \tilde{u}_1 \frac{\partial u_{g,2}}{\partial x_1} \right\}. \quad (\text{A.1})$$

The vertical gust component can be written in the Cartesian basis as:

$$u_{g,2}(x, t) = \begin{cases} \frac{v_{g,\max}}{2} \left(1 + \cos \left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}} \right) \right) & \text{for } (x_1 - x_{g,1}) \in \left[-\frac{L_g^{x_1}}{2}, \frac{L_g^{x_1}}{2} \right] \\ 0 & \text{else,} \end{cases} \quad (\text{A.2})$$

with the convected gust center defined as:

$$x_g(t) = x_g^0 + (t - t_0) u_\infty. \quad (\text{A.3})$$

The derivatives in Eq. (A.2) are then:

$$\begin{aligned} \frac{\partial u_{g,2}}{\partial t} &= + \frac{\pi v_{g,\max} u_\infty}{L_g^{x_1}} \sin \left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}} \right), \\ \frac{\partial u_{g,2}}{\partial x_1} &= - \frac{\pi v_{g,\max}}{L_g^{x_1}} \sin \left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}} \right). \end{aligned} \quad (\text{A.4})$$

The derivatives are therefore nearly identical but of opposite signs. Consequently, the source term (Eq. (A.1)) would vanish if the background velocity $\tilde{u}_1$ is equal to u_∞ . In fact, this is the case in the far-field region (far away from the plate), where the background velocity $\tilde{u}_1$ is equal to the free-stream velocity u_∞ . In that region, the source term plays no role and has no effect on the pressure field in contrast to the effects remarked in the horizontal gust configuration. As a result, no major deviations between FVM and SVM are realized for the vertical gust configuration (see Figs. 6 to 8).

A.2. Horizontal gusts

For a horizontal gust, the gust velocity points only in the x -direction and is given by $\mathbf{u}_g = [u_{g,1}(x, t), 0, 0]$. Hence, the source terms of the momentum equations simplify to $\mathbf{S} = [S_1(u_{g,1}(x, t)), 0, 0]$. Employing Eq. (21), the simplified source term of the x -momentum equation reads:

$$S_1 = \rho \left\{ \frac{\partial u_{g,1}}{\partial t} + (\tilde{u}_1 + u_{g,1}) \frac{\partial u_{g,1}}{\partial x_1} \right\}. \quad (\text{A.5})$$

The horizontal gust component can be written in the Cartesian basis as:

$$u_{g,1}(x, t) = \begin{cases} \frac{u_{g,\max}}{2} \left(1 + \cos \left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}} \right) \right) & \text{for } (x_1 - x_{g,1}) \in \left[-\frac{L_g^{x_1}}{2}, \frac{L_g^{x_1}}{2} \right] \\ 0 & \text{else,} \end{cases} \quad (\text{A.6})$$

with the convected gust center defined as:

$$x_g(t) = x_g^0 + (t - t_0) u_\infty. \quad (\text{A.7})$$

The derivatives in Eq. (A.2) are then:

$$\begin{aligned} \frac{\partial u_{g,1}}{\partial t} &= + \frac{\pi u_{g,\max} u_\infty}{L_g^{x_1}} \sin \left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}} \right), \\ \frac{\partial u_{g,1}}{\partial x_1} &= - \frac{\pi u_{g,\max}}{L_g^{x_1}} \sin \left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}} \right). \end{aligned} \quad (\text{A.8})$$

Table B.6

Fine and coarsened grids information and the relative error of the maximum lift coefficient.

Gust wavelength	Grid	Total no. of CVs	$C_{L,max}$	Relative error %
2c	Fine	42,883,200	2.6115	–
	xy-coarsened	10,720,800	2.4910	4.6
	xyz-coarsened	5,360,400	2.4962	4.4
4c	Fine	42,883,200	3.0178	–
	xy-coarsened	10,720,800	3.1153	3.2
	xyz-coarsened	5,360,400	3.1029	2.8

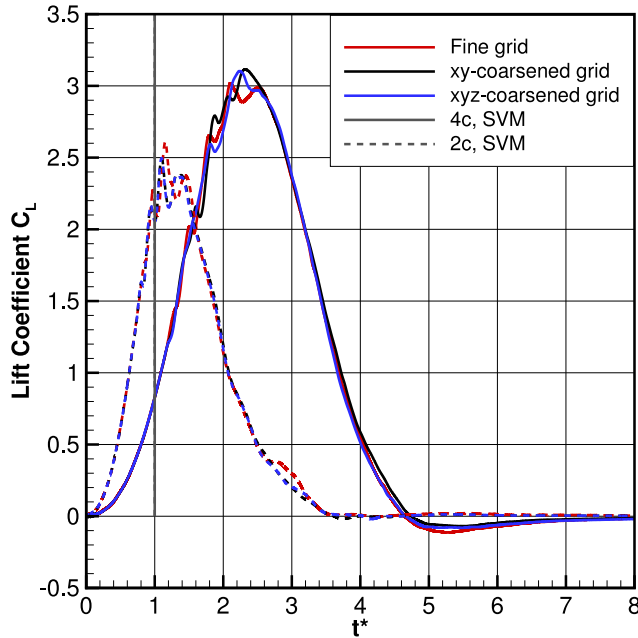


Fig. A.17. Vertical gust passing around the rounded flat plate: Instantaneous lift coefficients recorded on different grids for a grid sensitivity analysis. The gust shape is ECG with a gust amplitude of $A_g = 0.8$.

The derivatives are again nearly identical and of opposite signs as in Appendix A.1. However, this time the spatial derivative is multiplied by the total velocity $u_1 = \tilde{u}_1 + u_{g,1}$ which is different from u_∞ even in the far-field region. Consequently, one could clearly observe the role that the source term (Eq. (A.5)) plays in the early stages of the gust injection by its strong effect on the pressure field (see Fig. 12). As a result, major deviations between FVM and SVM are clearly visible for the horizontal gust configuration.

Appendix B. Grid sensitivity analysis

In order to examine the dependency of the results obtained on the grid used for the simulations, a grid sensitivity study is carried out. Two coarser grids are generated from the fine grid described in Section 5.1. Coarsening along a specific coordinate direction is performed by skipping every second grid line. The first grid is created by coarsening the blocks of the grid in the x - and y -directions only. This results in a computational grid containing 10,720,800 CVs. The second grid is generated by coarsening the blocks in all three coordinate directions resulting in a grid of 5,360,400 CVs. SVM simulations for a vertical gust configuration of different wavelengths (2c and 4c) are performed on the coarsened grids. The resulting lift coefficients obtained are plotted against those corresponding to the fine grid as a function of the convective time t^* shown in Fig. A.17. It could be observed that no significant variations among the lift coefficients are visible and the curves are, almost all times, coincident. The maximum lift coefficient recorded is chosen as a parameter for quantifying the error between the coarsened grids and the fine grid (see Table B.6). Although the difference between

the total number of control volumes of the grids is significantly large (compared to the fine grid, the xy -coarsened grid contains 25% of the total number of control volumes and the xyz -coarsened grid has only 12.5%), the relative error of the lift coefficient does not exceed 5%. This indicates that the flow fields are overall well resolved in the vicinity of the plate even upon the coarsening. Hence, one could conclude that the use of the fine grid is appropriate for the purpose of this study.

Note that the oscillations observed in the time histories of the lift coefficient can be physically explained. Looking at Figs. 6 to 8 the cause of these fluctuations becomes obvious. When the vertical gust impacts the flat plate, vortical structures are shed especially from the leading edge but also from the trailing edge. As visible in the pressure distributions the center of these vortices have low pressures influencing the local pressure distribution on the plate and finally the resulting lift coefficients.

References

- Apostolatos, A., De Nayer, G., Bletzinger, K.-U., Breuer, M., Wüchner, R., 2019. Systematic evaluation of the interface description for fluid-structure interaction simulations using the isogeometric mortar-based mapping. *J. Fluids Struct.* 86, 368–399.
- Badrya, C., Baeder, J.D., 2019. Numerical study of a flat plate wing response to large transverse gusts at low Reynolds number. *AIAA Scitech 2019 Forum*, San Diego, California, p. 0638, URL <https://arc.aiaa.org/doi/abs/10.2514/6.2019-0638>.
- Badrya, C., Biler, H., Jones, A.R., Baeder, J.D., 2021. Effect of gust width on flat-plate response in large transverse gust. *AIAA J.* 59 (1), 49–64.
- Biler, H., Badrya, C., Jones, A.R., 2019. Experimental and computational investigation of transverse gust encounters. *AIAA J.* 57 (11), 4608–4622.
- Breuer, M., 2002. *Direkte Numerische Simulation und Large-Eddy Simulation turbulenter Strömungen auf Hochleistungsrechnern*. In: Habilitationsschrift, Universität Erlangen-Nürnberg, Berichte aus der Strömungstechnik, Shaker Verlag, Aachen, Germany.
- Breuer, M., De Nayer, G., Münch, M., Gallinger, T., Wüchner, R., 2012. Fluid-structure interaction using a partitioned semi-implicit predictor-corrector coupling scheme for the application of large-eddy simulation. *J. Fluids Struct.* 29, 107–130.
- Burton, T., Sharpe, D., Jenkins, N., Bossanyi, E., 2001. *Wind Energy Handbook*. John Wiley & Sons.
- De Nayer, G., Apostolatos, A., Wood, J.N., Bletzinger, K.-U., Wüchner, R., Breuer, M., 2018. Numerical studies on the instantaneous fluid-structure interaction of an air-inflated flexible membrane in turbulent flow. *J. Fluids Struct.* 82, 577–609.
- De Nayer, G., Breuer, M., 2014. Numerical FSI investigation based on LES: Flow past a cylinder with a flexible splitter plate involving large deformations (FSI-PFS-2a). *Int. J. Heat Fluid Flow* 50, 300–315.
- De Nayer, G., Breuer, M., 2020. A source-term formulation for injecting wind gusts in CFD simulations. *J. Wind Eng. Ind. Aerodyn.* 207, 104405.
- De Nayer, G., Breuer, M., Perali, P., Grollmann, K., 2019. Modeling of wind gusts for large-eddy simulations related to fluid-structure interactions. In: Salvetti, M., Armenio, V., Fröhlich, J., Geurts, B.J., Kuerten, H. (Eds.), *ERCOFTAC Series, Direct and Large-Eddy Simulation XI*, 11th Int. ERCOFTAC Workshop on Direct and Large-Eddy Simulation: DLES-11, May 29–31, 2017. 25, Springer Nature Switzerland AG 2019, pp. 453–459.
- De Nayer, G., Kalmbach, A., Breuer, M., Sicklinger, S., Wüchner, R., 2014. Flow past a cylinder with a flexible splitter plate: A complementary experimental-numerical investigation and a new FSI test case (FSI-PFS-1a). *Comput. & Fluids* 99, 18–43.
- Demirdžić, I., Perić, M., 1988. Space conservation law in finite-volume calculations of fluid flow. *Int. J. Heat Fluid Flow* 8 (9), 1037–1050.
- Demirdžić, I., Perić, M., 1990. Finite-volume method for prediction of fluid flow in arbitrarily shaped domains with moving boundaries. *Int. J. Numer. Methods Fluids* 10 (7), 771–790.
- Durst, F., Schäfer, M., 1996. A parallel block-structured multigrid method for the prediction of incompressible flows. *Int. J. Numer. Methods Fluids* 22 (6), 549–565.
- Ferziger, J.H., Perić, M., 2002. *Computational Methods for Fluid Dynamics*, third ed. Springer Berlin.

- Franke, J., Baklanov, A., 2007. Best practice guideline for the CFD simulation of flows in the urban environment. In: COST Action 732 Quality Assurance and Improvement of Microscale Meteorological Models. University of Hamburg, Germany.
- Franke, J., Hellsten, A., Schlünzen, K.H., Carissimo, B., 2011. The COST 732 best practice guideline for CFD simulation of flows in the urban environment: A summary. *Int. J. Environ. Pollut.* 44 (1–4), 419–427.
- Frost, W., Long, B.H., Turner, R.E., 1978. Engineering handbook on the atmospheric environmental guidelines for use in wind turbine generator development. Tennessee Univ., Tullahoma (USA). Space Inst.; National Aeronautics and Space Administration, Huntsville, AL (USA). George C. Marshall Space Flight Center.
- Ghoreyshi, M., Jirasek, A., Greisz, I., Satchell, M., 2018. Indicial aerodynamic modeling for arbitrary gust responses, in: Applied Aerodynamics Conference, June 25–29, 2018, Atlanta, Georgia, AIAA-2018-3965.
- Gordnier, R.E., Visbal, M.R., 2015. Impact of a vortical gust on the aerodynamics of a finite aspect-ratio wing. In: 45th AIAA Fluid Dynamics Conference. pp. 1–18, URL <https://arc.aiaa.org/doi/abs/10.2514/6.2015-3072>.
- Granlund, K., Monnier, B., Ol, M., Williams, D., 2014. Airfoil longitudinal gust response in separated vs. attached flows. *Phys. Fluids* 26 (2), 027103.
- Gross, J.L., Main, J.A., Phan, L.T., Sadek, F.H., Cauffman, S.A., Jorgensen, D.P., 2013. Final report on the collapse of the Dallas Cowboys indoor practice facility, May 2, 2009.
- Gu, N., Lu, Z., Guo, T., 2012. Simulation of viscous flows around a moving airfoil by field velocity method with viscous flux correction. *Adv. Appl. Math. Mech.* 4 (3), 294–310.
- Hawbecker, P., Basu, S., Manuel, L., 2017. Realistic simulations of the July 1, 2011 severe wind event over the Buffalo Ridge Wind Farm. *Wind Energy* 20 (11), 1803–1822.
- Heinrich, R., 2014. Simulation of interaction of aircraft and gust using the TAU-code. In: Dillmann, A., Heller, G., Krämer, E., Kreplin, H.-P., Nitsche, W., Rist, U. (Eds.), *New Results in Numerical and Experimental Fluid Mechanics IX: Contributions to the 18th STAB/DGLR Symposium*, Stuttgart, Germany, 2012. Springer International Publishing, Cham, pp. 503–511.
- Heinrich, R., Reimer, L., 2013. Comparison of different approaches for gust modeling in the CFD code TAU. In: *International Forum on Aeroelasticity & Structural Dynamics 2013*. URL <https://elib.dlr.de/85834/>.
- Huntley, S., Jones, D., Gaitonde, A., 2016. 2D and 3D gust response using a prescribed velocity method in viscous flows, in: 46th AIAA Fluid Dynamics Conference.
- Huntley, S.J., Jones, D., Gaitonde, A., 2017. Aeroelastic gust response of an aircraft using a prescribed velocity method in viscous flows. In: 23rd AIAA Computational Fluid Dynamics Conference. Denver, Colorado, URL <https://arc.aiaa.org/doi/abs/10.2514/6.2017-3616>.
- IEC-Standard, 2002. 61400-21, Measurement and Assessment of Power Quality of Grid Connected Wind Turbines.
- Kasperski, M., 2007. A consistent model for the codification of wind loads. *J. Wind Eng. Ind. Aerodyn.* 95 (9), 1114–1124.
- Khosla, P.K., Rubin, S.G., 1974. A diagonally dominant second-order accurate implicit scheme. *Comput. & Fluids* 2 (2), 207–209.
- Knigge, C., Raasch, S., 2016. Improvement and development of one- and two-dimensional discrete gust models using a large-eddy simulation model. *J. Wind Eng. Ind. Aerodyn.* 153, 46–59.
- Küssner, H.G., 1936. Zusammenfassender Bericht über den instationären Auftrieb von Flügeln. *Luftfahrtforschung* 13 (12), 410–424.
- Kwon, D.K., Kareem, A., Butler, K., 2012. Gust-front loading effects on wind turbine tower systems. *J. Wind Eng. Ind. Aerodyn.* 104–106, 109–115.
- Lesoinne, M., Farhat, C., 1996. Geometric conservation laws for flow problems with moving boundaries and deformable meshes, and their impact on aeroelastic computations. *Comput. Methods Appl. Mech. Engrg.* 134 (1–2), 71–90.
- Montenegro, P.A., Heleno, R., Carvalho, H., Calçada, R., Baker, C.J., 2020. A comparative study on the running safety of trains subjected to crosswinds simulated with different wind models. *J. Wind Eng. Ind. Aerodyn.* 207, 104398.
- Münsch, M., 2015. Entwicklung und Anwendung eines semi-impliziten Kopplungsverfahrens zur numerischen Berechnung der Fluid-Struktur-Wechselwirkung in turbulenten Strömungen mittels Large-Eddy Simulationen. (Ph.D. thesis). University of Erlangen-Nuremberg, Germany.
- Norris, S.E., Cater, J.E., Stol, K.A., Unsworth, C.P., 2010. Wind turbine wake modelling using large-eddy simulation. In: *Proceedings of the 17th Australian Fluid Mechanics Conference*. Curran Associates, Inc., University of Auckland, Australia.
- Onol, A.O., Yesilyurt, S., 2017. Effects of wind gusts on a vertical axis wind turbine with trailing-edge separation. *J. Wind Eng. Ind. Aerodyn.* 162, 1–11.
- Parameswaran, V., Baeder, J.D., 1997. Indicial aerodynamics in compressible flow—Direct computational fluid dynamic calculations. *J. Aircr.* 34 (1), 131–133.
- Rhie, C.M., Chow, W.L., 1983. Numerical study of the turbulent flow past an airfoil with trailing-edge separation. *AIAA J.* 21 (11), 1525–1532.
- Sankar, L.N., Rou, S.Y., Malone, J.B., 1986. Application of surface transpiration in computational aerodynamics. In: 24th Aerospace Sciences Meeting. Reno, NV, USA, URL <https://arc.aiaa.org/doi/abs/10.2514/6.1986-511>.
- Singh, R., Baeder, J.D., 1996. On the significance of transonic effects on aerodynamics and acoustics of blade-vortex interactions. In: *Aeroacoustics Conference*. Pennsylvania, USA, URL <https://arc.aiaa.org/doi/abs/10.2514/6.1996-1697>.
- Singh, R., Baeder, J.D., 1997. Direct calculation of three-dimensional indicial lift response using computational fluid dynamics. *J. Aircr.* 34 (4), 465–471.
- Sitaraman, J., 2003. CFD Based unsteady aerodynamic modeling for rotor aeroelastic analysis. (Ph.D. thesis). University of Maryland, College Park, USA.
- Sitaraman, J., Baeder, J.D., 2000. Enhanced unsteady airload models using CFD. In: *Fluids 2000 Conference and Exhibit*. Denver, Colorado, URL <https://arc.aiaa.org/doi/abs/10.2514/6.2000-2465>.
- Sitaraman, J., Baeder, J.D., 2006. Field velocity approach and geometric conservation law for unsteady flow simulations. *AIAA J.* 44 (9), 2084–2094.
- Stone, H.L., 1968. Iterative solution of implicit approximations of multidimensional partial differential equations. *SIAM J. Numer. Anal.* 5 (3), 530–558.
- Wales, C., Jones, D., Gaitonde, A., 2014. Prescribed velocity method for simulation of aerofoil gust responses. *J. Aircr.* 52 (1), 64–76.
- Wu, Z., Cao, Y., Ismail, M., 2019. Gust loads on aircraft. *Aeronaut. J.* 123 (1266), 1216–1274.
- Zaide, A., Raveh, D.E., 2006. Numerical simulation and reduced-order modeling of airfoil gust response. *AIAA J.* 44 (8), 1826–1834.

Publication II: Aeroelastic response of an elastically mounted 2-DOF airfoil and its gust-induced oscillations

Reference

K. Boulbrachene and M. Breuer. Aeroelastic response of an elastically mounted 2-DOF airfoil and its gust-induced oscillations. *Journal of Fluids and Structures* 117 (2023), 103828. DOI: [10.1016/j.jfluidstructs.2022.103828](https://doi.org/10.1016/j.jfluidstructs.2022.103828)

Copyright

© 2022 The Authors. Published by Elsevier Ltd. This is an open access article distributed under the terms of the Creative Commons CC-BY license (<http://creativecommons.org/licenses/by/4.0/>), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.



Aeroelastic response of an elastically mounted 2-DOF airfoil and its gust-induced oscillations

K. Boulbrachene, M. Breuer*

Professur für Strömungsmechanik, Helmut-Schmidt-Universität Hamburg, D-22043 Hamburg, Germany

ARTICLE INFO

Article history:

Received 18 July 2022

Received in revised form 16 December 2022

Accepted 22 December 2022

Available online xxxx

Keywords:

Wind gust

Split velocity method

Fluid–structure interaction (FSI)

2-DOF aeroelastic oscillations

Limit-cycle oscillation (LCO)

Airfoil flutter

Large-eddy simulation (LES)

Energy analysis

ABSTRACT

The paper is concerned with numerical investigations on the effect of vertical wind gusts on airfoils in a parameter range relevant for Micro-Air Vehicles. Using a simplified substitute model instead of an elastic wing, a rigid but elastically mounted airfoil with two degrees of freedom (heave and pitch) is considered. The coupled problem is tackled by a partitioned fluid–structure interaction coupling scheme based on the large-eddy simulation (LES) technique and a rigid-body solver. In order to describe the effect of deterministic 1-cosine gusts of different gust lengths and gust strengths, the split velocity method (SVM) is incorporated into the simulation framework relying on the Arbitrary Lagrangian–Eulerian (ALE) formulation on temporally varying control volumes. First the flow fields and the corresponding aerodynamic forces during the direct airfoil–gust interaction are compared for a fixed and an elastically mounted airfoil. The intrinsic study on the elastic case includes nine different gust scenarios in the transitional Reynolds number regime in order to investigate the resulting flow fields and motion patterns and to answer the question whether limit-cycle oscillations (LCO) or even flutter can be induced. The results show that in seven of the studied cases, the airfoil–gust interaction leads to sustained heave and pitch oscillations of bounded amplitudes (i.e., LCO). Further investigations clarify that this can be physically attributed to the laminar separation taking place on the upper and lower surfaces of the airfoil. The two strongest gust cases, however, excite the airfoil to levels above its critical angle of attack and triggered a pitch-induced diverging flutter. An energy analysis of both characteristic scenarios (i.e., LCO and flutter) further elucidates the differences between both cases. The former case is driven by the heave motion, whereas the pitch DOF acts as an energy sink. Contrarily, in the case of flutter the pitching motion is powering the coupled system, whereas the heave motion dissipates energy.

© 2022 The Author(s). Published by Elsevier Ltd. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

1. Introduction

The unsteady aerodynamics of airfoils and their associated aeroelastic response have been an active field of research during the last century. The preceding transient aerodynamic models were potential flow-based models (Theodorsen, 1935; von Kármán and Sears, 1938) and were followed by studies addressing flow separation, formation and shedding of leading-edge vortices, and dynamic stall (McCroskey, 1982; Ekaterinaris and Platzer, 1998). While small deflections are typical for aerospace applications, energy harvesting applications, for instance, include large-amplitude oscillations. Depending on the application intended, the techniques employed to analyze and investigate these systems differ. For

* Corresponding author.

E-mail address: breuer@hsu-hh.de (M. Breuer).

example, linearized force models are found suitable for very small deflections, however, large-amplitude oscillations such as dynamic stall act as a source of nonlinearity. Moreover, the accompanying formation and shedding of leading-edge vortices results in highly transient loads which in turn pose considerable challenges to develop models capturing all the physics involved (Eldredge and Jones, 2019). Therefore, the accurate forecast of the aeroelastic response including all the possible extreme scenarios by means of experimental and numerical simulations is essential to ensure a reliable design process. A brief literature review focusing on the oscillatory behavior of airfoils subjected to different flow conditions is given in the following.

The wind tunnel experiments of a freely pitching NACA 0012 airfoil in the transitional Re regime conducted by Poirel et al. (2008) have extended the field in which the aerodynamic nonlinearities influence the aeroelastic stability. Unlike the stall flutter (known to be resulting from the periodic flow separation and reattachment at high angles-of-attack) or the transonic flutter (essentially resulting from shock waves moving along the wing's surface) (see Dowell and Tang (2002), Dowell et al. (2003)), Poirel et al. (2008) attributed the self-sustained oscillations found to the nonlinear aerodynamic load associated with the laminar boundary layer separation at the airfoil's trailing edge.

In order to obtain further insights into the physics involved in the course of the self-sustained oscillations, Poirel and Yuan (2010) performed large-eddy simulations (LES) of the flow around the airfoil undergoing a prescribed pitch motion taken from the pitch motion measured experimentally. It was found that the sustained oscillation is linked to the negative aerodynamic damping resulting from the unsteadiness of the trailing-edge laminar boundary-layer separation.

The complementary work of Poirel et al. (2011) conducted numerical simulations using an unsteady RANS solver and the SST $k-\omega$ model with low-Re number correction to study the laminar separation flutter (LSF). The results confirmed the key role of the laminar boundary layer separation on the initiation and maintenance of LSF. Moreover, the authors deduced that while high free-stream turbulence intensities tend to inhibit the sustained oscillations, the high frequency von Kármán shedding has no influence on the mechanisms maintaining the self-induced oscillations. Yuan et al. (2013) extended the simulation setup to include the heave degree of freedom (DOF). In contrast to the 1-DOF pitch system, minor contradictions with the experiments were found in the 2-DOF system which were attributed to the neglected dry friction in heave. Albeit the pitch-driven motion found (i.e., heave had relatively small amplitudes), the energy fed by the flow into the system was significantly amplified (three times larger).

Barnes and Visbal (2018b) coupled an implicit LES flow solver with a structural dynamics solver to explore pitching limit-cycle oscillations (LCO) on a rigid elastically mounted NACA 0012 airfoil operating at transitional Reynolds numbers. While for the low Reynolds numbers investigated sustained oscillations were found to grow naturally from rest, Reynolds numbers at the high end of the flutter regime demonstrated a bifurcation in different possible solution states. These appeared as emerging limit-cycle oscillations following an external disturbance or an initial condition and a static airfoil condition when no external excitation is applied. The latter case is essentially caused by the enhanced flow transition at higher Reynolds numbers which shortens the separation bubbles and diminishes their negative aerodynamic damping. Practically, external disturbances can be provided by atmospheric turbulence, abrupt changes in flight conditions, or wind gusts. For that reason, Barnes and Visbal (2018a) used the same 1-DOF aeroelastic configuration to subject the airfoil to a counterclockwise Taylor vortex gust. This transient vortical-gust/wing interaction was shown to be capable of eliciting laminar separation flutter which would not occur otherwise for the chosen Reynolds number.

Huntley et al. (2017) implemented two gust injection methods, namely, the split velocity method (SVM) and the field velocity method (FVM), into a flow solver and coupled that to a modal structural solver to predict the aeroelastic response of a flexible wing to a series of vertical gusts of the classical 1-cosine type. The results obtained by both methods were compared and noticeable differences in the solution were found for short wavelength gusts. Heinrich and Reimer (2013) implemented FVM (referred to as disturbance velocity approach in their work) and compared it to the resolved gust approach, a method in which the gust introduced at the far-field boundary is transported via an overset mesh. Based on these methods the interaction of a 1-cosine vertical gust penetrating a NACA 0012 wing with a horizontal tail plane (HTP) was simulated. The results of both methods were found to be in good agreement for gust wavelengths larger than two chord lengths. For short gusts, FVM over-predicted the lift peaks during the gust-HTP interaction as it failed to capture the wing's effect on the gust shape. Therefore, FVM was employed to perform aeroelastic simulations of a generic fighter aircraft and an A340 configuration encountering long vertical gusts. It is worth mentioning that for the last two aforementioned references, no flutter stability analysis was carried out. The advantages of SVM compared to FVM and its capabilities to deliver physically reasonable results was also worked out in Boulbrachene et al. (2021). Menon and Mittal (2019) studied flow-induced airfoil pitch oscillations of a rigid airfoil using a sharp-interface immersed boundary method. Beside investigating the parameters affecting the dynamics of wing flutter, the authors demonstrated the use of energy maps (maps describing the energy exchange between the flow and the airfoil over a range of selected parameters) as a tool to analyze the complex flow-induced flutter response of a 1-DOF system. Using the same computational model, Menon and Mittal (2020) demonstrated a method based on energy maps to accurately predict the aeroelastic response of a pitching airfoil to transverse gusts. The method predicts the final stationary state of the system given an initial angle-of-attack induced by a perturbation (i.e., a gust).

In this paper, the aeroelastic response of a two degrees of freedom (heave and pitch) NACA 0012 airfoil in the transitional Reynolds number regime subjected to strong vertical wind gusts is numerically investigated based on LES. This parameter combination of moderate Reynolds number and high gust ratio is highly relevant for Micro-Air Vehicles. The effect of different gust parameters (wavelengths and amplitudes) on the dynamics of the airfoil is examined. Especially

the question that should be answered is whether a certain gust can lead to limit-cycle oscillations or even flutter. Furthermore, the influence of different gust lengths and gust amplitudes on this phenomenon should be analyzed. The fluid–structure interaction setup without gusts has already been introduced by [De Nayer et al. \(2020\)](#) and validated against the experimental work of [Wood et al. \(2020\)](#). The simulation setup is slightly modified to accommodate the vertical gusts introduced. Wind gusts are applied by the split velocity method which was found in the previous studies to be capable of capturing the full interaction between the gust and the structure ([Wales et al., 2014](#); [Huntley et al., 2017](#); [Badrya et al., 2019](#); [Boulbrachene et al., 2021](#)).

2. Wind gust modeling

In a recent study by [Boulbrachene et al. \(2021\)](#) a literature review on different approaches to inject wind gusts into the computational domain was given. Besides far-field boundary conditions and source-term formulations prescribed velocity methods are a common methodology especially in aeronautics. Advantages of the latter are that the gust can be injected directly in front of the structure of interest, which avoids strong damping effects by numerical dissipation and makes highly resolved grids in the entire computational domain superfluous. Thus, it is the method of choice for the planned studies on the flutter behavior of an airfoil subjected to vertical gusts.

2.1. Split velocity method (SVM)

The split velocity method was originally proposed by [Wales et al. \(2014\)](#). Its main idea lies on the decomposition of the total velocity u_i into two components, namely, a background velocity $\tilde{u}_i$ and a prescribed gust velocity $u_{g,i}$. The governing equations are then rearranged in order to solve for the background velocity $\tilde{u}_i$. SVM is known in the literature to be an extension of the former field velocity method (FVM). While FVM solely employs the Arbitrary Lagrangian–Eulerian (ALE) formulation of the governing equations (but without actual grid movement), SVM extends that by an additional source term accounting for the feedback effect. The detailed derivation of SVM for the Euler equations is provided in [Wales et al. \(2014\)](#) and for the Navier–Stokes equations in [Huntley et al. \(2016\)](#) and [Boulbrachene et al. \(2021\)](#). The governing equations solved for SVM on a fixed grid read:

$$\int_{V^{\text{grid}}} \frac{\partial \rho}{\partial t} dV + \int_{S^{\text{grid}}} \rho (\tilde{u}_j + u_{g,j}) \cdot n_j dS = 0, \quad (1)$$

$$\int_{V^{\text{grid}}} \frac{\partial (\rho (\tilde{u}_i + u_{g,i}))}{\partial t} dV + \int_{S^{\text{grid}}} \rho (\tilde{u}_i + u_{g,i}) (\tilde{u}_j + u_{g,j}) \cdot n_j dS = - \int_{S^{\text{grid}}} \tau_{ij} \cdot n_j dS - \int_{S^{\text{grid}}} p \cdot n_i dS, \quad (2)$$

where ρ is the density of the fluid, τ_{ij} is the stress tensor based on the total velocity u_i , n_j describes the unit normal vector of the control volume pointing outwards and V^{grid} denotes the volume of the undeformed grid. Upon the rearrangement of the equations (see [Boulbrachene et al. \(2021\)](#)) one obtains:

$$\frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} \rho dV + \int_{S^{\text{grid}}} \rho \tilde{u}_j \cdot n_j dS + \gamma \int_{S^{\text{grid}}} \rho u_{g,j} \cdot n_j dS = 0, \quad (3)$$

$$\frac{d}{dt} \int_{V^{\text{grid}} \rightarrow V^*} \rho \tilde{u}_i dV + \int_{S^{\text{grid}}} \rho \tilde{u}_i \tilde{u}_j \cdot n_j dS + \gamma \int_{S^{\text{grid}}} \rho \tilde{u}_i u_{g,j} \cdot n_j dS = \int_{V^{\text{grid}}} S_{\text{momentum},i} dV - \int_{S^{\text{grid}}} \tau_{ij} \cdot n_j dS - \int_{S^{\text{grid}}} p \cdot n_i dS, \quad (4)$$

where the scaling factor γ is defined as $\gamma = \frac{V^*}{V^{\text{grid}}}$, with V^* being the pseudo updated volume caused by the apparent grid movement associated with the prescribed velocity methods. The source term $S_{\text{momentum},i}$ results from rearranging and simplifying Eq. (2) and reads:

$$S_{\text{momentum},i} = -\rho \left\{ \frac{\partial u_{g,i}}{\partial t} + (\tilde{u}_j + u_{g,j}) \frac{\partial u_{g,i}}{\partial x_j} \right\}. \quad (5)$$

The system of Eqs. (3)–(5) is applicable to fluid flow predictions relying on fixed grids as used in [Boulbrachene et al. \(2021\)](#) for the flow around a fixed flat plate. In case of fluid–structure interaction (FSI) the ALE formulation ([Breuer et al., 2012](#)) is applied in the present investigation. Hence, the equations have to be re-formulated for a temporally varying domain, i.e., control volumes (CV) with time-dependent volumes $V(t)$ and surfaces $S(t)$. For this purpose, the grid velocity $u_{\text{grid},j}$ with which the surface of a CV is moving has to be taken into account. The grid velocity is subtracted from the convective velocity (i.e., $\tilde{u}_j$) in the equations above. This yields the set of equations used to solve for the fluid flow:

$$\frac{d}{dt} \int_{V^n \rightarrow (V^{n+1})^*} \rho dV + \int_{S(t)} \rho (\tilde{u}_j - u_{\text{grid},j}) \cdot n_j dS + \gamma \int_{S(t)} \rho u_{g,j} \cdot n_j dS = 0, \quad (6)$$

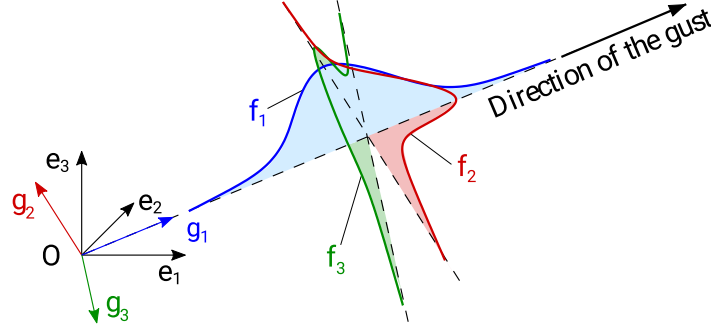


Fig. 1. Definition of the gust in the local basis $\mathcal{B}_1 = (O, \mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$.

$$\frac{d}{dt} \int_{V^n \rightarrow (V^{n+1})^*} \rho \tilde{u}_i dV + \int_{S(t)} \rho \tilde{u}_i (\tilde{u}_j - u_{\text{grid},j}) \cdot n_j dS + \gamma \int_{S(t)} \rho \tilde{u}_i u_{g,j} \cdot n_j dS = \int_{V(t)} S_{\text{momentum},i} dV - \int_{S(t)} \tau_{ij} \cdot n_j dS - \int_{S(t)} p \cdot n_i dS, \quad (7)$$

where V^n and V^{n+1} are the geometric cell volumes corresponding to the deformed grid at t^n and t^{n+1} , respectively. $(V^{n+1})^*$ is the updated deformed grid volume taking the additional pseudo volume change arising from the prescribed velocity method into account. It is worth mentioning that the geometric conservation law (GCL) (Demirdžić and Perić, 1988, 1990; Lesoinne and Farhat, 1996) is employed to suppress the contribution of the pseudo as well as the actual grid movement to the mass fluxes (by canceling out with the unsteady term in Eq. (6)). Hence, the mass conservation equation is reduced to:

$$\int_{S(t)} \rho \tilde{u}_j \cdot n_j dS = 0, \quad (8)$$

which is equivalent to the mass conservation equation for a fixed grid.

2.2. Gust velocity model

In the context of deterministic gust models the imposed analytical gust velocity $u_{g,i}$ is defined as (see Fig. 1):

$$u_{g,i}|_{\mathcal{B}_1}(t, \xi, \eta, \zeta) = A_g f_t(t) f_1(\xi) f_2(\eta) f_3(\zeta) g_1.$$

The constant A_g defines the strength of the gust and the function f_t the temporal evolution of the gust velocity. The functions f_1, f_2 and f_3 define the spatial distribution of the gust velocity in g_1, g_2 and g_3 direction, respectively.

For a vertical gust, the gust velocity consists of a single component pointing perpendicular to the streamwise direction. Therefore, the gust local basis is oriented such as $(\mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3) = (\mathbf{e}_3, \mathbf{e}_1, \mathbf{e}_2)$. In the present study, the prescribed gust is an Extreme Coherent gust (ECG) defined in the IEC-Standard (IEC-Standard, 2002) often denoted “1-cosine” shape. It varies in the streamwise direction and is convected downstream in time (i.e., $u_{g,i} = [0, 0, u_{g,3}(x_1, t)]$). Hence, f_t, f_1 and f_3 are equal to unity. The vertical gust component can be written in the Cartesian basis as:

$$u_{g,3}(x_1, t) = \begin{cases} \frac{A_g}{2} \left(1 + \cos \left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}} \right) \right) & \text{for } (x_1 - x_{g,1}) \in \left[-\frac{L_g^{x_1}}{2}, \frac{L_g^{x_1}}{2} \right] \\ 0 & \text{else,} \end{cases} \quad (9)$$

where A_g is the gust amplitude, $L_g^{x_1}$ is the gust length scale along the streamwise direction and $x_{g,1}(t)$ is the gust center coordinate. The introduction of this center coordinate (De Nayer et al., 2019) allows more control than the original shape. Based on the initial gust center coordinate $x_{g,1}^0$ and the gust convection velocity $u_{g,conv}$, the convected gust center is defined as:

$$x_{g,1}(t) = x_{g,1}^0 + (t - t_0) u_{g,conv}. \quad (10)$$

For a vertical gust, the source terms of the momentum equations are reduced to $S_{\text{momentum},i} = [0, 0, S_3(u_{g,3}(x_1, t))]$. Employing Eq. (5), the source term of the z-momentum equation reads:

$$S_3 = -\rho \left\{ \frac{\partial u_{g,3}}{\partial t} + \tilde{u}_1 \frac{\partial u_{g,3}}{\partial x_1} \right\}. \quad (11)$$

For the prescribed gust velocity (i.e., Eq. (9)), the temporal and spatial derivatives read:

$$\begin{aligned}\frac{\partial u_{g,3}}{\partial t} &= + \frac{\pi A_g u_{g,conv}}{L_g^{x_1}} \sin\left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}}\right), \\ \frac{\partial u_{g,3}}{\partial x_1} &= - \frac{\pi A_g}{L_g^{x_1}} \sin\left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}}\right).\end{aligned}\quad (12)$$

Substituting the derivatives (12) into Eq. (11) and simplifying the source term yields:

$$S_3(x_1, t) = -\rho \frac{\pi A_g}{L_g^{x_1}} \sin\left(\frac{2\pi (x_1 - x_{g,1}(t))}{L_g^{x_1}}\right) \{u_{g,conv} - \tilde{u}_1\}. \quad (13)$$

The source term is a function of the coordinate x_1 and the time and it is coupled to the background velocity $\tilde{u}_1$. This coupling relies on the difference between the gust convection velocity and the streamwise velocity of the flow. It can be clearly seen from Eq. (13) that the source term vanishes if these two quantities are identical. This is anticipated as, for instance, one would expect no interaction between a uniform horizontal parallel flow and a vertical jet traveling with the same horizontal velocity downstream. However, the situation is different when these velocities do not match (i.e., when $u_{g,conv} > \tilde{u}_1$ or $u_{g,conv} < \tilde{u}_1$). For the former case, the jet would normally tend to accelerate the horizontal background velocity, and since the total momentum of the fluid is conserved, this is achieved by decelerating the vertical background velocity through the addition of a negative source term. In contrast (i.e., when $u_{g,conv} < \tilde{u}_1$), the jet would resist and block the horizontal background velocity and this is modeled by accelerating the vertical background velocity by means of an added positive source term. The effect of the gust amplitude on the resulting source terms lies on amplifying these terms for stronger gusts and mitigating their effects for weaker ones.

2.3. Convective boundary condition treatment

Employing SVM special care has to be taken to properly apply a convective boundary condition at outlets. For this type of boundary condition the convective transport equation is solved at an outlet patch:

$$\frac{\partial u_i}{\partial t} + \frac{\partial U_j^c u_i}{\partial x_j} = 0, \quad (14)$$

where U_j^c is the outlet convective velocity vector. Since in SVM the velocity is decomposed into two components (i.e., background and gust velocity), Eq. (14) can be rewritten for a constant convective velocity vector as:

$$\frac{\partial (\tilde{u}_i + u_{g,i})}{\partial t} + U_j^c \frac{\partial (\tilde{u}_i + u_{g,i})}{\partial x_j} = 0. \quad (15)$$

Since SVM solves for the background velocity $\tilde{u}_i$, Eq. (15) is written for this variable as well:

$$\frac{\partial \tilde{u}_i}{\partial t} = - \left[\frac{\partial u_{g,i}}{\partial t} + U_j^c \frac{\partial (\tilde{u}_i + u_{g,i})}{\partial x_j} \right]. \quad (16)$$

When no gust is introduced into the computational domain and the free-stream flow is directed in x -direction, the convective velocity vector is $U_j^c = [u_\infty, 0, 0]$, where u_∞ is the free-stream velocity. However, when a vertical gust is applied, a vertical convective velocity component is required to allow for the convection of mass in that direction. The most suitable physically sound choice is $U_j^c = [u_\infty, 0, w_{g,mean}]$, where $w_{g,mean}$ denotes the mean of the imposed gust velocity profile (i.e., $w_{g,mean} = \frac{A_g}{2}$).

Due to the fact that the third component of the gust velocity (i.e., $u_{g,3}$) is the only non-zero component in vertical gusts, the application of the convective boundary condition using Eq. (16) differs from the classical application in the vertical direction only. Moreover, since the gust velocity solely varies along the streamwise direction, its spatial derivative vanishes for all other directions. Expanding Eq. (16) for $\tilde{u}_3$ with the defined convective velocity vector reads:

$$\frac{\partial \tilde{u}_3}{\partial t} = - \left[\frac{\partial u_{g,3}}{\partial t} + u_\infty \frac{\partial (\tilde{u}_3 + u_{g,3})}{\partial x_1} + w_{g,mean} \frac{\partial \tilde{u}_3}{\partial x_3} \right]. \quad (17)$$

Eq. (17) can be rearranged and written as:

$$\frac{\partial \tilde{u}_3}{\partial t} = -u_\infty \frac{\partial \tilde{u}_3}{\partial x_1} - w_{g,mean} \frac{\partial \tilde{u}_3}{\partial x_3} - \underbrace{\left[\frac{\partial u_{g,3}}{\partial t} + u_\infty \frac{\partial u_{g,3}}{\partial x_1} \right]}_*. \quad (18)$$

It is noticed that the term marked with $*$ is quite similar to the SVM source term in Eq. (11) except for the missing density and the free-stream velocity replacing the horizontal background velocity. Hence, this term proves to be essential

for achieving the conservation of momentum when the convective boundary condition is prescribed in the context of the split velocity method. Note that the corresponding $*$ -term in the equation for $\tilde{u}_1$ and $\tilde{u}_2$ is zero.

3. Numerical framework

3.1. Flow solver

The fully parallelized in-house finite-volume Navier–Stokes solver FASTEST-3D (Durst and Schäfer, 1996) is employed to solve for the incompressible fluid flow. The governing equations are discretized on a curvilinear, block-structured body-fitted grid with a collocated variable arrangement. The surface and volume integrals are approximated by the midpoint rule and the required flow variables at cell faces are linearly interpolated from neighboring cell-centered values leading to a second-order spatial accuracy. The time integration of the momentum equations is carried out using a second-order low-storage multi-stage Runge–Kutta method to obtain a provisional velocity. This predictor step yields an intermediate velocity field violating the continuity equation. Therefore, a Poisson equation for the pressure correction is solved and the coupled flow variables (i.e., velocities and pressure) are projected to the space of divergence-free vector fields in the so-called corrector step. This projection method (also known as predictor–corrector method) delivers velocities satisfying the mass conservation equation with a predefined accuracy (Breuer et al., 2012). Like in De Nayer et al. (2020), the large-eddy simulation (LES) technique is used to only resolve the large-scale turbulent flow field and model the small scales by a subgrid-scale (SGS) model. The standard Smagorinsky SGS model (Smagorinsky, 1963) with the well established constant $C_s = 0.1$ and van-Driest damping near solid walls is applied as it was shown to deliver reasonable results in De Nayer et al. (2018b).

In order to account for the multiphysics nature of a moving airfoil, the partitioned FSI approach presented in Breuer et al. (2012) is used to couple the fluid and the rigid-body solver. Moreover, the ALE formulation is used to account for two issues, namely, the time-dependent computational grid associated with the FSI problem and the pseudo-grid movement associated with the split velocity method. As the scope of this study is to investigate gust initiated airfoil oscillations, it is likely that the airfoil would undergo large translational and rotational movements. For this purpose, the hybrid grid adaption algorithm (IDW-TFI) developed by Sen et al. (2017) and particularly proposed for FSI-LES applications is used. This hybrid algorithm relies on two interpolation techniques, namely, the inverse distance weighting (IDW) interpolation for the block boundaries of the block-structured grid and a three-dimensional transfinite interpolation (TFI) for the inner grid points. The algorithm was found to be favorable as it combines two essential attributes, i.e., it retains the orthogonality of the grid in the vicinity of the moving body while maintaining acceptable CPU-time requirements. In the last decade this solver was intensively enhanced, tested and validated relying on complementary experimental and numerical pure CFD as well as FSI applications. With special focus on massively separated flows the following validation studies have to be mentioned.

In De Nayer et al. (2014) a complementary experimental–numerical investigation was carried out for the flow past a cylinder with a flexible splitter plate. The experimental setup was especially designed to be used as a well-defined but nevertheless challenging validation test case for fluid–structure interaction in turbulent flows. A detailed experimental investigation in a water tunnel using optical measurement techniques for both, the fluid flow and the structure deformation, were carried out and compared to LES predictions relying on the flow solver applied in the current study. The results were found to be in close agreement showing a quasi-periodic oscillating flexible structure in the first swiveling mode behind the bluff body. In Kalmbach and Breuer (2013) a second generic experimental FSI benchmark case was setup for a bluff-body flow combined with a flexible structure oscillating in the second swiveling mode. Again an enormous amount of flow field data as well as for the displacement of the structure were made available, which were used in De Nayer and Breuer (2014) to further validate the numerical methodology. The simulated phase-averaged data were found to be in close agreement with the experimental measurements concerning the form and the extrema of the deformation. Moreover, the frequency of the FSI phenomenon was well predicted with an error of less than 2.5%. Concerning other massively separated flows the complementary study by Wood et al. (2016) on the turbulent flow past a wall-mounted hemisphere ($Re = 50,000$) has to be mentioned. Here the flow detaches from the curved surface of the structure leading to extensive separations in the wake region comparable to the flow around airfoils at very high angles of attack. Comprehensive laser-Doppler measurements were conducted in a wind tunnel and compared with large-eddy simulations based on the present CFD code to validate the numerical simulations. Again the results of the measurements were found to be in close agreement with the predictions. Directly related to flows around airfoils are the combined studies of Wood et al. (2020) and De Nayer et al. (2020) which are the starting point of the current investigation. Here the flow around an elastically mounted NACA 0012 airfoil with two degrees of freedom was studied for different setups including large-amplitude oscillations and even flutter. Based on the numerical methodology also applied in the present study both limit-cycle oscillations and the flutter phenomenon yielding very high angles of attack could be reproduced by the coupled FSI predictions. For further validation studies, see Wood et al. (2018), De Nayer et al. (2018a), and Apostolatos et al. (2019).

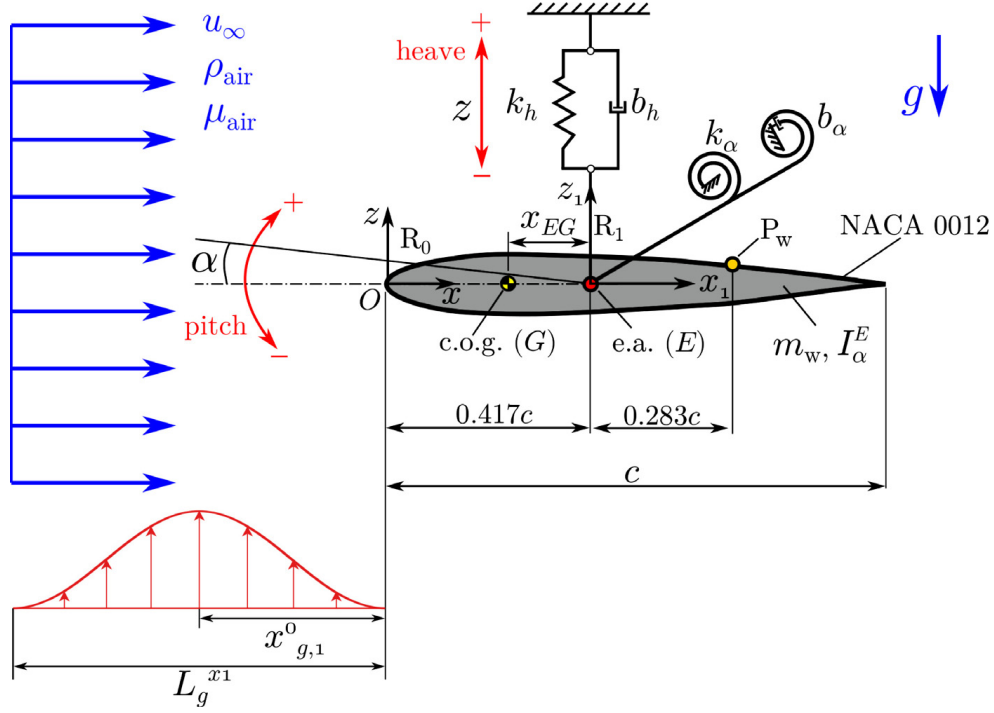


Fig. 2. Schematic representation of the investigated FSI case with all relevant parameters (notations from Wood et al. (2020)) and the introduced vertical gust velocity profile.

3.2. Rigid-body solver

The present FSI application is built on the precursor study of De Nayer et al. (2020) where the airfoil is treated as a rigid, but elastically mounted body (see Fig. 2). Consequently, fluid forces acting on the airfoil will cause its displacement as a single body without undergoing any structural deformations. In order to solve the dynamics of the airfoil experiencing fluid forces and moments, a solver describing the movement of the rigid body is applied. For this purpose, Newton's second law is considered in the Cartesian basis $R_0 = (x, y, z)$ for the translational and rotational motions, respectively. Leroyer and Visonneau (2005) wrote these equations in the form of a coupled system for the general case considering a moving deformable body. For the sake of brevity, only the necessary definitions, assumptions, and the resulting simplified equations are stated here (see De Nayer et al. (2020) for a detailed description). The airfoil has a local Cartesian coordinate system R_1 fixed to its elastic axis (e.a. or E). The governing equations for the rotational motions are written for this coordinate system. Therefore, the mass moment of inertia matrix is diagonal and time-independent. Moreover, the equations for the translational motion are formulated for the center of gravity denoted c.o.g. or G . Since the setup has only two degrees of freedom (2-DOF), i.e., translation in the z -direction and rotation around the y -axis, the system of equations describing the motion is reduced to two equations. This cancels out all other DOFs and their contributions to the inertial source terms. In fact, for the rigid airfoil considered, only a single source term associated with the heave DOF and arising from the rigid-body rotation remains.

The simplified coupled system of equations employed reads:

$$m_w \ddot{z} + b_h \dot{z} + k_h z - \underbrace{x_{EG} m_w}_{*} \ddot{\alpha} = F_{CFD, 3} - \underbrace{(-m_w \dot{\alpha}^2 z_{EG})}_{S_{R, 3}}, \quad (19)$$

$$I_{\alpha}^E \ddot{\alpha} + b_{\alpha} \dot{\alpha} + k_{\alpha} \alpha - \underbrace{x_{EG} m_w}_{*} \ddot{z} = M_{CFD, 2},$$

where z denotes the heave DOF, α the pitch DOF, b_h and b_{α} the material damping coefficients of the respective DOF and k_h and k_{α} the material stiffnesses of the respective DOF. m_w and I_{α}^E are the airfoil's mass and the mass moment of inertia regarding the elastic axis E . $F_{CFD, 3}$ is the lift force and $M_{CFD, 2}$ denotes the moment due to the fluid flow evaluated with respect to the elastic axis E . x_{EG} and z_{EG} are the horizontal and the vertical distance between E and G , respectively. The term marked with $*$ is the so-called "static moment" and $S_{R, 3}$ is the aforementioned source term. Both act as coupling terms between the heave and pitch motions.

It is worth mentioning that in order to circumvent potential singularities during the computation of the rotational DOF, Euler angles are replaced by *quaternions* as used in Leroyer and Visonneau (2005). In the present partitioned approach,

Table 1
Mechanical parameters of the coupled system.

Parameter	Symbol	Unit	Value
Distance between c.o.g. and e.a. (x_{EG})	x_{EG}/c	–	0.006
Mass of dynamic system	m_w	kg	0.33521
Mass moment of inertia based on f_α^{1-DOF}	I_α^E	kg m ²	1.381×10^{-4}
Bending stiffness	k_h	N/m	705
Torsional stiffness	k_α	Nm/rad	0.3832
Translational material damping ratio	D_h	–	2.30×10^{-3}
Rotational material damping ratio	D_α	–	1.15×10^{-3}

the time-step size is identical for the fluid and structural solvers. To obtain the solution of the motion at a new time step $n + 1$, the coupled system in Eq. (19) is solved sequentially, since the time step used by LES is very small. The static moment terms are moved to the right-hand side and taken from the previous time step n :

$$\begin{aligned} m_w \ddot{z}^{n+1} + b_h \dot{z}^{n+1} + k_h z^{n+1} &= F_{CFD, 3}^{n+1} + x_{EG}^n m_w \ddot{\alpha}^n + z_{EG}^n m_w (\dot{\alpha}^n)^2, \\ I_\alpha^E \ddot{\alpha}^{n+1} + b_\alpha \dot{\alpha}^{n+1} + k_\alpha \alpha^{n+1} &= M_{CFD, 2}^{n+1} + x_{EG}^n m_w \ddot{z}^n. \end{aligned} \quad (20)$$

The temporal discretization of the system of Eqs. (20) is implemented by using the *standard Newmark* method of second-order accuracy in time (Newmark, 1959). The Newmark parameters are set to $\beta = 0.25$ and $\gamma = 0.5$.

3.3. Coupling scheme

For the partitioned approach applied an exchange between both solvers is required. For this purpose, the pressure and shear forces on the airfoil are integrated up to determine the resulting force $F_{CFD, 3}$ and moment $M_{CFD, 2}$. At each time step these values are transferred to the rigid-body solver. In the other direction the structural displacements of the surface obtained by the superposition of the z -translation and the rotation around the y -axis are imposed at each fluid node of the FSI interface. An additional first-order estimation of the displacements at the beginning of each time step was found to be advantageous. The displacements of the surface of the airfoil are the input parameter for the grid adaption described in Section 3.1. Since in the present application the added-mass effect is negligible, a loose coupling algorithm is preferred. It delivers highly reliable results and saves a huge amount of CPU time in comparison to predictions based on a strong coupling scheme (De Nayer et al., 2020).

4. Description of the FSI case

The FSI case consists of an elastically mounted NACA 0012 airfoil of a chord length $c = 0.1$ m undergoing heave and pitch motions inspired by the experimental work of Wood et al. (2020) and the numerical counterpart by De Nayer et al. (2020). The airfoil is exposed to a constant free-stream velocity u_∞ and vertical gravity $\vec{g} = -g \vec{e}_z$ as sketched in Fig. 2. The wing which is initially at an angle of attack of 0° is allowed to pitch around its elastic axis located at a distance of $c_E = 0.417c$ from the leading edge. It is assumed that the structural setup can be modeled by a mass-spring-damper system for both degrees of freedom. Hence, six structural parameters are necessary to fully define the system. These parameters are the coefficients of Eq. (20) (i.e., m_w , I_α^E , k_h , k_α , $b_h = 2D_h\sqrt{k_h m_w}$ and $b_\alpha = 2D_\alpha\sqrt{k_\alpha I_\alpha^E}$). The mechanical properties used in the current study are those determined in De Nayer et al. (2020) for configuration **I**, in which the center of gravity nearly coincides with the center of rotation (i.e., $x_{EG} \approx 0$). In this configuration, the equations of motion (i.e., Eq. (20)) are almost uncoupled (in terms of inertia). Beside this setup two alternative configurations (**II** and **III**) were considered in these precursor studies mainly differing by the distance between the center of gravity and the center of rotation, i.e., $x_{EG} < 0$ and $x_{EG} > 0$. As shown in Wood et al. (2020) and De Nayer and Breuer (2020) the distance x_{EG} alters the dynamic characteristics of the system and is therefore an interesting parameter to study limit-cycle oscillations and flutter. Since most numerical simulations were carried out for case **I** and the system is considered to possess a higher stability compared to the case with $x_{EG} > 0$, which makes it more challenging with regard to the possible excitation by wind gusts, this configuration was chosen in the present study.

The parameters used are summarized in Table 1. The fluid properties defined are those of air at a constant temperature ($T = 288$ K) where the density and dynamic viscosity are $\rho_{air} = 1.225$ kg/m³ and $\mu_{air} = 18.27 \times 10^{-6}$ Pa s, respectively. For all cases studied, a constant free-stream velocity of $u_\infty = 5.06$ m/s is set yielding a Reynolds number $Re = \rho_{air} u_\infty c / \mu_{air} = 33,900$. In Wood et al. (2020) and De Nayer et al. (2020) this setup was investigated for a wide range of transitional Reynolds numbers ($9.66 \times 10^3 \leq Re \leq 3.60 \times 10^4$). While the overall kinematic state of the system was strongly damped for low Reynolds numbers, it got constantly amplified for the highest Reynolds number (i.e., flutter phenomenon). Within the range of $Re = 33,900$, the authors found the airfoil to be undergoing a series of sustained bounded amplitude oscillations with moderate angles upon the application of an external perturbation. Therefore, this Reynolds number is employed in the present study. Unlike the procedure followed in De Nayer et al. (2020), where the airfoil was initially excited by mechanical perturbations (initially prescribed lift force and pitching moment), this

Table 2
Vertical gust characteristics.

g_1	$f_t(t)$	$f_1(\xi)$	$f_2(\eta)$	$f_3(\zeta)$	$L_g^{x_1}/c$	$x_{g,1}^0/c$	A_g/u_∞		
e_3					2	-1	1	2	3
e_3	1	1	ECG	1	4	-2	1	2	3
e_3					6	-3	1	2	3

work aims to study the gust-induced perturbations and their effect on the airfoil's stability. Therefore, vertical wind gusts introduced upstream of the airfoil and convected downstream with a horizontal velocity $u_{g,conv} = u_\infty$ are used for this purpose. Gusts of different lengths (i.e., $L_g^{x_1}$) and several amplitudes (i.e., A_g) are investigated. The gust center coordinate (i.e., $x_{g,1}^0$) is chosen such that the leading edge of the gust and the airfoil coincide as shown in Fig. 2. The gust characteristics of the simulations performed are summarized in Table 2.

Considering commercial aircrafts operating at a cruise speed in the order of Mach number $Ma = 0.8$, the typical gust ratios relevant for this situation (defined by the certification rules of the European Aviation Safety Agency or the Federal Aviation Administration) are at least one order of magnitude smaller than those taken into account here. Nevertheless, high gust ratios make sense for Micro-Air Vehicles (MAVs), which are small aircrafts operating either remotely or autonomously at low flight velocities and thus moderate Reynolds numbers in the range considered in the present study. Here the gust response is a fundamental problem decisive for flight stability, maneuverability and control of MAVs. A recent study by Poudel et al. (2021) provides an interesting literature overview on the topic of gusts interacting with MAVs and motivates the necessity to study nonlinear, high magnitude gust-wing interactions. Other studies also considering large gust ratios are for example provided by Andreu-Angulo et al. (2020) and Perrotta and Jones (2017).

The computational domain around the NACA 0012 airfoil depicted in Fig. 3(a) is an extended version of that applied in the preceding study (De Nayer et al., 2020). The C-type grid of the inner domain (marked by black lines) is kept unchanged but extended in the radial direction (with r_{ex}) and in the wake of the airfoil (with w_{ex}). The extended sections have equidistant control volumes of cell heights identical to the respective last cell of the inner domain. This extension of the computational domain is intended to provide more space for the vortical structures resulting from the airfoil-gust interaction to develop above and past the airfoil. The extent of the flow domain is justified by previous investigations of similar flow cases, c.f. Breuer (2018) and the references cited therein. The radius of the semi-cylinder is now $r_{tot} = r + r_{ex} = (7.2 + 4.1)c = 11.3c$, the wake length is $w_{tot} = w + w_{ex} = (5 + 7)c = 12c$.

A critical issue for this kind of geometrically two-dimensional airfoils is the appropriate choice of the spanwise extension of the computational domain. Following the considerations in Schmidt and Breuer (2014, 2017) the width of the domain is set to $l_c = 0.25c$. According to Breuer (2018) this is a compromise between spanwise extension and spanwise resolution and based on grids of other research groups, e.g., Galbraith and Visbal (2009) and Visbal et al. (2009) with spanwise extensions of $l_c = 0.1-0.3c$. Galbraith and Visbal (2009) found only marginal differences between the different cases. Furthermore, Schmidt (2016) carried out additional simulations with a doubled spanwise domain size ($l_c = 0.5c$) evaluating two-point correlations. These investigations confirm the present choice. Note that these reference cases were carried out for flows in the same Reynolds number range. For flows at significantly higher $Re = \mathcal{O}(10^6)$ even much smaller spanwise extensions ($l_c/c = 0.012 - 0.06$) are typically used, see, e.g., Mellen et al. (2003) and Lobo et al. (2022).

The grid consists of 1,878,720 control volumes (CVs) and 60 CVs are equidistantly distributed in the spanwise direction. The distance between the first cell center and the wall in wall-normal direction is $4 \times 10^{-5} m$ with a geometrical stretching factor in the wall-normal direction of $q_s = 1.1$. A zoom of the grid in the vicinity of the airfoil is shown in Fig. 3(b). With this setup the viscous sublayer can be resolved and thus no-slip conditions are used on the surface of the airfoil. In order to prove this statement, additional LES predictions were carried out for the flow around the fixed airfoil at angles of attack in the range $0 \leq \alpha \leq 80^\circ$. In the Appendix these simulations are evaluated depicting the y^+ values of the wall-nearest grid points on the suction and pressure side of the airfoil. Obviously, at most points on the suction side the y^+ values are below unity. On the pressure side the values are slightly higher but even the peak values are below $y^+ = 5$. Thus, the dimensionless wall distance is in the range which allows to resolve the viscous sublayer.

Fig. 3(a) depicts the definition of the boundary conditions of the computational domain. At the inlet and the bottom boundary a constant free-stream velocity u_∞ is assumed, which means that the turbulence intensity is set to zero in order to concentrate on the effect of the wind gust on the airfoil behavior. Furthermore, in the spanwise direction periodic boundary conditions are applied. At the outlet and the top boundary a convective outflow boundary condition is prescribed as explained in Section 2.3.

To ensure the stability of the explicit time integration scheme used in the predictor step (see Section 3.1), a dimensionless time-step size of $\Delta t u_\infty/c = 5.06 \times 10^{-4}$ is applied. That corresponds to a maximum CFL number of about 0.5 and thus guarantees that the flow structures resolved in space by LES are also appropriately resolved in time.

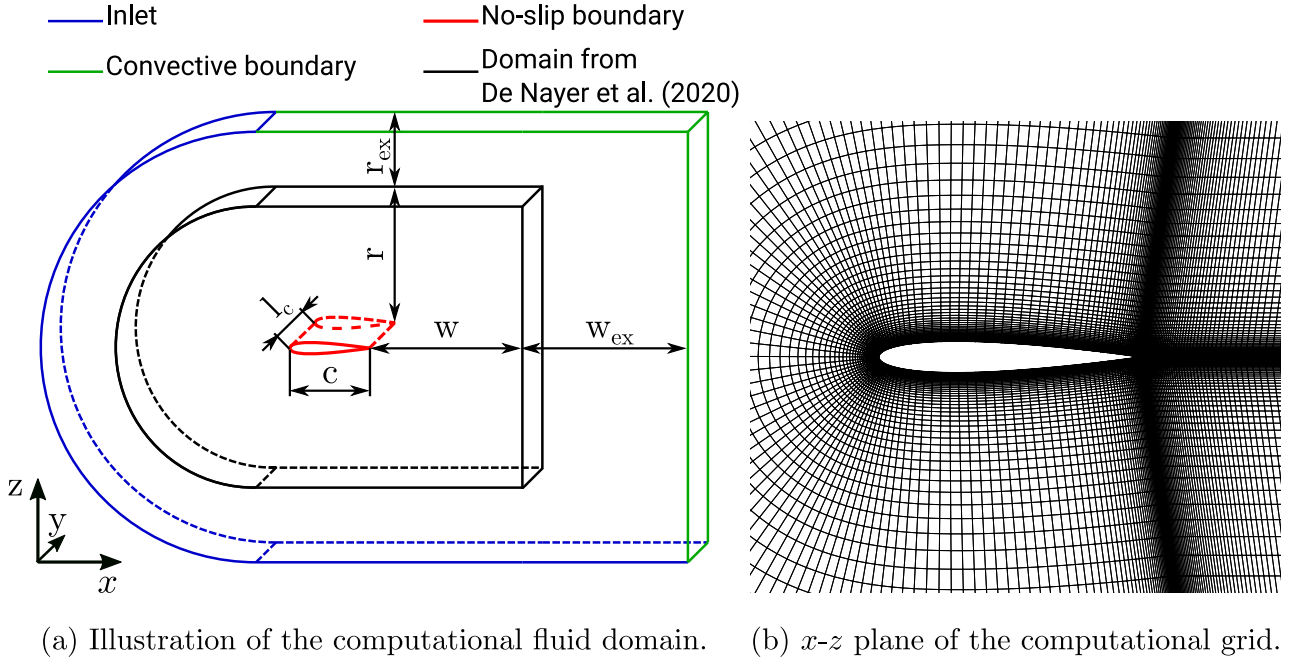


Fig. 3. CFD setup.

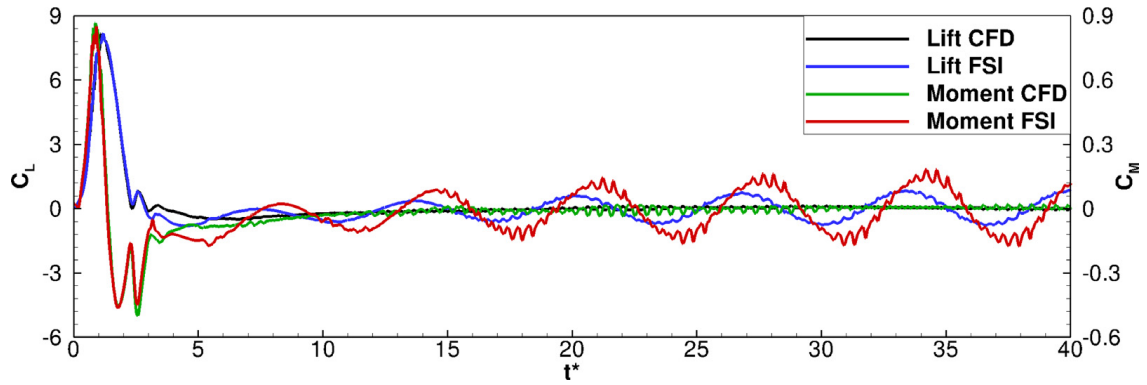


Fig. 4. Instantaneous lift and pitching moment coefficients of a fixed (CFD) and an elastically mounted airfoil (FSI) when subjected to a vertical gust of length $L_g^{x_1}/c = 2$ and amplitude $A_g/u_\infty = 2$. The sign definition is according to Fig. 2, i.e., the lift is positive in z -direction and the moment is positive in clockwise direction. (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

5. Results

5.1. CFD vs. FSI case

First results compare the aerodynamic forces exerted on the airfoil between two cases, namely, a fixed airfoil (CFD) and an elastically mounted airfoil (FSI). Fig. 4 depicts the instantaneous lift and pitching moment coefficients of the airfoil encountering a 1-cosine vertical gust of length $L_g^{x_1}/c = 2$ and strength $A_g/u_\infty = 2$. At $t^* = 0$ (where $t^* = u_\infty t/c$) the airfoil's leading edge starts penetrating the gust yielding an increase in the lift and the pitching moment coefficients. While the rise in the lift coefficient is attributed to an increase in the effective angle of attack of the airfoil, that of the pitching moment coefficient is due to the generated clockwise moment. This moment is a consequence of the high and low pressure fields acting concurrently to the left of the elastic axis on the airfoil's upper and lower surfaces, respectively (see pressure field snapshots at $t^* = 1$ in Figs. 5 and 6). During this phase of interaction a clockwise rotating leading-edge vortex (LEV) is formed. Obviously, the pressure and velocity distributions are quasi identical for the fixed and the elastically mounted airfoil at this instant in time. Since both the heave and the pitch motion are still very small at this early stage, the marginal deviations between both cases can be explained.

The steady increase in the magnitude of both coefficients continues until peak values at different instants in time are reached. While the pitching moment coefficient records its maximum magnitude when the gust center reaches the vicinity of the leading edge ($t^* \approx 0.9$), the peak of the lift occurs when the gust center is shortly behind the leading edge

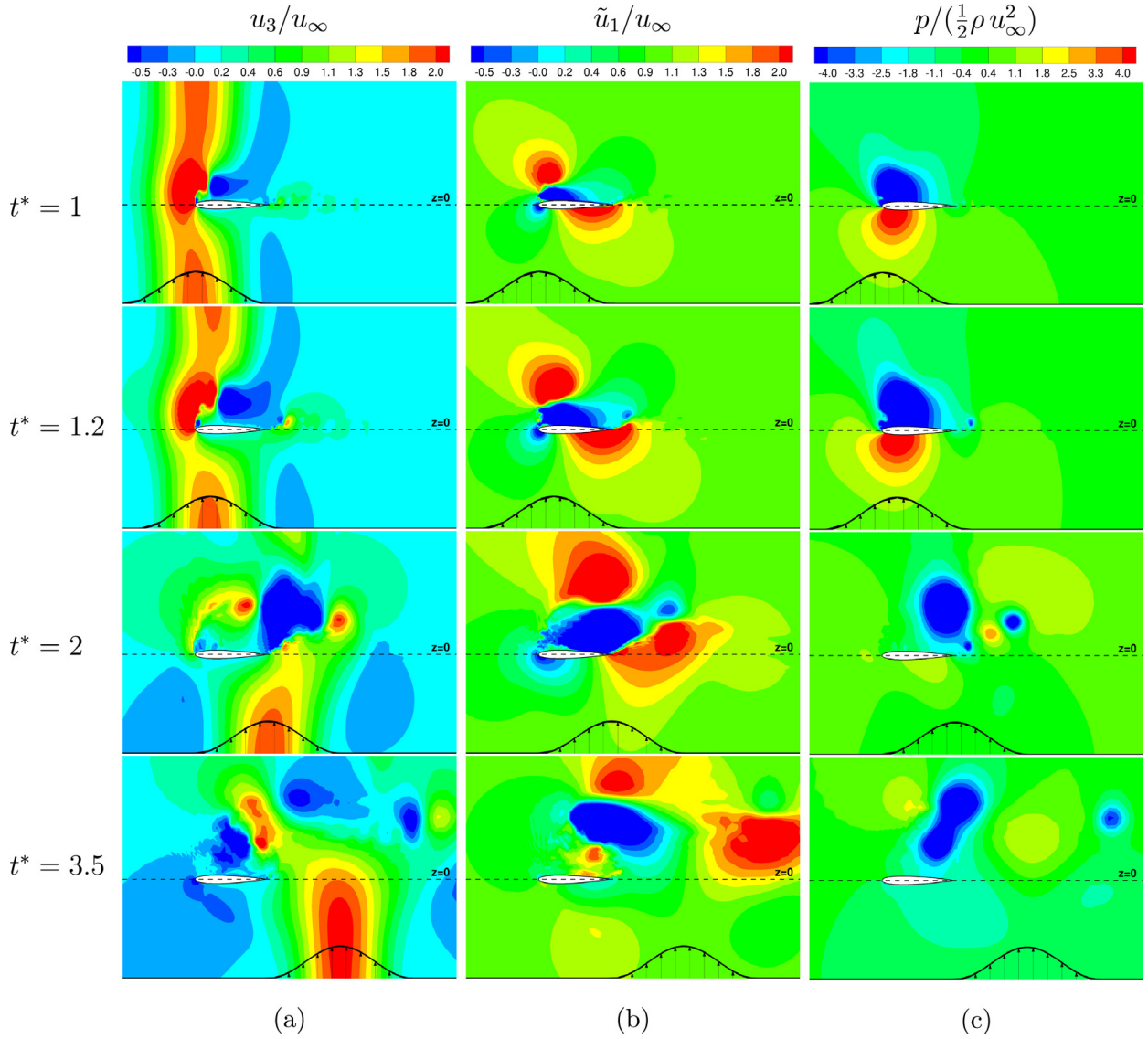


Fig. 5. Snapshots of the total vertical velocity (a), the background streamwise velocity (b) and the pressure (c) fields of the gust encounter ($L_g^{x1}/c = 2$, $A_g/u_\infty = 2$) on the fixed airfoil. $t^* = 1$: gust at leading edge, $t^* = 1.2$: instant at maximum lift, $t^* = 2$: gust center at trailing edge, $t^* = 3.5$: gust completely left the airfoil.

and before the mid-chord ($t^* \approx 1.2$ in Figs. 5 and 6). Again the deviations in the flow fields of the fixed and the elastically mounted airfoil are marginal. This can be explained with the same argument as before.

The lift coefficient then starts decaying to eventually recover the level of the steady-state values when the gust has completely left the plate. On the other hand, the pitching moment coefficient experiences a decrease to negative values as the gust center and the leading-edge vortex travel downstream along the chord of the airfoil (i.e., for $0.9 \leq t^* \leq 2.5$ as shown in Fig. 4). Although nearly identical histories are obtained during the airfoil–gust interaction (i.e., until $t^* = 3$) and thus the velocity and pressure fields in Figs. 5 and 6 are still in close agreement, the curves of the FSI case continue to oscillate (see discussion on limit-cycle oscillations in Section 5.3) while their CFD counterparts recover to the quasi steady-state values. Oscillating aerodynamic forces contribute to dampening as well as to amplifying the motion as found in later investigations. At the last instant in time ($t^* = 3.5$) depicted in Figs. 5 and 6 some small deviations in the resulting flow fields can be detected which are due to the induced onset of the pitch and heave motion.

The high-frequency components in the lift and moment coefficients, which in the FSI case are superimposed on the motion-dominated low-frequency components for $t^* > 10$, are the result of the laminar flow separation from the airfoil's lower and upper surface. To investigate this phenomenon, the spanwise y -vorticity ω_y (plotted on a mid-span slice), the skin-friction coefficient C_f (plotted on the upper surface of the airfoil), and the location of flow separation are depicted for different time instants in Fig. 7. The instants at which the snapshots (i.e., snapshots a–h) are taken correspond to the points in Fig. 8 showing the respective heave and pitch angle values. The snapshots depicted in Fig. 7a to d clearly show that when the airfoil starts to pitch up reaching its maximum angle of attack, the separation starting in the vicinity of

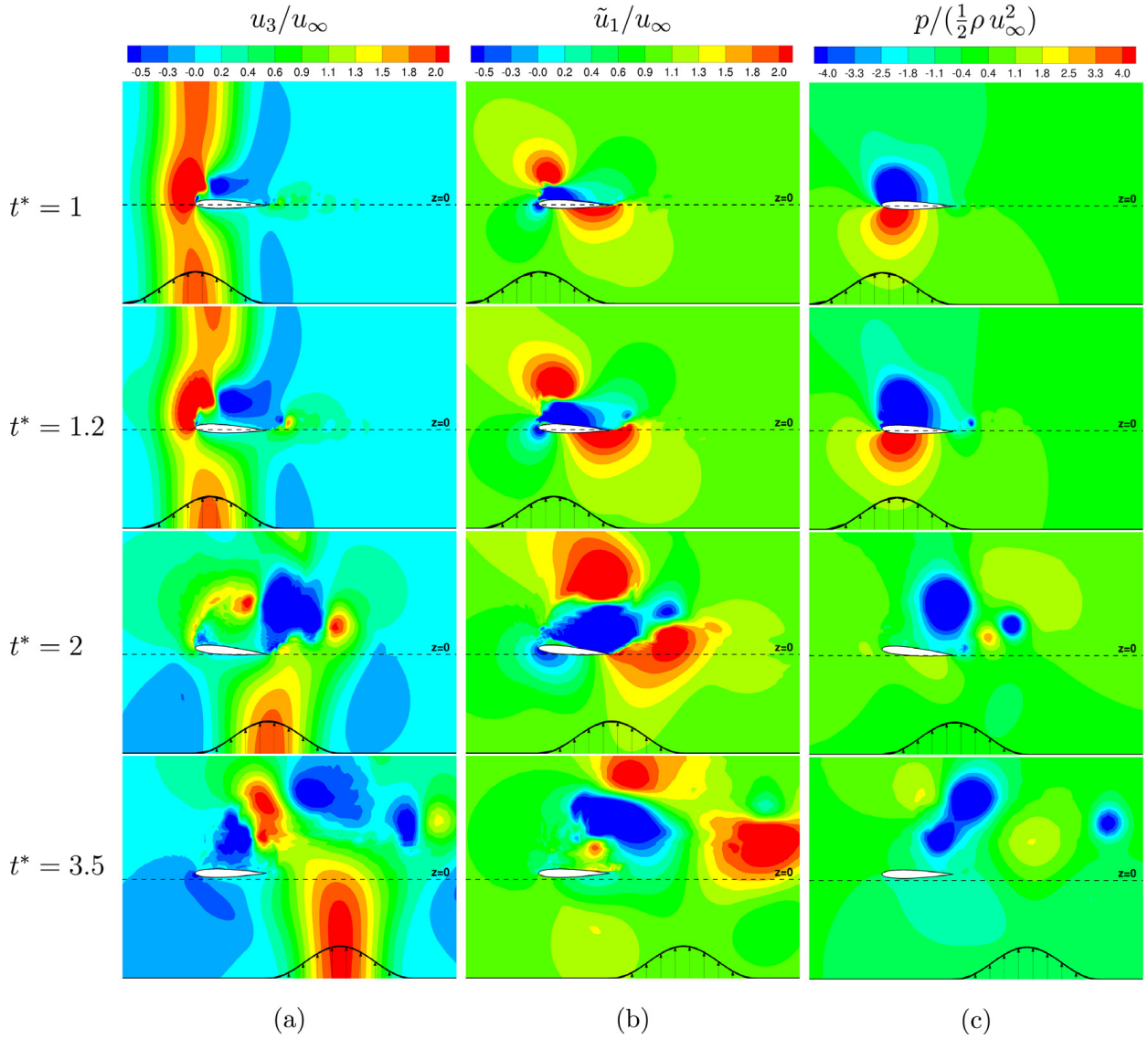


Fig. 6. Snapshots of the total vertical velocity (a), the background streamwise velocity (b) and the pressure (c) fields of the gust encounter ($L_g^{x1}/c = 2$, $A_g/u_\infty = 2$) on the elastically mounted airfoil. $t^* = 1$: gust at leading edge, $t^* = 1.2$: instant at maximum lift, $t^* = 2$: gust center at trailing edge, $t^* = 3.5$: gust completely left the airfoil.

the trailing edge moves upstream on the suction side. During the phase when the angle of attack is already decreasing again, vortices are shed from the suction side (see Fig. 7f and g). These vortices disappear when the angle of attack tends towards the other extremum (see Fig. 7h) and thus a favorable pressure gradient occurs on the corresponding side of the airfoil. At that stage the same process begins on the lower surface. Note that the flow phenomena described above are mostly two-dimensional which is typical for a laminar flow separation. The separated shear layers become unstable further downstream leading to the development of three-dimensional structures and later to the transition to turbulence. That explains the spanwise varying separation line visible in Fig. 7h.

Although similar high-frequency components are observed in the fixed case, those are rather caused by the continuous vortex-shedding process observed close to the trailing edge. Consequently, the phenomenon appears independent of the gust, but is augmented by the gust due to the resulting motion and especially the increase in the angle of attack.

5.2. Effect of the gust strength on the FSI case

As mentioned in Section 4, the effect of gust-induced perturbations on the dynamic behavior of the elastically mounted airfoil is intended to be studied. For that reason, a series of FSI simulations is carried out to investigate the influence of different gust strengths (at a fixed gust length) on the aeroelastic response of the airfoil. The time histories of the instantaneous dimensionless heave and pitch of the airfoil exposed to vertical gusts of the gust length $L_g^{x1}/c = 2$ (see Table 2) are depicted in Fig. 9.

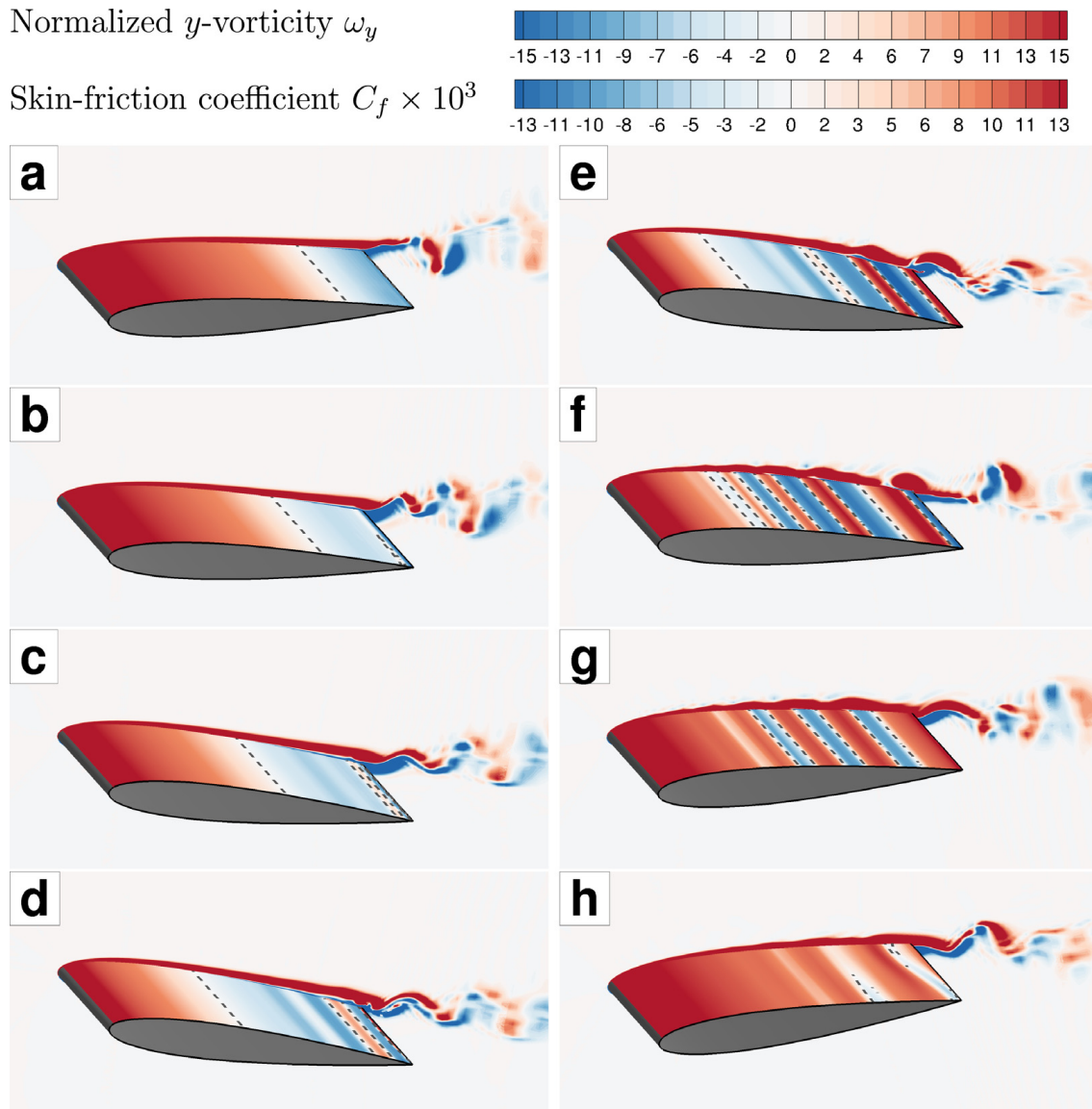


Fig. 7. Normalized y -vorticity ω_y (on the mid-span slice), skin-friction coefficient C_f (on the upper surface of the airfoil), and separation location (dashed line) for eight time instants (a–h) corresponding to the instants given in Fig. 8.

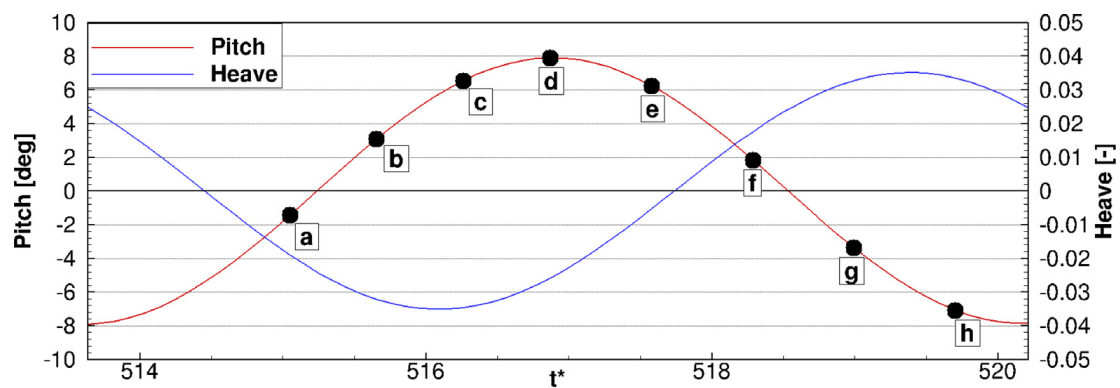


Fig. 8. Pitch angle and heave DOF during a LCO cycle.

A sudden increase in the magnitude of the heave and pitch as a reaction to the gust-imposed lift and pitching moment (depicted in Figs. 9(c) and 9(g), respectively) is observed for all three gust strengths. Especially the initial heave excitation strongly depends on A_g , whereas the first peak of the pitch marginally is nearly identical for all gust strengths. Pitch

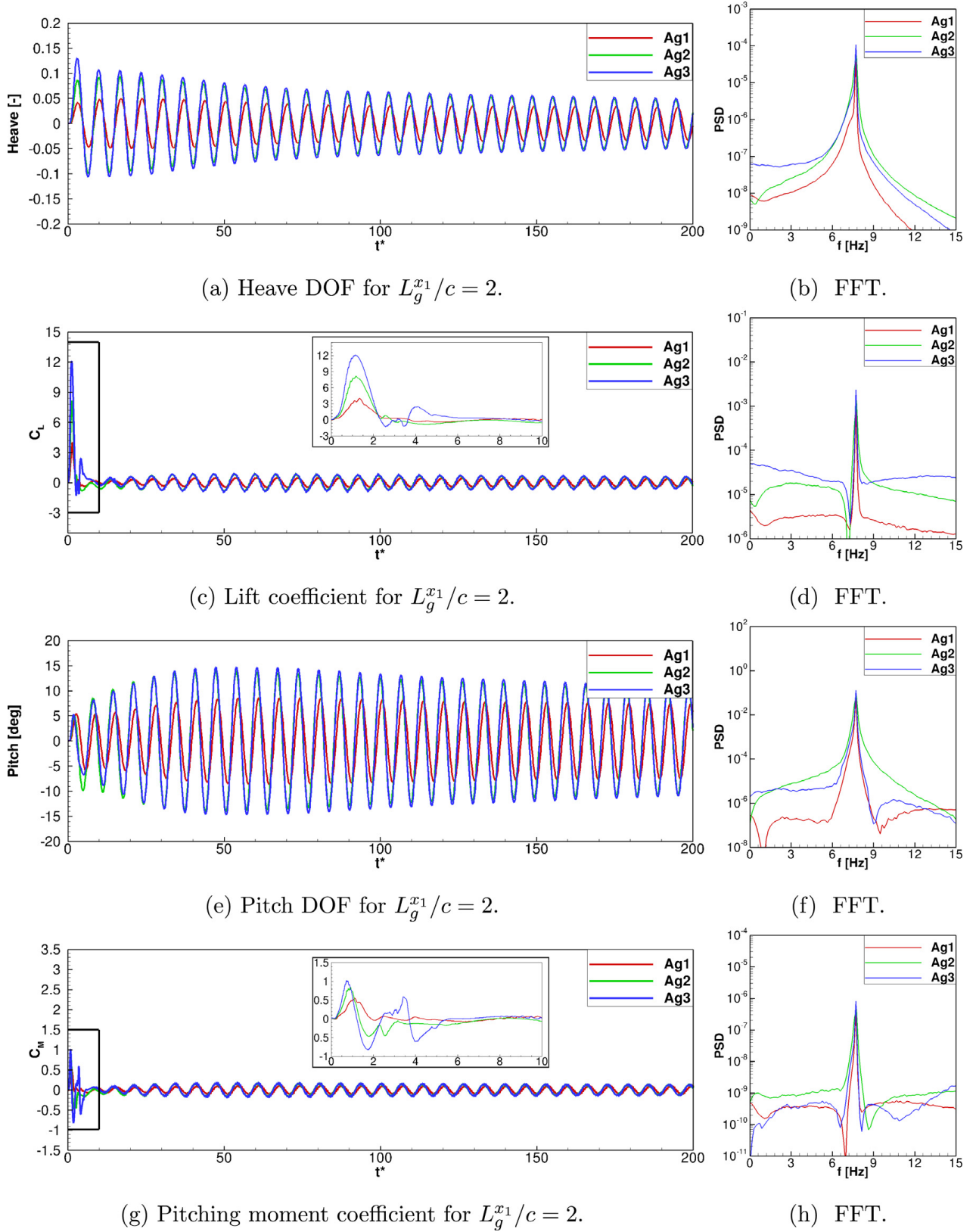


Fig. 9. Instantaneous heave and pitch DOFs (a,e), lift and pitching moment coefficients (c,g) of the elastically mounted airfoil when subjected to vertical gusts of length $L_g^{x1}/c = 2$ at three gust strengths $Ag/u_\infty = 1, 2$ and 3.

Table 3

Oscillation frequencies associated with the elastically mounted airfoil subjected to different vertical gusts. The cases in which flutter occurs, are given in boldface.

$L_g^{x_1}/c$	–	2			4			6		
A_g/u_∞	–	1	2	3	1	2	3	1	2	3
Heave frequency	[Hz]	7.70	7.75	7.70	7.63	7.63	7.81	7.66	7.68	7.92
Pitch frequency	[Hz]	7.70	7.75	7.70	7.63	7.63	8.10	7.66	7.68	7.92
Lift frequency	[Hz]	7.70	7.75	7.70	7.63	7.63	7.81	7.66	7.68	7.92
Moment frequency	[Hz]	7.70	7.75	7.70	7.63	7.63	8.10	7.66	7.68	7.92
Phase angle	deg	135.2	138.6	138.9	138.3	140.8	162.8	138.4	139.3	–

oscillations continue to gain amplitude until $t^* \approx 54$ (i.e., the 9th cycle) for all prescribed gust strengths. The oscillations then start to steadily decay finally arriving at a bounded amplitude (i.e., a limit-cycle oscillation). For the heave DOF, a similar trend is observed for $A_g/u_\infty = 1$ and 2, where a maximum amplitude is reached at about $t^* = 16.7$ (i.e., the 3rd cycle). However, for the strongest gust (i.e., $A_g/u_\infty = 3$) the maximum heave amplitude is recorded within the first cycle and the oscillatory behavior is then seen to undergo a damped motion approaching the limit-cycle oscillation. Considering the aerodynamic forces it is found that the stronger the gust is, the higher are the lift and pitching moment coefficients observed. This is of course expected as stronger gusts induce higher aerodynamic forces and moments which describe the right-hand side of Eq. (19). Overall, a direct relationship between the gust strength and the maximum heave and pitch amplitudes reached during the gust impact and the immediate phase thereafter is also obvious. However, additional simulations continuing the present case far beyond $t^* > 200$ have provided clear evidence that the amplitudes of heave and pitch aim for the same final bounded values.

5.3. Effect of the gust length on the FSI case

The heave and pitch DOF responses of the further investigated gust lengths (i.e., $L_g^{x_1}/c = 4$ and $L_g^{x_1}/c = 6$) shown in Figs. 10 and 11 exhibit a similar tendency for gust strengths of $A_g/u_\infty = 1$ and 2. However, it is worth mentioning that for the same gust strength, wider gusts are found to cause higher amplitudes across the aerodynamic forces and their respective degrees of freedom during the gust encounter. This is expected as, for wider gusts, the airfoil experiences a higher effective angle of attack along its chord for a longer period of time. Again, both DOFs show an initial increase upon the gust encounter followed by a continuous decrease to eventually settle after several oscillation cycles at a stage of sustained bounded amplitudes. Note that in the long term the amplitudes observed for excitations by gusts of different length scales are identical for both gust strengths of $A_g/u_\infty = 1$ and 2. In accordance with the previously discussed case of $L_g^{x_1}/c = 2$ a limit-cycle oscillation is induced by the gusts of these strengths ($A_g/u_\infty \leq 2$).

In contrast, for a gust strength of $A_g/u_\infty = 3$ the prescribed gusts tend to induce significantly larger excitations which trigger a diverging pitch motion. While the amplitude of the heave oscillations experiences a steady decay (see $A_g/u_\infty = 3$ in Figs. 10(a) and 11(a)), the pitch DOF engages in a series of oscillations in which the amplitude is constantly increasing (see Figs. 10(e) and 11(e)). Obviously, these two cases do not show a limit-cycle oscillation anymore, but represent the flutter case. Except the heave motion the amplitudes of all other quantities (lift, pitch, pitching moment) continuously increase until finally the predictions could not be continued and blow up due to an overwhelming pitch motion. Thus, in case of an excitation by a strong wind gust ($A_g/u_\infty \geq 3$) with a sufficiently large length ($L_g^{x_1}/c \geq 4$) the flutter scenario can already be reached at $Re = 33,900$, whereas in the previous study of the identical configuration using artificial initial perturbations (De Nayer et al., 2020) the flutter case was only achieved at a higher Reynolds number of $Re = 36,000$.

Moreover, despite the preserved periodicity of the heave and pitch motion, the aerodynamic coefficients show more complex time histories which is the outcome of additional higher frequencies observed in the signals. A frequency analysis carried out for both DOFs along with their respective driving forces for all gust lengths and strengths is depicted on the right columns of Figs. 9 to 11. The frequencies corresponding to the maximum power spectral density of each case are summarized in Table 3. It can be observed that for all cases in which the system converges to the limit-cycle oscillation, all signals oscillate at nearly the same frequency. The frequency of oscillation varies slightly across these cases in a small range of (7.63, 7.75). That is at its maximum about 1.5% and the deviations are attributed to inaccuracies in determining the peak values. The dominant frequency is also in close agreement with the preceding study (De Nayer et al., 2020), in which a frequency of 7.69 Hz was determined for the oscillations with moderate amplitudes. However, for the cases in which flutter occurs, an increase in the frequency of oscillation of approximately 5% (especially for the pitch DOF) is found. Table 3 also provides the observed phase angles. For the LCO case the values predicted in the current study are in close agreement with the value of 138° found in De Nayer et al. (2020). In the flutter case the determination of the phase angle is more challenging. In accordance with the previous work, an increase of the phase angle is observed.

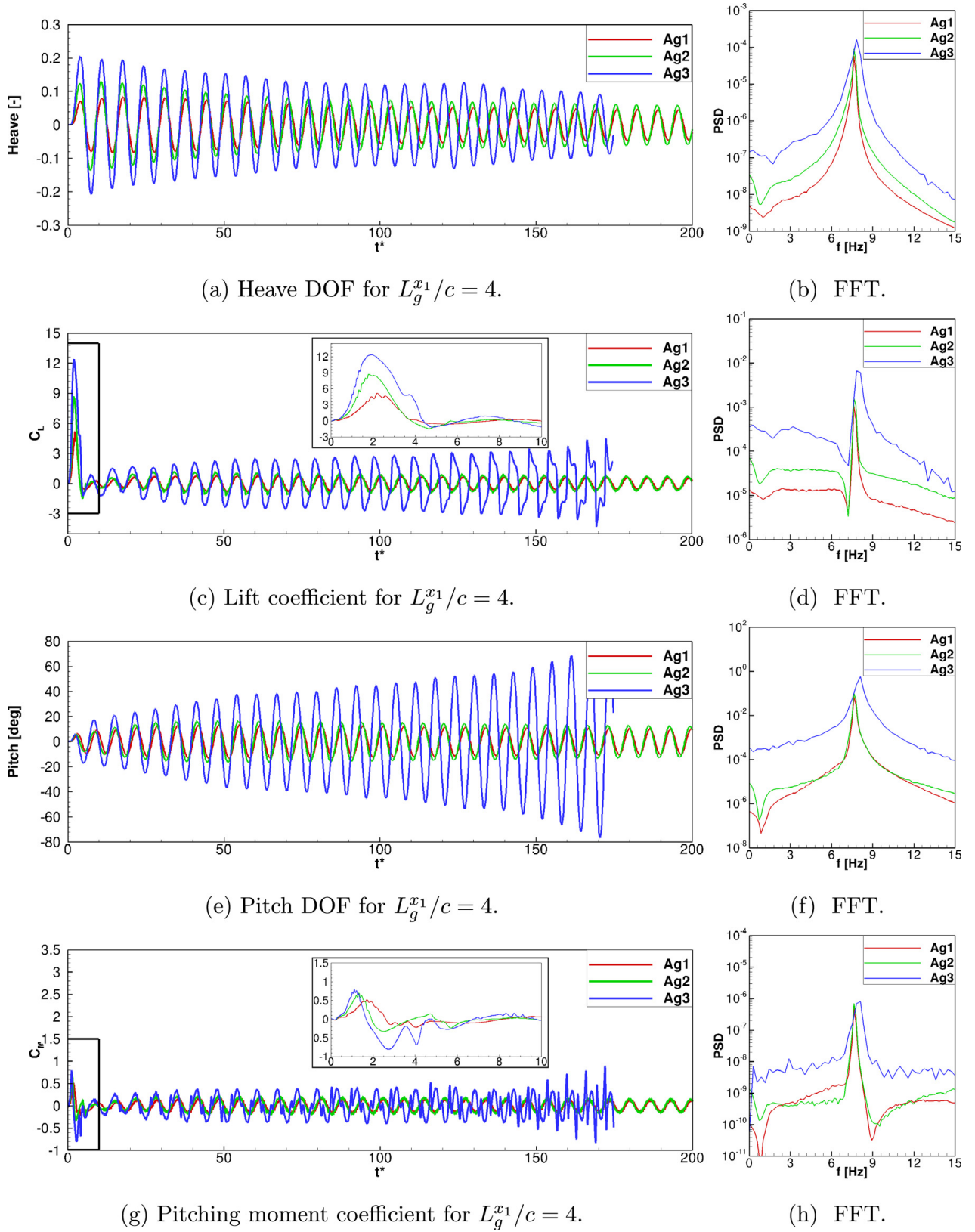


Fig. 10. Instantaneous heave and pitch DOFs (a,e), lift and pitching moment coefficients (c,g) of the elastically mounted airfoil when subjected to vertical gusts of length $L_g^{x1}/c = 4$ at three gust strengths $Ag/u_\infty = 1, 2$ and 3.

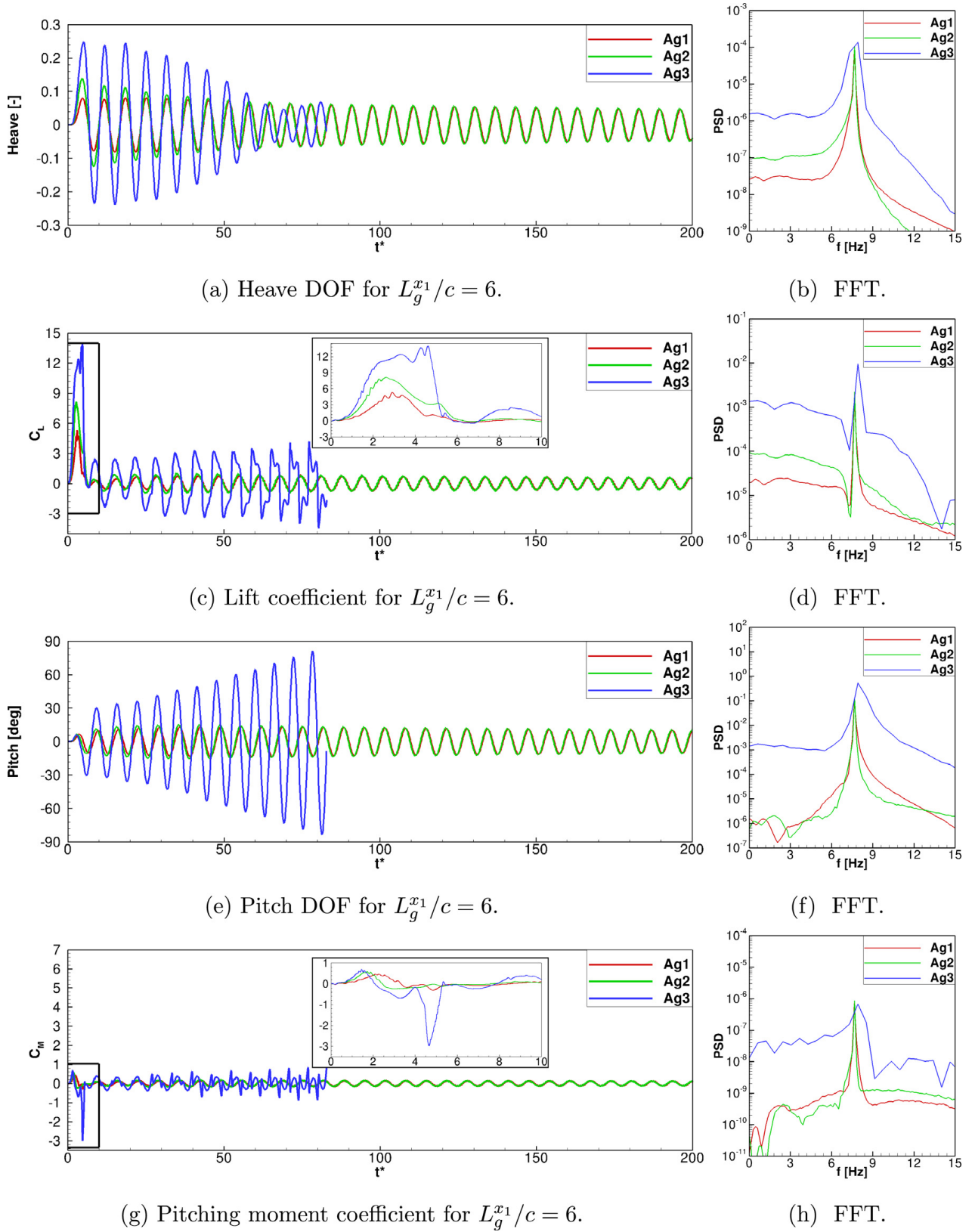


Fig. 11. Instantaneous heave and pitch DOFs (a,e), lift and pitching moment coefficients (c,g) of the elastically mounted airfoil when subjected to vertical gusts of length $L_g^{x1}/c = 6$ at three gust strengths $A_g/u_\infty = 1, 2$ and 3 .

5.4. LCO vs. flutter gust interaction

The y -vorticity field ω_y resulting from the gust encounter with the elastically mounted airfoil is compared for two cases in Fig. 12. The first is the LCO case ($L_g^{x1}/c = 2$, $A_g/u_\infty = 2$) and the second is the flutter scenario ($L_g^{x1}/c = 4$, $A_g/u_\infty = 3$). The snapshots of both cases are taken in such a way that the gust center is located at the same x -coordinate. In the first instant (i.e., snapshot I) the gust center is $0.2c$ in front of the leading edge. While the LEV is in its initial development stage for the shorter gust, both the LEV and the trailing-edge vortex (TEV) are formed and shed in the longer gust case. This is expected as the gust fully covers the airfoil in the latter case causing an increase in the effective angle of attack throughout the chord length.

In snapshots II and III, the gust center travels and reaches the mid-chord. During this time interval, the TEV resulting from the short gust is generated and shed far from the airfoil in the streamwise direction, whereas the vortices of the long gust are enlarged and vertically convected above the airfoil. The structures formed are similar to those found behind an airfoil immersed in a flow at different angles of attack. At this instant in time, the evaluated average effective angles of attack along the chord length are 58.5° and 70.6° for the short and the long gust, respectively.

In snapshot IV, the gust center arrives at the trailing edge pushing the trailing-edge vortices further from the airfoil. Moreover, it is visible that the gust reverses its influence on the pitch angle as a pitch angle of 0° is restored.

As the gust center starts leaving the TE in snapshot V (gust $0.6c$ behind TE), a second TEV is formed in the $L_g^{x1}/c = 2$ case. Due to the high counterclockwise moment generated by the long gust, a stronger downward pitching motion is observed leading to the shedding of a second LEV.

In snapshot VI, the airfoil–gust encounter is about to end in the $L_g^{x1}/c = 2$ case and the second TEV is seen to travel in the upstream direction and below the LEV. In the $L_g^{x1}/c = 4$ case, the second LEV collides with the TEV forming an area of highly disturbed flow. Moreover, due to the longer interaction time lasting 5 dimensionless time units, a second TEV starts to form.

Finally, in snapshots VII and VIII the vortices start to convect downstream in both cases. However, one could notice that for the short gust case nearly no vortices are present within the gust region discernible in Fig. 12 based on the sketch of the gust profile at the bottom of each figure. Instead, main vortices exist either behind or in front of the gust. In contrast, the dispersed vortices of the long gust case coexist with the gust and are convected slightly upwards. This is of great significance when the feedback effect of the structure on the gust is discussed.

Turning the attention to the post-interaction phase, Fig. 13 depicts three snapshots of the total vertical velocity field for both cases when the gust has already left the airfoil. In each instant, the trailing edge of both gusts is at the same distance x_{EE} away from the trailing edge of the airfoil. It can be clearly observed how the shape of the gust velocity profile is radically distorted due to the airfoil–gust interaction. The jet has no longer a continuous 1-cosine shape especially in the shielded area above the airfoil, i.e., $z > 0$. Instead the jet is cut into two halves (see instant I of Fig. 13). While the upper half of the jet freely convects in vertical direction and leaves the computational domain, the lower half of the jet faces the strongly disturbed field in the wake of the airfoil. The flow problem becomes similar to the situation when injecting a transverse jet into a cross-flow but with an injection plane which is continuously moving in streamwise direction. Since the prescribed gust is infinite (equivalent to having a jet supplying nozzle) the lower jet continues to pave its path back in the region above the airfoil (visible in instants II and III of Fig. 13). However, this vertical convection is accompanied by a disturbance-induced jet curvature further downstream. This signifies the capability of the SVM in capturing the full physics of the interaction involved.

5.5. Energy analysis

In order to analyze the motion that the airfoil undergoes due to its interaction with the vertical gust, an energy analysis is conducted. The energies involved are differentiated into the extracted and damped energies. The energy differential extracted by the airfoil from the flow for a degree of freedom z and its corresponding driving force F_z can be written as:

$$dE_{extr} = F_z dz = F_z \dot{z} dt. \quad (21)$$

For an oscillatory motion, the energy extracted per period (T) can be written as:

$$E_{extr} = \int_t^{t+T} F_z \dot{z} dt. \quad (22)$$

The product $(F_z \dot{z})$ shall be referred to as the integrand. The extracted energy for heave and pitch degrees of freedom then reads:

$$E_{h,extr} = \int_t^{t+T} F_L \dot{z} dt, \quad (23)$$

$$E_{p,extr} = \int_t^{t+T} M_p \dot{\alpha} dt, \quad (24)$$

where $F_L = F_{CFD,3}$ is the lift force, $\dot{z}$ is the heave velocity, $M_p = M_{CFD,2}$ is the pitching moment, and $\dot{\alpha}$ is the pitch velocity. It is obvious from Eqs. (23) and (24) that the phase shift between the velocity of the degree of freedom and its

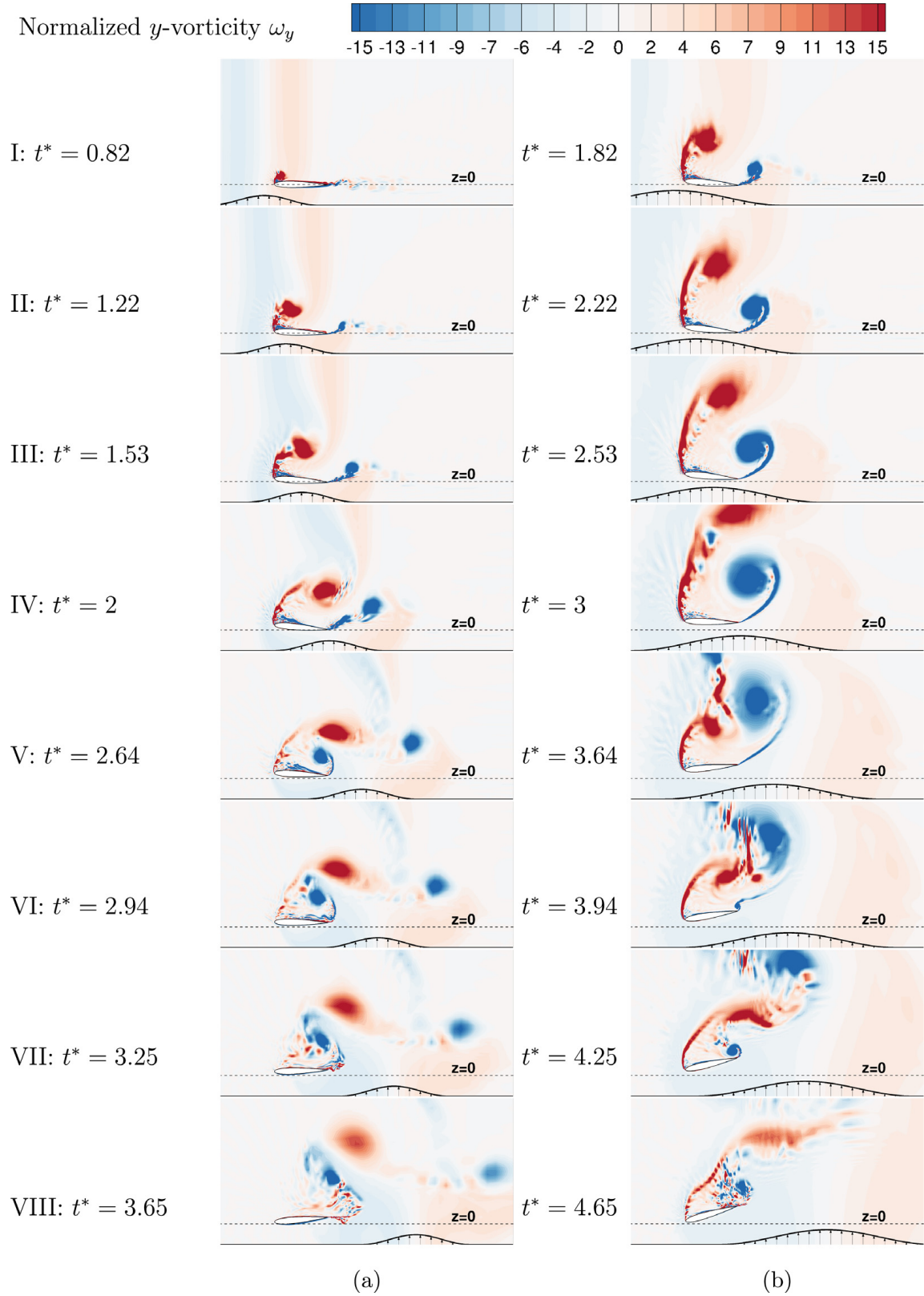


Fig. 12. Eight snapshots of the y -vorticity ω_y of the elastically mounted airfoil. (a) Short gust: $L_g^{\chi_1}/c = 2$, $A_g/u_\infty = 2$; (b) Long gust: $L_g^{\chi_1}/c = 4$, $A_g/u_\infty = 3$.

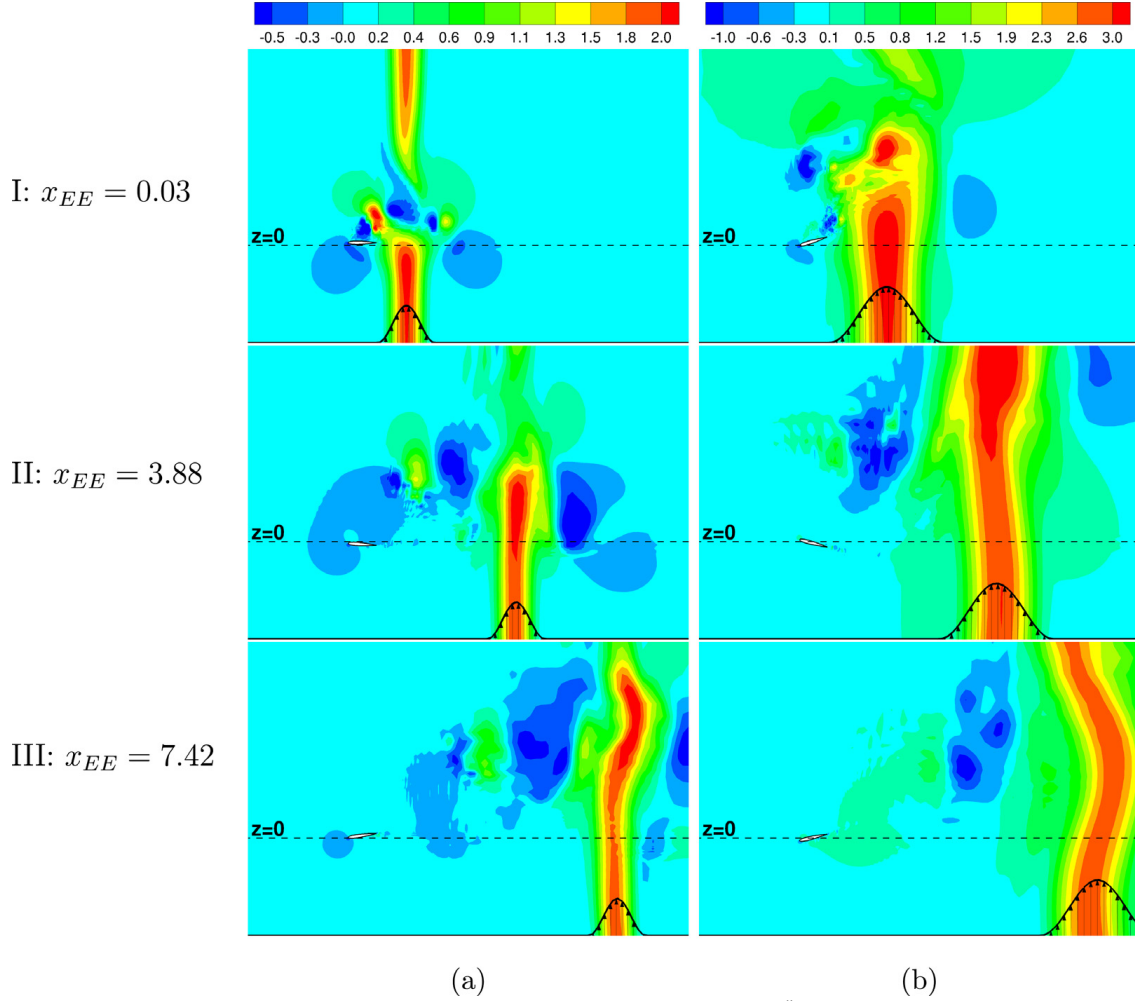


Fig. 13. Post-gust interaction. Total vertical velocity at three time instants. (a) Short gust: $L_g^{x1}/c = 2$, $A_g/u_\infty = 2$; (b) Long gust: $L_g^{x1}/c = 4$, $A_g/u_\infty = 3$.

driving force determines whether the flow is feeding the system with energy (i.e., positive E_{extr}) or damping its energy (i.e., negative E_{extr}) (Menon and Mittal, 2019). Similarly, the structural damping energy of each degree of freedom during a single oscillation could be computed by evaluating the integral:

$$E_{h,damp} = \int_t^{t+T} b_h \dot{z}^2 dt, \quad (25)$$

$$E_{p,damp} = \int_t^{t+T} b_\alpha \dot{\alpha}^2 dt, \quad (26)$$

where b_h and b_α are the material damping coefficients defined in Section 4. The difference between the extracted and damped energy of a specific degree of freedom defines its total energy:

$$E_{h,total} = E_{h,extr} - E_{h,damp}, \quad (27)$$

$$E_{p,total} = E_{p,extr} - E_{p,damp}. \quad (28)$$

Finally, since energy is a scalar quantity, the total energy of the system is defined by the summation of the total energy of both degrees of freedom:

$$E_{sys,total} = E_{h,total} + E_{p,total}. \quad (29)$$

Note that in the following the energy is made dimensionless with $\frac{1}{2} \rho u_\infty^2 A c$, where A is the projected area given by $A = l_c c$. Figs. 14(a) and 14(c) show the dimensionless DOF, the dimensionless velocity of the corresponding DOF, the driving force coefficient and the dimensionless integrand during one period of the limit-cycle oscillation (here for $L_g^{x1}/c = 2$ and $A_g/u_\infty = 2$) discussed in Section 5.1. It is worth mentioning that both lift and pitching moment coefficients are low-pass filtered with a cut-off frequency of $f_c = 25$ Hz. Filtering is carried out to facilitate the visualization of the relevant low-frequency dynamics (as this suppresses the short-term fluctuations visible in Fig. 4 and highlights the long-term trend) and to enable a low-frequency range analysis where dominant frequencies exist. Performing such a filtering

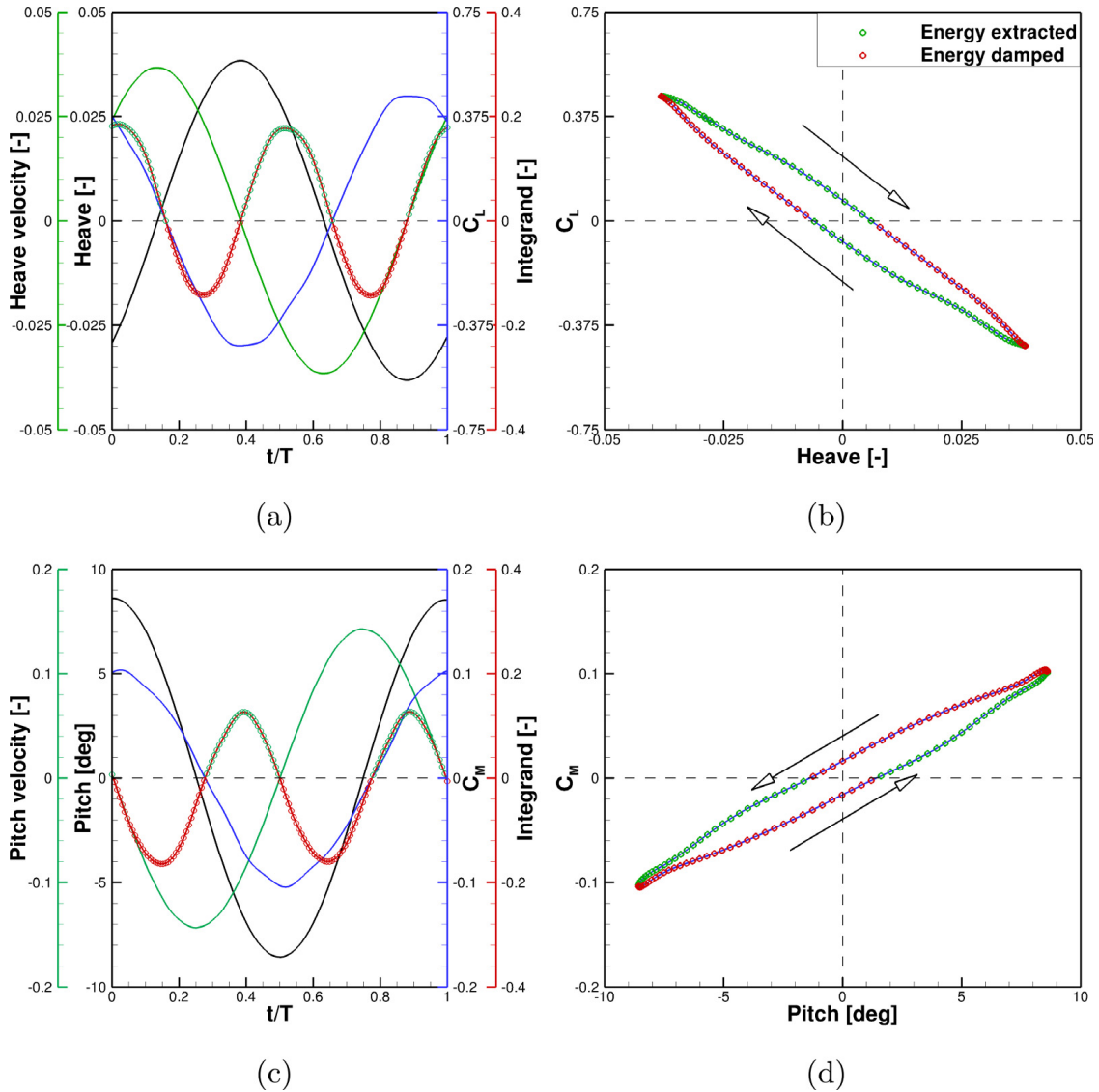


Fig. 14. Analysis within a single LCO cycle for $L_g^{x1}/c = 2$ and $A_g/u_\infty = 2$. Dimensionless DOF, velocity of the corresponding DOF, aerodynamic coefficient and integrand (a,c), aerodynamic coefficient vs. its respective DOF (b,d). (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

was found to be of no influence on the work done by the aerodynamic force and moment as it is insensitive to the high-frequency content contained within these quantities (Poirel et al., 2011). As mentioned earlier, the integrand is a derived quantity obtained by the product of the respective velocity of the DOF by its driving force (i.e., the green curve times the blue curve). The integral of this product (or the area confined between the curve and the x-axis) represents the total dimensionless energy extracted within that period. While the value of this integral is found to be positive for the heave DOF, the interaction of the pitch motion with the flow yields a negative value. This indicates that the flow feeds the airfoil with energy through the heave DOF (i.e., acting as an energy source) and absorbs this energy back from the pitch DOF (i.e., acting as an energy sink). The sum of the interchanged energy between the flow and the airfoil in combination with the total structural damping energies leads to a vanishing total energy as will be shown later.

Figs. 14(b) and 14(d) portray the aerodynamic hysteresis curves by plotting the lift coefficient as a function of the heave and the pitching moment coefficient as a function of the pitch angle during one cycle, respectively. The arrows indicate the direction of advancing time. The green and red circles correspond to those plotted in the integrand curves in Figs. 14(a) and 14(c). They are used to identify the phases in which energy is being extracted from the flow or delivered to the flow. The direction in which the ellipse is formed (clockwise or counterclockwise) indicates the sign of the work done by the aerodynamic coefficients (Poirel et al., 2008). The clockwise rotation in Fig. 14(b) signifies that positive work (or a negative aerodynamic damping) is being imposed by the lift coefficient on the airfoil. In contrast, the counterclockwise rotation seen in Fig. 14(d) points to the overall damping effect that the pitching moment coefficient poses on the airfoil. Thus, the directions of energy transfers for heave and pitch are consistent with the analysis above. Both aerodynamic coefficients mainly show a linear dependency on their respective degrees of freedom. This is unfolded in Figs. 14(a) and 14(c) where the aerodynamic coefficients and the degrees of freedom are sinusoidal functions separated by a phase shift.

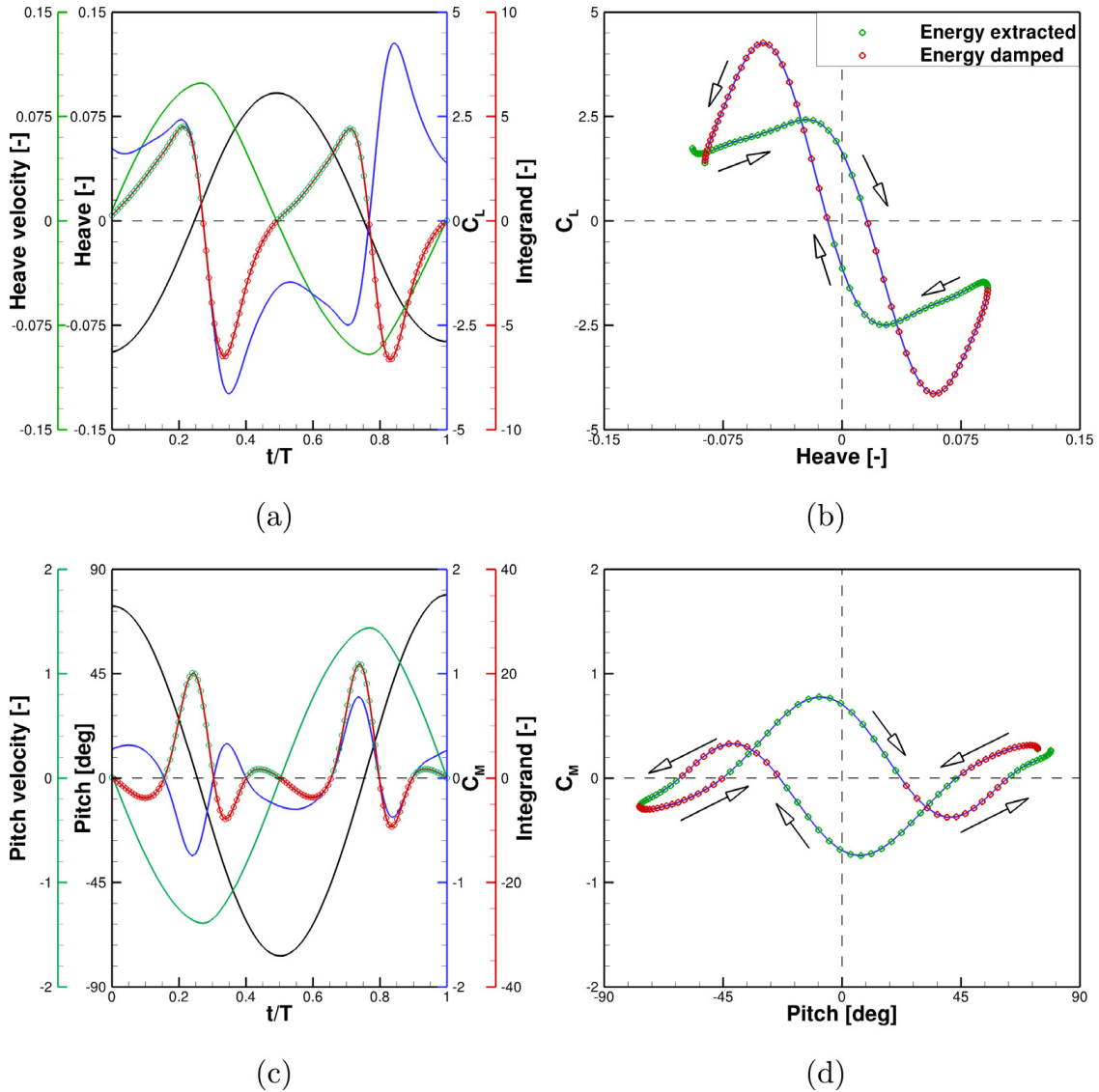


Fig. 15. Analysis within a single diverging cycle for $L_g^{x1}/c = 4$ and $A_g/u_\infty = 3$. Dimensionless DOF, velocity of the corresponding DOF, aerodynamic coefficient and integrand (a,c), aerodynamic coefficient vs. its respective DOF (b,d).

The linear relationship is even more obvious in Figs. 14(b) and 14(d) where the slope of the formed ring is mainly constant. Note that the product of the slope with $\frac{1}{2}\rho u_\infty^2 A$ for the heave and $\frac{1}{2}\rho u_\infty^2 A c$ for the pitch is interpreted as an equivalent aerodynamic stiffness coefficient by Poirel et al. (2008).

Fig. 15 demonstrates identical relations as those presented in Fig. 14 but for a diverging oscillation (here for $L_g^{x1}/c = 4$ and $A_g/u_\infty = 3$) leading to flutter. In contrast to the LCO case, the areas evaluated under the heave and pitch integrands are found to be negative and positive, respectively. This indicates that the areas evaluated under the heave and pitch integrands supply and energy dissipation. Focusing on Figs. 15(a) and 15(c) one could directly distinguish the high non-linearity (non-sinusoidal curves) of the aerodynamic coefficients and the complex distribution of the extracted energy integrands when displayed as a function of their respective DOFs. This is even more evident in Figs. 15(b) and 15(d) where the aforementioned equivalent aerodynamic stiffness coefficients experience sudden changes in their sign and magnitude within one cycle. The source of this non-linear response is attributed to the dynamic stall that the airfoil is undergoing. Indeed, the imposed strong gust excites the pitch DOF of the airfoil to levels exceeding its critical angle of attack (also known as stall angle). It is also worth mentioning that, unlike the LCO case, the loops formed by plotting the aerodynamic coefficients as a function of their respective DOFs switch their direction of turning during the cycle. This indicates that the flow is alternately amplifying (clockwise middle loop) and damping (counterclockwise end loops) the energy of the system within a single cycle.

The energy analysis is now further extended to cover the entire evolution of the different energy components defined in Eqs. (27) to (29) for each DOF. Figs. 16(a) and 16(c) depict the energy components of the heave and pitch DOFs evaluated for their respective cycles of a LCO ($L_g^{x1}/c = 2$, $A_g/u_\infty = 2$) and a diverging motion ($L_g^{x1}/c = 4$, $A_g/u_\infty = 3$), respectively. A single period of an oscillation is defined as the time required for a DOF to record two consecutive peaks. Since the DOFs are not in phase, energy components of different DOFs were computed separately. It is also worth mentioning that

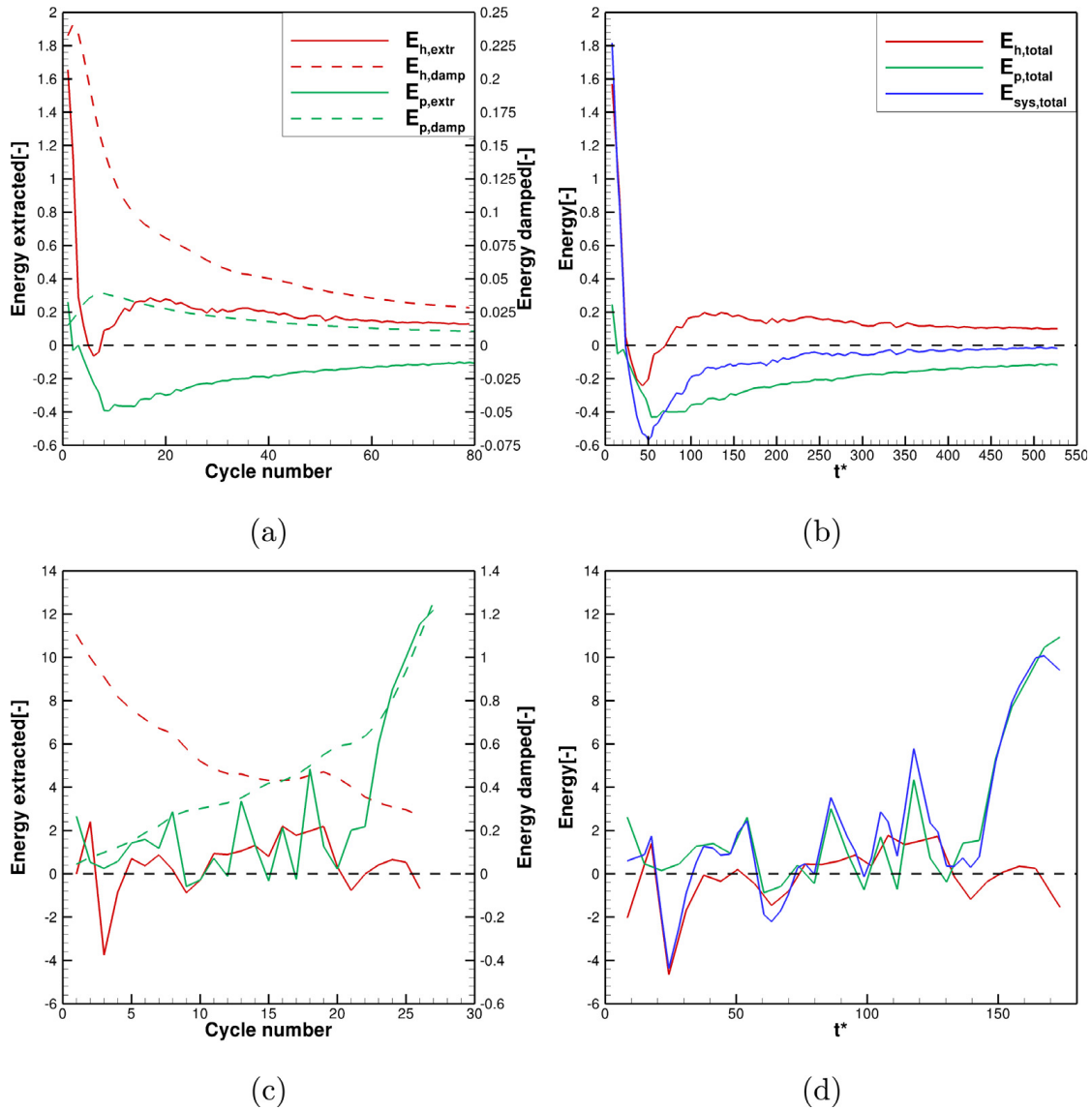


Fig. 16. Analysis of the dimensionless energy for a limit-cycle oscillation case ($L_g^{x_1}/c = 2$ and $A_g/u_\infty = 2$) and a diverging motion case ($L_g^{x_1}/c = 4$ and $A_g/u_\infty = 3$). Heave and pitch extracted and damped energies computed for each respective cycle (a,c), the dimensionless time histories of the instantaneous total energies of both DOFs and the total energy of the system (b,d).

the initial airfoil–gust interaction phase is excluded from the analysis. This phase is eliminated since it contains high energy peaks from which the system’s energy starts decaying and rather complex aerodynamic coefficients not following a simple harmonic motion (see zoom-in in Figs. 9 to 11) are observed. Eqs. (27) to (29) are then employed to plot the total heave, pitch and system’s energy as a function of the dimensionless time in Figs. 16(b) and 16(d). Focusing on the LCO case, it can be clearly seen in Fig. 16(a) that both DOFs possess an initially positive extracted energy which undergoes a steep decay. This is foreseeable as the flow (including the gust) initially excites and feeds the system with energy in favor of the definition of positive degrees of freedom. The subsequent sharp decrease in the extracted energy is attributed to the airfoil’s high kinetic and potential energies being transferred back to the flow in an attempt to achieve a global energy equilibrium. While the extracted energy of the pitch DOF declines below zero and remains negative throughout the simulation time, its heave DOF counterpart stabilizes at positive values. This in turn is in agreement with the finding discussed in Figs. 14(b) and 14(d). The structural damping energies of the heave and pitch DOFs, despite their different magnitudes, show the same behavior. Both energies experience an initial increase attributed to the temporary increase in the velocity of the DOFs followed by a steady decrease to a constant value. The total energies shown in Fig. 16(b) further prove that the system undergoes a LCO motion as the heave and pitch DOFs energies converge to identical values of opposite signs leading to a vanishing total energy of the system.

A completely different outcome is obtained when the same analysis is applied to the diverging motion case. Energy extraction of both DOFs is initially found to be chaotic and not following any perceivable direction (see Fig. 16(c)). The system is seen to gain energy from the flow in some cycles and losing it in others. This is attributed to the highly transient dynamics (the complex aerodynamic coefficients and the varying phase angle between these coefficients and

the velocities of the DOFs) involved in this case caused by the strong gust-induced instability. This behavior is seen until the extracted energy by the pitch DOF starts accumulating. On the other hand, the structural damping energies dictated by the velocities show a clearly recognizable trend. The structural damping of the heave DOF experiences a continuous decrease indicating that its velocity is reducing across the cycles. In contrast, the structural damping of the pitch DOF (and hence its velocity) witnesses a sustained increase until the simulation diverges. The total energies depicted in Fig. 16(d) clearly show that in the case of the diverging motion the main part of the energy is extracted by the pitch motion. That is in clear contrast to the LCO case, where the heave motion was found to be responsible for the energy extraction.

6. Conclusions

The split velocity method combined with a partitioned FSI solver is applied to study the gust-induced oscillations of an elastically mounted NACA 0012 airfoil. This high-fidelity simulation concept relies on an eddy-resolving approach (LES) to guarantee that the most important flow structures are captured, body-fitted block-structured grids (ALE) to resolve the boundary layers and an advanced grid adaptation algorithm (IDW-TFI) to ensure that the high grid quality required for LES is conserved. Choosing a transitional Reynolds number and high gust ratios relevant for MAVs, and a 2-DOF setup with both heave and pitch motion, the following conclusions can be drawn:

- A vertical gust hitting either a fixed ($\alpha = 0^\circ$) or an elastically mounted airfoil from the bottom leads to a strong clockwise rotating leading-edge vortex. That induces a positive peak in the lift and in the pitching moment. During the airfoil–gust interaction (i.e., $t^* \leq 3$ for a gust length $L_g^x/c = 2$) both the flow fields as well as the forces and moments are nearly identical for the fixed and the elastically mounted airfoil. After the gust has left the airfoil, a limit-cycle oscillation is observed for the heave and pitch motion, whereas in the fixed case the quasi steady-state values are recovered.
- An analysis of the effect of different gust strengths A_g confirms the expectation that the initial heave excitation strongly depends on A_g . Astonishingly that is not the same for the pitch motion.
- For the same gust strength, wider gusts lead to higher amplitudes of the aerodynamic forces and the heave and pitch motion during the airfoil–gust interaction.
- Seven out of nine investigated cases finally end up in limit-cycle oscillations of the airfoil. Although the initial excitation during the gust impact depends on the gust strength and gust length, the amplitudes of the heave and pitch oscillations, which are observed after a long transition period, are all identical for these LCO cases. Thus, the final outcome of these gust impacts is independent of the gust and purely influenced by the non-gust flow interaction with the airfoil dynamics.
- Two out of the nine cases finally induce flutter with a diverging pitch motion. These are the setups with the strongest gusts given by $A_g/u_\infty = 3$ and $L_g^x/c \geq 4$. The longer the gust is, the faster the extreme pitch excitation is reached.
- The above-mentioned limit between both modes is obviously depending on the parameters defining the system, i.e., inertia, stiffness, and damping parameters as well as x_{EG} and Re number. Nevertheless, the results shown are generic in a sense that a configuration is considered which depending on the strength of excitation due to different gusts ends up in either LCO or flutter.
- By means of an energy analysis the significant differences between the LCO and the flutter case are worked out. The former case is driven by the heave motion, whereas the pitch DOF acts as an energy sink. In the flutter scenario the roles of energy supply and energy dissipation are opposite. Now the pitching motion is powering the coupled system, whereas the heave motion dissipates energy.

Declaration of competing interest

The authors declare the following financial interests/personal relationships which may be considered as potential competing interests: Michael Breuer reports financial support was provided by German Research Foundation.

Data availability

Data will be made available on request.

Acknowledgement

The project is financially supported by the German Research Foundation (*Deutsche Forschungsgemeinschaft*) under the contract number BR 1847/17-1.

Appendix. Evaluation of the near-wall resolution

To evaluate the near-wall resolution of the applied grid, additional LES predictions were carried out for the flow around the fixed airfoil at various angles of attack in the range $0 \leq \alpha \leq 80^\circ$. For this purpose, the inner part of the grid was rotated using the hybrid grid adaption algorithm IDW-TFI described in Section 3.1. Of course, Re and all other parameters

of the setup were unchanged. The instantaneous simulations were carried out over a time interval of about twenty vortex shedding cycles. The results were averaged in time and additionally in the homogeneous spanwise direction. Based on the predicted local wall shear stress τ_w , the shear stress velocity u_τ is determined, finally leading to the $y^+ = \Delta y u_\tau / \nu$ values of the wall-nearest grid points shown in Fig. A.17. Except for the case of $\alpha = 0^\circ$ the outcome is different for the suction and the pressure side of the airfoil. In the separated flow regions the wall shear stresses are lower leading to y^+

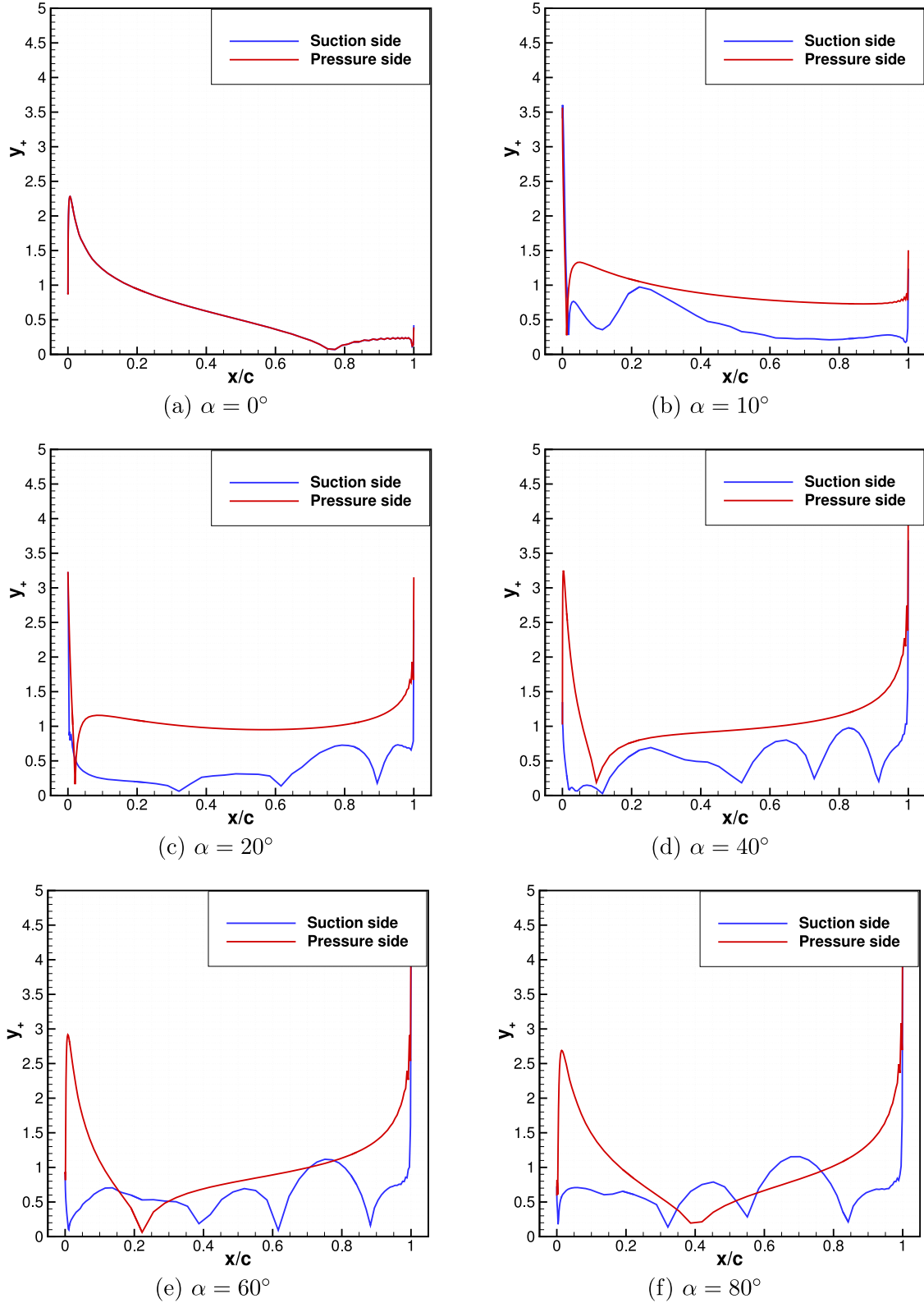


Fig. A.17. Distribution of y^+ of the wall-nearest grid points on the airfoil for six different angles of attack.

values below unity in most regions with the exception of the trailing edge. On the pressure side the values are in general slightly higher but even the peak values are below $y^+ = 5$. Thus, it can be concluded that the dimensionless wall distance is in the range which allows to resolve the viscous sublayer even for the cases with high angles of attack.

References

- Andreu-Angulo, I., Babinsky, H., Biler, H., Sedky, G., Jones, A.R., 2020. Effect of transverse gust velocity profiles. *AIAA J.* 58 (12), 5123–5133.
- Apostolatos, A., De Nayer, G., Bletzinger, K.-U., Breuer, M., Wüchner, R., 2019. Systematic evaluation of the interface description for fluid–structure interaction simulations using the isogeometric mortar-based mapping. *J. Fluids Struct.* 86, 368–399.
- Badrya, C., Baeder, J., Jones, A., 2019. Application of prescribed velocity methods to a large-amplitude flat-plate gust encounter. *AIAA J.* 57 (8), 1–13.
- Barnes, C., Visbal, M., 2018a. Gust response of rigid and elastically mounted airfoils at a transitional Reynolds number. *Aerosp. Sci. Technol.* 74, 112–119.
- Barnes, C., Visbal, M., 2018b. On the role of flow transition in laminar separation flutter. *J. Fluids Struct.* 77, 213–230.
- Boulbrachene, K., De Nayer, G., Breuer, M., 2021. Assessment of two wind gust injection methods: Field velocity vs. split velocity method. *J. Wind Eng. Ind. Aerodyn.* 218, 104790.
- Breuer, M., 2018. Effect of inflow turbulence on an airfoil flow with laminar separation bubble: An LES study. *Flow Turbul. Combust.* 101 (2), 433–456.
- Breuer, M., De Nayer, G., Münsch, M., Gallinger, T., Wüchner, R., 2012. Fluid-structure interaction using a partitioned semi-implicit predictor-corrector coupling scheme for the application of large-eddy simulation. *J. Fluids Struct.* 29, 107–130.
- De Nayer, G., Apostolatos, A., Wood, J.N., Bletzinger, K.-U., Wüchner, R., Breuer, M., 2018a. Numerical studies on the instantaneous fluid–structure interaction of an air-inflated flexible membrane in turbulent flow. *J. Fluids Struct.* 82, 577–609.
- De Nayer, G., Breuer, M., 2014. Numerical FSI investigation based on LES: Flow past a cylinder with a flexible splitter plate involving large deformations (FSI-PfS-2a). *Int. J. Heat Fluid Flow* 50, 300–315.
- De Nayer, G., Breuer, M., 2020. A source-term formulation for injecting wind gusts in CFD simulations. *J. Wind Eng. Ind. Aerodyn.* 207, 104405.
- De Nayer, G., Breuer, M., Perali, P., Grollmann, K., 2019. Modeling of wind gusts for large-eddy simulations related to fluid-structure interactions. In: Salvetti, M., Armenio, V., Fröhlich, J., Geurts, B.J., Kuerten, H. (Eds.), *ERCOFTAC Series, Direct and Large-Eddy Simulation XI*, 11th Int. ERCOFTAC Workshop on Direct and Large-Eddy Simulation: DLES-11, May 29–31, 2017, Vol. 25. Springer Nature Switzerland AG 2019, pp. 453–459.
- De Nayer, G., Breuer, M., Wood, J.N., 2020. Numerical investigations on the dynamic behavior of a 2-DOF airfoil in the transitional Re number regime based on fully coupled simulations relying on an eddy-resolving technique. *Int. J. Heat Fluid Flow* 85, 108631.
- De Nayer, G., Kalmbach, A., Breuer, M., Sicklinger, S., Wüchner, R., 2014. Flow past a cylinder with a flexible splitter plate: A complementary experimental-numerical investigation and a new FSI test case (FSI-PfS-1a). *Comput. & Fluids* 99, 18–43.
- De Nayer, G., Schmidt, S., Wood, J.N., Breuer, M., 2018b. Enhanced injection method for synthetically generated turbulence within the flow domain of eddy-resolving simulations. *Comput. Math. Appl.* 75 (7), 2338–2355.
- Demirdžić, I., Perić, M., 1988. Space conservation law in finite-volume calculations of fluid flow. *Int. J. Numer. Methods Fluids* 8 (9), 1037–1050.
- Demirdžić, I., Perić, M., 1990. Finite-volume method for prediction of fluid flow in arbitrarily shaped domains with moving boundaries. *Int. J. Numer. Methods Fluids* 10 (7), 771–790.
- Dowell, E., Edwards, J., Strganac, T., 2003. Nonlinear aeroelasticity. *J. Aircr.* 40 (5), 857–874.
- Dowell, E.H., Tang, D., 2002. Nonlinear aeroelasticity and unsteady aerodynamics. *AIAA J.* 40 (9), 1697–1707.
- Durst, F., Schäfer, M., 1996. A parallel block-structured multigrid method for the prediction of incompressible flows. *Int. J. Numer. Methods Fluids* 22 (6), 549–565.
- Ekaterinaris, J.A., Platzer, M.F., 1998. Computational prediction of airfoil dynamic stall. *Prog. Aerosp. Sci.* 33 (11), 759–846.
- Eldredge, J.D., Jones, A.R., 2019. Leading-edge vortices: Mechanics and modeling. *Annu. Rev. Fluid Mech.* 51 (1), 75–104.
- Galbraith, M.C., Visbal, M.R., 2009. Implicit large-eddy simulation of low-Reynolds number transitional flow past the SD7003 airfoil. In: 46th AIAA Aerospace Sciences Meeting and Exhibit, 7–10 January 2008, Reno, Nevada. URL <https://doi.org/10.2514/6.2008-225>.
- Heinrich, R., Reimer, L., 2013. Comparison of different approaches for gust modeling in the CFD code TAU. In: *International Forum on Aeroelasticity & Structural Dynamics 2013*. URL <https://elib.dlr.de/85834/>.
- Huntley, S.J., Jones, D., Gaitonde, A., 2016. 2D and 3D gust response using a prescribed velocity method in viscous flows. In: 46th AIAA Fluid Dynamics Conference. American Institute of Aeronautics and Astronautics, pp. 1–15.
- Huntley, S.J., Jones, D., Gaitonde, A., 2017. Aeroelastic gust response of an aircraft using a prescribed velocity method in viscous flows. In: 23rd AIAA Computational Fluid Dynamics Conference. American Institute of Aeronautics and Astronautics, Denver, Colorado, pp. 1–18.
- IEC-Standard, 2002. 61400-21. Tech. rep..
- Kalmbach, A., Breuer, M., 2013. Experimental PIV/V3V measurements of vortex-induced fluid–structure interaction in turbulent flow – A new benchmark FSI-PfS-2a. *J. Fluids Struct.* 42, 369–387.
- Lerooy, A., Visonneau, M., 2005. Numerical methods for RANSE simulations of a self-propelled fish-like body. *J. Fluids Struct.* 20 (7), 975–991.
- Lesoinne, M., Farhat, C., 1996. Geometric conservation laws for flow problems with moving boundaries and deformable meshes, and their impact on aeroelastic computations. *Comput. Methods Appl. Mech. Engrg.* 134 (1–2), 71–90.
- Lobo, B.A., Özçakmak, Ö.S., Madsen, H.A., Schaffarczyk, A.P., Breuer, M., Sørensen, N.N., 2022. On the laminar-turbulent transition mechanism on Megawatt wind turbine blades operating in atmospheric flow. *Wind Energy Sci.* URL <https://doi.org/10.5194/wes-2022-42>.
- McCroskey, W.J., 1982. Unsteady airfoils. *Annu. Rev. Fluid Mech.* 14 (1), 285–311.
- Mellen, C.P., Fröhlich, J., Rodi, W., 2003. Lessons from LESFOIL project on large-eddy simulation of flow around an airfoil. *AIAA J.* 41 (4), 573–581.
- Menon, K., Mittal, R., 2019. Flow physics and dynamics of flow-induced pitch oscillations of an airfoil. *J. Fluid Mech.* 877, 582–613.
- Menon, K., Mittal, R., 2020. Aeroelastic response of an airfoil to gusts: Prediction and control strategies from computed energy maps. *J. Fluids Struct.* 97, 103078.
- Newmark, N.M., 1959. A method of computation for structural dynamics. *J. Eng. Mech. Div.* 85 (3), 67–94.
- Perrotta, G., Jones, A.R., 2017. Unsteady forcing on a flat-plate wing in large transverse gusts. *Exp. Fluids* 58 (101).
- Poirel, D., Harris, Y., Benaissa, A., 2008. Self-sustained aeroelastic oscillations of a NACA 0012 airfoil at low-to-moderate Reynolds numbers. *J. Fluids Struct.* 24, 700–719.
- Poirel, D., Métivier, V., Dumas, G., 2011. Computational aeroelastic simulations of self-sustained pitch oscillations of a NACA 0012 at transitional Reynolds numbers. *J. Fluids Struct.* 27, 1262–1277.
- Poirel, D., Yuan, W., 2010. Aerodynamics of laminar separation flutter at a transitional Reynolds number. *J. Fluids Struct.* 26, 1174–1194.
- Poudel, N., Yu, M., Hrynyuk, J.T., 2021. Gust mitigation with an oscillating airfoil at low Reynolds number. *Phys. Fluids* 33 (10), 101905.
- Schmidt, S., 2016. Entwicklung einer hybriden LES-URANS Methode für interne und externe Strömungen (Ph.D. thesis). Professur für Strömungsmechanik, Helmut-Schmidt-Universität, Hamburg, Germany, URL <https://openhsu.ub.hsu-hh.de/handle/10.24405/4295>.

- Schmidt, S., Breuer, M., 2014. Hybrid LES–URANS methodology for the prediction of non–equilibrium wall–bounded internal and external flows. *Comput. & Fluids* 96, 226–252.
- Schmidt, S., Breuer, M., 2017. Source term based synthetic turbulence inflow generator for eddy–resolving predictions of an airfoil flow including a laminar separation bubble. *Comput. & Fluids* 146, 1–22.
- Sen, S., De Nayer, G., Breuer, M., 2017. A fast and robust hybrid method for block-structured mesh deformation with emphasis on FSI-LES applications. *Int. J. Numer. Methods Eng.* 111 (3), 273–300.
- Smagorinsky, J., 1963. General circulation experiments with the primitive equations I: The basic experiment. *Mon. Weather Rev.* 91 (3), 99–165.
- Theodorsen, T., 1935. General Theory of Aerodynamic Instability and the Mechanism of Flutter. NACA Report 496.
- Visbal, M.R., Gordnier, R.E., Galbraith, M.C., 2009. High-fidelity simulations of moving and flexible airfoils at low Reynolds numbers. *Exp. Fluids* 46, 903–922.
- von Kármán, T., Sears, W.R., 1938. Airfoil theory for non-uniform motion. *J. Aeronaut. Sci.* 5 (10), 379–390.
- Wales, C., Jones, D., Gaitonde, A., 2014. Prescribed velocity method for simulation of aerofoil gust responses. *J. Aircr.* 52 (1), 64–76.
- Wood, J.N., Breuer, M., De Nayer, G., 2018. Experimental studies on the instantaneous fluid–structure interaction of an air–inflated flexible membrane in turbulent flow. *J. Fluids Struct.* 80, 405–440.
- Wood, J.N., Breuer, M., De Nayer, G., 2020. Experimental investigations on the dynamic behavior of a 2-DOF airfoil in the transitional Re number regime based on digital-image correlation measurements. *J. Fluids Struct.* 96, 103052.
- Wood, J.N., De Nayer, G., Schmidt, S., Breuer, M., 2016. Experimental investigation and large-eddy simulation of the turbulent flow past a smooth and rigid hemisphere. *Flow Turbul. Combust.* 97 (1), 79–119.
- Yuan, W., Poirel, D., Wang, B., 2013. Simulations of pitch-heave limit-cycle oscillations at a transitional Reynolds number. *AIAA J.* 51 (7), 1716–1732.

Publication III: FSI Simulations of Wind Gusts Impacting an Air-Inflated Flexible Membrane at $Re = 100,000$

Reference

G. De Nayer, M. Breuer and K. Boulbrachene. FSI simulations of wind gusts impacting an air-inflated flexible membrane at $Re = 100,000$. *Journal of Fluids and Structures* 109 (2022), 103462. DOI: [10.1016/j.jfluidstructs.2021.103462](https://doi.org/10.1016/j.jfluidstructs.2021.103462)

Copyright

© 2021 The Authors. Published by Elsevier Ltd. This is an open access article distributed under the terms of the Creative Commons CC-BY license (<http://creativecommons.org/licenses/by/4.0/>), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.



FSI simulations of wind gusts impacting an air-inflated flexible membrane at $Re = 100,000$

G. De Nayer, M. Breuer*, K. Boulbrachene

Professur für Strömungsmechanik, Helmut-Schmidt-Universität Hamburg, D-22043 Hamburg, Germany

ARTICLE INFO

Article history:

Received 13 July 2021

Received in revised form 26 November 2021

Accepted 3 December 2021

Available online 28 December 2021

Keywords:

Wind gust

Fluid–structure interaction (FSI)

Hemisphere

Membrane

Turbulent flow

Large-eddy simulation (LES)

Source-term formulation

ABSTRACT

The paper addresses the simulation of turbulent wind gusts hitting rigid and flexible structures. The purpose is to show that such kind of complex fluid–structure interaction (FSI) problems can be simulated by high-fidelity numerical techniques with reasonable computational effort. The main ingredients required for this objective are an efficient method to inject wind gusts within the computational domain by the application of a recently developed source-term formulation, an equally effective method to prescribe the incoming turbulent flow and last but not least a reliable FSI simulation methodology to predict coupled problems based on a partitioned solution approach combining an LES fluid solver with a FEM/IGA solver for the structure. The present application is concerned with a rigid and a membranous hemisphere installed in a turbulent boundary layer and impacted by wind gusts of different strength. The methodology suggested allows to inject the gusts in close vicinity of the object of interest, which is typically well resolved. Therefore, the launch and transport of the wind gust can be realized without visible numerical dissipation and without large computational effort. The effect of the gusts on the flow field, the resulting forces on the structure and the corresponding deformations in case of the flexible structure are analyzed in detail. A comparison between the rigid and the flexible case makes it possible to work out the direct reaction of the deformations on the force histories during the impact. Furthermore, in case of the flexible structure the temporal relationships between local or global force developments and the local deformations are evaluated. Such predictions pinpoint the areas of high stresses and strains, where the material is susceptible to failure.

© 2021 The Author(s). Published by Elsevier Ltd. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

1. Introduction

Fluid–structure interaction (FSI) problems constitute a category of problems belonging to interdisciplinary subjects which have drawn the interest of scientists due to its crucial relevance to many industrial and practical applications. The fields in which these applications are found include aeronautics (e.g., wings), wind engineering (e.g., wind turbines), civil engineering (bridges, skyscrapers and lightweight thin structures) and even biomedical engineering (e.g., cardiac valves).

Among the wide range of applications, this work concentrates on the interaction of fluid flows with flexible membranous structures. Such structures have become widely used in the modern civil engineering industry due to their practical characteristics. Large expanses of space could be optimally enclosed by such structures as they are solely subjected to tensile membrane loads. They are also featured by their formability enabling the creative minds to come up with designs

* Corresponding author.

E-mail address: breuer@hsu-hh.de (M. Breuer).

which were not feasible with traditional construction materials while maintaining a cost-efficient solution. In a structure of this type, a pre-stress defining its shape and tensile state is required. For a tensile membrane structure, a pre-stress is imposed by the unique selection of boundary conditions, whereas for an air-inflated structure, a pre-stress is imposed by internal pressurized air. A careful consideration of this design parameter is necessary since it determines the stability and the integrity of a structure undergoing an interaction with different wind load scenarios. To this end, a realistic modeling of wind loads acting on this structure becomes mandatory to achieve reliable structural designs (Newman et al., 1984).

In nature, most extreme wind events are characterized by being non-stationary and highly transient (e.g., thunderstorms, tornados or wind gusts). The physics associated with convective gusts (e.g., winds spawned by thunderstorms) is fundamentally different from that involving conventional momentum driven turbulence (e.g., winds in neutrally stable atmospheric boundary layers) in both, kinematics and dynamics. The main distinguishing aspects are the dissimilar velocity profiles and the distinct statistical behavior (Kwon and Kareem, 2013).

In contrast to traditional logarithmic or power-law velocity profiles developing in an atmospheric boundary layer, the presence of a gust yields an inverted velocity profile with a peak value near the surface. Such a variation in the velocity profile certainly leads to a considerable change in the interaction between the fluid and the structure. This change could be further compounded by an increase in the parameters of the inherently transient gusts studied (i.e., gust amplitude and length scales). Moreover, a careful examination of the statistical nature of such extreme wind events is required. Typically, design wind loads are estimated based on a velocity profile (depending on the terrain category) evaluated by superimposing a mean wind velocity profile and gust-induced fluctuations characterized by a statistically stationary process. This assumption is, however, highly questionable as in the event of a wind gust, sharp changes in the wind speed are pronounced (Kareem and Wu, 2013).

Although design standards are established providing guidelines to prevent structural damages due to wind from happening (c.f., American Society of Civil Engineers, 1997; Frost et al., 1978; IEC-Standard, 2002; Burton et al., 2001; Kasperski, 2007), these are based on assumptions to alleviate the efforts of the design process. Therefore, the development of methods that help in establishing a framework for investigating the effect of realistic wind loads (transient, highly dynamic wind events), which significantly affect the structural response, would substantially contribute to the design procedure. Highly resolved numerical simulations of the flow field and the resulting deformations are seen as an effective tool to gain a deeper insight into the complex physics involved in such problems (Michalski et al., 2011, 2015).

The available literature concerned with the numerical simulation of the FSI problem comprising the evaluation of wind effects on flexible structures is vast and includes a wide range of applications. In the course of demonstrating the reliability of the coupling algorithm employed for conducting FSI simulations, Glück et al. (2003) simulated unsteady deformations of a membranous roof exposed to a basic flow superimposed by a time-dependent wind gust at the top inlet boundary. In Wüchner et al. (2007), transient FSI simulations were carried out and the maximum inflow wind speed was varied in a periodic manner to imitate the effect of a gust on a four-point membranous tent. Wu et al. (2008) investigated the interaction between the flow over a two-dimensional low-rise building and its flexible membranous roof. The inflow of the considered configuration was defined by a power law and the FSI dominant factors were highlighted by modifying flow and structural parameters. The results of the flexible case were analyzed and compared to its rigid counterpart. Yuan et al. (2010) carried out FSI simulations to describe the dynamic response of a flexible liquid-storage container under strong wind actions. A penalty based finite-element method combined with the Arbitrary Lagrangian–Eulerian description was used to solve the FSI problem. Several turbulent wind cases were accomplished and design recommendations were provided. The relevant studies of Michalski et al. (2011, 2015) simulated the structural response of a large membranous umbrella subjected to transient wind loads, where the effects of the gust were implicitly incorporated in the synthetically generated wind field. The study was supplemented by a statistical validation of the numerical methodology adopted based on a real-scale experimental setup. Augier et al. (2012) provided a quantitative experimental validation of a dynamic FSI model devoted to simulate the aeroelastic problem of yacht sails and rig. The FSI model coupled an inviscid flow solver based on the vortex-lattice method with a finite-element method based structural solver. The experimental and numerical results of the flying shapes of the sails and the loads on the rigs showed good agreement. Mokin et al. (2017) simulated the aeroelastic behavior of an air-supported structure by a monolithic and a partitioned approach. An inflated shell in form of a clipped 3/4 sphere with a prescribed internal pressure was subjected to an air flow of high Reynolds number to mimic the wind interaction during storms. Results of different wind speeds were compared against experimental data and a good agreement was achieved. In the scope of the recent advancements in the prediction of the aeroelastic response of wind turbines, Santo et al. (2020) carried out high-fidelity FSI simulations of a horizontal-axis wind turbine immersed in an atmospheric boundary layer based on the RANS approach. Wind gusts of different shapes and strengths were modeled and superimposed on the atmospheric boundary layer to locally impact a single blade when pointing upwards. The loads and deflections of the blade were monitored through its rotation.

Despite the extensive FSI studies conducted, it can be stated that according to the best knowledge of the authors, no study had an emphasis on the dynamic effects of an explicitly defined local wind gusts on flexible membranous structures. For this purpose, this work focuses on establishing a computational FSI framework to assess the effect of a wind gust convecting through a turbulent boundary layer and impacting on an air-inflated spherically shaped flexible membranous structure. The present test case is a further development step of two subsequent setups. In Wood et al. (2016) the flow around a rigid hemisphere placed within a turbulent boundary layer was studied experimentally and numerically. In Wood et al. (2018) the rigid hemisphere was replaced by a flexible membrane with nominally the same

shape and experimentally studied in the same wind tunnel as the rigid counterpart before. This FSI test case was also studied in detail based on coupled numerical simulations in De Nayer et al. (2018a) and Apostolatos et al. (2019a). The new contribution here is that the turbulent inflow is now superimposed by a wind gust in order to investigate the effect of such short-term but strong phenomena on the forces and deformations of the structure.

The paper is organized as follows: In Section 2, the source-term wind gust modeling technique employed for the injection of the wind gust inside the computational domain is presented. Section 3 describes the numerical framework applied to solve the FSI problem. Section 4 and Section 5 provide the details about the computational setup of the rigid and the flexible hemisphere followed by a description of the applied wind gusts parameters in Section 6. The results obtained for different wind gusts on the rigid and flexible case are discussed in Section 7 and summarized in Section 8.

2. Wind gust modeling

2.1. Overview

Modeled wind gust profiles can be classified into two main categories, namely, deterministic and stochastic wind gusts. The scope of this work is restricted to deterministic gusts relying on the superposition of an analytical function defining the mean shape of the gust and synthetically generated turbulence. In the literature, there exist different alternatives for the injection of wind gusts into the computational domains. One of the conventional approaches is referred to as the far-field boundary condition (FBC) method, where the wind gust is introduced at the inlet patch and can freely propagate through the domain (Norris et al., 2010). The gust is, however, susceptible to numerical dissipation owing to the often coarse grid resolution in the upstream region. As a remedy, a fine overset grid is often used to transport the gust at the expense of higher computational effort (Heinrich and Reimer, 2013). On the other hand, prescribed velocity methods (field velocity method, FVM (Singh and Baeder, 1997), and split velocity method, SVM (Wales et al., 2014)), define the imposed gust velocity inside the computational domain at each point in space and each instant in time by suitably modifying the governing equations. Such a method is found to be advantageous for many aeronautics applications (e.g., an airfoil penetrating a vertical gust). As shown in Boulbrachene et al. (2021), for horizontally oriented gusts the FVM and the SVM methodologies are not an appropriate choice. Recently, De Nayer and Breuer (2020) proposed a pure source-term (ST) formulation to inject horizontal wind gusts close to the region of interest. Upon injection, the gust is allowed to freely propagate through the domain and, consequently, the full coupling (i.e., including the feedback effect) between the gust and the surrounding field is captured. Therefore, this method is employed in this work.

2.2. Source-term formulation

For simulating fluid–structure interaction problems based on a finite-volume scheme, the Arbitrary Lagrangian–Eulerian (ALE) formulation of the Navier–Stokes equations is applied. Inserting the momentum source-term (ST) formulation for a gust (De Nayer and Breuer, 2020) the integral form of the governing equations describing the fluid flow are written based on the fluid velocity $\tilde{\mathbf{u}}$ as:

$$\begin{aligned} \frac{d}{dt} \int_{V^{\text{grid}}} \rho dV + \int_{S^{\text{grid}}} \rho (\tilde{\mathbf{u}} - \mathbf{u}^{\text{grid}}) \cdot \mathbf{n} dS &= 0, \\ \frac{d}{dt} \int_{V^{\text{grid}}} \rho \tilde{\mathbf{u}} dV + \int_{S^{\text{grid}}} \rho (\tilde{\mathbf{u}} - \mathbf{u}^{\text{grid}}) \tilde{\mathbf{u}} \cdot \mathbf{n} dS &= \mathbf{S}_{\text{turb}}^{\text{syn}} + \mathbf{S}_{\text{gust}}^{\text{ST}} - \int_{S^{\text{grid}}} \tilde{\boldsymbol{\tau}} \cdot \mathbf{n} dS - \int_{S^{\text{grid}}} p \mathbf{n} dS, \end{aligned}$$

where ρ is the density of the fluid, $\tilde{\boldsymbol{\tau}}$ is the stress tensor based on the flow velocity $\tilde{\mathbf{u}}$, and $\mathbf{n}$ is the surface normal vector of the considered control volume. Here, the volume and surface integrals are taken over the current time-dependent volume $V^{\text{grid}}(t)$ and the surface $S^{\text{grid}}(t)$ of a grid cell. $\mathbf{u}^{\text{grid}}$ denotes the grid velocity with which the surface of a control volume (CV) is moving. The momentum source term $\mathbf{S}_{\text{turb}}^{\text{syn}}$ describes the synthetically generated turbulent fluctuations to mimic the inflow turbulence as described in Sections 3 and 4. Correspondingly, the momentum source term $\mathbf{S}_{\text{gust}}^{\text{ST}}$ models the contribution of the gust based on the *total* velocity $\mathbf{u}$ and is formulated on the special local basis $\mathcal{B}_1 = (\mathbf{O}, \mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$ (see Fig. 1) as follows:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{\mathcal{B}_1} = \int_{V^{\text{grid}}} (\mathbf{u}|_{\mathcal{B}_1} \cdot \nabla) (\rho \mathbf{u}|_{\mathcal{B}_1}) dV.$$

The *total* velocity vector $\mathbf{u}$ reads:

$$\mathbf{u} = \tilde{\mathbf{u}} + \mathbf{u}_g,$$

where $\tilde{\mathbf{u}}$ is the flow velocity and the primary variable solved for during the simulation. $\mathbf{u}_g$ describes the imposed analytical gust velocity in the context of deterministic gust models defined as:

$$\mathbf{u}_g|_{\mathcal{B}_1}(t, \xi, \eta, \zeta) = A_g f_t(t) f_1(\xi) f_2(\eta) f_3(\zeta) \mathbf{g}_1.$$

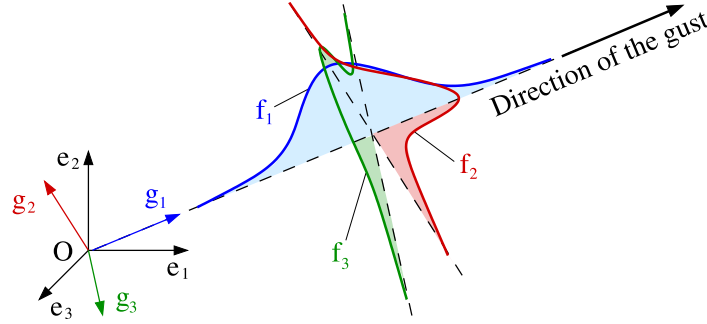


Fig. 1. Definition of the gust in the local basis $B_1 = (O, \mathbf{g}_1, \mathbf{g}_2, \mathbf{g}_3)$.

The constant A_g defines the strength of the gust and the function f_t the temporal evolution of the gust velocity during the injection phase. The functions f_1, f_2 and f_3 denote the spatial distribution of the gust velocity in $\mathbf{g}_1, \mathbf{g}_2$ and $\mathbf{g}_3$ direction, respectively.

The source term for describing the gust is simplified by taking into account that the only non-zero gust velocity component is that of the first vector of the local basis B_1 (i.e., $\mathbf{g}_1$), and assuming that the gradient of the flow velocity along this direction is negligible (i.e., $\frac{\partial \tilde{\mathbf{u}}|_{B_1}}{\partial \xi} \approx 0$) (see De Nayer and Breuer, 2020). Based on the gust shapes employed in this work, a spatial restriction of the source term in hand is necessary to achieve a favorable positive momentum (see Pasquetti and Peres, 2015; De Nayer and Breuer, 2020). Compiling the points mentioned and approximating the volume integral by means of the midpoint rule, the source term written in the Cartesian basis $B_0 = (O, \mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3)$ reads:

$$\mathbf{S}_{\text{gust}}^{\text{ST}}|_{B_0} \approx \begin{cases} \rho A_g f_t(t) f_2(t_\eta(x_{cc}, y_{cc}, z_{cc})) f_3(t_\zeta(x_{cc}, y_{cc}, z_{cc})) \frac{\partial f_1}{\partial \xi} \Big|_{cc} V^{\text{grid}} \mathbf{u}_g|_{B_0, cc} & \text{for } \xi < \xi_g \\ 0 & \text{else} \end{cases} \quad (1)$$

Thus, a distinction is made between the so-called expulsion phase ($\xi < \xi_g$) and the ingestion phase ($\xi > \xi_g$). Here the subscript cc is an abbreviation for *cell center*. The functions t_ξ, t_η and t_ζ find the local coordinates ξ, η and ζ based on given Cartesian coordinates, respectively. Finally, the vector $\mathbf{u}_g|_{B_0}$ is expressed as:

$$\mathbf{u}_g|_{B_0}(t, x, y, z) = A_g f_t(t) f_1(t_\xi(x, y, z)) f_2(t_\eta(x, y, z)) f_3(t_\zeta(x, y, z)) \mathbf{n}^g|_{B_0}, \quad (2)$$

where $\mathbf{n}^g|_{B_0}$ is the user-defined unit vector defining the direction of the gust in the Cartesian basis.

2.3. Gust shapes

In the following, the spatial functions (f_1, f_2 and f_3) as well as the temporal function f_t characterizing the deterministic gust imposed in the cases presented below are described. For the sake of generality, we let the variable ϕ be equal to the coordinate $\phi = \{x, y, z\}$ or the time t and the subscript i equal to 1, 2, 3 or t , respectively. The constant ϕ_g is the central value of the gust distribution for the corresponding ϕ and L_g^ϕ its length scale.

- **Adapted “1-cosine” shape** (Extreme Coherent Gust, ECG):

The original “1-cosine” shape found in the literature can be adapted by introducing the central value ϕ_g . That allows to achieve more control than for the original shape:

$$f_i(\phi) = \begin{cases} \frac{1}{2} \left(1 + \cos \left(\frac{2\pi (\phi - \phi_g)}{L_g^\phi} \right) \right) & \text{for } (\phi - \phi_g) \in \left[-\frac{L_g^\phi}{2}, \frac{L_g^\phi}{2} \right] \\ 0 & \text{else} \end{cases}$$

In the definition of $\mathbf{S}_{\text{gust}}^{\text{ST}}|_{B_0}$ given by Eq. (1), the first derivative $\partial f_1 / \partial \xi$ appears during the expulsion phase, i.e., for $(\phi - \phi_g) \in \left[-\frac{L_g^\phi}{2}, 0 \right]$, whereas the source term is forced to zero during the ingestion phase, i.e., for $(\phi - \phi_g) \in \left[0, \frac{L_g^\phi}{2} \right]$.

For further details refer to De Nayer and Breuer (2020). In order to guarantee a smooth continuous source term between the expulsion and the ingestion phase at $\phi = \phi_g$, the ECG shape is slightly adapted to deliver a C^2 -continuous function for the applied source-term formulation (see the zoomed view in Fig. 2(b)) as follows:

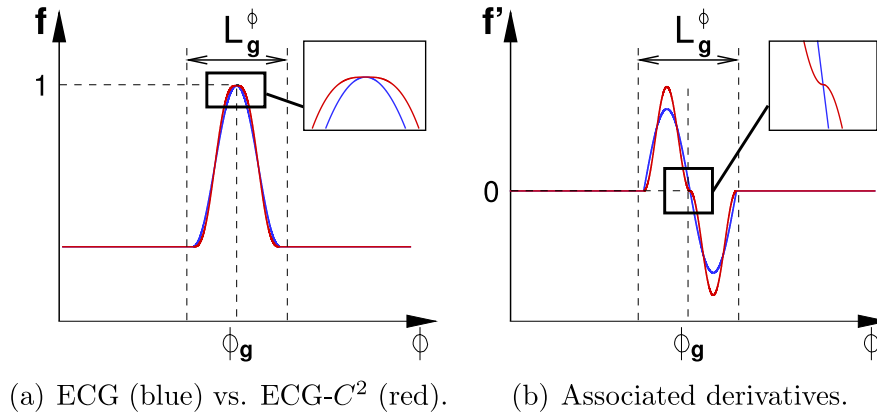


Fig. 2. Adapted “1-cosine” vs. modified C^2 -“1-cosine” shapes and their derivatives for a given scale L_g^ϕ and central value ϕ_g .

• **Modified C^2 -“1-cosine” shape** (Extreme Coherent Gust C^2 , ECG- C^2):

$$f_i(\phi) = \begin{cases} \frac{2}{L_g^\phi} \left(\phi - \left(\phi_g - \frac{L_g^\phi}{2} \right) + \frac{L_g^\phi}{4\pi} \sin \left(\frac{4\pi (\phi - \phi_g)}{L_g^\phi} + \pi \right) \right) & \text{for } \left(\phi - \left(\phi_g - \frac{L_g^\phi}{4} \right) \right) \in \left[-\frac{L_g^\phi}{4}, \frac{L_g^\phi}{4} \right] \\ -\frac{2}{L_g^\phi} \left(\phi - \left(\phi_g + \frac{L_g^\phi}{2} \right) + \frac{L_g^\phi}{4\pi} \sin \left(\frac{4\pi (\phi - \phi_g)}{L_g^\phi} - \pi \right) \right) & \text{for } \left(\phi - \left(\phi_g + \frac{L_g^\phi}{4} \right) \right) \in \left[-\frac{L_g^\phi}{4}, \frac{L_g^\phi}{4} \right] \\ 0 & \text{else .} \end{cases}$$

Fig. 2 illustrates both distributions for a given scale L_g^ϕ and a central value ϕ_g .

The duration of the gust L_g^t is linked to the spatial length scale of the gust in streamwise direction L_g^ξ by the expression:

$$L_g^\xi = L_g^t u_g^{\text{conv}},$$

where u_g^{conv} is an approximation of the convective velocity of the wind gust in the surrounding flow field. In the following, the default value is the free-stream velocity $u_g^{\text{conv}} = u_\infty$.

3. Simulation framework for CFD and FSI problems within turbulent flows

In order to solve pure CFD problems, the 3D fully parallelized finite-volume fluid solver FASTEST-3D is applied (Durst and Schäfer, 1996). Since modern realistic test cases imply turbulent flows, FASTEST-3D includes different techniques to predict this kind of flows. In the present study the large-eddy simulation (LES) technique is used, i.e., the large scales of the turbulent flow field are resolved directly, whereas the non-resolvable small scales have to be taken into account by a subgrid-scale (SGS) model. The classical Smagorinsky (1963) SGS model applying the standard parameter $C_s = 0.1$ combined with the Van-Driest damping function is used, since preliminary work in De Nayer et al. (2018b) shows that for the test case considered the model delivers reasonable results.

Since LES is applied, a semi-implicit predictor–corrector scheme (projection method) of second-order accuracy in space and time is preferred over the standard implicit SIMPLE scheme for the solution of the pressure–velocity coupling problem (Breuer et al., 2012). The filtered Navier–Stokes equations are discretized on a curvilinear, block-structured body-fitted grid with a collocated variable arrangement. Standard discretization methods such as mid-point rule and blended central scheme (5% upwind) are set as in Wood et al. (2016).

In order to mimic a realistic turbulent inflow, synthetic turbulent perturbations are injected into the computational domain upstream of the zone of interest (here the hemispherical structure). These synthetic data are generated using the so-called Synthetic Turbulent Inflow Generator (STIG) relying on the digital filter concept of Klein et al. (2003) and introduced into the domain based on the source-term method by Schmidt and Breuer (2017), Breuer (2018) and De Nayer et al. (2018b).

Facing an FSI problem, the simulation framework has to be extended. Based on a partitioned procedure detailed in Breuer et al. (2012), the fluid solver FASTEST-3D is coupled with the computational structure dynamics (CSD) solver Carat++ (Bletzinger et al., 2006). The coupling and the mapping between the two diverse surface discretizations at the FSI

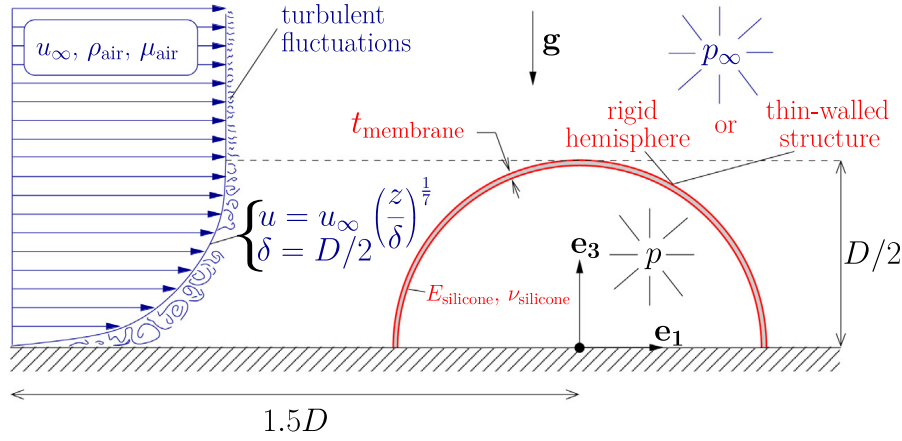


Fig. 3. Surface-mounted rigid (see Section 4) or air-inflated flexible hemisphere (see Section 5) within a turbulent boundary layer.

interface are carried out by the open-source software EMPIRE (Sicklinger et al., 2014). Validation cases can be found in De Nayer et al. (2014) and De Nayer and Breuer (2014).

Since the FSI simulation framework relies on body-fitted meshes, FASTEST-3D is written in the ALE formulation given in Section 2. To describe the fluid motion in temporally varying domains, the fluid mesh has to be adapted at each time step. Different techniques are available: In case of small deformations a very fast algebraic procedure is performed based on a combination of linear and transfinite interpolations (TFI) (Thompson et al., 1985). If the deformations become significant, a hybrid adaption method especially developed for FSI problems within LES (Sen et al., 2017) is applied. It relies on a combination of an inverse distance weighting interpolation for the block boundaries of the block-structured grid and a three-dimensional TFI for the inner mesh.

The deformation of the structure due to the fluid loads at the FSI interface is predicted by Carat++. It is a finite-element and an isogeometric solver for the structure developed with emphasis on the prediction of the mechanical behavior of thin-walled structures (i.e., shells and membranes). Furthermore, structural optimization and form finding procedures for pre-stressed tensile structures are available in the software (Breitenberger et al., 2015; Philipp et al., 2016). The momentum equation written in a Lagrangian frame of reference is applied to describe the dynamic equilibrium of the structure. A St. Venant–Kirchhoff material law is assumed and links the Piola–Kirchhoff stress tensor with the Green–Lagrange strain tensor (Basar and Weichert, 2013).

EMPIRE (Enhanced Multi Physics Interface Research Engine) is an open-source coupling tool dedicated to co-simulations with n clients ($n \geq 2$). This library allows the user to generate its own coupling algorithms by writing a user-defined xml input file. Standard coupling algorithms such as loose or strong staggered schemes or more complex approaches such as Jacobian-based schemes can be easily defined. EMPIRE does the mapping between the non-matching CFD and CSD grids and the exchange of data with its clients. Details about the presently applied mortar mapping in case of FEM or IGA discretizations can be found in Wang et al. (2016) or Apostolatos et al. (2019b).

4. Rigid hemisphere within a turbulent boundary layer

4.1. Description of the CFD case

The investigated hemispherical rigid body (diameter D) is mounted on an uniform smooth wall as depicted in Fig. 3. The origin of the frame of reference is taken at the center of the base area of the hemisphere, where $x = \mathbf{e}_1$ denotes the streamwise, $y = \mathbf{e}_2$ the spanwise and $z = \mathbf{e}_3$ the vertical (wall-normal) direction. The structure is put into a thick turbulent boundary layer, where the distribution of the time-averaged velocity in main flow direction closely follows the 1/7 power law. At a distance of 1.5 diameters upstream of the bluff body the thickness of the boundary layer δ corresponds to the height of the hemisphere ($\delta = D/2$) (see Wood et al., 2016).

Contrary to the case presented in Wood et al. (2016) the Reynolds number of the air flow ($\rho = \rho_{\text{air}} = 1.225 \text{ kg/m}^3$, $\mu_{\text{air}} = 18.27 \times 10^{-6} \text{ kg/(m s)}$ at $\vartheta = 20^\circ \text{ C}$) is set to $\text{Re} = \rho_{\text{air}} D u_\infty / \mu_{\text{air}} = 100,000$ and corresponds to one of the cases of Wood et al. (2018). u_∞ is the undisturbed free-stream mean velocity in x -direction at standard atmospheric conditions. The Mach number is low ($\text{Ma} \leq 0.03$). The air flow can be assumed to be incompressible and isotherm. The results will be given in dimensionless form based on D , u_∞ and ρ_{air} .

4.2. Computational setup

To simulate the flow, the chosen computational domain is a large hemispherical expansion (radius $10D$) with its origin at the center of the hemisphere. A medium block-structured mesh is used composed of 4.3×10^6 control volumes.

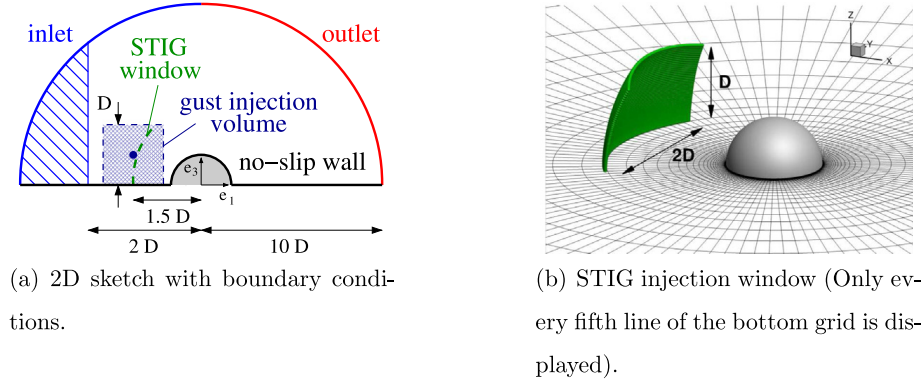


Fig. 4. Boundary conditions, STIG injection window and gust injection window used for the simulation.

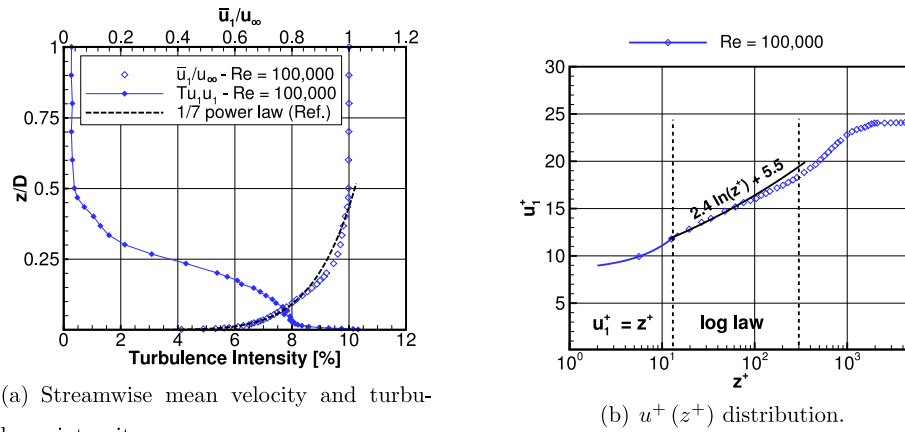


Fig. 5. Velocity and turbulence intensity in streamwise direction for the turbulent boundary layer at $Re = 100,000$. Measured by Wood et al. (2018) at $-1.5D$ in front of the hemisphere.

Comparisons with a fine mesh were carried out in De Nayer et al. (2018b). The viscous sublayer is fully resolved and the first cell center is approximately located at a distance of $\Delta z/D \approx 5 \times 10^{-5}$ from the wall. The aspect ratio of the cells on the hemispherical body is in the range between 1 and 10. The geometric stretching ratios are kept below 1.1. Based on these parameters the dimensionless cell sizes in the wall-normal and tangential directions still fit to the recommendations by Piomelli and Chasnov (1996) for wall-resolved LES. The time step is chosen equal to $\Delta t^* = \Delta t u_\infty / D = 3.317 \times 10^{-5}$.

Since the predictions are wall-resolved, the bottom of the domain and the surface of the hemisphere are no-slip walls. The outer surface of the hemispherical expansion is divided into inlet and outlet patches (see Fig. 4). At the outlet a combination of a convective boundary condition and a zero velocity-gradient boundary is applied. The latter is used in those regions which are not strongly influenced by vortical flow structures. The convective velocity is set according to the 1/7 power law.

In order to prescribe a realistic turbulent inflow ignoring the wind gust at the moment, synthetic perturbations are injected into the computational domain upstream of the rigid body. For the generation of the synthetic inflow data the time-averaged velocities and Reynolds stresses measured in the experiment (see Fig. 5 and Wood et al., 2018) are taken. Since it was not possible to determine all Reynolds stress components in the direct vicinity of the wall, these measurements have been complemented by DNS data of Schlatter et al. (2009).

The other important parameters to be defined are the integral time and length scales. The digital filter concept of Klein et al. (2003) solely allows to define a constant value within the entire boundary layer. Thus, a reasonable compromise is required. In the present case, only the turbulent structures injected at $z/D < 0.5$ ($z^+ < 1140$) will have an impact on the bluff body and therefore will mainly drive the flow around the hemisphere. Moreover, looking at the distribution of the turbulence intensity, the most intensive structures are at $z/D \lesssim 0.3$ ($z^+ \lesssim 680$). The dominant structures present in a turbulent boundary layer are the streaks and the hairpin vortices. Since the streaks are confined near the wall in the buffer layer, it was decided to generate synthetic turbulent structures of the size of typical hairpin vortices. These structures have a characteristic streamwise dimension of $\Delta x^+ = 200$ (Bailly and Comte-Bellot, 2015) or 300 (Lesieur, 2008) near the wall and of $\Delta x^+ = 140$ in the logarithmic region, a spanwise extension of about $\Delta y^+ = 100$ and a height between $\Delta z^+ = 200$ and 400 (Adrian, 2007).

The STIG injection window is located approximately $-1.5D$ upstream of the body (see Fig. 4). It has an height of approximately $1D$ and a width of $2D$. Based on the grid resolution of the STIG injection window ($-1 \leq y/D \leq 1$ with

$\Delta y^{\text{STIG}}/D = 1.021 \times 10^{-2}$ and $0 \leq z/D \leq 1$ with $\Delta z^{\text{STIG}}/D = 2.191 \times 10^{-3}$) and the fact that the support of the filters as defined in Klein et al. (2003) should be at least 2, i.e., $n_y = L_y^{\text{STIG}}/\Delta y^{\text{STIG}} = 2$, it leads to a minimum length scale $L_y^{\text{STIG}}/D = 2.04 \times 10^{-2}$. Since the integral length scales are assumed to be equal in all directions, the integral length scale in wall-normal direction is $L_z^{\text{STIG}}/D = L_y^{\text{STIG}}/D = L^{\text{STIG}}/D = 2.04 \times 10^{-2}$. Using the classical length scale specification for the log-law region $L = \kappa z$ applied in the mixing length concept by Prandtl, the height where this integral length scale is reached is equal to $z^+ = 165$ with $u_\tau = 0.374$ m/s at $\text{Re} = 100,000$. The dimensionless integral time scale is deduced with the help of the Taylor hypothesis and set to $T^{\text{STIG}} u_\infty/D = L^{\text{STIG}} u_\infty/(D u_1(z^+ = 165)) = 2.85 \times 10^{-2}$. With this value the STIG produces synthetic turbulent structures with a dimensionless size of about $\Delta x^+ = \Delta y^+ = \Delta z^+ = 4 L^{\text{STIG}} \rho_{\text{air}} u_\tau / \mu_{\text{air}} \approx 300$ in each direction¹. This is a good compromise between the size of typical hairpin structures in streamwise, spanwise and wall-normal direction.

For the STIG-R injection method (see De Nayer et al., 2018b), the 3D influence area is defined by a Gaussian distribution. In the present setup, the integral length scale in streamwise direction L^{STIG} is rather small and shorter than the cell size of the injection plane located approximately $-1.5D$ upstream of the body. Therefore, the source term has to be scaled according to the second scenario given in De Nayer et al. (2018b). In the present case the influence area is $L_{\text{inf}}/D = 2 L^{\text{STIG}}/D = 4.08 \times 10^{-2}$ and thus smaller than $\Delta d_0/D = 8. \times 10^{-2}$, where Δd_0 is the width of the grid cell corresponding to the injection patch. It leads to $S_\phi^{\text{CFD}}(x_i) = \sqrt{2} \text{erf}(\sqrt{\pi}/2) \frac{L^{\text{STIG}}}{\Delta d_0} S_\phi^{\text{syn}}$.

A total of 480,000 time steps of synthetic turbulent inflow data corresponding to about 15 dimensionless time units are generated.

5. Air-inflated flexible hemisphere within a turbulent boundary layer

5.1. Description of the FSI case

The rigid hemisphere used in the above case is now replaced by a thin-walled membranous structure pressurized on the inside by air in a form of a hemisphere as shown in Fig. 3. As before it is mounted on a smooth flat plate and plunged into the same turbulent boundary layer as for the rigid case (see Section 4.1 and Wood et al. (2018) for details). Additionally, the setup is now exposed to the gravitational acceleration $\mathbf{g} = -9.81 \mathbf{e}_3$.

In the experimental setup of Wood et al. (2018) the thin membrane is made of silicone (Wacker Elastosil 625). To model this material using the St. Venant–Kirchhoff assumption, its density, Young's modulus and Poisson's ratio were experimentally determined in Wood et al. (2018) to $\rho_{\text{silicone}} = 1050 \text{ kg/m}^3$, $E_{\text{silicone}} = 7 \times 10^5 \text{ Pa}$ and $\nu_{\text{silicone}} \approx 0.45$, respectively. More details about these material assumptions can be found in De Nayer et al. (2018a). The thickness of the membrane t_{membrane} is not exactly constant along the structure ($t_{\text{membrane}} \in [1.2 \times 10^{-4} \text{ m}, 2 \times 10^{-4} \text{ m}]$). Preliminary predictions made on pure structural test cases show that an average thickness of $t_{\text{membrane}} = 1.65 \times 10^{-4} \text{ m}$ delivers good agreement with the experiments (De Nayer et al., 2018a).

One important parameter has also to be set: The pressure difference between the inner gauge pressure of the structure and the pressure outside $\Delta p_{\text{FSI}} = p - p_\infty$. In order to set a sufficiently high inner pressure to stabilize the structure, but also not a too strong gauge pressure to match the desired shape of the hemisphere, a value of $\Delta p_{\text{FSI}} = 43 \text{ Pa}$ is chosen through the calibration process described in Wood et al. (2018).

5.2. Computational FSI setup

Based on the FSI simulation framework described in Section 3, the computational setup for the current test case is as follows. The setup for the fluid part is exactly the same as for the rigid case detailed in Section 4.2. The CSD computational setup relies on the preliminary works of De Nayer et al. (2018a) and Apostolatos et al. (2019b). Two different kinds of simulations will be presented and compared in the result part:

- The first is based on the FEM discretization of the thin-walled structure with a grid containing 1926 *Constant Strain Triangle* (CST) elements and 999 nodes. All nodes have three degrees of freedom except the ones connected with the smooth plate at $z/D = 0$. These nodes cannot move in any translational direction. Mesh studies can be found in De Nayer et al. (2018a).
- The second relies on the IGA discretization of the membrane by a single patch containing 625 bi-quadratic C^1 -continuous elements. The hemispherical shape is attained by a form-finding analysis. Details of preceding investigations and visualizations of the mesh can be found in Apostolatos et al. (2019b).

The static pressure difference Δp_{FSI} is considered constant during the simulation and is modeled in the CSD solver by a follower pressure load acting on each surface element. With this simplification the air flow in the inner domain has not to be solved by the fluid solver. Additionally, the gravitational acceleration is taken into account in the CSD solver as a dead load acting on the volume of each element. Moreover, an initial homogeneous and isotropic pre-stress

¹ For more details, we refer to De Nayer et al. (2018a).

Table 1
Gusts characteristics for ST formulation.

gust	$\mathbf{g}_1$	A_g	$f_t(t) = \text{ECG}-C^2$	$f_1(\xi) = \text{ECG}-C^2$	$f_2(\eta) = \text{ECG}$	$f_3(\zeta) = \text{ECG}$
			L_g^t	L_g^ξ	L_g^η	L_g^ζ
<i>weak</i>	$\mathbf{e}_1$	$0.5 u_\infty$	$L_g^\xi / u_g^{\text{conv}}$	$1 D$	$1 D$	$1 D$
<i>medium</i>	$\mathbf{e}_1$	u_∞	$L_g^\xi / u_g^{\text{conv}}$	$1 D$	$1 D$	$1 D$
<i>strong</i>	$\mathbf{e}_1$	$1.5 u_\infty$	$L_g^\xi / u_g^{\text{conv}}$	$1 D$	$1 D$	$1 D$

tensor field $\mathbf{n}_0$ is applied as usual with membrane elements. A previous investigation (De Nayer et al., 2018a) based on the experimental steady calibration case of Wood et al. (2018) delivered a value of $n_{\text{membrane}} = 7794.5$ Pa. The standard second-order non-linear Newmark scheme is applied with the same dimensionless time step as the fluid solver, i.e., $\Delta t^* = \Delta t U_\infty / D = 3.317 \times 10^{-5}$. In addition, the structural damping of the hemisphere has to be taken into account, since the material is silicone. A Rayleigh damping with the mass- and stiffness-proportional parameters set to $\alpha_r = 17.47$ and $\beta_r = 1.89 \times 10^{-4}$, is applied (De Nayer et al., 2018a).

Concerning the FSI coupling between fluid and structure parts, a loose coupling scheme is preferred to minimize the data exchange and the mesh adaption, since the ratio of the structural density to the fluid density is large ($\rho_{\text{silicone}} / \rho_{\text{air}} \approx 857$).

6. Wind gust settings

For a practical application of wind gusts hitting rigid and flexible structures as investigated in the present study, the gusts are directed parallel to the streamwise direction. An appropriate gust injection region is selected in the upstream area of the structure in such a way that it is not too close in order to give the gust a room to develop. At the same time it should be not too far in order to avoid numerical dissipation and long simulation times. Therefore, the center of the gust at the beginning of the ST injection is located at the point $(-1.5 D, 0, D/2)$. Based on the definition of the gust introduced in Section 2, the vector $\mathbf{n}^g|_{\mathcal{B}_0}$ defining the gust direction in the Cartesian basis $\mathcal{B}_0$ is equal to the basis vector $\mathbf{e}_1$, which represents the streamwise direction. Consequently, the vector $\mathbf{g}_1$ of the local gust basis is identical to $\mathbf{e}_1$. The gust velocity component given by Eq. (2) is defined as follows:

The temporal shape function $f_t(t)$ follows the same analytical function as the spatial shape $f_1(\xi)$ oriented in streamwise direction in the present case. The special ECG- C^2 shape is selected for both. In the spanwise and vertical directions (i.e., $\mathbf{g}_2 = \mathbf{e}_2$ and $\mathbf{g}_3 = \mathbf{e}_3$, respectively), a standard ECG shape is selected, i.e., $f_2(\eta) = f_3(\zeta) = \text{ECG}$.

The parameters to be defined next are the temporal and spatial scales in each direction. Since the temporal scale is deduced from the spatial length scale in the streamwise direction with $L_g^t = L_g^\xi / u_g^{\text{conv}}$, only three spatial scales are required (i.e., L_g^ξ , L_g^η and L_g^ζ). In this study, these length scales are chosen in such a way that the body of the wind gust can completely cover the whole structure. Since the body has a length and a width of $1 D$, the gust length scales along the streamwise and the spanwise directions (i.e., L_g^ξ and L_g^η) are set to $1 D$. It is also worth mentioning that a streamwise gust length scale of $1 D$ is sufficient since it is much longer than the largest vortical structures injected by the synthetic turbulent generator (see Section 4.2). Similarly, given that the height of the rigid hemisphere is $D/2$, the half of the gust length scale along the vertical direction is set to a value of $D/2$ (or $L_g^\zeta = 1 D$). Here, three gust strengths are investigated, a *weak* gust with an amplitude A_g of $u_\infty/2$, a *medium* gust with an amplitude of u_∞ and a *strong* gust with $A_g = 1.5 u_\infty$. The gust characteristics of the simulations performed for the rigid and flexible cases are summarized in Table 1.

7. Results and discussion

7.1. Propagation of the gusts upstream of the rigid or flexible hemisphere

Fig. 6 compares the temporal evolution of the dimensionless streamwise velocity u_1/u_∞ and the dimensionless pressure $p/(\rho_{\text{air}} u_\infty^2)$ for three different wind gusts injected by the ST method. The case without gust is plotted as a reference and the fluctuations observed in this case are due to the incoming turbulent flow. The four selected monitoring points are located upstream of the rigid or flexible body and their positions are at the height of the gust center ($z/D = 0.5$) in the symmetry plane ($y/D = 0$) (see Fig. 7(d)). The temporal evolutions in the approaching phase of the gusts are displayed in Fig. 6 as a function of the dimensionless time $t^* = t u_\infty / D$. $t^* = 0$ corresponds to the beginning of the gust injection, whereas $t^* = 1$ defines the end of the injection. The gusts differ in terms of their amplitudes A_g . As expected, the stronger the amplitude of the gust is, the stronger is the maximum of the total velocity obtained, where the latter can be roughly estimated by $u_1^{\text{max}}/u_\infty = 1 + A_g/u_\infty$. Furthermore, the asymmetry in the obtained temporal gust histories increases with rising gust amplitudes A_g . The variation of the pressure at the location of the gust is as follows. In the phase, when the fluid velocity increases for example at point P01 or P02, the pressure is higher than without the gust. However, when the center of the gust has already passed a certain location and the fluid velocity decreases again, the pressure level is smaller than without the gust and then returns to atmospheric pressure. Obviously, the stronger the gust

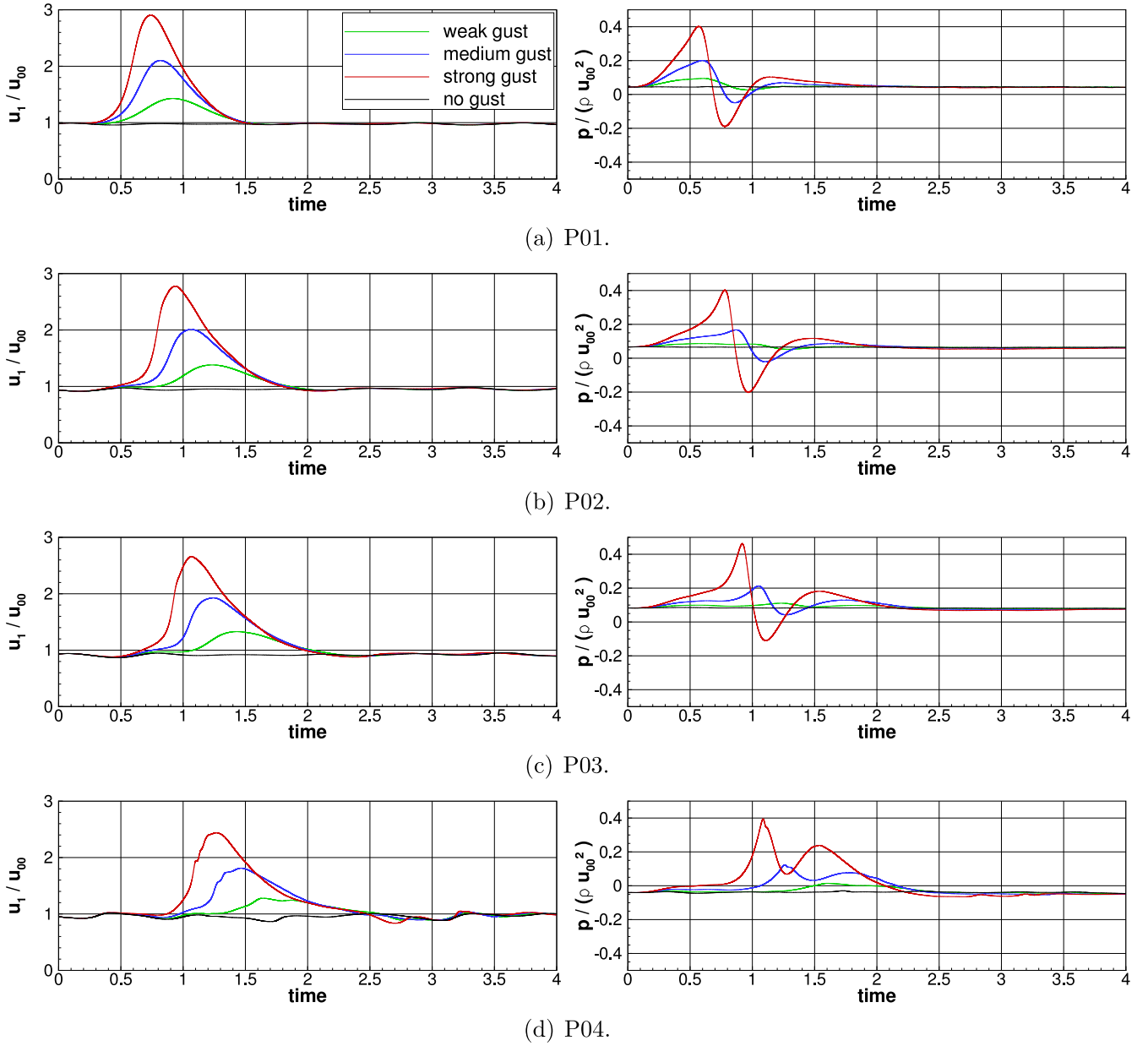


Fig. 6. Comparison of wind gusts with varying amplitudes upstream of the rigid body at four monitoring points ($L_g^x = 1D$, $L_g^y = 1D$, $L_g^z = 1D$, $u_g^{\text{conv}} = u_\infty$). *Left:* Streamwise velocity; *Right:* Pressure.

is, the stronger is the rise and fall of the pressure, where a quadratic relationship between the pressure changes and the gust amplitude has to be expected.

Figs. 7(a) to 7(c) depict the *medium* gust approaching the rigid hemisphere under three different views at $t^* \approx 0.95$. This instant in time is just before the impact of the gust on the body. Since the gust is defined as a short but strong variation of the velocity, the streamwise velocity u_1/u_∞ is chosen for the visualization. Additionally, the gust structure itself is highlighted by the red iso-surface with $u_1/u_\infty = 1.4$. For a direct comparison with the predicted flow field without a wind gust using the same turbulent inflow data, Figs. 7(d) to 7(f) depict the corresponding data for exactly the same views and the same instant in time. This contrasting juxtaposition shows very clearly that the flow field around the hemisphere and in its wake is not at all influenced by the wind gust at this point in time. Solely in front of the bluff body the effect of the gust can apparently be seen in form of an increased level of the wind speed in the main flow direction. Similar to the streamwise velocity the pressure distribution around the hemisphere and in its wake is not affected by the gust (see Fig. 8). Upstream of the body the increase of the velocity due to the presence of the gust leads to a local decrease of the pressure at the position of the gust.

At approximately $t^* \approx 1$ the gust hits the body. The effects of this impact on the forces, the gust structure and later on the deformations are discussed in the subsequent sections.

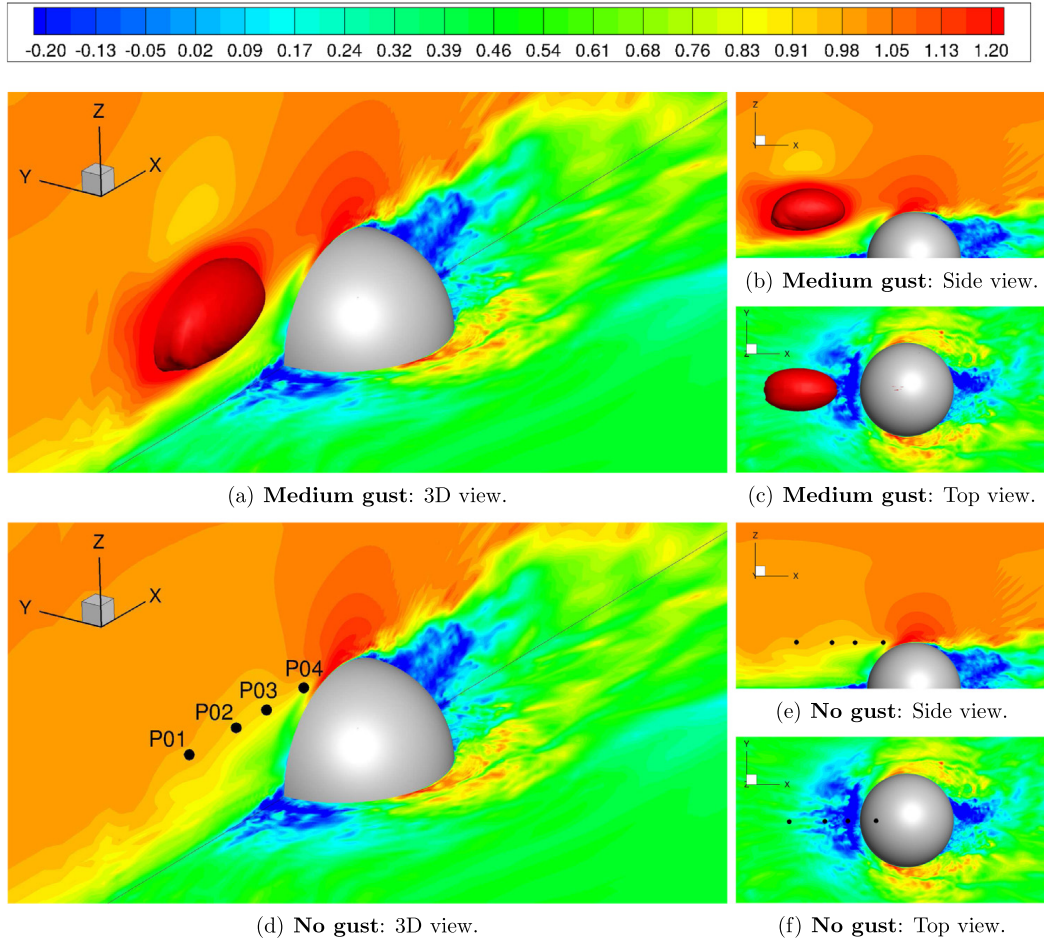


Fig. 7. Instantaneous snapshot of the *medium* gust approaching the rigid hemisphere at $t^* \approx 0.95$ (top: (a)–(c)) compared with the same snapshot without gust (bottom: (d)–(f)). Streamwise velocity u_1/u_∞ in a slice near the bottom wall and in the symmetry plane $y = 0$. The gust structure is highlighted by an iso-surface of the streamwise velocity $u_1/u_\infty = 1.4$.

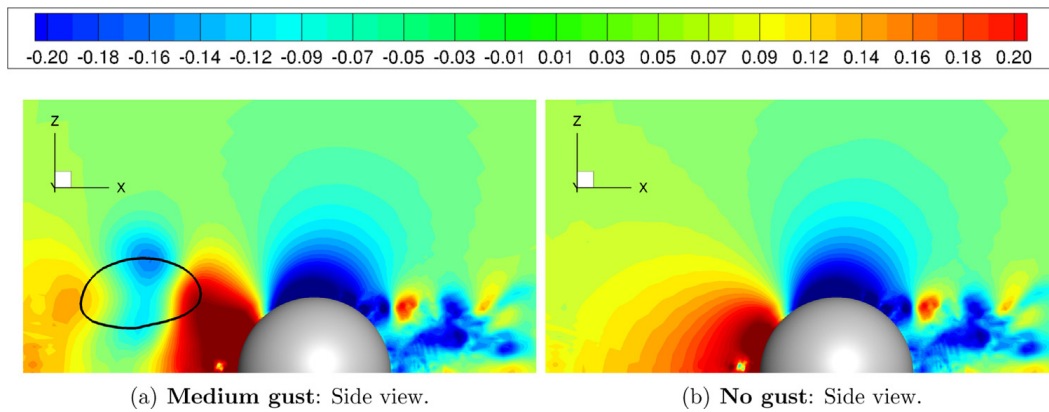


Fig. 8. Instantaneous snapshot of the *medium* gust approaching the rigid hemisphere at $t^* \approx 0.95$ (a) compared with the same snapshot without gust (b). Pressure $p/(\rho_{\text{air}} u_\infty^2)$ in the symmetry plane $y = 0$. The gust position is highlighted by a black iso-line of the streamwise velocity $u_1/u_\infty = 1.4$.

7.2. Impact of the gusts on the rigid hemisphere and resulting fluid forces

The entire development of the flow field during the approach and impact of the gust is shown for the case of the *medium* gust strength in Figs. 9 and 10. Again the streamwise velocity in a slice near the bottom wall and in the symmetry plane is displayed in Fig. 9 but now for six instants in time, which describe the entire process of the impact and disappearance of the gust. Furthermore, the gust structure is highlighted again by an iso-surface with $u_1/u_\infty = 1.4$. The pressure distribution is additionally presented in Fig. 10, where the iso-surface tracking of the gust structure is replaced by an

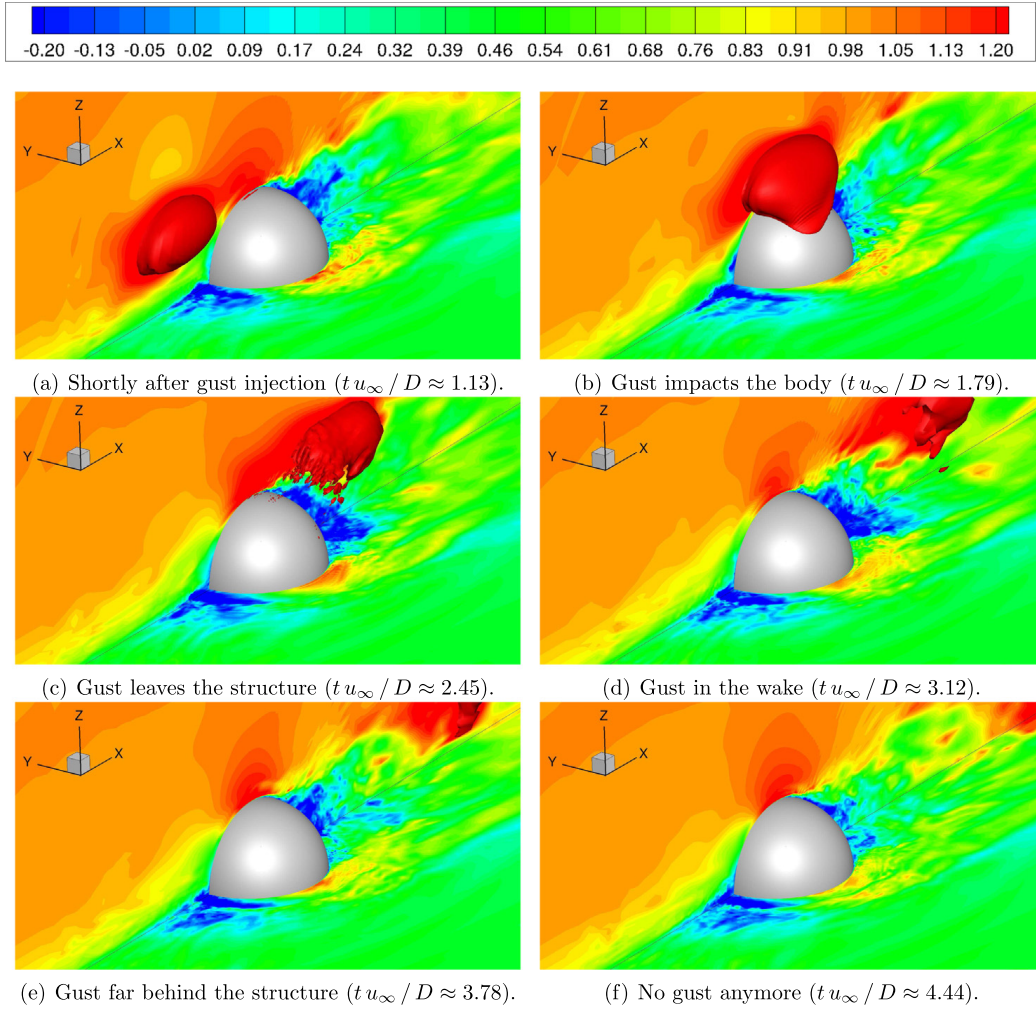


Fig. 9. Rigid case: Medium wind gust moving downstream and impacting the rigid hemisphere (streamwise velocity u_1/u_∞ in a slice near the bottom wall and in the symmetry plane $y = 0$). The gust structure is highlighted by an iso-surface of the streamwise velocity $u_1/u_\infty = 1.4$.

iso-line in the symmetry plane to see the pressure changes hidden behind the gust structure. In order to quantify the effect of the gust on the structure, the resulting forces in streamwise and vertical direction are shown in Fig. 11 as a function of the dimensionless time. Note that the forces are also made dimensionless leading to the force coefficients c_x and c_z defined as the forces divided by the stagnation pressure $\rho_{\text{air}} u_\infty^2 / 2$ and the reference area $\pi D^2 / 8$. In addition, the pressure force related parts of the total force coefficients, c_x^p and c_z^p , are plotted. It can be clearly seen in Fig. 11 that the major contribution of the force coefficient for this geometry is the pressure. Especially for the vertical component the shear stresses only marginally contribute to the total force.

After its injection the *medium* wind gust impacts the front of the rigid hemisphere at about $t^* \approx 1.13$ (see Fig. 9(a)) leading to an increase of the streamwise force coefficient c_x (blue curve in Fig. 11(a)). At this first instant in time the fluid load on the hemisphere is only a bit higher than for the case of the turbulent flow without gust. After reaching this first maximum at $t^* \approx 1.2$, c_x decreases, since the high pressure area in front of the body alleviates as the gust approaches the tip of the rigid body. In the next time instant displayed in Fig. 9(b) the gust has moved downstream and hits the tip at about $t^* \approx 1.8$. At this time the vertical force coefficient c_z (blue curve in Fig. 11(b)) reaches a maximum, since the pressure minimum is above the hemisphere (see Fig. 10(b)). Simultaneously the streamwise force coefficient c_x tends towards its global maximum at $t^* \approx 2$, since at that time the pressure minimum is on the back side of the structure (see the low pressure zone moving from the tip to the back in Figs. 10(b) and 10(c)). At the third instant in time (see Fig. 9(c)) the wind gust detaches from the body. The vertical force coefficient c_z reaches its minimum at that time and also c_x decreases since the effect of the pressure minimum gets weaker with increasing distance between the gust and the hemisphere (see the differences in the low pressure area behind the hemisphere in Figs. 10(c) and 10(d)). At $t^* \approx 3.12$ (Fig. 9(d)) the gust is still visible in the wake of the body. The two force components on the structure show a different behavior. The streamwise force coefficient c_x drops to a level, which is below the value of the case without the gust, whereas c_z is still slightly higher than in the case without the gust. The last two instants in time displayed in Figs. 9(e) and 9(f) depict how the gust leaves the region of interest. The zone of low pressure induced by the gust in the vicinity

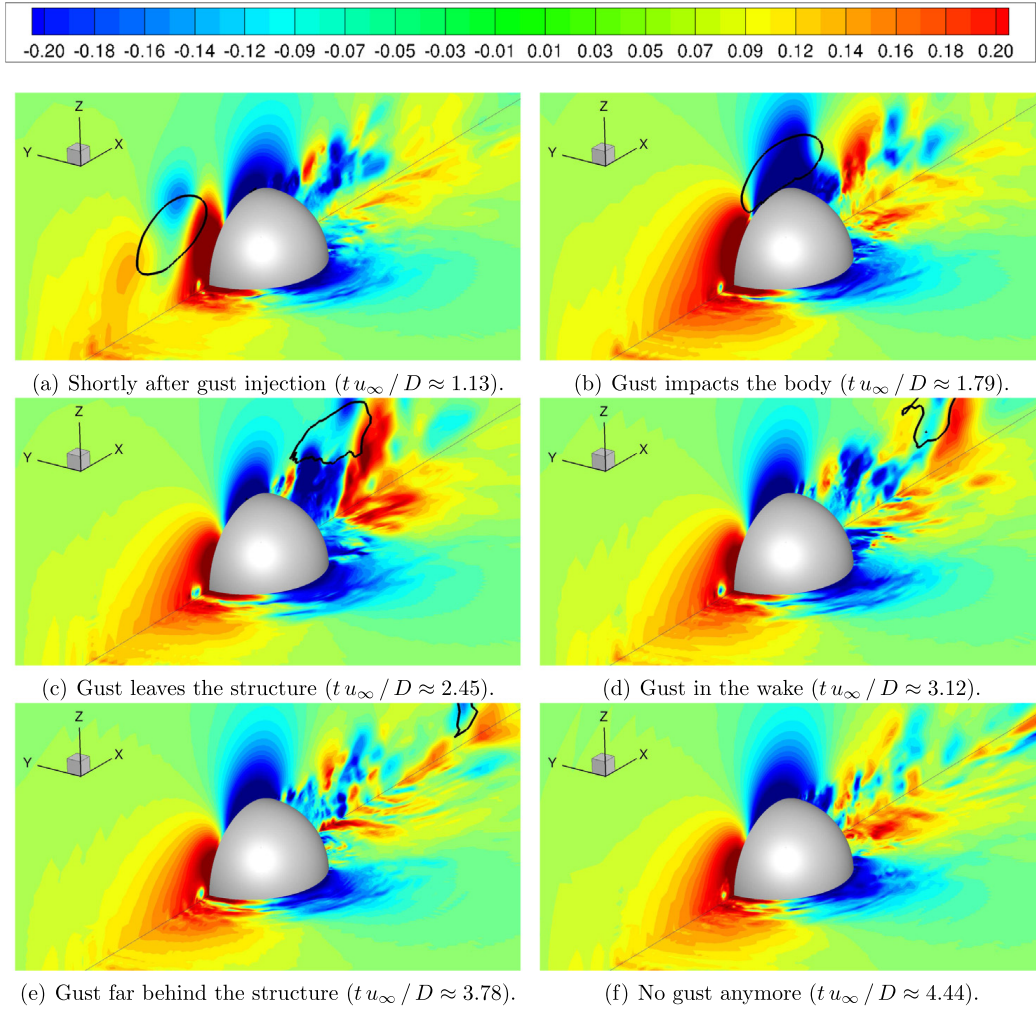


Fig. 10. Rigid case: Medium wind gust moving downstream and impacting the rigid hemisphere (pressure $p/(\rho_{\text{air}} u_\infty^2)$) in a slice near the bottom wall and in the symmetry plane $y = 0$). The gust position is highlighted by a black iso-line of the streamwise velocity $u_1/u_\infty = 1.4$ in the symmetry plane.

of the body has vanished (see Figs. 10(e) and 10(f)). Consequently, both force coefficients reach a similar level as for the case without the gust again. Since the wind gust has significantly affected the turbulent flow field around the bluff body, one cannot expect the time histories to converge when the gust has passed the hemisphere.

Fig. 12 gathers the streamwise and vertical force coefficients obtained for different gust strengths. The effects of the *weak* gust on c_x and c_z are minor. Both local maxima of c_x are present, but with low amplitudes and close to each other. Concerning c_z the local maximum is reached at about $t^* \approx 2$ and thus about 0.23 dimensionless time units later than for the *medium* gust discussed before. This observation can be explained by the lower traveling velocity of the *weak* gust compared with the *medium* gust. Similarly, the local maximum of c_z for the *strong* gust is about 0.24 time units earlier than for the *medium* gust.

By increasing the strength of the gust the amplitudes of both local c_x maxima increase. Additionally, the duration between both maxima decreases with intensified gust strength, since the traveling velocity of the gust is higher. As expected, the maximum of c_z augments as well, since the low pressure area at the tip of the hemisphere becomes stronger in case of a more powerful gust. As mentioned before for the *medium* gust, the pressure force is the major contribution to the total force in both streamwise and vertical direction for all gust strengths. Therefore, the peak values of the force coefficients quadratically change with the oncoming velocity, which is given by the superposition of the approaching boundary layer velocity and the velocity of the wind gust.

7.3. Impact of the gusts on the flexible hemisphere and resulting fluid forces and deformations

7.3.1. Detailed analysis for the medium gust

In the following, the setup changes to investigate the FSI case of an air-inflated flexible hemisphere under different wind gust loads. As in the previous section, the global force coefficients c_x and c_z are considered first. Fig. 13 compares the

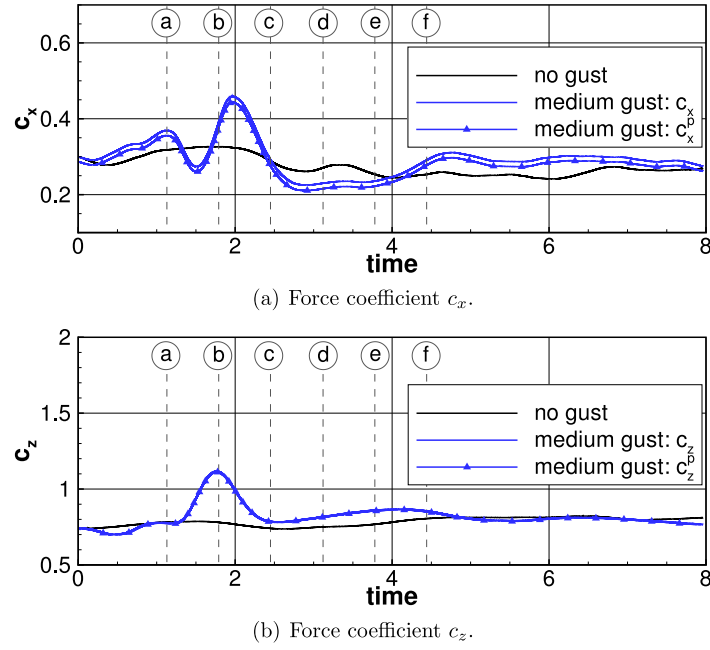


Fig. 11. Rigid case (medium gust): Streamwise and vertical force coefficients (solid lines) with the corresponding pressure force related parts (solid lines with symbols).

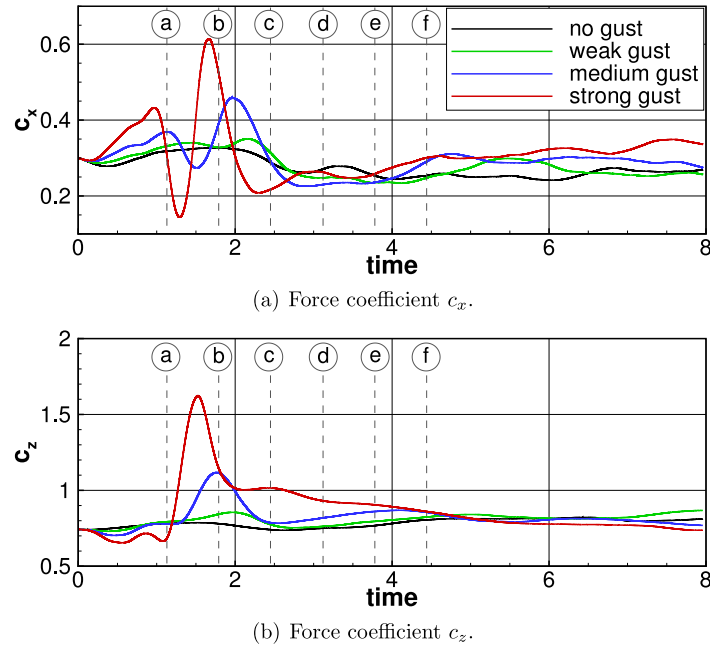


Fig. 12. Rigid case: Streamwise and vertical force coefficients predicted for different gust strengths.

streamwise and vertical force coefficients obtained for the flexible case and the *medium* gust with those predicted for the rigid case. The temporal evolutions of both cases are not identical, but also not so far from each other. Two findings can be derived from this observation. First, the deflection of the structure has a minor effect on these global surface-integrated quantities especially on the resulting peak values. Second, for highly unsteady FSI problems such a representation is insufficient. In that case instantaneous contour plots such as the streamwise velocity and the pressure are mandatory to apprehend the whole FSI phenomenon. Figs. 14 and 15 gather the distributions of u_1/u_∞ and $p/(\rho_{\text{air}}u_\infty^2)$ obtained by the FSI simulation relying on a FEM membrane discretization at the same six time instants as in Section 7.2. Again, a horizontal slice near the wall and a vertical slice located in the symmetry plane are chosen to depict the flow development. The position of the gust structure is tracked either by the iso-surface $u_1/u_\infty = 1.4$ or by its corresponding iso-line in the plane $y = 0$. The FSI-interface is gray with some lighting effects for a better representation of the membrane deflection.

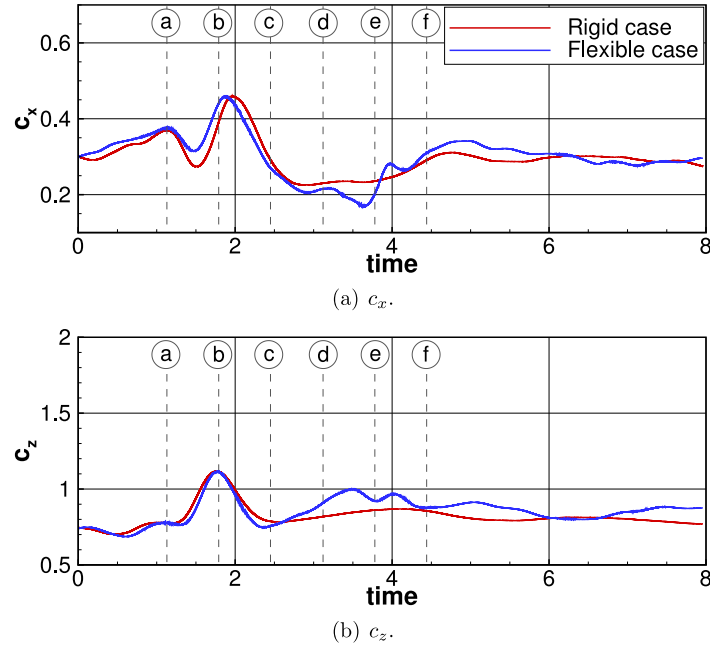


Fig. 13. Flexible FEM vs. rigid case (medium gust): Streamwise and vertical force coefficients.

To complete the visualization of the FSI phenomenon obtained for the *medium* gust, deflections of the membrane in the symmetry plane $y = 0$ are plotted in Fig. 16(b) for the same six time instants.

At the first time instant (Figs. 14(a) and 15(a)) the injection of the gust has just finished and it arrives at the flexible body. At this instant in time the membrane is already slightly deformed due to the loads of the turbulent flow without gust (see black line (a) in Fig. 16(b) and refer to De Nayer et al. (2018a) for more details). The gust at its current position has no significant effect on the deflection. Then, the gust moves downstream. At about $t^* \approx 1.8$ it hits the tip of the deformed membrane. As for the rigid case this time instant coincides with the maximum of the vertical force coefficient c_z (see Fig. 13(b)) due to the strong low pressure area above the hemisphere. However, this time the structure starts to deform (see red line (b) in Fig. 16(b)). Due to the impact of the gust, the front of the membrane grows hollow as visible in Figs. 14(b) and 16(b). In the indentation, a high pressure region is found (see Fig. 15(b)) and this zone is clearly larger than in the rigid case (see Fig. 9(b)). The gust structure continues to pass downstream and leaves the body. During this phase the forming indentation at the front still gets larger and deeper as depicted in Fig. 14(c) for $t^* \approx 2.45$ and with the blue line (c) in Fig. 16(b). Consequently, a certain delay between the maxima of the total forces at $t^* \approx 1.8$ and the maxima of the local deflections of the front is observed. In the next time instants the gust is convecting downstream in the wake and vanishes. The maximum of the deflection is reached in Fig. 14(c), when the gust has already passed the hemisphere completely. After this time instant the front indentation starts to decrease again (see green line (d) in Fig. 16(b)). Figs. 14(d) and 14(e) show how the passing of the gust into the wake strongly alters the recirculation area behind the hemisphere. Consequently, a pronounced and a relatively long phase, in which the streamwise force coefficient is lower than for the rigid case, is observed in Fig. 13(a). In this time interval an increase of c_z is found. An oscillation of the membrane front indentation occurs before it reaches its original deflected state again (see the evolution of the orange line (e) to the gray line (f) and then to the black line (a) (original state) in Fig. 16(b)). This phenomenon is responsible for the oscillation of the global force coefficients c_x and c_z observable in Figs. 13(a) and 13(b) at about $t^* \approx 3.8$. In the last time instant presented in Fig. 14(f) the gust has already vanished in the wake and the turbulent flow around the bluff body is back to the original state without gust. Consequently, the deflections of the inflated hemisphere return to their original level.

The injection of the wind gust occurs at an arbitrary time decided by the user. The wind gust is then superposed by the underlying turbulent fluctuations. In order to evaluate the effect of this injection time on the FSI phenomenon, five coupled FSI simulations applying different injection times are compared in Appendix.

7.3.2. Effect of varying gust strength

Gusts with different strengths are injected in front of the flexible structure. The force coefficients in streamwise and vertical directions obtained with the *weak*, *medium* and *strong* gusts are compared in Fig. 17. The force coefficients predicted for the different gusts on the flexible case look similar to those obtained for the rigid body. The histories of c_x only slightly differ concerning the amplitudes of the extrema. For the *strong* gust with the flexible body, c_x reaches a first local maximum of 0.460, then a local minimum of 0.245 followed by the second local maximum of 0.679. Considering

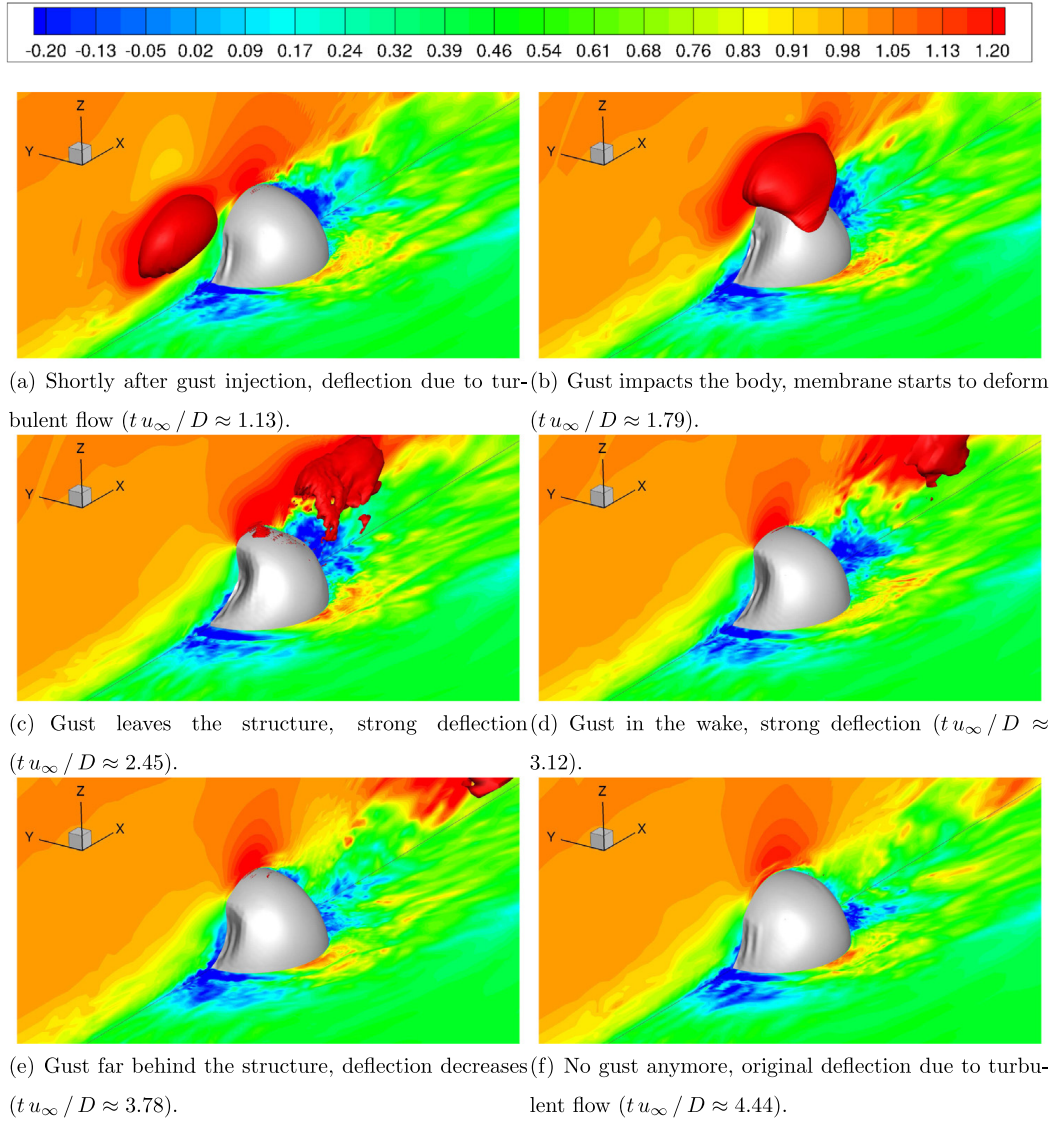


Fig. 14. Flexible FEM case: Injected *medium* wind gust moving downstream and impacting an air-inflated flexible membrane in form of a hemisphere (streamwise velocity u_1/u_{∞} in a slice near the bottom wall and in the symmetry plane $y = 0$). The gust structure is highlighted by an iso-surface of the streamwise velocity $u_1/u_{\infty} = 1.4$.

the rigid body, these extrema are 0.432, 0.144 and 0.614, respectively. Hence, the flexibility of the membrane tends to reduce the differences between the extrema, smoothing the slope of c_x . This is also valid for the *weak* and *medium* gusts.

Flow fields predicted for the *weak* and *strong* gusts are not plotted, since the spatial distributions of the streamwise velocity and the pressure do not really differ from those induced by the *medium* gust. Only their magnitudes change. However, a comparison of the corresponding structural deflections is of interest. Fig. 16 compares the deflections obtained for the *weak*, *medium* and *strong* gusts at the same six time instants as in Fig. 14. In case of the *weak* gust the indentation at the front increases until a deflection of minor amplitude is reached and, then, decreases with time (see Fig. 16(a)). In case of the *medium* and *strong* gusts, the deflections of the indentation are much larger and lead to waves along the vertical direction. This is visible in Figs. 16(b) and 16(c), where some lines cross each others.

A comparison of the maximally deflected membrane shapes is visualized in Fig. 18. Note that due to the time delays discussed above based on the histories of the forces, the maximal deflections observed for the three gust strengths are reached at different instants in time. Obviously, major changes are observed in the membrane deflections. As expected, the stronger the gust is, the deeper is the deflection at the front of the body (see Fig. 18). The formation of this indentation at the front has consequences on the rest of the structural shape: With increasing gust strength the tip of the body tends to be higher and the sides to be wider. These changes in the shape occur due to outer pressure distribution combined with the constant gauge pressure applied inside the closed volume. When the gust is far away, the indentation at the front reduces and the tip and sides of the hemispherical body return to their original positions.

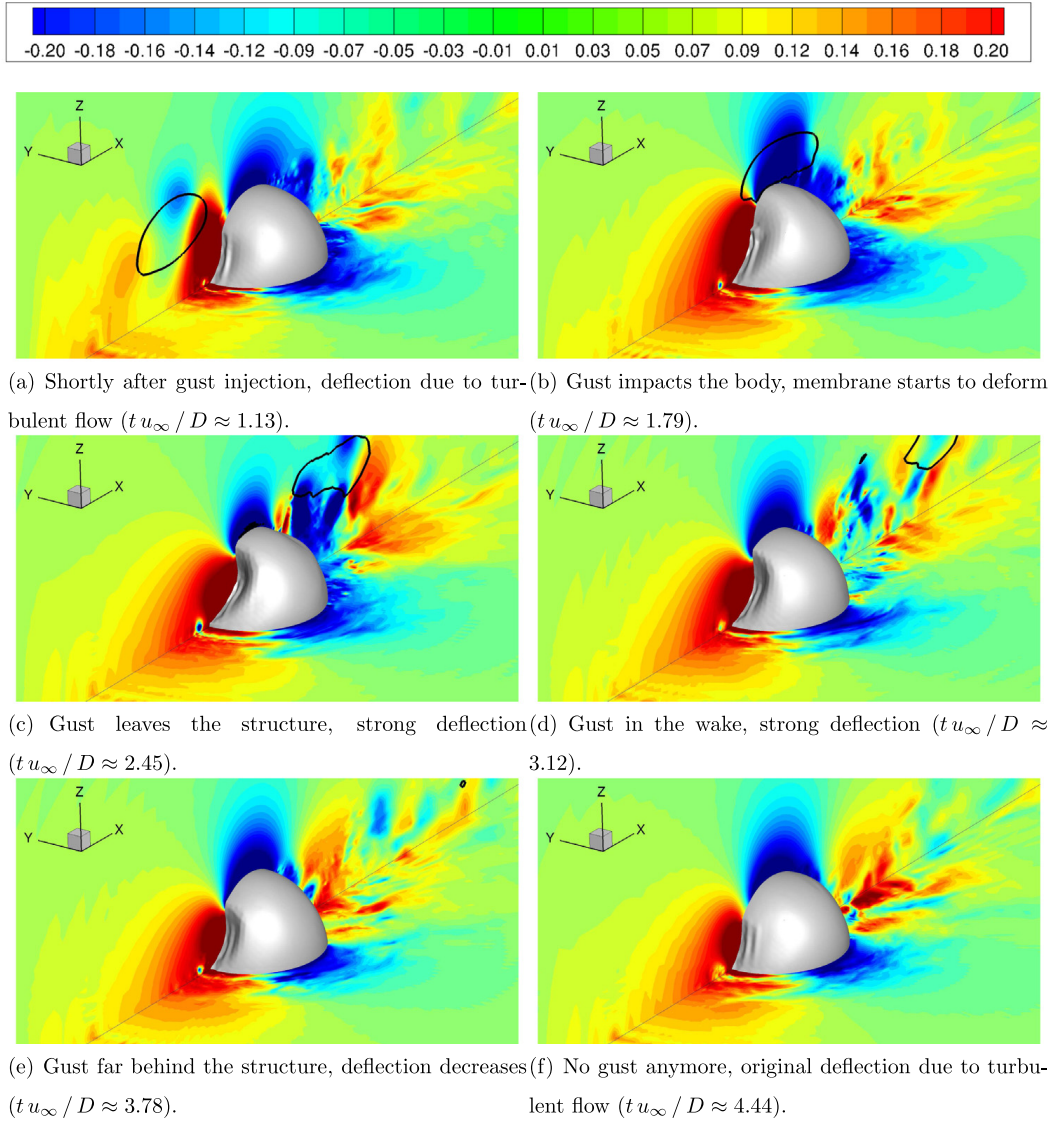


Fig. 15. Flexible FEM case: Injected *medium* wind gust moving downstream and impacting an air-inflated flexible membrane in form of a hemisphere (pressure $p/(\rho_{\text{air}} u_{\infty}^2)$) in a slice near the bottom wall and in the symmetry plane $y = 0$). The gust position is highlighted by a black iso-line of the streamwise velocity $u_1/u_{\infty} = 1.4$ in the symmetry plane.

For a deeper analysis of the structural behavior and the possible failure of the material, the principal strain values ε_1 and ε_2 are computed according to:

$$\varepsilon_1, \varepsilon_2 = \frac{\varepsilon_{11} + \varepsilon_{22}}{2} \pm \sqrt{\left(\frac{\varepsilon_{11} - \varepsilon_{22}}{2}\right)^2 + \varepsilon_{12}^2},$$

where ε_{ij} stands for the covariant components of the Green–Lagrange strain tensor defined over the surface of each finite element. Fig. 19 depicts these principal strains for the maximally deflected structure at all three gust strengths. The maxima of ε_1 are found in all cases at the apex of the membranous structure, whereas the absolute values are augmented by a factor of about 1.75 between the weak and the strong gust. Note that in the case of the strong gust ε_1 reaches values of about 6.5% and thus slightly exceeds the recommended classical limit of 5% for the application of the St. Venant–Kirchhoff material model.

For the second principal strain ε_2 the minima are of interest. These appear at the border of the front indentation, which at least in the case of the strong gust partially coincides with the apex of the hemisphere. For this strain the extrema between the weak and the strong gust differ by a factor of about 2.7.

Furthermore, a classical criterion to analyze failure of ductile materials is to predict the von Mises stress and to compare this equivalent uni-axial tensile stress with the yield stress of the material. Fig. 20 depicts the distribution of the von Mises stress for the same instants in time which are shown for the principal strains in Fig. 19. In agreement with the first principal strain ε_1 the largest von Mises stresses are observed at the apex of the hemispherical membrane. Of course,

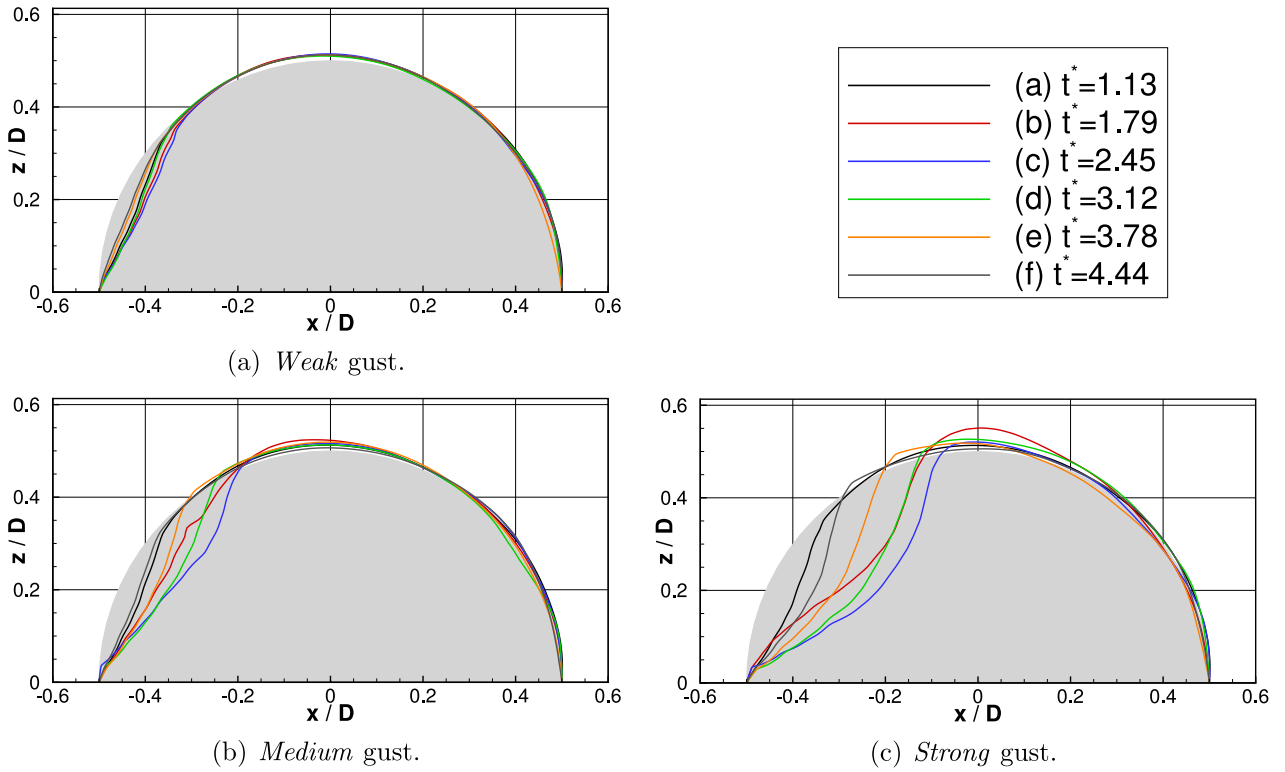


Fig. 16. Flexible FEM case: Structure deflections in the symmetry plane at six time instants.

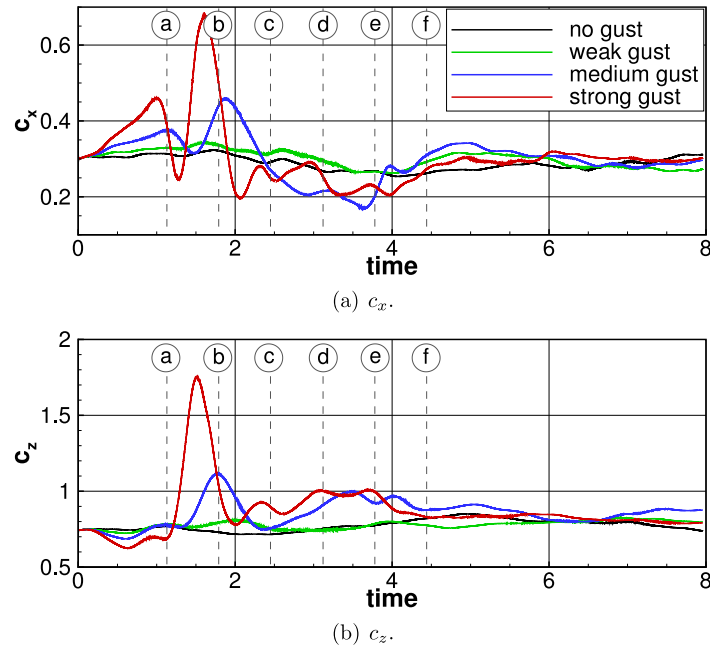


Fig. 17. Flexible FEM case: Streamwise and vertical force coefficients.

a rising gust strength leads to an increase of the von Mises stress, where again a factor of about 1.73 similar to ε_1 is found between the maxima of the weak and the strong gust. It has to be mentioned that in the present generic test case the von Mises stress does not reach the yield stress of silicone. According to the standardized plastic tensile test (EN ISO 527-2, ASTM D638) carried out in [Wood et al. \(2018\)](#) the yield stress is expected to be at least one order of magnitude higher than the currently occurring von Mises stress. Nevertheless, this issue does not play a role here, since the main purpose was to demonstrate that such kind of failure analysis is now possible for highly instantaneous loads acting on the structure.

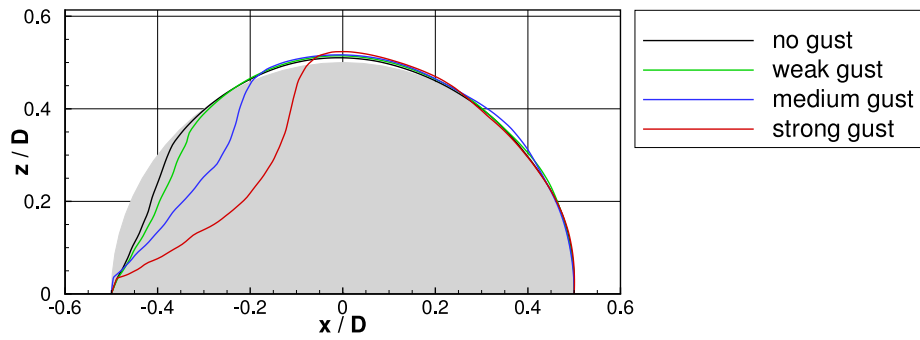


Fig. 18. Flexible FEM case: Maximum structural deflections in the symmetry plane for different gust strengths.

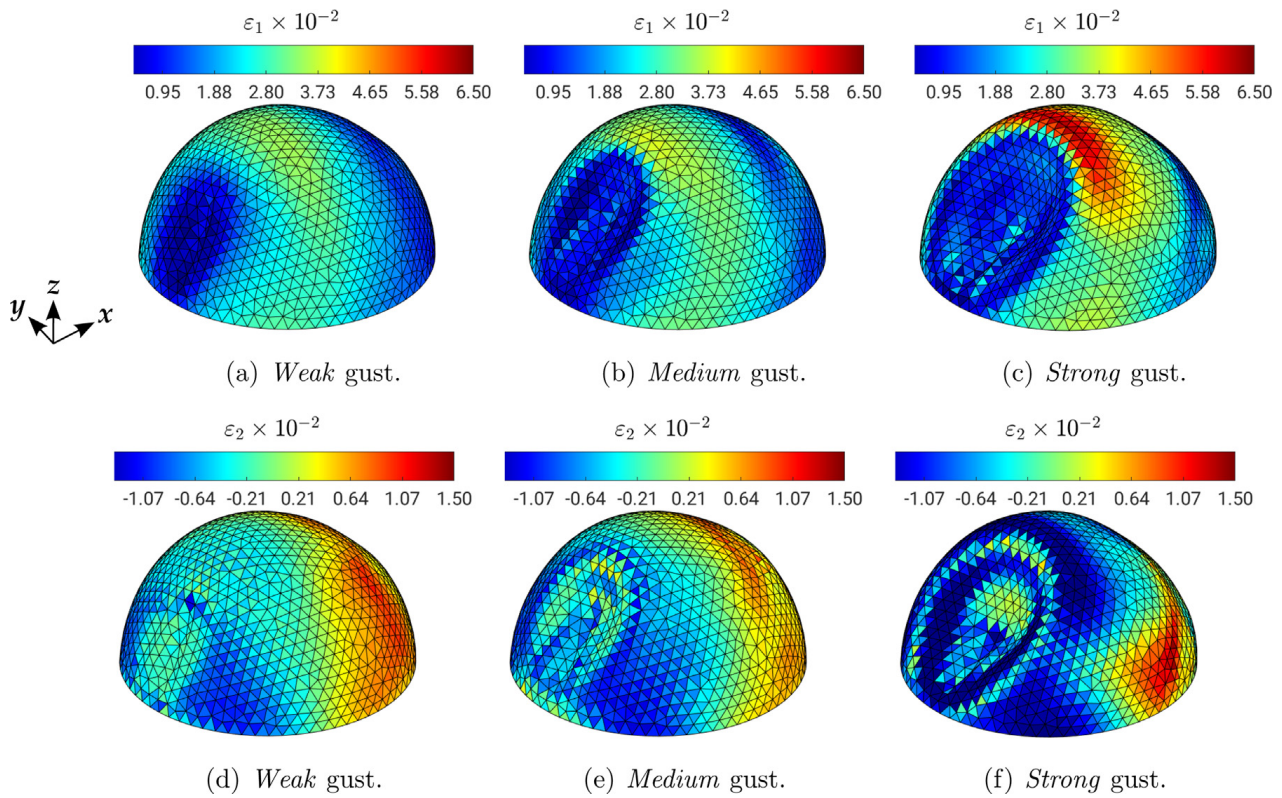


Fig. 19. Flexible FEM case: Distribution of the principal strain values ε_1 ((a) to (c)) and ε_2 ((d) to (f)) for the maximally deflected structure at different gust strengths.

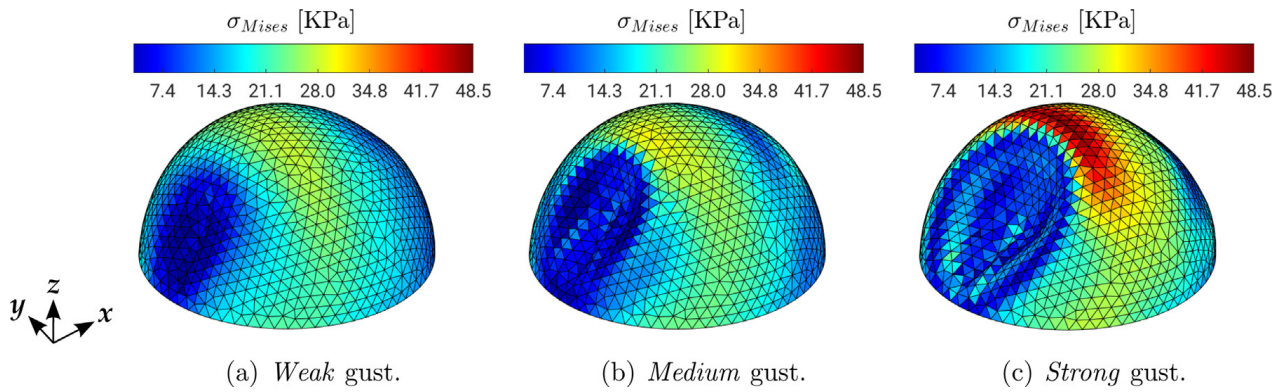


Fig. 20. Flexible FEM case: Von Mises stress distributions for the maximally deflected structure at different gust strengths.

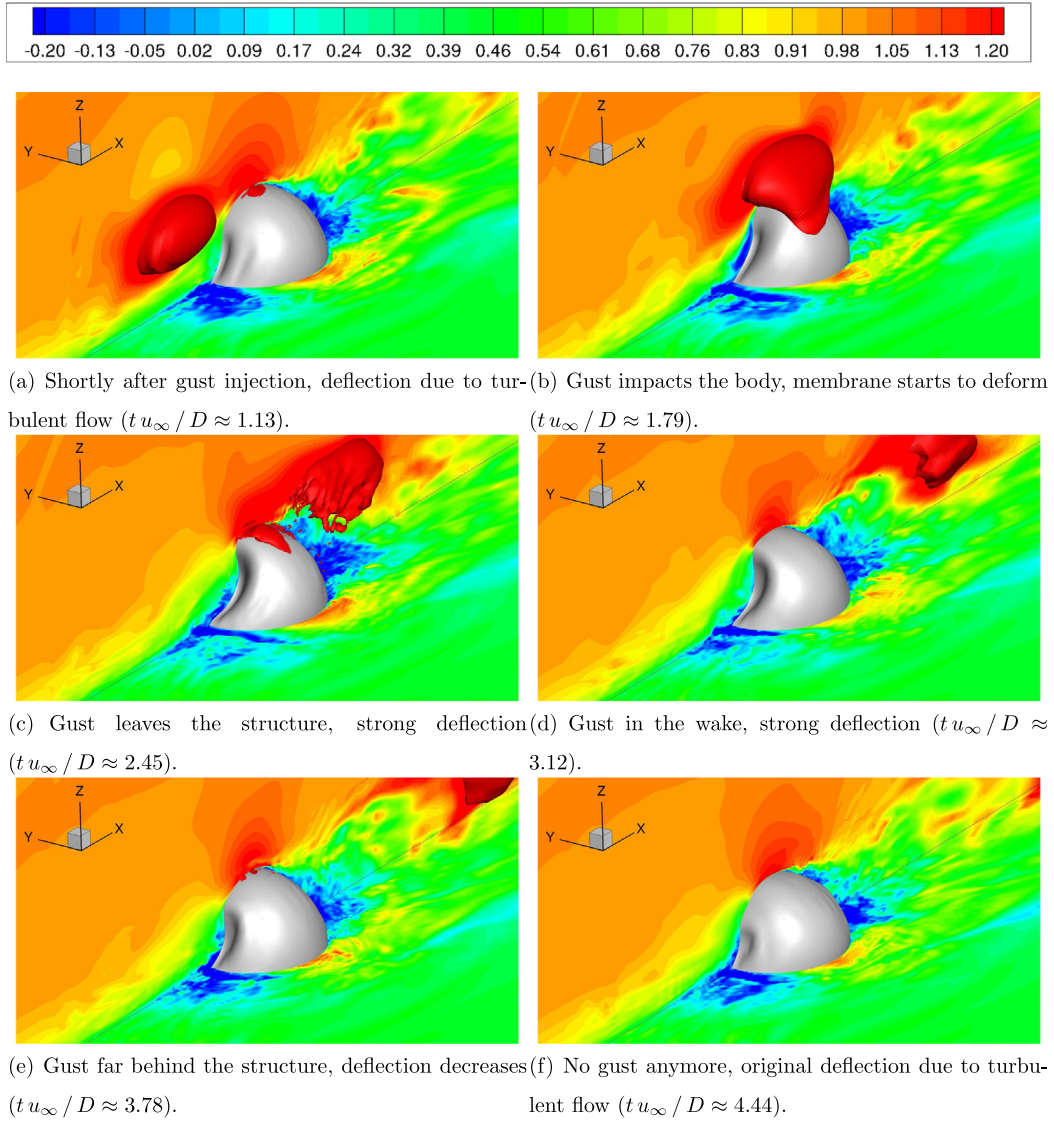
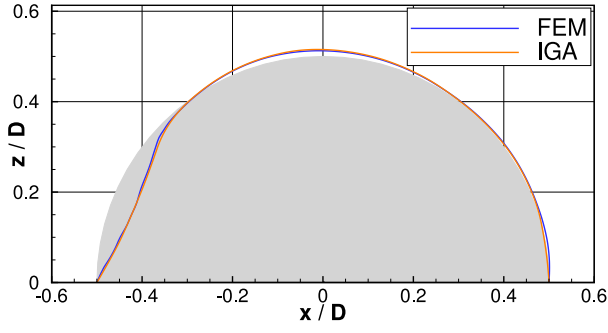


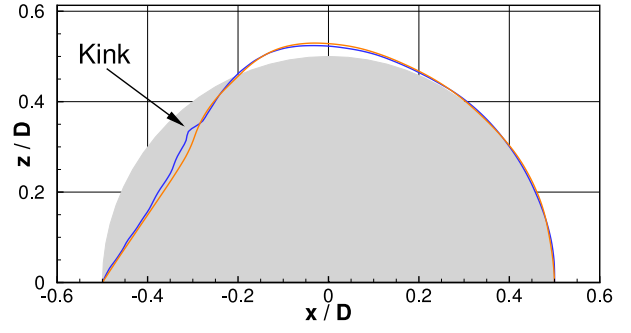
Fig. 21. Flexible IGA case (medium gust) Injected *medium* wind gust moving downstream and impacting an air-inflated flexible membrane in form of a hemisphere (streamwise velocity u_1/u_∞ in a slice near the bottom wall and in the symmetry plane $y = 0$). The gust structure is highlighted by an iso-surface of the streamwise velocity $u_1/u_\infty = 1.4$.

7.3.3. FEM vs. IGA

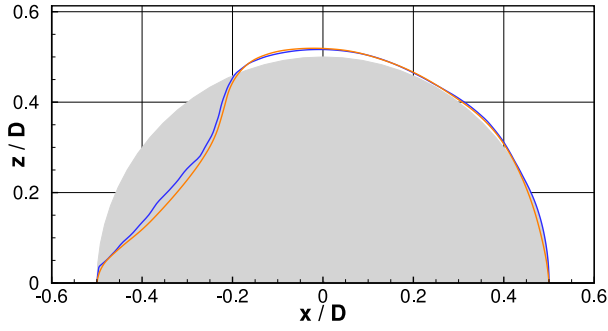
As detailed in Section 5.2 the thin-walled structure is either discretized by finite elements or an IGA discretization. The results analyzed up to now are all based on the FEM discretization. The results of the IGA method are exemplarily shown in Fig. 21, which is the counterpart to Fig. 14. Obviously, both methods deliver very similar deflections of the membranous structure, where the main difference is found in the front indentation. In order to visualize this difference, the structural deflections along the symmetry plane obtained by IGA and FEM are plotted for the same six instants in time (see Fig. 22). For the first, fourth, fifth and sixth time instants the spatial evolution of the membrane predicted by IGA is very similar to its FEM counterpart. Small but rather visible differences appear for $t^* \approx 1.79$ and 2.45 , where the deflections of the membrane are maximal. At $t^* \approx 1.79$ a kink in the deflections of the membrane predicted by FEM is present. Such artificial kinks are a consequence of the linear triangular membrane finite elements, which are large compared to the fluid cells at the FSI interface. In the case of large structural deflections, as those resulting from wind gust loading, the coarse low-order FEM mesh mapped to the fluid grid leads to an artificial roughness of the FSI interface. The coarser the FEM mesh is, the stronger is the artificial roughness (Apostolatos et al., 2019b). This artificial roughness can have a significant impact on the flow prediction, particularly on the location of the flow separation. Note that the FEM mesh currently employed for the FSI-LES simulation leads to flow predictions similar to those obtained by IGA as proved in Apostolatos et al. (2019b). Contrary to the FEM method, the isogeometric mortar-based mapping method predicts a smooth surface of the membrane (see Fig. 23(a)) since the artificial kinks arising from the FEM mesh (see Fig. 23(b)) are not transferred into the fluid domain. However, the IGA method leads to significantly higher computational costs as discussed in the next section.



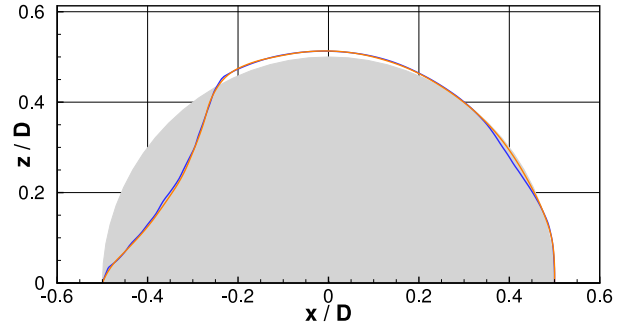
(a) Shortly after gust injection, deflection due to turbulent flow ($tu_\infty/D \approx 1.13$).



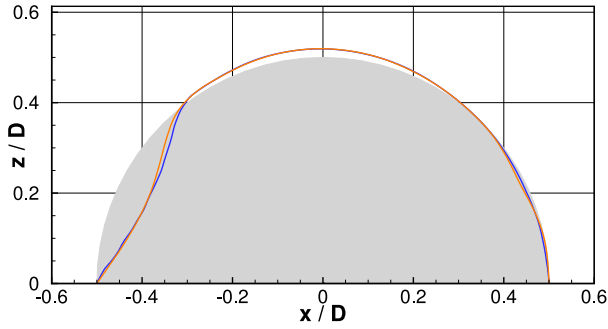
(b) Gust impacts the body, membrane starts to deform ($tu_\infty/D \approx 1.79$).



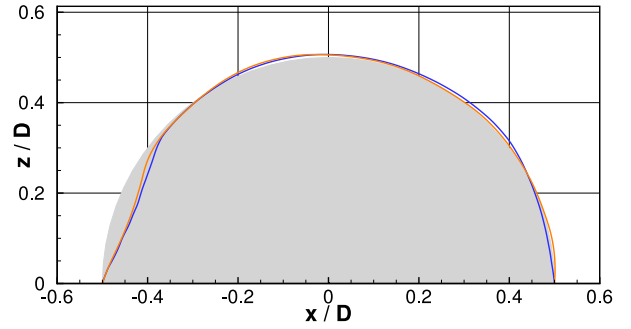
(c) Gust leaves the structure, strong deflection ($tu_\infty/D \approx 2.45$).



(d) Gust in the wake, strong deflection ($tu_\infty/D \approx 3.12$).

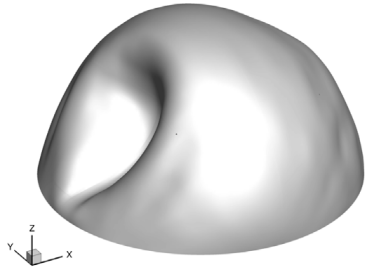


(e) Gust far behind the structure, deflection decreases ($tu_\infty/D \approx 3.78$).

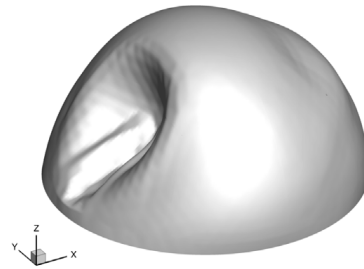


(f) No gust anymore, original deflection due to turbulent flow ($tu_\infty/D \approx 4.44$).

Fig. 22. Flexible IGA case vs. flexible FEM case: Structure deflections in the symmetry plane.



(a) IGA.



(b) FEM.

Fig. 23. Flexible IGA case vs. flexible FEM case: Deformed shape of the FSI interface at $t u_\infty/D \approx 2.45$.

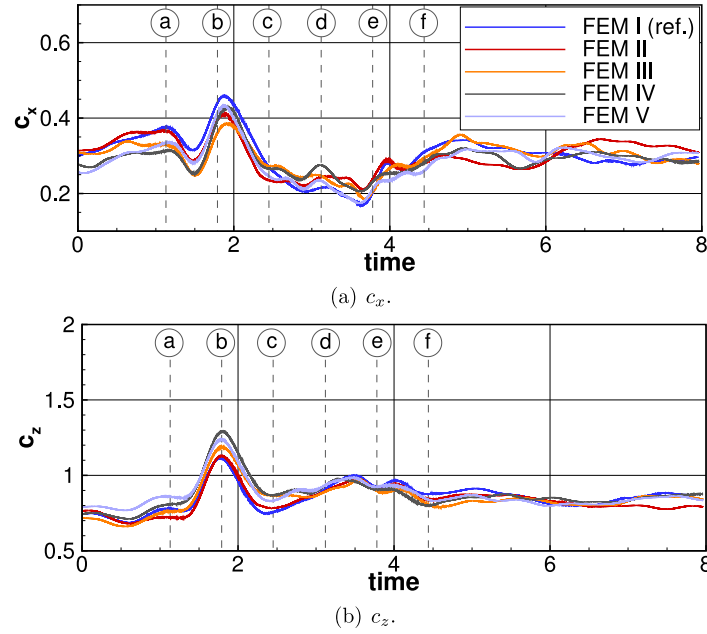


Fig. A.24. Flexible FEM case (medium gust): Streamwise and vertical force coefficients obtained with gusts injected at different time instants to demonstrate the effect of the underlying inflow turbulence.

7.4. Computational costs

The LES predictions including the different gusts conducted for the CFD case based on the rigid hemisphere took about 20 h wall-clock time on 40 processors (Intel Xeon Ivy Bridge E5-2650 with QDR Infiniband). The strength of the gust has a minimal impact on the computational costs (less than 0.5% of the total CPU time). The fully-coupled FSI-LES predictions including the gust and with a FEM discretization of the structure requires about 47 h on 42 processors of the same machine. This increase in CPU costs is only due to the FSI coupling (data exchange and fluid mesh adaption) and not due to the gust itself. The corresponding FSI-LES predictions applying the IGA discretization of the structure run during 230 h on 42 processors of the same machine. The total CPU-time increases about a factor of 5 compared to the FEM discretization. The IGA modeling alone is responsible for this strong increase and not the gust injection.

8. Conclusions

A recently developed novel gust injection technique was applied to simulate a challenging turbulent FSI case, i.e., a flexible membranous structure within a turbulent boundary layer undergoing highly transient wind events. The simulation methodology relies on a partitioned solution approach (LES/FEM/IGA) and takes the incoming turbulence into account. The purpose of this contribution was to show that an extension towards the consideration of wind gusts is straightforward and can be efficiently realized by the source-term formulation proposed in [De Nayer and Breuer \(2020\)](#), but up to now only validated based on laminar and turbulent background flows predicted on Cartesian grids. Similar to these generic test cases, the present real FSI application using a block-structured curvilinear grid confirms that it is possible to place the injection area of the gust close to the region of interest, which renders the method highly efficient. The numerical dissipation of the gust and time-consuming simulations for the transport of the gust from the inlet to the actual location of the FSI region are circumvented. Furthermore, the present practical application ratifies that the convergence behavior of the mass conservation equation does not deteriorate, since no source term is added to the continuity equation and thus the divergence-free condition for incompressible fluids is still satisfied. Consequently, a powerful methodology arises to tackle turbulent wind gusts in FSI applications.

The test case of the rigid and flexible hemisphere under different wind gust loads has brought the following findings:

- Horizontal wind gusts induce a significant increase of the forces in streamwise and vertical directions, i.e., a wind gust with an amplitude of $A_g = 1.5 u_\infty$ already leads to a doubling of the global forces on the structure compared to the pure boundary layer flow.
- In the present case of the hemispherical structure the flexibility of the membrane slightly dampens the resulting peak values of the global forces compared to the rigid case.
- However, in case of the flexible structure the local fluid loads play a more important role than the global forces. In the current configuration the pressure force is identified as the dominant contribution and the shear force is only of minor importance.

- The pressure forces induced by the gust hitting the structure lead to strong local deflections especially on the front part but not exclusively. These deflections are found to appear with a certain delay with respect to the local and global forces.
- Based on the predicted deflections of the membrane it is possible to determine the local stresses and strains in the material. As a classical failure criterion for ductile materials the von Mises stress can be compared with the yield stress of the material in order to check the structural integrity.
- With the help of a fully-coupled FSI-LES prediction applying a FEM discretization for the structure, the safety of a design under the influence of highly transient wind events can be assessed in a time frame of less than two days on a small cluster.
- The same fully-coupled FSI-LES simulation including gust, but applying an IGA discretization of the membrane delivers similar results as the FEM one in terms of observed FSI phenomena, flow fields and deflections. As expected, the deformed shape of the membrane is smoother. However, the computational time is about five times higher.

CRedit authorship contribution statement

G. De Nayer: Conceptualization, Methodology, Software, Validation, Investigation, Visualization, Writing – original draft, Writing – review & editing. **M. Breuer:** Conceptualization, Methodology, Resources, Writing – original draft, Writing – review & editing, Supervision, Project administration, Funding acquisition. **K. Boulbrachene:** Conceptualization, Methodology, Investigation, Visualization, Writing – original draft, Writing – review & editing.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgments

The project is financially supported by the *Deutsche Forschungsgemeinschaft*, Germany under the contract number BR 1847/17-1. Additionally, the authors want to thank the Chair of Structural Analysis of the Technical University of Munich for providing the computational structure dynamics solver Carat++ (Bletzinger et al., 2006) and the open-source coupling software EMPIRE (Sicklinger et al., 2014). Finally, the authors are grateful to A. Apostolatos for his support on the post-processing.

Appendix. Reproducibility of the wind gust generation by the ST method

In order to evaluate the reproducibility of the wind gust generation by the ST method, five coupled FSI simulations with the FEM membrane discretization are carried out. In the reference simulation (denoted FEM I) discussed in the previous sections, the gust is injected directly after the restart, which continues the preceding simulation with continuous turbulent inflow fluctuations. In the other simulations (II to V) again the same restart file for the fluid flow and the structure as well as the turbulent inflow data are used. Thus, the initial conditions are identical to the reference simulation. The only difference is that the gusts are injected with a certain time delay of $\Delta t_{\text{delay}}^* = 1, 2, 3$ and 4 compared to the gust injection time of FEM I, respectively. Consequently, the same gust is superimposed on five different background flows allowing to evaluate the variation of the resulting forces as a result of these superpositions.

The predicted streamwise and vertical global force coefficients are compared in Fig. A.24. Note that in the figures the time axes for FEM II to V are accordingly shifted to fit the time of the reference simulation for the sake of a direct comparison.

Obviously, the histories of the force coefficients obtained for the five different cases differ in terms of the absolute values, whereas the instants in time at which the extrema appear are more or less the same. Nevertheless, visible differences in the extreme values of c_x and c_z can be observed. For example, the peak value of c_z varies between 1.11 (FEM I) and 1.29 (FEM IV), which is about 16% and thus not negligible. Variations of the same order of magnitude are visible for the streamwise force coefficient c_x . However, evaluating the differences between the local minima and the local maxima in the interval (a) to (c), the deviations are even stronger between $\Delta c_x = 0.123$ for FEM II and 0.175 for FEM IV. This observed variability requires deeper investigations and will be the topic of ongoing studies.

References

- Adrian, R.J., 2007. Hairpin vortex organization in wall turbulence. *Phys. Fluids* 19 (4), 041301.
- American Society of Civil Engineers, 1997. Standard Guidelines for Air Supported Structures. In: ASCE standard, American Society of Civil Engineers, URL <https://books.google.de/books?id=srVxQgAACAAJ>.
- Apostolatos, A., Bletzinger, K.-U., Wüchner, R., 2019a. Weak imposition of constraints for structural membranes in transient geometrically nonlinear isogeometric analysis on multipatch surfaces. *Comput. Methods Appl. Mech. Engrg.* 350, 938–994.

- Apostolatos, A., De Nayer, G., Bletzinger, K.-U., Breuer, M., Wüchner, R., 2019b. Systematic evaluation of the interface description for fluid–structure interaction simulations using the isogeometric mortar-based mapping. *J. Fluids Struct.* 86, 368–399.
- Augier, B., Bot, P., Hauville, F., Durand, M., 2012. Experimental validation of unsteady models for fluid structure interaction: Application to yacht sails and rigs. *J. Wind Eng. Ind. Aerodyn.* 101, 53–66.
- Bailly, C., Comte-Bellot, G., 2015. *Turbulence*. Springer.
- Basar, Y., Weichert, D., 2013. *Nonlinear Continuum Mechanics of Solids: Fundamental Mathematical and Physical Concepts*. Springer Science & Business Media.
- Bletzinger, K.-U., Wüchner, R., Kupzok, A., 2006. Algorithmic treatment of shells and free form-membranes in FSI. In: Bungartz, H.-J., Schäfer, M. (Eds.), *Fluid-Structure Interaction*. In: *Lecture Notes in Computational Science and Engineering*, LNCSE, vol. 53, Springer, Heidelberg, pp. 336–355.
- Boulbrachene, K., De Nayer, G., Breuer, M., 2021. Assessment of two wind gust injection methods: Field velocity vs. split velocity method. *J. Wind Eng. Ind. Aerodyn.* 218, 104790.
- Breitenberger, M., Apostolatos, A., Philipp, B., Wüchner, R., Bletzinger, K.-U., 2015. Analysis in computer aided design: Nonlinear isogeometric b-rep analysis of shell structures. *Comput. Methods Appl. Mech. Engrg.* 284, 401–457.
- Breuer, M., 2018. Effect of inflow turbulence on an airfoil flow with laminar separation bubble: An LES study. *Flow Turbul. Combust.* 101 (2), 433–456.
- Breuer, M., De Nayer, G., Münsch, M., Gallinger, T., Wüchner, R., 2012. Fluid-structure interaction using a partitioned semi-implicit predictor-corrector coupling scheme for the application of large-eddy simulation. *J. Fluids Struct.* 29, 107–130.
- Burton, T., Sharpe, D., Jenkins, N., Bossanyi, E., 2001. *Wind Energy Handbook*. John Wiley & Sons.
- De Nayer, G., Apostolatos, A., Wood, J.N., Bletzinger, K.-U., Wüchner, R., Breuer, M., 2018a. Numerical studies on the instantaneous fluid–structure interaction of an air-Inflated flexible membrane in turbulent flow. *J. Fluids Struct.* 82, 577–609.
- De Nayer, G., Breuer, M., 2014. Numerical FSI investigation based on LES: Flow past a cylinder with a flexible splitter plate involving large deformations (FSI-PfS-2a). *Int. J. Heat Fluid Flow* 50, 300–315.
- De Nayer, G., Breuer, M., 2020. A source-term formulation for injecting wind gusts in CFD simulations. *J. Wind Eng. Ind. Aerodyn.* 207, 104405.
- De Nayer, G., Kalmbach, A., Breuer, M., Sicklinger, S., Wüchner, R., 2014. Flow past a cylinder with a flexible splitter plate: A complementary experimental-numerical investigation and a new FSI test case (FSI-PfS-1a). *Comput. & Fluids* 99, 18–43.
- De Nayer, G., Schmidt, S., Wood, J.N., Breuer, M., 2018b. Enhanced injection method for synthetically generated turbulence within the flow domain of eddy-resolving simulations. *Comput. Math. Appl.* 75 (7), 2338–2355.
- Durst, F., Schäfer, M., 1996. A parallel block-structured multigrid method for the prediction of incompressible flows. *Int. J. Numer. Methods Fluids* 22 (6), 549–565.
- Frost, W., Long, B.H., Turner, R.E., 1978. *Engineering Handbook on the Atmospheric Environmental Guidelines for Use in Wind Turbine Generator Development*. Tech. rep., Tennessee Univ., Tullahoma (USA). Space Inst.; National Aeronautics and Space Administration, Huntsville, AL (USA). George C. Marshall Space Flight Center.
- Glück, M., Breuer, M., Durst, F., Halfmann, A., Rank, E., 2003. Computation of wind-induced vibrations of flexible shells and membranous structures. *J. Fluids Struct.* 17 (5), 739–765.
- Heinrich, R., Reimer, L., 2013. Comparison of different approaches for gust modeling in the CFD code TAU. In: *Int. Forum on Aeroelasticity & Structural Dynamics 2013*. URL <https://elib.dlr.de/85834/>.
- IEC-Standard, 2002. 61400-21. Tech. report.
- Kareem, A., Wu, T., 2013. Wind-induced effects on bluff bodies in turbulent flows: Nonstationary, non-Gaussian and nonlinear features. *J. Wind Eng. Ind. Aerodyn.* 122, 21–37.
- Kasperski, M., 2007. A consistent model for the codification of wind loads. *J. Wind Eng. Ind. Aerodyn.* 95 (9), 1114–1124.
- Klein, M., Sadiki, A., Janicka, J., 2003. A digital filter based generation of inflow data for spatially-developing direct numerical or large-eddy simulations. *J. Comput. Phys.* 186, 652–665.
- Kwon, D.K., Kareem, A., 2013. Generalized gust-front factor: A computational framework for wind load effects. *Eng. Struct.* 48, 635–644.
- Lesieur, M., 2008. *Turbulence in Fluids*, 4th edition Springer.
- Michalski, A., Gawenat, B., Gellenne, P., Haug, E., 2015. Computational wind engineering of large umbrella structures. *J. Wind Eng. Ind. Aerodyn.* 144, 96–107.
- Michalski, A., Kermel, P.D., Haug, E., Löhner, R., Wüchner, R., Bletzinger, K.-U., 2011. Validation of the computational fluid–structure interaction simulation at real-scale tests of a flexible 29 m umbrella in natural wind flow. *J. Wind Eng. Ind. Aerodyn.* 99 (4), 400–413.
- Mokin, N.A., Kustov, A.A., Trushin, S.I., 2017. Numerical simulation of an air-supported structure in the air flow. In: Bletzinger, K.-U. and Onate, E. and Kröplin, B. (Eds.), *VIII Int. Conf. on Textile Composites and Inflatable Structures, Structural Membranes 2017*.
- Newman, B.G., Ganguli, U., Shrivastava, S.C., 1984. Flow over spherical inflated buildings. *J. Wind Eng. Ind. Aerodyn.* 17 (3), 305–327.
- Norris, S.E., Cater, J.E., Stol, K.A., Unsworth, C.P., 2010. Wind turbine wake modelling using large-eddy simulation. In: *Proceedings of the 17th Australian Fluid Mechanics Conference*. Curran Associates, Inc., University of Auckland, Australia.
- Pasquetti, R., Peres, N., 2015. A penalty model of synthetic micro-jet actuator with application to the control of wake flows. *Comput. & Fluids* 114, 203–217.
- Philipp, B., Wüchner, R., Bletzinger, K.-U., 2016. Advances in the form-finding of structural membranes. *Procedia Eng.* 155, 332–341.
- Piomelli, U., Chasnov, J.R., 1996. Large-eddy simulations: Theory and applications. In: Hallböck, M., Henningson, D.S., Johansson, A.V., Alfredson, P.H. (Eds.), *Turbulence and Transition Modeling*. Kluwer, pp. 269–331.
- Santo, G., Peeters, M., Van Paepegem, W., Degroote, J., 2020. Fluid-structure interaction simulations of a wind gust impacting on the blades of a large horizontal axis wind turbine. *Energies* 13, 509.
- Schlatter, P., Orlu, R., Li, Q., Brethouwer, G., Fransson, J.H.M., Johansson, A.V., Alfredsson, P.H., Henningson, D.S., 2009. Turbulent boundary layers up to $Re_\theta = 2500$ studied through simulation and experiment. *Phys. Fluids* 21 (5), 51702.
- Schmidt, S., Breuer, M., 2017. Source term based synthetic turbulence inflow generator for eddy-resolving predictions of an airfoil flow including a laminar separation bubble. *Comput. & Fluids* 146, 1–22.
- Sen, S., De Nayer, G., Breuer, M., 2017. A fast and robust hybrid method for block-structured mesh deformation with emphasis on FSI-LES applications. *Int. J. Numer. Methods Eng.* 111 (3), 273–300.
- Sicklinger, S., Belsky, V., Engelmann, B., Elmqvist, H., Olsson, H., Wüchner, R., Bletzinger, K.-U., 2014. Interface Jacobian-based co-simulation. *Int. J. Numer. Methods Eng.* 98 (6), 418–444.
- Singh, R., Baeder, J.D., 1997. Direct calculation of three-dimensional indicial lift response using computational fluid dynamics. *J. Aircr.* 34 (4), 465–471.
- Smagorinsky, J., 1963. General circulation experiments with the primitive equations I: The basic experiment. *Mon. Weather Rev.* 91 (3), 99–165.
- Thompson, J.F., Warsi, Z.U.A., Wayne Mastin, C., 1985. *Numerical Grid Generation: Foundations and Applications*, Vol. 45. North-Holland.
- Wales, C., Jones, D., Gaitonde, A., 2014. Prescribed velocity method for simulation of aerofoil gust responses. *J. Aircr.* 52 (1), 64–76.
- Wang, T., Wüchner, R., Sicklinger, S., Bletzinger, K.-U., 2016. Assessment and improvement of mapping algorithms for non-matching meshes and geometries in computational FSI. *Comput. Mech.* 57 (5), 793–816.
- Wood, J.N., Breuer, M., De Nayer, G., 2018. Experimental studies on the instantaneous fluid–structure interaction of an air-Inflated flexible membrane in turbulent flow. *J. Fluids Struct.* 80, 405–440.

- Wood, J.N., De Nayer, G., Schmidt, S., Breuer, M., 2016. Experimental investigation and large-eddy simulation of the turbulent flow past a smooth and rigid hemisphere. *Flow Turbul. Combust.* 97 (1), 79–119.
- Wu, Y., Sun, X., Shen, S., 2008. Computation of wind-structure interaction on tension structures. *J. Wind Eng. Ind. Aerodyn.* 96, 2019–2032.
- Wüchner, R., Kupzok, A., Bletzinger, K.-U., 2007. A framework for stabilized partitioned analysis of thin membrane-wind interaction. *Int. J. Numer. Methods Fluids* 54 (6–8), 945–963.
- Yuan, C., Puyong, W., Xianlong, J., 2010. Dynamic analysis of flexible container under wind actions by ALE finite-element method. *J. Wind Eng. Ind. Aerodyn.* 98, 881–887.

Publication IV: A numerical method to mimic an experimental wind gust generator: The immersed boundary gust generator

Reference

K. Boulbrachene and M. Breuer. A numerical method to mimic an experimental wind gust generator: The immersed boundary gust generator. *Physics of Fluids* 36 (2024), 015146. DOI: [10.1063/5.0187787](https://doi.org/10.1063/5.0187787)

Copyright

© 2024 The Authors. Published by AIP Publishing. This is an open access article distributed under the terms of the Creative Commons CC-BY license (<http://creativecommons.org/licenses/by/4.0/>), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

A numerical method to mimic an experimental wind gust generator: The immersed boundary gust generator

Cite as: Phys. Fluids **36**, 015146 (2024); doi: [10.1063/5.0187787](https://doi.org/10.1063/5.0187787)

Submitted: 16 November 2023 · Accepted: 27 December 2023 ·

Published Online: 24 January 2024



View Online



Export Citation



CrossMark

K. Boulbrachene and M. Breuer^{a)}

AFFILIATIONS

Professur für Strömungsmechanik, Helmut-Schmidt-Universität Hamburg, D-22043 Hamburg, Germany

^{a)} Author to whom correspondence should be addressed: breuer@hsu-hh.de

ABSTRACT

To generate horizontal wind gusts in a classical wind tunnel, Wood, Breuer, and Neumann [A novel approach for artificially generating horizontal wind gusts based on a movable plate: The paddle,” J. Wind Eng. Ind. Aerodyn. 230, 105170 (2022)] developed a new wind gust generator denoted the “paddle.” The working principle relies on the partial blocking of the outlet of the wind tunnel nozzle by a plate that vertically moves into the free-stream. Based on laser-Doppler anemometer measurements of the velocity at only a few locations, the basic functionality of the device was proven. The objective of the present contribution is to numerically mimic the gust generator and the flow field induced by the paddle in the test section. Contrary to the single-point measurements, the three-dimensional time-resolved simulation delivers the entire flow field and thus allows to investigate all details of the generated gust. To describe the paddle motion, the immersed boundary method with a continuous and direct forcing approach is implemented into a finite-volume flow solver for large-eddy simulations. A uniform and a non-uniform distribution of the Lagrangian markers are investigated where the latter ensures that an excessive increase in the computational resources required can be avoided. The predictions allow to characterize the resulting flow features induced by the paddle in great detail. Furthermore, a comparison of the numerical and experimental results is carried out based on the time histories of the streamwise and vertical velocity components at certain positions showing a close agreement. Finally, the forces acting on the fluid by the moving paddle are evaluated.

© 2024 Author(s). All article content, except where otherwise noted, is licensed under a Creative Commons Attribution (CC BY) license (<http://creativecommons.org/licenses/by/4.0/>). <https://doi.org/10.1063/5.0187787>

I. INTRODUCTION

Wind gusts are a meteorological phenomenon that is also of high interest for engineers due to their devastating consequences on structures. Especially constructions made of lightweight thin membranes such as large umbrellas, tents, or stadium roofs are vulnerable to wind events. In order to better understand the effect of strong wind gusts on such constructions, investigations on this fluid-structure interaction (FSI) scenario are necessary. Measurements at real scales can only be carried out if the building has already been constructed. Furthermore, they are extremely challenging and costly. Thus, investigations at a smaller scale are an alternative, but require to artificially generate the wind gust in the wind tunnel.

Recently, Wood, Breuer, and Neumann¹ carried out a literature review on the topic of experimental devices devised for the generation of artificial gusts. They distinguished the different methods into two classes: methods for the generation of low and high amplitude gusts.

The latter class also includes the working principle of substantially blocking the wind tunnel inflow section. For example, Neunaber and Braud² developed a rotating bar called the *chopper* cutting through a wind tunnel inlet to model extreme gust events for wind turbines. The flow velocity depicts a spike followed by an inverse gust due to the blockage of the inlet. Thus, rapid and strong velocity fluctuations with high turbulence intensities are generated.

In order to better control the shape and the intensity of the provoked wind gust, to reach a high reproducibility and to allow the superposition of the wind gust to the free-stream flow, Wood, Breuer, and Neumann¹ developed an artificial wind gust generator (WGG) denoted the “paddle.” In Sec. II, the main features of this new device are explained. By partially blocking the outlet of the wind tunnel nozzle by the paddle, the flow is strongly accelerated leading to a horizontal gust. Based on laser-Doppler anemometer (LDA) measurements, the instantaneous and phase-averaged streamwise and wall-normal

velocities were determined at 11 specific points in the flow field. That allowed to get a first impression of the flow field during the gust generation. However, a full two-dimensional (e.g., geometric symmetry plane) or even a full three-dimensional (e.g., entire test section) representation of the gust event could not be achieved. This gave rise to the idea that this obvious gap can be closed through appropriate numerical simulations leading to the current study.

Since the experimental setup includes the moving paddle, which blocks a part of the outlet of the nozzle for a short time interval, this specific feature has to be taken into account. One way to model this is by using a body-conforming mesh and to directly enforce the boundary conditions on the interface. This is the so-called Arbitrary Lagrangian–Eulerian (ALE) method³ known for its great capability in resolving the flow and stress distributions accurately along the fluid-structure interface. However, for fluid-structure interaction (FSI) problems involving large displacements (which is the case here), the necessary time-consuming remeshing during the course of the simulation leads to a substantial computational costs. In addition, in applications with large deformations, the quality of the adapted grid and therefore the solution can suffer significantly. An alternative method to the body-fitted approach is the immersed boundary method (IBM). The IBM, originally introduced by Peskin,⁴ found its initial application in simulating cardiac mechanics and the associated blood flow. Unlike the body-conforming approach, this method uses an overlapping (i.e., non-conforming) discretizations for the fluid and structural domains. Eulerian and Lagrangian frames of references are used to describe physical quantities associated with the fluid and the structure, respectively. The no-slip boundary condition is enforced on the fluid–solid interface by formulating an appropriate boundary forcing term.

Mittal and Iaccarino⁵ classified the existing IBMs into two categories based on the way this forcing term is introduced into the governing equations. The first is termed as the *continuous forcing approach* in which the boundary forcing is incorporated into the continuous governing equations of the fluid flow (i.e., before discretization). This gives this class of IBMs the attractive feature of being able to formulate the forcing term independent of the employed spatial discretization. In the second class denoted the *discrete forcing approach*, the governing equations are first discretized without considering the IB. Then, its influence is brought up by adjusting the discretization in the cells in its vicinity. Although the formulated forcing term is intimately connected to the spatial discretization scheme, the discrete forcing enables a sharp representation of the IB. Examples of this class of IBMs employing a direct imposition of boundary conditions are the ghost-cell finite-difference approach⁶ and the cut-cell finite-volume approach.⁷ It is worth mentioning that this class of IBMs encounters the challenge of the “freshly cleared” cells when the boundary is moving. In simple terms, this issue arises when a spatial discontinuity linked to the sharp representation of the IB becomes apparent within fluid cells that were within the solid phase during the previous time step. Although remedies to this issue were suggested in Udaykumar, Mittal, and Shyy,⁸ Udaykumar *et al.*,⁹ this challenge is not met in the continuous forcing approach. This is due to the spreading effect across a few grid cells on both sides of the boundary, facilitating a gradual transition between the fluid and solid phases, effectively mitigating temporal discontinuities for cells transitioning into the fluid phase. Since this work is concerned with the inclusion of a moving IB (here the paddle) into the

computational domain, continuous forcing IBMs are favored and discussed hereafter.

The first IBM with a continuous forcing approach is the original IBM.^{4,10} At the continuous level, the boundary forces are represented by a source term added to the governing equations as singular functions acting along the immersed surface. At the discrete level, the IB is described by a set of Lagrangian markers conceived as material points and the boundary forces are localized at these discrete Lagrangian locations. In order to avoid numerical instabilities, these localized forces are introduced in conjunction with a regularized delta function of finite support, effectively spreading the force smoothly over the neighboring Eulerian cells. Depending on the material type of the IB (i.e., elastic or rigid), different methods were used to compute the forcing function. For flexible boundaries, Peskin^{4,11} used Hooke’s law to compute the IB forcing in the simulation of blood flow inside a heart with flexible valves. Different approaches were proposed for the incorporation of rigid immersed boundaries. Beyer and LeVeque¹² and Lai and Peskin¹³ suggested to consider a structure attached to an equilibrium location by a spring of large stiffness. This formulation of the singular force is viewed as a specific version of the feedback mechanism outlined in the virtual boundary method, initially proposed by Goldstein, Handler, and Sirovich¹⁴ and subsequently applied in Saiki and Biringen.¹⁵ In this technique, forces (conceived as a system of virtual springs and dampers) are computed based on the difference between the actual and the desired values of the velocity. These are then set to act in an opposing manner to reinstate the desired velocity. This indirect formulation of the forces introduces user-defined (spring and damper) parameters that are chosen as a compromise between achieving acceptable boundary conditions and sufficiently large time step sizes.^{13,16} To circumvent the drawback of the time step restriction associated with these indirect penalty formulations (also see Angot, Bruneau, and Fabrie,¹⁷ Khadra *et al.*¹⁸), direct force formulations were proposed. Fadlun, Verzicco, and Orlandi¹⁹ introduced a direct forcing scheme that alters the discretized momentum equation at Eulerian points in the proximity of the IB such that the desired velocity at the boundary points is attained. However, undesirable force oscillations in flows involving moving boundaries were observed due to the interpolation procedure relating values at fixed grid nodes and values at arbitrary located boundaries. Uhlmann²⁰ addressed this issue by refining an alternative direct forcing scheme. His approach relies on computing the force on the Lagrangian marker first and subsequently spreading it onto the neighboring Eulerian cells. This renders this method particularly well-suited for the intended application in this study. To enable the application of this method on a general, possibly curvilinear, non-uniform Eulerian grid without compromising either its conservation properties or the order of the underlying spatial discretization, it is employed according to the work proposed by Pinelli *et al.*²¹

The paper is organized as follows: Section II describes the experimental setup of the wind gust generator. The immersed boundary method used to simulate the flow in the gust generator is explained in Sec. III, while the numerical framework relying on the large-eddy simulation (LES) technique is detailed in Sec. IV. All details of the setup including the kinematics of the paddle are given in Sec. V. Starting with a general description of the arising flow field, a comparison of the measured and predicted velocity signals is presented in Sec. VI. Finally, the forces acting by the moving paddle on the flow are analyzed.

II. EXPERIMENTAL SETUP OF THE WIND GUST GENERATOR

Since one objective of the current study is to numerically mimic the novel wind gust generator (WGG) denoted the “*paddle*,” in the following, the setup of the WGG is briefly described. The description is based on the work of Wood, Breuer, and Neumann¹ and Wood and Breuer.²² The concept relies on the idea to design a device that allows to artificially generate horizontal wind gusts within a classical Göttingen-type wind tunnel. It should be possible to cost-efficiently retrofit the WGG to any wind tunnel with an open test section and to build it from standard components.

The working principle sketched in Fig. 1 is simple: Initially, the nozzle is fully open and the fluid flow from the left to the right between the nozzle and the receiver defines the measurement section in which the object of interest can be placed. The horizontal gust is generated by partially blocking the outlet of the wind tunnel nozzle by the paddle. For this purpose, a rigid aluminum plate (red) is screwed onto the movable saddle of a tooth belt axis. That is clamped onto a massive rig and positioned above the outlet of the nozzle. The saddle is powered by a servo motor, a gear box, and an axial kit (Festo). All relevant kinematic parameters (stroke length, velocity, and acceleration) of the tooth belt axis can be fully controlled by the software tool Festo Automation Suite.

The starting point is the situation when the wind tunnel is operating in its normal mode. Then, the paddle of the WGG is set into motion, moving vertically downward into the free-stream. By reducing the outlet area, the flow is strongly accelerated underneath the moving paddle leading to the unsteady flow velocity. The maximum velocity of the artificially generated horizontal gust is approximately achieved at the time when the paddle reaches its maximum displacement and thus the maximum blocking of the nozzle. After this instant in time, the plate is set to move back upward causing an expansion of the flow. Consequently, the flow velocity decreases again and after a transition

period the free-stream velocity is recovered. Since the movement of the paddle can be arbitrary defined by the distance it covers, its velocity and acceleration during the downward and upward stroke, the gust can be set as desired within certain limits. In Wood, Breuer, and Neumann,¹ five different motion patterns of the paddle were investigated, and their effects on the generated gusts were analyzed. For gust type 1 relying on a fast and symmetric motion pattern, the parameters are set to mimic the 1-cosine gust shape considered by the IEC-Standard²³ and denoted as Extreme Coherent Gust (ECG). This case is considered in the present study.

III. IMMERSED BOUNDARY METHOD WITH A DIRECT FORCING APPROACH

In this work, the paddle is modeled as a moving boundary employing the immersed boundary (IB) method with a direct forcing approach proposed by Uhlmann.²⁰ On the continuous level, the presence of the immersed boundary (Γ) is incorporated into the continuous governing equations by the addition of a forcing term $\mathbf{f}$ to their right-hand side. The Navier–Stokes equations for an incompressible fluid on the domain Ω then read

$$\left. \begin{aligned} \rho \frac{\partial \mathbf{u}}{\partial t} &= \rho \nabla \cdot (\mathbf{u}\mathbf{u}) + \mu \Delta \mathbf{u} - \nabla p + \mathbf{f} \\ \nabla \cdot \mathbf{u} &= 0 \end{aligned} \right\} \mathbf{x} \in \Omega, \quad (1)$$

where $\mathbf{f}$ is a volume force that represents the influence of the solid object on the surrounding fluid. On the discrete level, quantities at the Eulerian (i.e., fluid grid) and the Lagrangian (i.e., solid's boundary points) locations are distinguished by lower- and upper-case letters, respectively. In the scope of the collocated grid arrangement employed in the finite-volume solver used, Eulerian locations refer to the cell centers $\mathbf{x}_{i,j,k}$. The Lagrangian locations $\mathbf{X}_l$ are the coordinates of the distributed Lagrangian force points defining the immersed boundary Γ

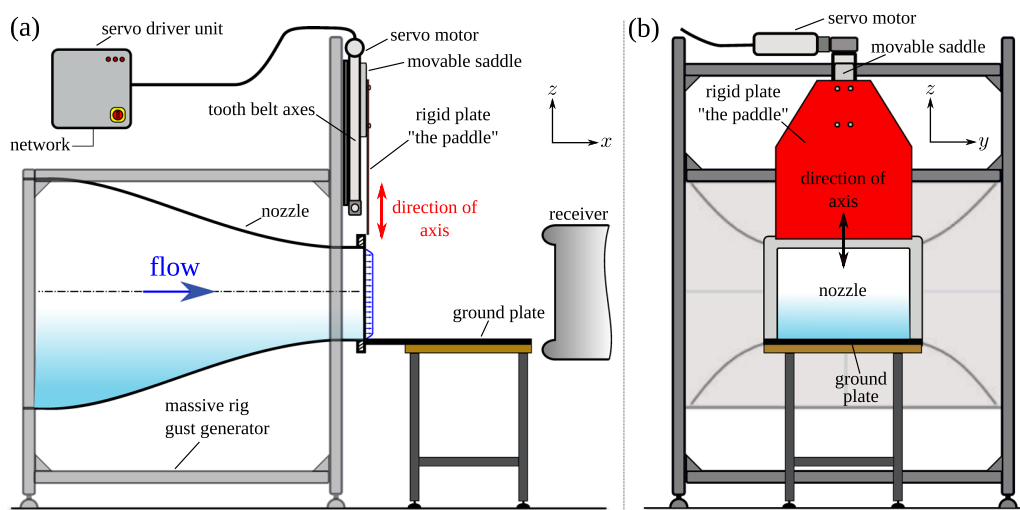


FIG. 1. Experimental setup of the wind tunnel with the new wind gust generator: (a) side view showing the test section with the additional ground plate and (b) front view depicting the paddle and the outlet of the nozzle. Reprinted with permission from Wood *et al.*, J. Wind Eng. Ind. Aerodyn. **230**, 105170 (2022). Copyright 2022 Author(s), licensed under a Creative Commons Attribution (CC BY) license.

(i.e., $\mathbf{X}_l \in \Gamma \forall 1 \leq l \leq N_L$, where N_L denotes the total number of Lagrangian markers).

The following describes the formulation of the volume force adopted in the framework of the predictor–corrector method used to solve the Navier–Stokes equations. The predictor step consists of explicitly marching the solution in time to obtain an intermediate velocity $\mathbf{u}^*$. To keep the description compact, the procedure is explained based on the simple explicit Euler scheme. The final formulation relies on a three-substeps low-storage Runge–Kutta scheme of second-order accuracy

$$\rho \frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = \mathbf{rhs}^n. \quad (2)$$

Here, $\mathbf{rhs}^n$ is a short-hand for the summation of the convective, diffusive, and pressure terms at time step t^n . When considering the immersed boundary force term appearing in Eq. (1), Eq. (2) becomes

$$\rho \frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = \mathbf{rhs}^n + \mathbf{f}^n. \quad (3)$$

One way to find the force $\mathbf{f}^n$ that would directly set the intermediate velocity to the desired velocity $\mathbf{u}_d$ on the fluid–solid interface is to simply write

$$\mathbf{f}^n = \rho \frac{\mathbf{u}_d - \mathbf{u}^n}{\Delta t} - \mathbf{rhs}^n. \quad (4)$$

However, this will only hold true if the Eulerian and Lagrangian locations are coincident. When dealing with time-dependent immersed boundaries, it is often the case that the fluid–solid interface is not conforming to the cell centers of the fluid grid. Here, the distinction previously made between the Eulerian and Lagrangian positions becomes necessary.

Fadlun, Verzicco, and Orlandi¹⁹ suggested three interpolation procedures with an ascending order of accuracy when this direct forcing approach is used. The first is to assign the force in Eq. (4) to all Eulerian points closest to the immersed boundary as if they are coincident to it. This results in a diffused interface rather than a sharp one. The second interpolation procedure is an improved version of the first, where a volume-fraction weighting is considered to scale the forcing applied to the momentum cells closest to the boundary. The last proposition is to linearly interpolate the velocity of the immersed boundary $\mathbf{u}_d$ to the Eulerian points closest to the boundary to obtain $\bar{\mathbf{u}}_d$ and substitute it in Eq. (4) instead of $\mathbf{u}_d$. However, Uhlmann²⁰ reported that the use of this direct forcing approach might cause serious oscillations of the hydrodynamic forces due to inadequate smoothing.

In the direct forcing scheme proposed by Uhlmann,²⁰ the force term is evaluated at the Lagrangian force points instead

$$\mathbf{F}^n = \rho \frac{\mathbf{U}_d - \mathbf{U}^n}{\Delta t} - \mathbf{RHS}^n \quad \forall \quad \mathbf{X}_l, \quad (5)$$

with $\mathbf{U}_d$ being the velocity at the fluid–solid interface. This velocity is to be explicitly prescribed by the rigid-body motion of the moving paddle considered in this work. In Eq. (5), the two unknown terms (i.e., $\mathbf{U}^n$ and $\mathbf{RHS}^n$) can both be combined into a single unknown, namely, the intermediate velocity $\mathbf{U}^*$ at a Lagrangian location

$$\mathbf{U}^* = \mathbf{U}^n + \frac{\Delta t}{\rho} \mathbf{RHS}^n. \quad (6)$$

An Eulerian analogue to Eq. (6) exists in the projection method used in this work [by solving Eq. (2) for $\mathbf{u}^*$]. Thus, by solving Eq. (6) for $\mathbf{U}^n$ and substituting it into Eq. (5), the following expression of the Lagrangian force results:

$$\mathbf{F}^n = \rho \frac{\mathbf{U}_d - \mathbf{U}^*}{\Delta t} \quad \forall \quad \mathbf{X}_l. \quad (7)$$

In order to couple the fluid and the solid phases, one has to conveniently transfer quantities between the two frames of reference. Therefore, the original IB method introduced by Peskin¹⁰ depending on the use of a bounded continuous approximation of the Dirac delta function δ_h (also referred to as the regularized delta function) is employed. The transfer equations for a uniform Cartesian grid g_h of grid size h read

$$\mathbf{U}^*(\mathbf{X}_l) = \sum_{i,j,k \in g_h} \mathbf{u}^*(\mathbf{x}_{i,j,k}) \delta_h(\mathbf{x}_{i,j,k} - \mathbf{X}_l) \Delta v_{i,j,k} \quad \forall 1 \leq l \leq N_L, \quad (8)$$

$$\mathbf{f}(\mathbf{x}_{i,j,k}) = \sum_{l=1}^{N_L} \mathbf{F}(\mathbf{X}_l) \delta_h(\mathbf{x}_{i,j,k} - \mathbf{X}_l) \Delta V_l \quad \forall \mathbf{x}_{i,j,k} \in g_h, \quad (9)$$

where $\Delta v_{i,j,k}$ is the volume of the Eulerian cell (i, j, k) , and ΔV_l is the volume associated with the Lagrangian force point (l) . The operation performed by Eq. (8) is called the interpolation operation $\mathcal{I}(\mathbf{u}^*)$, owing to the fact that it computes a Lagrangian quantity as a weighted average of the neighboring Eulerian quantities [see Fig. 2(a)]. Contrarily, Eq. (9) is referred to as the spreading operation $\mathcal{S}(\mathbf{F})$, where the force is spread out from a Lagrangian force point to the Eulerian points in its vicinity [see Fig. 2(b)]. Hence, the steps of the predictor–corrector method accounting for the immersed boundary can be summarized by the following equations:

$$1. \quad \mathbf{u}^* = \mathbf{u}^n + \frac{\Delta t}{\rho} \mathbf{rhs}^n \quad \forall \mathbf{x}_{i,j,k} \in g_h, \quad (10)$$

$$2. \quad \mathbf{U}^* = \mathcal{I}(\mathbf{u}^*) \quad \forall \mathbf{X}_l \in \Gamma, \quad (11)$$

$$3. \quad \mathbf{F} = \rho \frac{\mathbf{U}_d - \mathbf{U}^*}{\Delta t} \quad \forall \mathbf{X}_l \in \Gamma, \quad (12)$$

$$4. \quad \mathbf{f} = \mathcal{S}(\mathbf{F}) \quad \forall \mathbf{x}_{i,j,k} \in g_h, \quad (13)$$

$$5. \quad \mathbf{u}^{**} = \mathbf{u}^* + \frac{\Delta t}{\rho} \mathbf{f} \quad \forall \mathbf{x}_{i,j,k} \in g_h, \quad (14)$$

$$6. \quad \Delta p' = \frac{\rho}{\Delta t} \nabla \cdot \mathbf{u}^{**} \quad \forall \mathbf{x}_{i,j,k} \in g_h, \quad (15)$$

$$7. \quad \mathbf{u}^{n+1} = \mathbf{u}^n - \frac{\Delta t}{\rho} \nabla p' \quad \forall \mathbf{x}_{i,j,k} \in g_h. \quad (16)$$

Compared to a classic predictor–corrector scheme, steps 2 to 5 have been added and lead to a second intermediate velocity $\mathbf{u}^{**}$. That builds the basis for the right-hand side of the Poisson equation (15) for the pressure correction p' (step 6) and the final determination of the new velocity $\mathbf{u}^{n+1}$ (step 7).

The regularized delta function considered is that defined by Roma, Peskin, and Berger²⁴ (see Fig. 3), and it involves three grid points in each coordinate direction

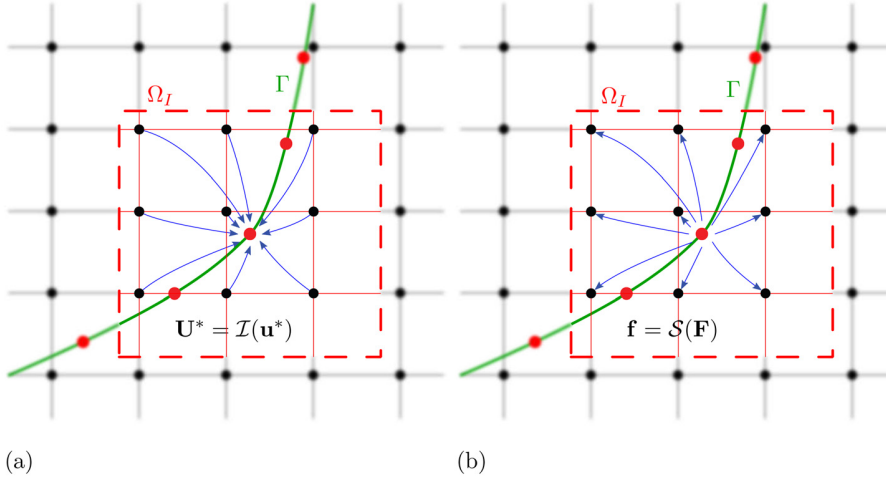


FIG. 2. (a) Interpolation of Eulerian velocities and (b) spreading of a Lagrangian force.

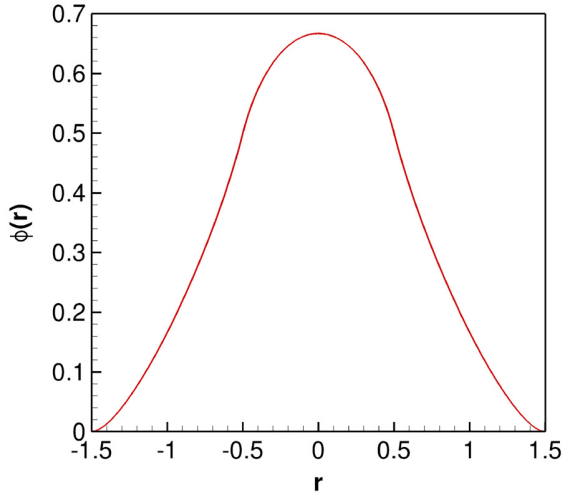


FIG. 3. Regularized delta function by Roma, Peskin, and Berger.²⁴

$$\delta_d(r) = \frac{1}{d} \phi_d(r)$$

$$= \frac{1}{d} \begin{cases} \frac{1}{3} (1 + \sqrt{-3r^2 + 1}), & \text{if } |r| \leq \frac{1}{2}, \\ \frac{1}{6} (5 - 3|r| - \sqrt{-3(1 - |r|)^2 + 1}), & \text{if } \frac{1}{2} \leq |r| \leq \frac{3}{2}, \\ 0, & \text{otherwise,} \end{cases} \quad (17)$$

where the subscript d is the dilation parameter ($d = h$ for uniform grids). It defines the dimensions of the kernel's support Ω_b , and the parameter r is the distance between an Eulerian and a Lagrangian coordinate normalized by the dilation parameter [$r = (x - X)/d$].

The smoothing kernel is continuous, of compact support, and designed to satisfy a set of discrete moment conditions (the zeroth and first moments)

$$\sum_{i,j,k \in g_h} \delta_h(\mathbf{x}_{i,j,k} - \mathbf{X}_l) \Delta v_{i,j,k} = 1, \quad (18)$$

$$\sum_{i,j,k \in g_h} (\mathbf{x}_{i,j,k} - \mathbf{X}_l) \delta_h(\mathbf{x}_{i,j,k} - \mathbf{X}_l) \Delta v_{i,j,k} = 0. \quad (19)$$

These discrete analogues of the basic properties of the Dirac delta function ensure the integral conservation of the force and moment in the spreading operation

$$\begin{aligned} \sum_{i,j,k \in g_h} \mathbf{f}(\mathbf{x}_{i,j,k}) \Delta v_{i,j,k} &= \sum_{l=1}^{N_L} \mathbf{F}(\mathbf{X}_l) \left(\sum_{i,j,k \in g_h} \delta_h(\mathbf{x}_{i,j,k} - \mathbf{X}_l) \Delta v_{i,j,k} \right) \Delta V_l \\ &= \sum_{l=1}^{N_L} \mathbf{F}(\mathbf{X}_l) \Delta V_l, \end{aligned} \quad (20)$$

$$\begin{aligned} \sum_{i,j,k \in g_h} \mathbf{x}_{i,j,k} \times \mathbf{f}(\mathbf{x}_{i,j,k}) \Delta v_{i,j,k} &= \sum_{l=1}^{N_L} \left(\sum_{i,j,k \in g_h} \mathbf{x}_{i,j,k} \delta_h(\mathbf{x}_{i,j,k} - \mathbf{X}_l) \Delta v_{i,j,k} \right) \times \mathbf{F}(\mathbf{X}_l) \Delta V_l \\ &= \sum_{l=1}^{N_L} \mathbf{X}_l \times \mathbf{F}(\mathbf{X}_l) \Delta V_l. \end{aligned} \quad (21)$$

The wind tunnel geometry simulated in this work requires a stretched grid that clusters the grid points close to walls in order to accurately capture the boundary layer. The grid is therefore Cartesian non-uniform. For the sake of writing the equations in a compact manner, in the following, the shift between an Eulerian and a Lagrangian coordinate (e.g., $x - X$) shall be referred to by the subscript "s" (i.e., $x_s = x - X$, $y_s = y - Y$, and $z_s = z - Z$).

In 1 D, the non-uniformity of the grid is accounted for by defining a modified window function according to Pinelli *et al.*²¹

$$\tilde{\delta}_d(x_s) = \sum_{l=1}^n b_l(X, d) x_s^{(l-1)} \delta_d(x_s), \quad (22)$$

where $b_l(X, d)$ are the unknown polynomial coefficients, and their number n is to be defined later. The unknown polynomial coefficients are determined by imposing a set of modified moment conditions

$$\tilde{m}_{(i-1)}(X) = \int_{\Omega_l} x_s^{(i-1)} \tilde{\delta}_d(x_s) dx = \delta_{i1}, \quad (23)$$

where δ_{ij} is the Kronecker delta. By substituting Eq. (22) in Eq. (23), one can write the modified moment conditions in terms of the original moments

$$\begin{aligned} \tilde{m}_{(i-1)}(X) &= \int_{\Omega_l} x_s^{(i-1)} \sum_{l=1}^n b_l(X, d) x_s^{(l-1)} \delta_d(x_s) dx \\ &= \sum_{l=1}^n b_l(X, d) \int_{\Omega_l} x_s^{(i+l-2)} \delta_d(x_s) dx \\ &= \sum_{l=1}^n b_l(X, d) m_{(i+l-2)}(X) = \delta_{i1}. \end{aligned} \quad (24)$$

The number of the required polynomial coefficients in Eq. (22) (i.e., n) primarily depends on the required number of the reproducing moment conditions to be satisfied, the dimensionality of the problem, and the order of the underlying Eulerian spatial discretization scheme.²¹ In 3D, the linear combinations of $\{1, x, y, z\}$ guarantees the integral conservation of the force and moment [i.e., Eqs. (20) and (21)]. Since the governing equations are discretized in space by a second-order accurate scheme (i.e., surface and volume integrals are approximated by the midpoint rule), the modified window function then reads

$$\begin{aligned} \tilde{\delta}_{\xi, \eta, \zeta}(x_s, y_s, z_s) &= [b_1 + b_2 x_s + b_3 y_s + b_4 z_s \\ &\quad + b_5 (x_s y_s) + b_6 (y_s z_s) + b_7 (z_s x_s) \\ &\quad + b_8 x_s^2 + b_9 y_s^2 + b_{10} z_s^2] \delta_{\xi, \eta, \zeta}(x_s, y_s, z_s), \end{aligned} \quad (25)$$

where the dependency of the coefficients can be expressed by $b_l = b_l(X, Y, Z, \xi, \eta, \zeta)$, $l = 1, \dots, 10$, and the original window function is defined as the product

$$\delta_{\xi, \eta, \zeta}(x_s, y_s, z_s) = \delta_{\xi}(x_s, y_s, z_s) \delta_{\eta}(x_s, y_s, z_s) \delta_{\zeta}(x_s, y_s, z_s), \quad (26)$$

with the subscripts ξ , η , and ζ being the dilatation parameters in each coordinate direction. Hence, 10 modified moment conditions need to be satisfied in order to evaluate the unknown polynomial coefficients. The continuous form of these conditions read

$$\begin{aligned} \tilde{m}_{\mathbf{i}(h), \mathbf{j}(h), \mathbf{k}(h)}(X, Y, Z) \\ = \int_{\Omega_l} x_s^{\mathbf{i}(h)} y_s^{\mathbf{j}(h)} z_s^{\mathbf{k}(h)} \sum_{l=1}^{n=10} b_l x_s^{l(l)} y_s^{l(l)} z_s^{l(l)} \delta_{\xi, \eta, \zeta}(x_s, y_s, z_s) dx dy dz = \delta_{h1}, \end{aligned} \quad (27)$$

where the arguments $h, l = 1, \dots, 10$ and the vectors $\mathbf{i}$, $\mathbf{j}$, and $\mathbf{k}$ are defined in such a way that the coordinate shifts (x_s , y_s and z_s) within the summation are raised to their respective powers according to Eq. (25)

$$\mathbf{i} = [0, 1, 0, 0, 1, 0, 1, 2, 0, 0], \quad (28)$$

$$\mathbf{j} = [0, 0, 1, 0, 1, 1, 0, 0, 2, 0], \quad (29)$$

$$\mathbf{k} = [0, 0, 0, 1, 0, 1, 1, 0, 0, 2]. \quad (30)$$

Further simplification yields

$$\begin{aligned} \tilde{m}_{\mathbf{i}(h), \mathbf{j}(h), \mathbf{k}(h)}(X, Y, Z) \\ = \sum_{l=1}^{n=10} b_l \int_{\Omega_l} x_s^{\mathbf{i}(h)+\mathbf{i}(l)} y_s^{\mathbf{j}(h)+\mathbf{j}(l)} z_s^{\mathbf{k}(h)+\mathbf{k}(l)} \delta_{\xi, \eta, \zeta}(x_s, y_s, z_s) dx dy dz \\ = \sum_{l=1}^{n=10} b_l m_{\mathbf{i}(h)+\mathbf{i}(l), \mathbf{j}(h)+\mathbf{j}(l), \mathbf{k}(h)+\mathbf{k}(l)}(X, Y, Z) = \delta_{h1}. \end{aligned} \quad (31)$$

Letting h and l refer to the row and column number, respectively, the resulting symmetric linear system of equations that needs to be solved for each Lagrangian location reads

$$\begin{bmatrix} \tilde{m}_{0,0,0} \\ \tilde{m}_{1,0,0} \\ \tilde{m}_{0,1,0} \\ \tilde{m}_{0,0,1} \\ \tilde{m}_{1,1,0} \\ \tilde{m}_{0,1,1} \\ \tilde{m}_{1,0,1} \\ \tilde{m}_{2,0,0} \\ \tilde{m}_{0,2,0} \\ \tilde{m}_{0,0,2} \end{bmatrix} \begin{bmatrix} m_{0,0,0} & m_{1,0,0} & m_{0,1,0} & m_{0,0,1} & m_{1,1,0} & m_{0,1,1} & m_{1,0,1} & m_{2,0,0} & m_{0,2,0} & m_{0,0,2} \\ & m_{2,0,0} & m_{1,1,0} & m_{1,0,1} & m_{2,1,0} & m_{1,1,1} & m_{2,0,1} & m_{3,0,0} & m_{1,2,0} & m_{1,0,2} \\ & & m_{0,2,0} & m_{0,1,1} & m_{1,2,0} & m_{0,2,1} & m_{1,1,1} & m_{2,1,0} & m_{0,3,0} & m_{0,1,2} \\ & & & m_{0,0,2} & m_{1,1,1} & m_{0,1,2} & m_{1,0,2} & m_{2,0,1} & m_{0,2,1} & m_{0,0,3} \\ & & & & m_{2,2,0} & m_{1,2,1} & m_{2,1,1} & m_{3,1,0} & m_{1,3,0} & m_{1,1,2} \\ & & & & & m_{0,2,2} & m_{1,1,2} & m_{2,1,1} & m_{0,3,1} & m_{0,1,3} \\ & & & & & & m_{2,0,2} & m_{3,0,1} & m_{1,2,1} & m_{1,0,3} \\ & & & & & & & m_{4,0,0} & m_{2,2,0} & m_{2,0,2} \\ & & & & & & & & m_{0,4,0} & m_{0,2,2} \\ & & & & & & & & & m_{0,0,4} \end{bmatrix} \begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ b_4 \\ b_5 \\ b_6 \\ b_7 \\ b_8 \\ b_9 \\ b_{10} \end{bmatrix} = \begin{bmatrix} 1 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}. \quad (32)$$

The entries of the symmetric matrix are numerically evaluated for each Lagrangian marker $\mathbf{X} = (X, Y, Z)$ by transforming the integral in Eq. (31) to a summation over the Eulerian cells falling within

the cage $\Omega_l(\xi, \eta, \zeta)$. First, the closest cell center $\mathbf{x}_{i,j,k} = (x_{i,j,k}, y_{i,j,k}, z_{i,j,k})$ is sought, and two of its neighbors in each direction are identified (i.e., $\mathcal{S}_l = \{\mathbf{x}_{i+l,j,k}, \mathbf{x}_{i-l,j,k}\}$ for $l = -1, 0, 1$). Maximum and

minimum local grid spacings in each direction are then evaluated according to Pinelli *et al.*²¹

$$\begin{aligned} h_x^+(\mathbf{X}) &= \max\{|x_{i,j,k} - x_{i-1,j,k}|\}, x_{i,j,k}, x_{i-1,j,k} \in \mathcal{S}_I, \\ h_x^-(\mathbf{X}) &= \min\{|x_{i,j,k} - x_{i-1,j,k}|\}, x_{i,j,k}, x_{i-1,j,k} \in \mathcal{S}_I, \\ h_y^+(\mathbf{X}) &= \max\{|y_{i,j,k} - y_{i,j-1,k}|\}, y_{i,j,k}, y_{i,j-1,k} \in \mathcal{S}_I, \\ h_y^-(\mathbf{X}) &= \min\{|y_{i,j,k} - y_{i,j-1,k}|\}, y_{i,j,k}, y_{i,j-1,k} \in \mathcal{S}_I, \\ h_z^+(\mathbf{X}) &= \max\{|z_{i,j,k} - z_{i,j,k-1}|\}, z_{i,j,k}, z_{i,j,k-1} \in \mathcal{S}_I, \\ h_z^-(\mathbf{X}) &= \min\{|z_{i,j,k} - z_{i,j,k-1}|\}, z_{i,j,k}, z_{i,j,k-1} \in \mathcal{S}_I, \end{aligned} \quad (33)$$

and the dilation parameters depending on the local grid spacing are defined as

$$\xi_I = \frac{5}{6} h_x^+(\mathbf{X}) + \frac{1}{6} h_x^-(\mathbf{X}), \quad (34)$$

$$\eta_I = \frac{5}{6} h_y^+(\mathbf{X}) + \frac{1}{6} h_y^-(\mathbf{X}), \quad (35)$$

$$\zeta_I = \frac{5}{6} h_z^+(\mathbf{X}) + \frac{1}{6} h_z^-(\mathbf{X}). \quad (36)$$

The discrete counterpart of the matrix entries can now be written as

$$\begin{aligned} m_{i,j,k}(X, Y, Z) \\ = \sum_{h,l,q \in \mathcal{S}_I} (x_{h,l,q} - X)^i (y_{h,l,q} - Y)^j (z_{h,l,q} - Z)^k \delta_{\xi_I, \eta_I, \zeta_I}(x_s, y_s, z_s) \Delta v_{h,l,q}. \end{aligned} \quad (37)$$

The symmetric linear system of equations [i.e., Eq. (32)] is solved using the “dsysv” subroutine provided by the linear algebra package LAPACK.²⁵

IV. NUMERICAL FRAMEWORK

A. Flow solver

The present study employs the in-house Navier–Stokes solver FASTEST-3D²⁶ to solve for the incompressible fluid flow. The governing equations are discretized in space using the finite-volume technique on a curvilinear, block-structured body-fitted grid with a collocated cell-centered variable arrangement. Parallelization is achieved by domain decomposition and explicit message passing based on the Message Passing Interface (MPI). The midpoint rule is used to approximate the surface and volume integrals and the required flow variables at cell faces are linearly interpolated from neighboring cell-centered values leading to a second-order spatial accuracy. The convective fluxes are approximated using the flux blending technique,^{27,28} combining 3% of a first-order accurate upwind scheme with 97% of a second-order accurate central scheme. To avoid pressure field oscillations, the momentum interpolation technique of Rhie and Chow²⁹ is employed for non-staggered grids to couple the pressure and velocity fields.

For the pressure–velocity coupling problem, a semi-implicit predictor–corrector scheme³⁰ with second-order accuracy is used. The momentum equations are first advanced in time using a low-storage multi-stage Runge–Kutta method to obtain an intermediate velocity. This velocity is known for not satisfying the continuity equation (i.e., not a solenoidal vector field). The corrector step then ensures mass conservation by solving a Poisson equation for the pressure correction

(here through an incomplete lower–upper (LU) decomposition method³¹) and projecting the coupled flow variables (i.e., velocities and pressure) to the space of divergence-free vector fields. This procedure provides second-order accuracy in both space and time. The large-eddy simulation (LES) technique is used to only resolve the large-scale turbulent flow field and model the small scales by a subgrid-scale (SGS) model. The standard Smagorinsky SGS model³² with the well-established constant $C_S = 0.1$ and van-Driest damping near solid walls is applied. Since the near-wall grid resolution is sufficient to resolve the viscous sublayer as will be shown in Sec. V, wall functions are not required. Instead, wall-resolved LES is carried out. To incorporate the immersed boundary method, steps 2 to 5 described in Sec. III are additionally carried out.

B. Lagrangian marker tracking

The tracking of the Lagrangian markers is performed in the computational space (c-space) obtained by transforming the structured grid connecting the cell centers [i.e., z-grid in Fig. 4(a)] to an orthonormal grid. The time-consuming search process in the physical space (p-space) associated with finding the new computational cell containing the Lagrangian marker can therefore be avoided. The location of the Lagrangian marker in the c-space ξ_I [see Fig. 4(b)] is defined as the summation of the index triple of the cell containing it and a fractional offset as

$$\xi_I = [\xi_I, \eta_I, \zeta_I] = [i_I, j_I, k_I] + [\Delta \xi_I^+, \Delta \eta_I^+, \Delta \zeta_I^+]. \quad (38)$$

The use of such a definition is also advantageous for the efficient search of the nearest cell center to the Lagrangian marker required for defining the bounds of the regularized delta function (i.e., for finding $\mathbf{x}_{i,j,k}$). The index triples $[i_I, j_I, k_I]$ and $[\hat{i}_I, \hat{j}_I, \hat{k}_I]$ can be obtained by applying the integer operators “int” and “nint” provided by programming languages on the computational coordinates $[\xi_I, \eta_I, \zeta_I]$, respectively. The fractional offset can be solved for using Eq. (38) and its complement thereafter reads

$$\Delta \xi_I^- = 1 - \Delta \xi_I^+, \quad \Delta \eta_I^- = 1 - \Delta \eta_I^+, \quad \Delta \zeta_I^- = 1 - \Delta \zeta_I^+. \quad (39)$$

The fractional offset and its complement can be employed to write the transformation T of the marker’s position ξ_I to the p-space by means of the trilinear interpolation using the cell centered values $\mathbf{x}_{i,j,k}$ ³³

$$\begin{aligned} \mathbf{X}_I = T(\xi_I) &= \mathbf{x}_{i,j,k} [\Delta \xi_I^- \Delta \eta_I^- \Delta \zeta_I^-] + \mathbf{x}_{i+1,j,k} [\Delta \xi_I^+ \Delta \eta_I^- \Delta \zeta_I^-] \\ &+ \mathbf{x}_{i,j+1,k} [\Delta \xi_I^- \Delta \eta_I^+ \Delta \zeta_I^-] + \mathbf{x}_{i,j,k+1} [\Delta \xi_I^- \Delta \eta_I^- \Delta \zeta_I^+] \\ &+ \mathbf{x}_{i+1,j+1,k} [\Delta \xi_I^+ \Delta \eta_I^+ \Delta \zeta_I^-] + \mathbf{x}_{i+1,j,k+1} [\Delta \xi_I^+ \Delta \eta_I^+ \Delta \zeta_I^+] \\ &+ \mathbf{x}_{i,j+1,k+1} [\Delta \xi_I^- \Delta \eta_I^+ \Delta \zeta_I^+] + \mathbf{x}_{i+1,j+1,k+1} [\Delta \xi_I^+ \Delta \eta_I^+ \Delta \zeta_I^+]. \end{aligned} \quad (40)$$

In addition to the visualization purpose of the immersed boundary, this interpolation is used to evaluate the Lagrangian location in physical coordinates required in Eqs. (8) and (9).

As previously mentioned in Sec. III, the immersed boundary (i.e., the paddle) is set into motion by a prescribed velocity $\mathbf{U}_d(t) = [U_d, V_d, W_d]$. This velocity is of course defined with respect to the p-space. In order to update the marker’s location in the c-space, one

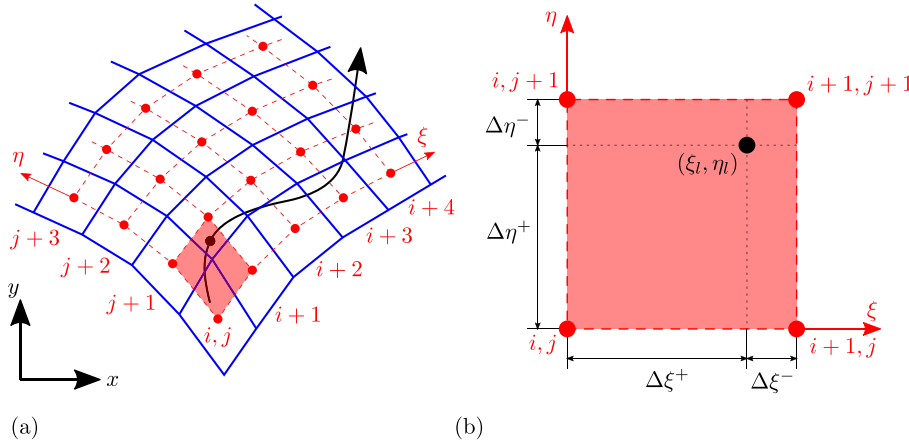


FIG. 4. (a) Two-dimensional p-space schematic sketch showing a computational grid (blue solid lines), cell centers (red circles), z-grid (red dashed lines), and Lagrangian marker's trajectory (black solid line) and (b) corresponding c-space representation showing the coefficients of the trilinear interpolation.

has to first transform the prescribed velocity components to the c-space by

$$\frac{d\xi_l}{dt} = \left(\frac{\partial \xi}{\partial x} \right)_l U_d + \left(\frac{\partial \xi}{\partial y} \right)_l V_d + \left(\frac{\partial \xi}{\partial z} \right)_l W_d, \quad (41)$$

$$\frac{d\eta_l}{dt} = \left(\frac{\partial \eta}{\partial x} \right)_l U_d + \left(\frac{\partial \eta}{\partial y} \right)_l V_d + \left(\frac{\partial \eta}{\partial z} \right)_l W_d, \quad (42)$$

$$\frac{d\zeta_l}{dt} = \left(\frac{\partial \zeta}{\partial x} \right)_l U_d + \left(\frac{\partial \zeta}{\partial y} \right)_l V_d + \left(\frac{\partial \zeta}{\partial z} \right)_l W_d. \quad (43)$$

Hence, the location of a Lagrangian marker is updated by carrying out the temporal integration

$$\xi_l^{n+1} = \xi_l^n + \left(\frac{\partial \xi}{\partial x} \right)_l \int_{t^n}^{t^{n+1}} U_d dt + \left(\frac{\partial \xi}{\partial y} \right)_l \int_{t^n}^{t^{n+1}} V_d dt + \left(\frac{\partial \xi}{\partial z} \right)_l \int_{t^n}^{t^{n+1}} W_d dt, \quad (44)$$

$$\eta_l^{n+1} = \eta_l^n + \left(\frac{\partial \eta}{\partial x} \right)_l \int_{t^n}^{t^{n+1}} U_d dt + \left(\frac{\partial \eta}{\partial y} \right)_l \int_{t^n}^{t^{n+1}} V_d dt + \left(\frac{\partial \eta}{\partial z} \right)_l \int_{t^n}^{t^{n+1}} W_d dt, \quad (45)$$

$$\zeta_l^{n+1} = \zeta_l^n + \left(\frac{\partial \zeta}{\partial x} \right)_l \int_{t^n}^{t^{n+1}} U_d dt + \left(\frac{\partial \zeta}{\partial y} \right)_l \int_{t^n}^{t^{n+1}} V_d dt + \left(\frac{\partial \zeta}{\partial z} \right)_l \int_{t^n}^{t^{n+1}} W_d dt. \quad (46)$$

The time-independent spatial metric coefficients (here for a fixed grid) appearing in Eqs. (44)–(46) are the entries of the Jacobian of the p-space to the c-space coordinate transformation $\mathbf{J}$ and are evaluated at the Lagrangian location. For this purpose, the CZ+ tracking scheme proposed by Schäfer and Breuer³³ is employed. First, the Jacobian of the inverse coordinate transformation $\mathbf{J}^{-1}$ (i.e., c-space to p-space) is analytically evaluated by differentiating Eq. (40). The first column of $\mathbf{J}^{-1}$ at a position ξ_l reads

$$\begin{aligned} \mathbf{J}_1^{-1}(\xi_l) = \frac{\partial \mathbf{X}_l}{\partial \xi_1} = \frac{\partial T(\xi_l)}{\partial \xi_1} = & \Delta \eta_l^- \Delta \zeta_l^- (\mathbf{x}_{i+1,j,k} - \mathbf{x}_{i,j,k}) \\ & + \Delta \eta_l^+ \Delta \zeta_l^- (\mathbf{x}_{i+1,j+1,k} - \mathbf{x}_{i,j+1,k}) \\ & + \Delta \eta_l^- \Delta \zeta_l^+ (\mathbf{x}_{i+1,j,k+1} - \mathbf{x}_{i,j,k+1}) \\ & + \Delta \eta_l^+ \Delta \zeta_l^+ (\mathbf{x}_{i+1,j+1,k+1} - \mathbf{x}_{i,j+1,k+1}). \end{aligned} \quad (47)$$

Following the evaluation of $\mathbf{J}_m^{-1}(\xi_l)$ where $m = 1, 2$, and 3 and inverting it, the spatial metrics in Eqs. (44)–(46) are obtained.

V. DESCRIPTION OF THE SETUP

The aim of the present setup is to create a numerical counterpart of the experimental wind gust generator of Wood, Breuer, and Neumann¹ (i.e., the paddle). This shall facilitate a deeper analysis of the complex flow generated by the paddle as all flow field variables (i.e., velocities and pressure) are solved for throughout the entire computational domain. The complex wind tunnel geometry depicted in Fig. 1 is simplified to model the nozzle, the ground plate and the moving paddle as shown in Fig. 5. The geometry of the nozzle of the wind tunnel is simplified by neglecting its contraction ratio. It is replaced by a duct of length $l = 1.5$ m but with a rectangular cross-section which is identical to the outlet of the nozzle in the experimental setup ($h = 0.375$ m and $w = 0.5$ m). A uniform horizontal velocity field of $u_{\text{inlet}}^{\text{steady}} = 4.914$ m/s is defined at the inlet patch. The flow and especially

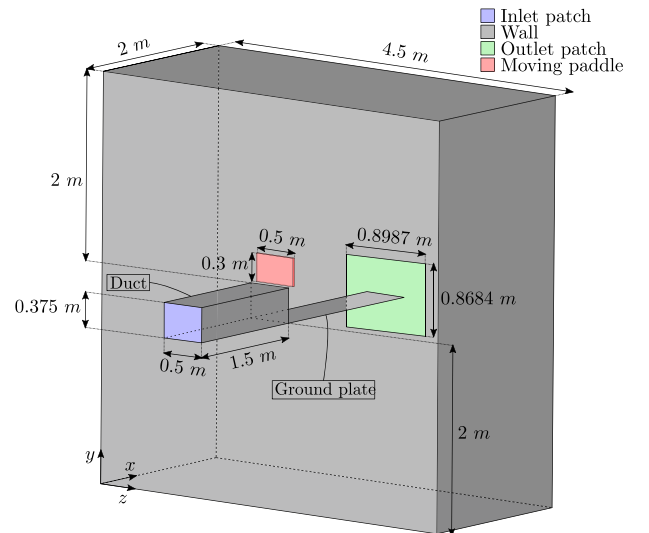


FIG. 5. Geometry of the test case and definition of the boundary conditions.

the boundary layers at the walls develop within the duct achieving a free-stream velocity $u_\infty = 5.14$ m/s at its outlet (identical to that of the experimental setup). No-slip and impermeability boundary conditions are applied on the walls of the duct, the far-field walls and the ground plate. A convective boundary condition is defined at the outlet patch (modeling the receiver of the wind tunnel) with a convective velocity of $u_{conv} = u_{inlet}^{steady}$.

The Eulerian grid is a block-structured Cartesian grid consisting of 36.6×10^6 control volumes. The total number of control volumes and their distribution is motivated by the following considerations. The geometric block-structure consisting of 10 blocks is split up into a parallel block-structure with 340 blocks. That allows a parallel execution on 340 cores with a load balancing efficiency of 100%. Grid refinement near relevant rigid walls (i.e., duct walls and ground plate) is essential to capture the significant viscous effects leading to the formation of thin boundary layers. The maximum first cell size positioning the first node (i.e., its cell center) within the viscous sublayer (i.e., $y^+ < 5$) is therefore sought. The hydraulic diameter of the rectangular duct (i.e., $D_h = 4wh/2(w+h) = 0.4286$) is used as a characteristic length to calculate the Reynolds number $Re = \rho D_h u_\infty / \mu = 147,701$. The Darcy friction factor found by Moody³⁴ is

$$f = 0.0055 \left[1 + \left(2 \times 10^{-4} \frac{k}{D_h} + \frac{10^6}{Re} \right)^{\frac{1}{2}} \right], \quad (48)$$

where k is the pipe's effective roughness height. Assuming a smooth pipe (i.e., $k=0$), the skin friction coefficient is evaluated by $c_f = f/4 = 3.97 \times 10^{-3}$. Hence, to resolve the near-wall region accurately, one could approximate the maximum height of the first grid cell off the wall (Δy) resulting in $y_{cc}^+ = 2$ at the cell center by $\Delta y/D_h = (4/Re) \cdot \sqrt{2/c_f} \approx 6 \times 10^{-4}$ (i.e., $\Delta y \approx 2.6 \times 10^{-4}$ m). In order to reduce the computational costs, the grid is mildly stretched in the off wall directions with an expansion factor of 1.04. The same stretching factor is also applied in the streamwise direction. Starting with the finest grid at the outlet of the nozzle, where the paddle is located, the grid is mildly coarsened in the upstream and downstream directions. Based on the free-stream velocity, the stability of the explicit time integration scheme is ensured by choosing a time step size of $\Delta t = 2 \times 10^{-5}$ s leading to a Courant–Friedrichs–Lewy number less than 0.4.

A. Distribution of Lagrangian points describing the paddle

The geometry of the paddle is simplified by a parallelepiped of right angles with $L_2 = 0.3$ m and $L_3 = 0.5$ m. This simplification is assumed to have no effect on the results as in both cases (i.e., experiment and simulation) a rectangular cross-section will penetrate the flow exiting the nozzle. The moving paddle of thickness $L_1 = 3 \times 10^{-3}$ m is placed 0.015 m away from the outlet of the nozzle.

Two distributions of the Lagrangian markers used for the description of the paddle are investigated, namely, a uniform distribution and a one-dimensional non-uniform distribution described below. The uniformly distributed set of Lagrangian markers considered is shown in Fig. 6. The spacing between the markers is defined (in each coordinate direction) based on the minimum local Eulerian grid spacing (i.e., $\Delta x_{i,min}$ for $i = 1, 2$, and 3) that the paddle will pass by during its closing and opening stroke. This is to avoid any potential fluid penetration through the surface of the paddle arising from inadequate Eulerian

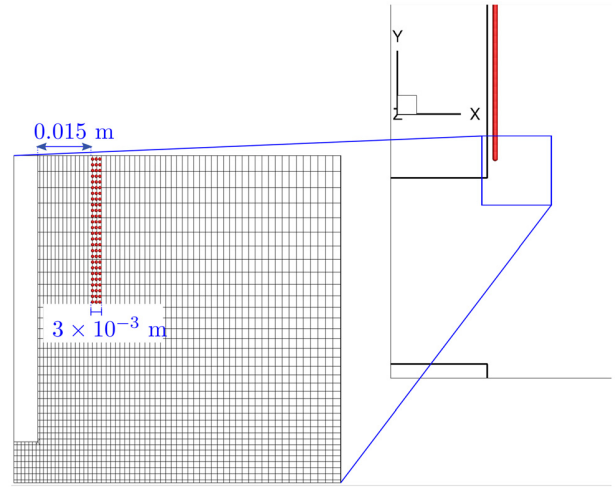


FIG. 6. Zoom of the nozzle outlet with the Eulerian grid and the uniformly distributed Lagrangian markers.

cells coverage by the Lagrangian markers window function. Thus, the Lagrangian grid spacings are defined based on the following two steps. The minimum local Eulerian grid spacing in each coordinate direction (here, it is the first cell size at a no-slip wall for y and z coordinates) is determined and used to evaluate the number of Lagrangian markers in each respective direction

$$N_i = \frac{L_i - \Delta x_{i,min}}{\Delta x_{i,min}} + 1. \quad (49)$$

The outcome of Eq. (49) (N_i) can be a decimal number. Since the number of Lagrangian markers can only be a natural number, Lagrangian grid spacings are therefore defined by rounding N_i up to the nearest integer as

$$\Delta X_i = \frac{L_i}{\lceil N_i \rceil}. \quad (50)$$

Based on the Eulerian grid employed, the number of Lagrangian markers in each direction is $\lceil N_1 \rceil = 3$, $\lceil N_2 \rceil = 1154$, and $\lceil N_3 \rceil = 1924$ (i.e., $N_i = \lceil N_1 \rceil \lceil N_2 \rceil \lceil N_3 \rceil = 6\,660\,888$). The corresponding Lagrangian grid spacings are $\Delta X = 1 \times 10^{-3}$ m and $\Delta Y = \Delta Z = 2.59 \times 10^{-4}$ m setting the volume associated with each Lagrangian force point to $\Delta V = \Delta X \Delta Y \Delta Z = 6.75 \times 10^{-11}$ m³.

The simulation is seen to undergo a remarkable slowdown (8 times slower) due to the abundance of Lagrangian markers resulting from the uniform spatial distribution. This is primarily a consequence of the uneven distribution of Lagrangian markers among the parallel processors. Lagrangian markers are predominantly shared by a maximum of 12 out of the total 340 processors throughout the simulation. For each processor containing Lagrangian markers, an additional computational load arises from the interpolation, spreading, and tracking processes. Consequently, the time taken to perform one time step on the heavily burdened processors is significantly increased leading to an adverse affect on the overall performance.

Therefore, an alternative given by the one-dimensional non-uniform distribution is examined. In this enhanced variant, the

distribution of the Lagrangian markers is left identical to the uniform distribution along the x and y axes. However, along the z axis, the distribution is defined in such a way that for each cell center a Lagrangian marker is assigned. In other words, Lagrangian markers are set to follow the grid stretching employed along the z axis. The reason for this choice lies in the fact that, during the paddle's opening and closing stroke, the distribution of Eulerian points along the z axis remains unchanged. This leads to a significantly reduced total number of Lagrangian markers of $N_L = 761\,640$ where $N_3 = 220$ and thus decreased by a factor of about 9. It is worth mentioning that with this approach the volume associated with the Lagrangian marker varies along the z axis which has to be taken into account in the algorithm.

B. Definition of the paddle's velocity

Wood, Breuer, and Neumann¹ investigated five different paddle motion patterns and analyzed their effects on the generated gusts. Each pattern is characterized by its own settings of the paddle's acceleration, velocity, and displacement during the downward and upward stroke. The main gust pattern used in their work (also denoted *Type 1* pattern) is considered in this study as it imitates the standard 1-cosine symmetric gust shape defined in the IEC-Standard²³ as the Extreme Coherent Gust (ECG).

A smooth (differentiable) analytical function is fitted to the vertical velocity curve defined by the experiment as shown in Fig. 7. The fitted velocity function found reads

$$V_d(t) = \begin{cases} -A_1(3\phi_1^2 - 2\phi_1^3), & \text{if } t_{s,1} = 0 \leq t < t_{e,1} = 0.07 \text{ s}, \\ -A_1, & \text{if } t_{e,1} \leq t < t_{s,2} = 0.16 \text{ s}, \\ A_2(3\phi_2^2 - 2\phi_2^3) - A_1, & \text{if } t_{s,2} \leq t < t_{e,2} = 0.33 \text{ s}, \\ A_2 - A_1, & \text{if } t_{e,2} \leq t < t_{s,3} = 0.438 \text{ s}, \\ (A_2 - A_1) & \\ -(A_2 - A_1)(3\phi_3^2 - 2\phi_3^3), & \text{if } t_{s,3} \leq t < t_{e,3} = 0.517 \text{ s}, \\ 0, & \text{otherwise,} \end{cases} \quad (51)$$

where $\phi_i = (t - t_{mid,i})/t_{w,i}$, $t_{mid,i} = (t_{e,i} + t_{s,i})/2$, $t_{w,i} = t_{e,i} - t_{s,i}$, $A_1 = 1.05$, and $A_2 = 2$. Like in the experiment, the offset displacement $\Delta y_{os} = 0.045 \text{ m}$ is the distance that the paddle needs to travel (when it is initially at its idle position) in order to start blocking the nozzle's outlet. This hinders the unwanted initial interaction between the paddle and the undisturbed free-stream flow and allows for the gradual buildup of the paddle's velocity preceding the interaction. The bell-shaped displacement pattern is then prescribed letting the paddle

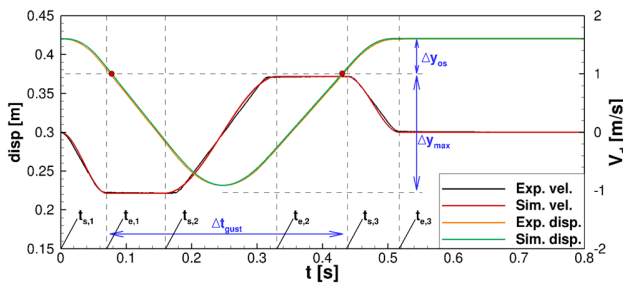


FIG. 7. Kinematic characteristics of the fitted *Type 1* paddle movement pattern.

block a maximum of $\Delta y_{max} = 0.1437 \text{ m}$ of the nozzle's outlet [i.e., a blocking ratio (BR) of 0.383] and have a total interaction time of $\Delta t_{gust} = 0.352 \text{ s}$ (i.e., the time span between the two red circles in Fig. 7). The paddle is eventually brought back to rest at its idle position. Note that the present values of Δy_{max} and Δt_{gust} slightly deviate from those given in Wood, Breuer, and Neumann.¹ The reason is that the latter corresponds to the prescribed values of the tooth belt axis in the experimental setup, whereas the present values better describe the real measured motion pattern. It is worth mentioning that in the simulation it is sufficient to prescribe the no-slip vertical velocity of the paddle as its displacement is a derived value found by Eqs. (44)–(46).

C. Definition of the inlet velocity variation

In Wood, Breuer, and Neumann,¹ an undershooting of the streamwise velocity below the free-stream velocity was observed at the nozzle's outlet during and after the paddle encounter. The authors explained this observation by conducting an additional closing stroke test (i.e., a sudden downward movement of the paddle without a subsequent upward movement). This undershooting phenomenon is attributed to the paddle acting as a bluff body blocking the fluid flow and thus causing high pressure losses due to the paddle in the fluid stream. Since the blower is not operated in a mode that allows to compensate these higher pressure losses appearing in the wind tunnel, less air is blown through the wind tunnel causing an undershoot below the free-stream velocity. Note that even a fully controlled operation of the blower might not be able to avoid this phenomenon owing to its long response time. Further opening stroke tests (i.e., a sudden upward movement of the paddle after staying in the fluid flow over a long time period) have clearly shown that the recovery time until the free-stream velocity is reached again, is much longer than Δt_{gust} . In order to account for this inlet velocity drop, Constant Temperature Anemometry (CTA) measurements recorded inside the nozzle during the gust generation process³⁵ are used to define a time-dependent inlet boundary condition as shown in Fig. 8. The inlet velocity function defined reads

$$u_{inlet}(t) = \begin{cases} u_{inlet}^{steady}, & \text{if } 0 \leq t < t_{s,1}^{in} = 0.076 \text{ s}, \\ u_{inlet}^{steady} - A_3(3\phi_1^2 - 2\phi_1^3), & \text{if } t_{s,1}^{in} \leq t < t_{e,1}^{in} = t_{s,2}^{in} = 0.3 \text{ s}, \\ (u_{inlet}^{steady} - A_3) + A_3(3\phi_2^2 - 2\phi_2^3), & \text{if } t_{s,2}^{in} \leq t < t_{e,2}^{in} = 1.588 \text{ s}, \\ u_{inlet}^{steady}, & \text{otherwise,} \end{cases} \quad (52)$$

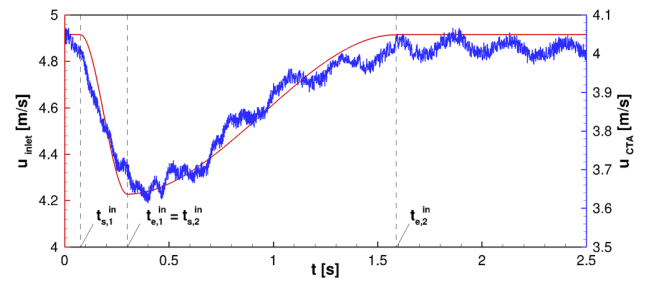


FIG. 8. Time-dependent inlet boundary condition and CTA measurements in the nozzle.³⁵

where ϕ_i is defined in a similar manner as in Eq. (51) and A_3 is set to 0.686. Note that a direct comparison of the amplitudes between the prescribed inlet velocity function and the CTA measurements is not meaningful, since the latter were carried out somewhere in the converging part of the wind tunnel nozzle.

VI. RESULTS

The results presented are mainly based on the simulation with the one-dimensional non-uniform distribution of Lagrangian markers. This is due to the fact that both alternatives deliver nearly the same results. Therefore, in order to save computational resources, the second alternative of Lagrangian markers distribution is primarily employed. A comparison of the results obtained by both distributions is carried out in the Appendix.

A. Flow field induced by the wind gust generator

In the following, results showing the evolution of the flow during the gust generation process by means of the moving paddle are provided. Figure 9 points out the time instants at which the fields in Figs. 10 and 11 are taken. These visualizations depict the streamwise and vertical velocity components as well as the pressure and the vorticity component in z -direction for a x - y cross-section in the center of the duct. At instant 1, the paddle is at its idle position, and the fields represent the initial conditions used for the simulations where the flow is initially allowed to fully develop for 10 s. Two flow phenomena are clearly visible. On the one hand, boundary layers develop along the top and bottom wall of the duct. For the latter, the development of the boundary layer continues along the ground plate. On the other hand, a free shear layer is formed in the region resulting from the horizontal continuation of the upper duct wall. At instant 2, the flow starts to deviate from its original stream as the paddle blocks 15.8% of the nozzle's outlet. This causes the streamwise velocity to accelerate and the vertical velocity to build up negative values below the paddle. The positive vertical velocity at the small gap between the nozzle's outlet and the paddle indicates fluid escaping from that region as well. While a pressure rise is observed in front of the paddle, a strong vortical structure with a pressure minimum in its center starts to form at the backside of the paddle as also visible in the vorticity distribution in Fig. 11(b). That is due to the fluid separation taking place at its bottom edge. At instant 3, the paddle continues its downstroke reaching a

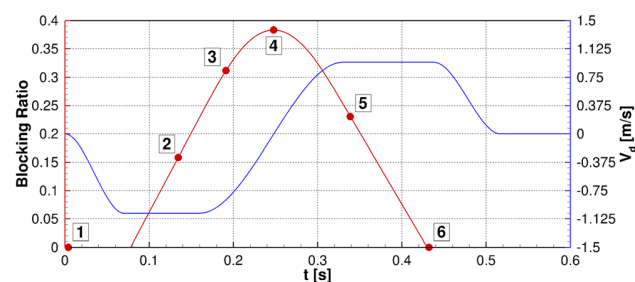


FIG. 9. Nozzle's blocking ratio (red line) annotated with visualization time instants and paddle's vertical velocity (blue line) during the downward and upward strokes. Instant 1: paddle at idle position, instant 2: 15.8% blocking ratio, instant 3: 31.2% blocking ratio, instant 4: 38.3% blocking ratio (maximum), instant 5: 22.0% blocking ratio, and instant 6: paddle just left nozzle's outlet.

blocking ratio of 31.2%. A higher streamwise velocity extending to the ground plate and higher pressure values extending upstream of the duct are recorded as the flow in the outlet is further contracted. Note that this pressure rise and the associated decrease in the streamwise velocity in front of the paddle is the reason for defining a transient inlet velocity function provided in Sec. V C. The size of the strong vortex originating from the bottom edge of the paddle increases, and the vortical structure is convected downstream. A new shear layer develops behind the paddle's edge which follows its motion. At instant 4, the paddle arrives at its maximum blocking ratio of 38.3%. Obviously, in the remaining outlet region above the ground plate, the streamwise velocity reaches high values which characterize the resulting strong horizontal gust. This describes the main effect to be achieved by the paddle motion. A side effect is that the pushed fluid stream also possesses a downward velocity component directed toward the ground plate. At this instant in time, the area unaffected by the shear layer reaches its minimum. The new shear layer developing at the paddle's bottom edge breaks up into a series of small vortices while the large vortex is further convected downstream. At instant 5, the paddle starts traveling back to its idle position (i.e., upward stroke) where its blocking ratio reduces to 22.0%. At this stage, the large vortical structure hits the ground plate and the strength of the generated gust already decreases again resulting in highly transient velocity fields. The approach of the vortex at the ground plate has certain consequences for the remaining part of the flow field. When the vortical structure reaches the plate, it lifts up the boundary layer on the ground plate. Thus, a small region of low streamwise velocity is observed close to the bottom wall, while at the same time, the vertical velocity is positive in this area pointing away from the wall. At instant 6, the paddle leaves the nozzle's outlet and the process of flow recovery to the initial state starts.

B. Spanwise characteristics of the generated gust

In the following, the spanwise characteristics of the gust are discussed. Given the results in Figs. 10 and 11 and the experimental analysis of Wood, Breuer, and Neumann,¹ the schematic sketch shown in Fig. 12 illustrates the regions formed during the gust generation process using the paddle. The zones are denoted the free shear layer region and the undisturbed gust region. The latter zone indicates the region where a stable gust forms with a good experimental reproducibility. The size of these regions strongly depends on the maximum displacement of the paddle where larger displacements lead to a smaller undisturbed gust region. For feasibility purposes, the origin of the coordinate system chosen for the Laser-Doppler Anemometer (LDA) measurements in Wood, Breuer, and Neumann¹ was shifted by an offset $\Delta x_{\text{LDA}} = 0.06$ m away from the nozzle's outlet. The shifted and normalized coordinate system (normalized by the width of the nozzle's outlet) shall now be employed to describe different positions in the following. Figures 13 and 14 depict the evolution of velocity components at the x -planes $x/w = 0$ and $x/w = 0.2$, respectively. The latter cross-section is specifically chosen, since it corresponds to the beginning of the region typically used to place an FSI model in the test section. Although the edges of the paddle are illustrated on the same plane, it is important to note that they are positioned behind the plane. They are projected both to enhance visualization and to provide a reference for tracking the position of the paddle.

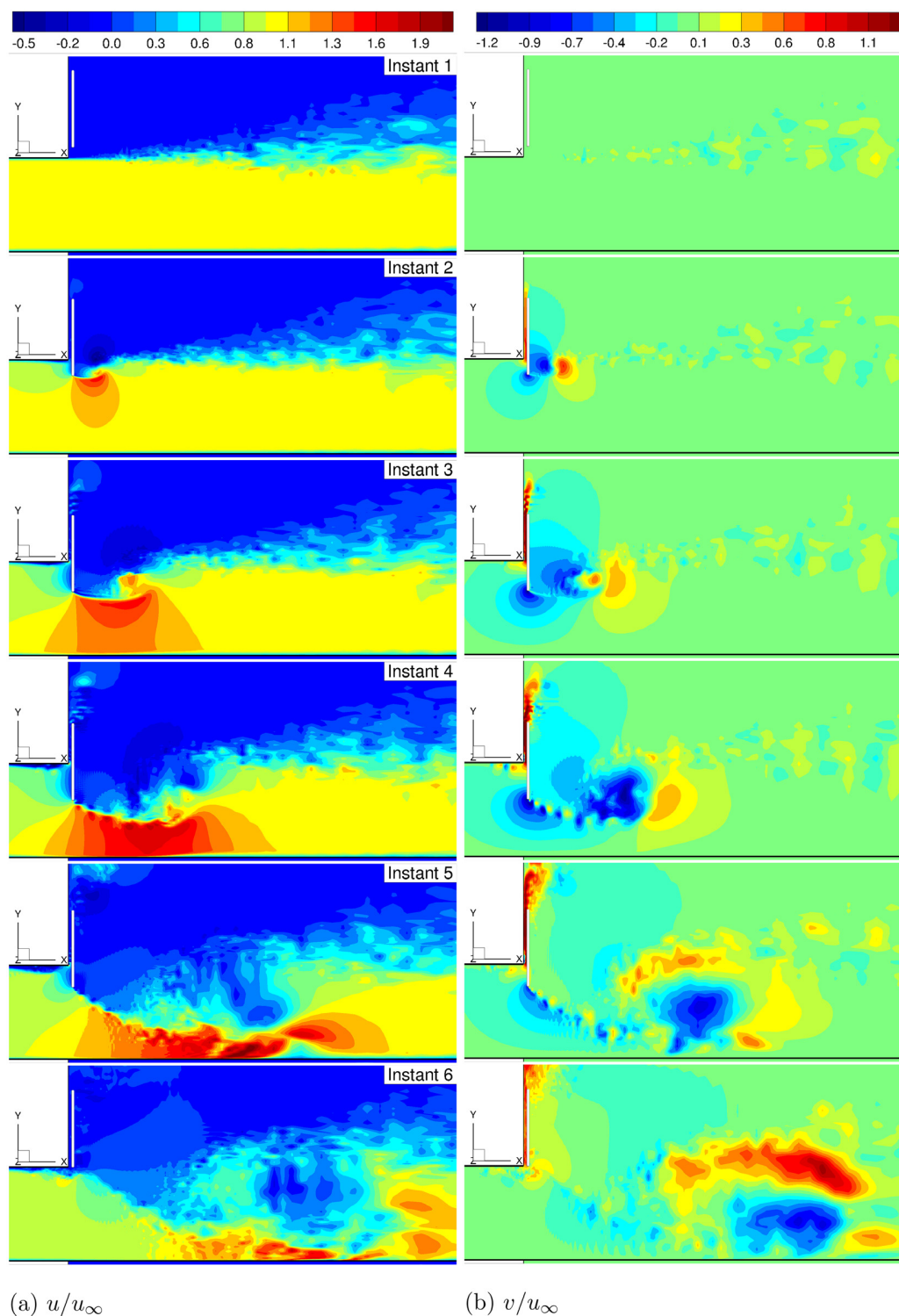


FIG. 10. Snapshots of the (a) streamwise velocity and the (b) vertical velocity during the paddle's downward and upward strokes. Instants correspond to those depicted in Fig. 9.

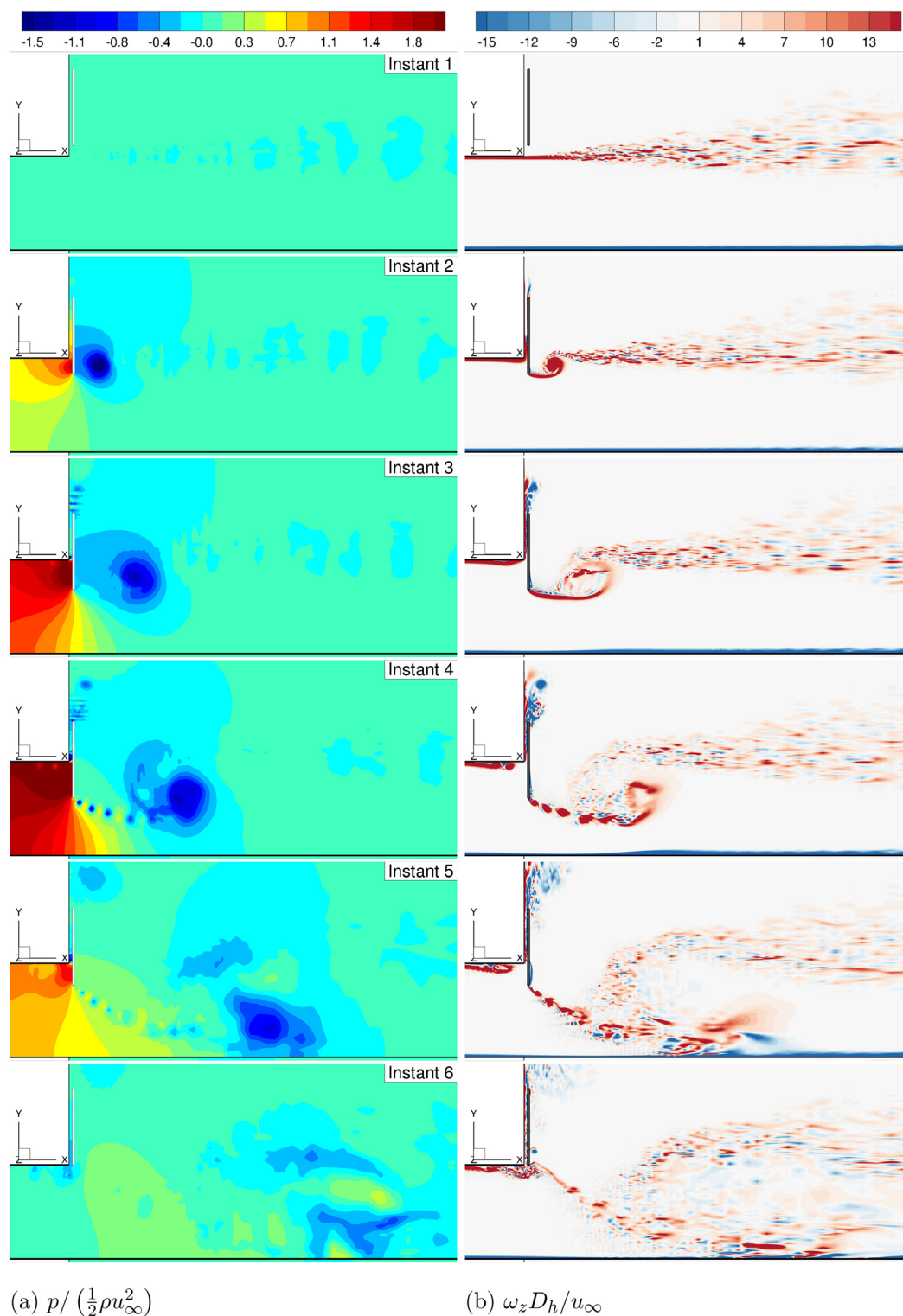


FIG. 11. Snapshots of the (a) pressure and the (b) z-vorticity ω_z during the paddle's downward and upward strokes. Instants correspond to those depicted in Fig. 9.

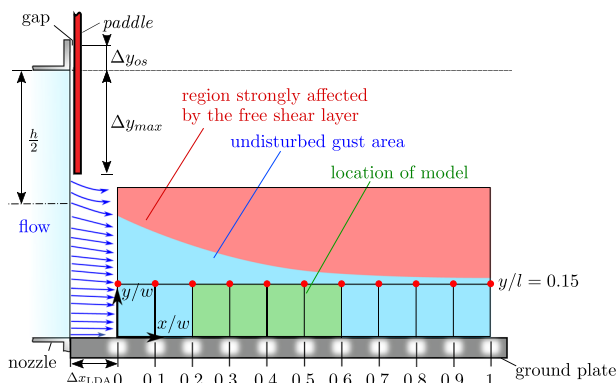


FIG. 12. Schematic sketch of flow regions and the usable space for gust investigations (adjusted with permission from Wood *et al.*, *J. Wind Eng. Ind. Aerodyn.* **230**, 105170 (2022). Copyright 2022 Author(s), licensed under a Creative Commons Attribution (CC BY) license.

First, the spanwise characteristics of the gust at the x -plane $x/w = 0$ are discussed (see Fig. 13). At instant 1, the flow is predominantly influenced by the free-stream velocity, which experiences a rapid decline to zero in the proximity of the duct's walls. Throughout this region, both the y - and z -components of the velocity are zero as expected since presently no inflow turbulence is taken into account. Advancing to instant 2, as the paddle begins its penetration of the nozzle's outlet, a symmetric velocity pattern emerges for all three components. The streamwise velocity exhibits a nearly uniform increase along the z axis beneath the paddle. However, closer to the ground plate, this uniformity is disturbed, revealing slightly higher streamwise velocities at the free ends (i.e., the left and right edges). This behavior can be attributed to the fluid's tendency to follow the path of least resistance. Simultaneously, a negative vertical velocity emerges due to the downward movement of the paddle and the formation of a strong vortical structure on the backside of the paddle. Except at the free ends the contour lines of the vertical velocity are approximately parallel to the ground plate. Additionally, a spanwise velocity component initiates directing the fluid toward the left and right edges of the ground plate. A careful examination of the y - and z -velocity fields in the lower left and right corners of the paddle reveals the development of clockwise and counterclockwise rotating vortices, respectively. Note that an additional simulation with a paddle elongated in both positive and negative z -directions has shown that the relevant flow field in the measurement region of the wind tunnel is only marginally influenced by this extension. At instants 3 and 4, all velocity components persistently increase in magnitude within the region unobstructed by the paddle. A highly transient flow characterizes the paddle's left and right edges due to the fluid separation taking place. To provide a visual impression of the generated gust, Fig. 15 depicts an isometric view based on the iso-surface of the streamwise velocity ($u/u_\infty = 1.38$). At instant 5, as the paddle is returning to its idle position, an overall reduction in the streamwise velocity is observed. However, this time, the maximum streamwise velocity is not positioned directly beneath the paddle but close to the ground plate. This shift (w.r.t. instant 2) is attributed to the strongly decelerated flow field in front of the paddle visible in Fig. 10(a) (instants 3 to 5) which is released when the paddle leaves the nozzle. While the unobstructed region below the paddle experiences a

decrease in the magnitude of the negative vertical velocity, the region above the paddle reaches positive velocities, following the shear layer generated by the paddle's upward motion. Finally, at instant 6, as the paddle is completely withdrawn from the nozzle's outlet, all velocity fields undergo a gradual return to their initial states.

An analysis of the flow field further downstream at the x -plane $x/w = 0.2$ (see Fig. 14) reveals a noteworthy finding regarding the spanwise characteristics of the generated gust. It can be observed that the flow field possesses an increased level of homogeneity in z -direction, particularly in its streamwise component. Another observation is the symmetric distribution of the z -component of the velocity at the free ends driving the flow out of the x - y plane (i.e., expanding the flow in the spanwise direction). Compared to the cross-section at $x/w = 0$ (Fig. 13), the level increases further downstream at $x/w = 0.2$ (Fig. 14).

Concluding the observation described above, the following can be stated. Excluding from both spanwise edges about 20% of the total width ($w = 0.5$ m) and despite the minor effect of the z -velocity component in that region, the flow field generated by the paddle can be considered as nearly homogeneous in the spanwise direction. That is in full agreement with the experimental outcome of Wood, Breuer, and Neumann¹ who detected nearly constant streamwise velocity component of the generated gust for this spanwise range. Nevertheless, two important issues have to be mentioned here when confronting the predictions with the experimental data. First, the LDA measurements were restricted to two velocity components, and thus, the spanwise velocity was not considered at all. Second, the main measurements were solely carried out for the geometric symmetry plane ($z = w/2$) where also in the predictions vanishing spanwise velocity components are found. Thus, the simulations have provided new insight into the flow field generated by the paddle which were not obvious from the measurements. For the planned FSI experiments and the corresponding simulations, these characteristic features of the gust (homogeneity in spanwise direction in the kernel of the gust and outward pointing spanwise velocity) have to be taken into account.

C. Comparison of numerical and experimental results

A validation of the proposed numerical method is carried out by a direct comparison with the experimental results of Wood, Breuer, and Neumann.¹ The comparison is performed in the symmetry plane at seven streamwise positions in the range $0 \leq x/w \leq 0.6$ and a height of $y/w = 0.15$ (see Fig. 12). This covers the region in which a wind tunnel model is typically placed in. Due to the presence of a shear layer at the top surface of the jet exiting the nozzle and the high Reynolds number considered, small vortical structures are irregularly formed and convected downstream as shown at instant 1 in Fig. 11(b). Consequently, the flow generated by the paddle's motion is dependent on the state of the flow at which the paddle is entering the nozzle's outlet. Henceforth, ensemble-averaged numerical results based on 25 initial condition states spanning from 10 to 22 s of the initial simulation without the paddle are considered. An interval of 0.5 s is chosen to separate the different initial conditions. The choice is based on a detailed examination of the flow variables at the shear layer, which revealed that for the chosen time interval the flow states exhibit statistical independence. The ensemble-averaged LDA measurements of the streamwise velocity and its numerical counterpart are depicted in Fig. 16.

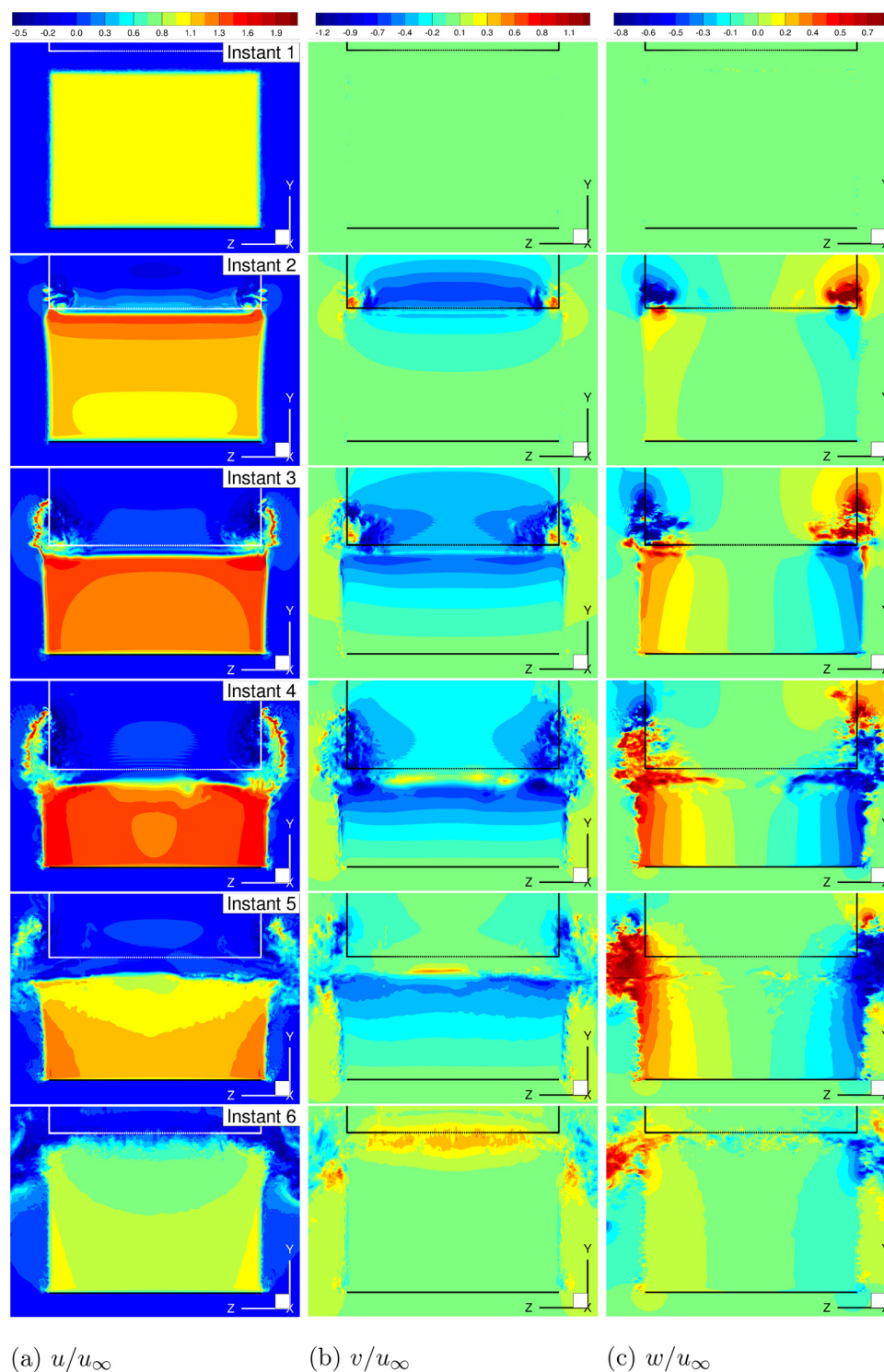


FIG. 13. Snapshots of the (a) streamwise velocity, the (b) vertical velocity, and the (c) spanwise velocity fields at the x -plane $x/w = 0$ during the paddle's downward and upward strokes. Instants correspond to those depicted in Fig. 9.

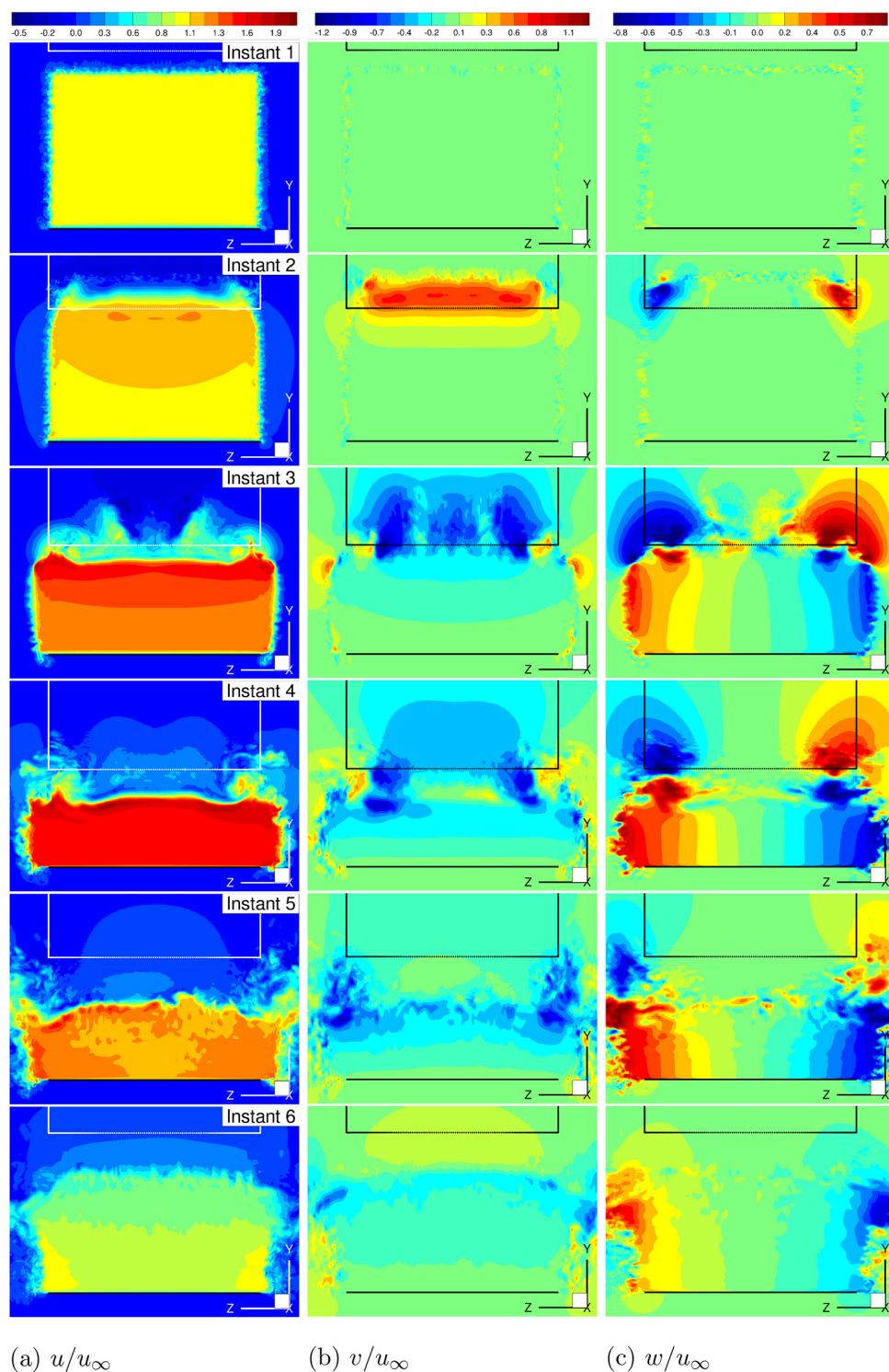


FIG. 14. Snapshots of the (a) streamwise velocity, the (b) vertical velocity, and the (c) spanwise velocity fields at the x -plane $x/w = 0.2$ during the paddle's downward and upward strokes. Instants correspond to those depicted in Fig. 9.

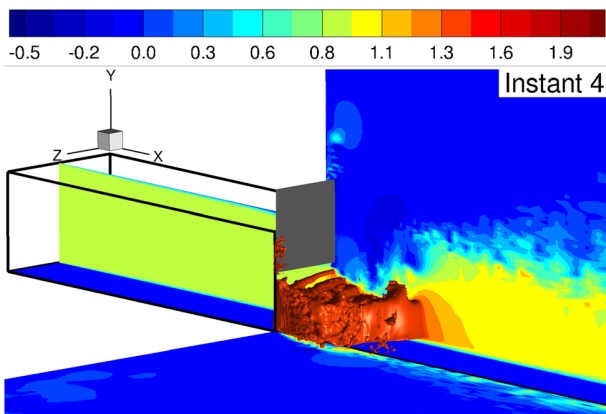


FIG. 15. An isometric view of the generated gust. All planes depict the streamwise velocity, and the gust structure is highlighted at instant 4 with the iso-surface $u/u_\infty = 1.38$.

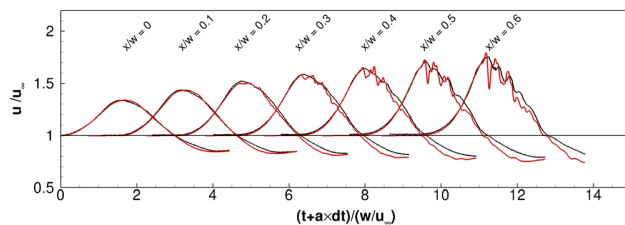
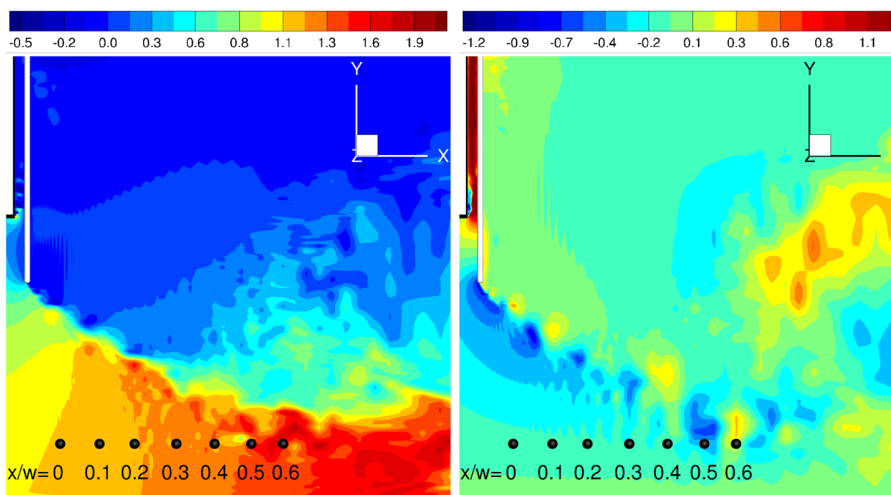


FIG. 16. Comparison between the ensemble-averaged experimental streamwise velocity u/u_∞ (black) reprinted with permission from Wood *et al.*, *J. Wind Eng. Ind. Aerodyn.* **230**, 105170 (2022). Copyright 2022 Author(s), licensed under a Creative Commons Attribution (CC BY) license and its numerical counterpart (red) of the gust generated by the paddle at the first seven points in Fig. 12.



(a) u/u_∞ .

(b) v/u_∞ .

FIG. 17. Snapshot of the (a) instantaneous streamwise velocity and the (b) instantaneous vertical velocity fields at instant 5. The first seven points in Fig. 12 are marked for reference.

The data are initially time-shifted to align the instant $t^* = t/(w/u_\infty) = 0$ with the beginning of the paddle blocking the outlet of the nozzle. To enhance the visibility of the data for the different streamwise locations, the time histories of the flow variables are shifted by adding the product of a freely chosen time increment of $\Delta t_{inc} = 0.15$ s and a spacing factor $a \in (0, \dots, 6)$.

Focusing on the time history at a single point (e.g., $x/w = 0$), the streamwise velocity possesses a bell-shaped pattern that is consistent with the displacement curve of the paddle (see Fig. 7). As the paddle initially blocks the nozzle's outlet, it effectively reduces the available outlet area. According to the principle of mass conservation, the flow beneath the paddle is forced to accelerate to counterbalance this reduction in the outlet area. The flow acceleration continues until the maximum displacement of the paddle is reached. Subsequently, as the paddle starts traveling back to its idle position, the streamwise velocity decelerates (due to the increase in the outlet area) and even drops below the free-stream velocity (due to the transient inlet velocity defined in Sec. V C). Eventually, the free-stream velocity is recovered.

It can be observed that the numerical results exhibit a remarkable coincidence with the experimental results, affirming the fidelity of the numerical method applied. Two key observations confirm this agreement. In the range $0 \leq x/w \leq 0.3$, the streamwise velocity curves closely mirror those observed in the experimental data. Moreover, both results consistently illustrate the prominent increase in the maximum streamwise velocity as one moves along the downstream direction. This increase will be elaborated upon when the time history of the pressure at these points is discussed. As one progresses toward positions within the range $x/w \geq 0.4$, an additional phenomenon occurs. The descending slope of the streamwise velocity begins to exhibit a more chaotic and turbulent character. Comparable oscillations are observed in the ensemble-averaged data of Wood, Breuer, and Neumann,¹ but those mainly initiated slightly further downstream at $x/w = 0.5$. Looking at the time histories of the instantaneous data

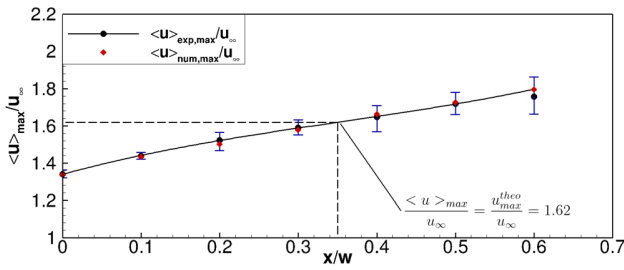


FIG. 18. Comparison between the maximum ensemble-averaged experimental streamwise velocity u/u_∞ , including the velocity range (error bars) reprinted with permission from Wood *et al.*, J. Wind Eng. Ind. Aerodyn. **230**, 105170 (2022). Copyright 2022 Author(s), licensed under a Creative Commons Attribution (CC BY) license and its numerical counterpart at the first seven points in Fig. 12.

of single events (not shown here for brevity), the amplitudes of the fluctuations are even significantly larger than for the ensemble-averaged data. That holds true for both the simulations and the measurements. These velocity fluctuations are attributed to the free shear layer and the vortices shed from the bottom edge of the paddle as shown in Fig. 17(a). Obviously, even 25 simulations are not sufficient to average these chaotic fluctuations out.

Figure 18 depicts the maximum ensemble-averaged experimental streamwise velocity (i.e., $\langle u \rangle_{\text{exp,max}}/u_\infty$) and its numerical counterpart (i.e., $\langle u \rangle_{\text{num,max}}/u_\infty$) at the selected positions. Based on $\langle u \rangle_{\text{exp,max}}/u_\infty$, Wood, Breuer, and Neumann¹ employed a third-order polynomial curve fitting (black line) to predict the maximum streamwise velocity at locations within the range $0 \leq x/w \leq 0.6$. At each measurement point and based on the 25 experimental realizations, error bars are employed to define a velocity range by marking the extremal experimental values. Obviously, the numerically predicted mean values $\langle u \rangle_{\text{num,max}}/u_\infty$ fit very well to the experimental counterparts for $x/w \leq 0.5$. At the last point evaluated ($x/w = 0.6$), a slight deviation is visible, which can be attributed to the less realizations in the simulations. Nevertheless, the predicted value stays within the measured velocity range. Furthermore, two previously discussed characteristics can be highlighted. The first is concerned with the increase in the maximum streamwise velocity along the streamwise direction, evident by the positive slope of the curve. The second characteristic is related to the increased chaotic behavior within the region $x/w \geq 0.4$ as indicated by the expansion of the error bars. Furthermore,

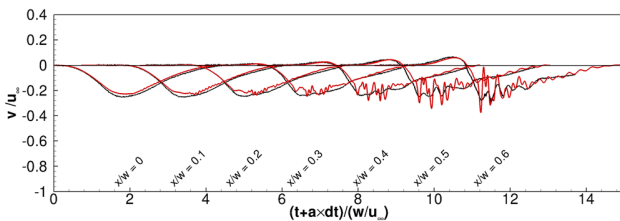


FIG. 19. Comparison between the ensemble-averaged experimental vertical velocity v/u_∞ (black) reprinted with permission from Wood *et al.*, J. Wind Eng. Ind. Aerodyn. **230**, 105170 (2022). Copyright 2022 Author(s), licensed under a Creative Commons Attribution (CC BY) license and its numerical counterpart (red) of the gust generated by the paddle at the first seven points in Fig. 12.

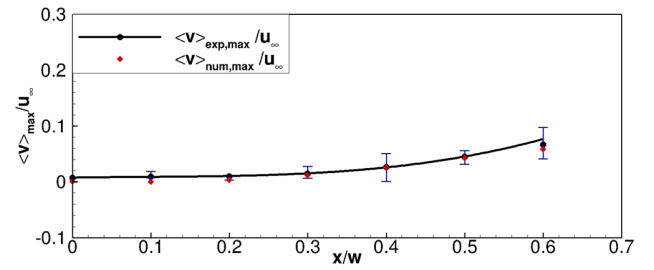


FIG. 20. Comparison between the maximum ensemble-averaged experimental vertical velocity v/u_∞ including the velocity range (error bars) reprinted with permission from Wood *et al.*, J. Wind Eng. Ind. Aerodyn. **230**, 105170 (2022). Copyright 2022 Author(s), licensed under a Creative Commons Attribution (CC BY) license and its numerical counterpart at the first seven points in Fig. 12.

one can express the theoretical maximum streamwise velocity as a function of the maximum blocking ratio [here $\max(\text{BR}) = 0.383$] by $u_{\text{max}}^{\text{theo}}/u_\infty = (1 - \max(\text{BR}))^{-1} = 1.62$. According to the fitted curve, this value falls at $x/w = 0.35$ (i.e., within the region where a FSI model is commonly positioned). Note that these values are slightly different from the data in Wood, Breuer, and Neumann¹ (maximum theoretical velocity of 1.63 and its corresponding location at 0.36). The reason is that Wood, Breuer, and Neumann¹ used the prescribed paddle displacement for their calculation leading to a slightly different blocking ratio of $\text{BR}_{\text{exp}} = 0.387$. However, the recorded (real) displacement used in the prediction is slightly less.

An analogous agreement of the experimental and the numerical results is evident in the wall-normal velocity component (see Fig. 19). The vertical velocity mainly exhibits negative values, indicating that the flow is directed toward the ground plate. This behavior is consistent with the displacement curve of the paddle. During its downstroke, the paddle gradually guides the jet exiting the nozzle's outlet downward. However, during its upward stroke, it steadily reduces its steering effect on the flow, allowing the jet to return to its original trajectory. Beyond the range $x/w \geq 0.3$, increasing positive vertical velocities are initially observed. These are attributed to the strong counterclockwise rotating vortex forming at the bottom edge of the paddle during its downstroke [see instants 2 to 4 in Fig. 10(b)]. Similar to the velocity in the streamwise direction, a strong increase in the fluctuations becomes obvious in this region. This can be again attributed to the strong shear layer developing from the lower edge of the paddle as shown in Fig. 17(b). Even in the ensemble-averaged LDA data these fluctuations are still visible in Fig. 19.

Analogous to Fig. 18, Fig. 20 depicts the maximum values of the vertical velocity. It is important to mention that the maximum

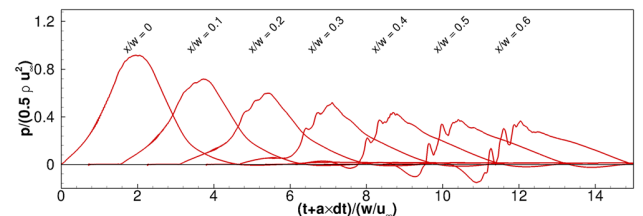


FIG. 21. Predicted ensemble-averaged pressure at the first seven points in Fig. 12.

velocities under consideration pertain solely to the initial surge phase, rather than encompassing the entire curve. The maximum wall-normal velocity (see Fig. 20) displays values near zero at points in the close proximity to the paddle (i.e., for $x/w \leq 0.2$). However, in the region $x/w \geq 0.3$, a steady increase along the streamwise direction is observed. Again, predictions and measurements are found to be in close agreement. Note that the minimum values are not depicted here for the following reason. Despite phase averaging both datasets still show very large fluctuations, so that a direct comparison makes no sense.

Employing numerical simulations facilitates the extension of the analysis to examine the time history of the pressure depicted in Fig. 21. It is important to emphasize that a direct comparison with the experimental data is unattainable due to the exclusive provision of velocity signals by the LDA measurements. A contrasting trend of the pressure curves is observed compared to the streamwise velocity signals, i.e., pressure peaks decrease in downstream direction. This behavior can be explained as follows. As clearly visible in Fig. 11(a), the blocking of the nozzle's outlet by the paddle leads to a strong increase in the pressure in the upstream region. Obviously, this effect decreases again as the distance from the nozzle increases [see pressure contour lines at the instants 2 to 5 in Fig. 11(a)]. This explains the inverse correlation between the pressure peaks and the distance of the points from the nozzle's outlet. Furthermore, this decrease in the pressure generates a negative pressure gradient that evokes the streamwise velocity to accelerate in the downstream direction. The pressure minima recorded in the region $x/w > 0.5$ are attributed to the large vortical structure originating from the bottom edge of the paddle during its downstroke [see instant 3 in Fig. 11(a)]. This vortical structure impacts the data evaluation points within this region before it gets convected downstream and pressure maxima emanating from the outlet are attained.

Finally, it should be mentioned that the IB method introduced in Sec. III does not lead to any numerically induced pressure oscillations. This is due to the compliance of the numerical method employed with the basic laws of physics. The external forces (represented by the hydrodynamic interactions) acting on a material point (i.e., a Lagrangian marker) are first evaluated based on Newton's second law. Then, these forces are transferred to the flow by smoothly distributing the Lagrangian forces to the neighboring Eulerian cells. This transfer is accomplished by the use of a regularized delta function that fulfills a set of conservation properties (including force and moment). Consequently, it ensures the satisfaction of Newton's third law of motion while keeping the incompressibility constraint intact.

Such smooth pressure curves are unattainable when less sophisticated approaches are used to simulate the paddle's motion. For example, the abrupt stepwise blocking of cells existing within the paddle's volume, achieved by the brutal prescription of all velocity components, leads to significant pressure oscillations as the paddle begins blocking the outlet. This issue emerges as the pressure solver struggles with the challenge of preserving the pressure-velocity coupling and adhering to the incompressibility constraint within the blocked cells.

D. Forces acting on the fluid by the moving paddle

Analyzing the force coefficients provides deeper insight into the underlying physics of the interaction between the flow and the paddle. This is facilitated by the fact that interpolation and spreading operators maintain the conservation of both force and moment [see Eqs. (20)

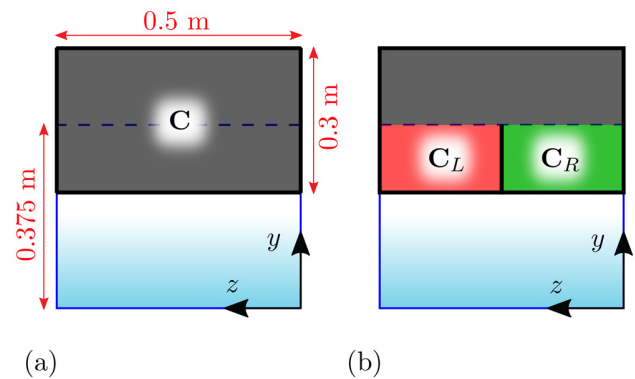


FIG. 22. Schematic sketch showing the different regions of Lagrangian markers considered in which the force coefficients are evaluated. (a) The entire paddle and (b) the direct interaction region split up into left and right halves.

and (21)]. Therefore, the total force can be readily calculated by summing the contributions from all Lagrangian points. Defining the reference area as the maximum area blocked by the paddle (i.e., $A_{ref} = w \Delta y_{max}$), the Lagrangian force coefficients read

$$\mathbf{C} = \frac{\sum_{i=1}^{N_i} \mathbf{F}(\mathbf{X}_i) \Delta V_i}{\frac{1}{2} \rho u_\infty^2 A_{ref}}. \quad (53)$$

Identical force coefficients must be obtained if the force is evaluated with respect to the Eulerian frame of reference (i.e., $\mathbf{C} = \mathbf{c}$). This can be numerically verified by replacing the numerator of Eq. (53) with the left-hand side of Eq. (20). When analyzing the force coefficients, it is crucial to consider how the Lagrangian forces are defined. As per Eq. (12), the sign (hence, the direction) of a Lagrangian force is determined by the difference between the prescribed velocity of the immersed boundary (i.e., $\mathbf{U}_d(t) = [0, V_d(t), 0]$) and the interpolated fluid velocity in its vicinity [i.e., $\mathcal{I}(\mathbf{u}^*)$]. Figure 23 depicts the temporal evolution of the ensemble-averaged Lagrangian force coefficients and their Eulerian counterpart for the entire paddle [see Fig. 22(a)]. It is important to note that these coefficients pertain to the force exerted by the paddle on the flow (in contrast to the classical perspective). For all force coefficients, both curves (i.e., Lagrangian and Eulerian) align perfectly. This verifies the rigorous conservation of the force in the numerical method employed. Note that the conservation of the moments was also checked (not depicted here) and found to be fully satisfied.

To facilitate a better understanding of the results, the temporal evolution of the ensemble-averaged Lagrangian force coefficients focusing exclusively on the Lagrangian markers located in the region of the direct flow-paddle interaction (i.e., excluding those above the nozzle's outlet) is depicted in Fig. 24. These Lagrangian markers are further split up into the left and right half of the paddle depending on their spanwise location [see Fig. 22(b)].

The x -component of the force coefficient on the entire paddle [see Fig. 23(a)] has initially a zero value until the paddle begins blocking the nozzle's outlet. Then, this drops below zero indicating the

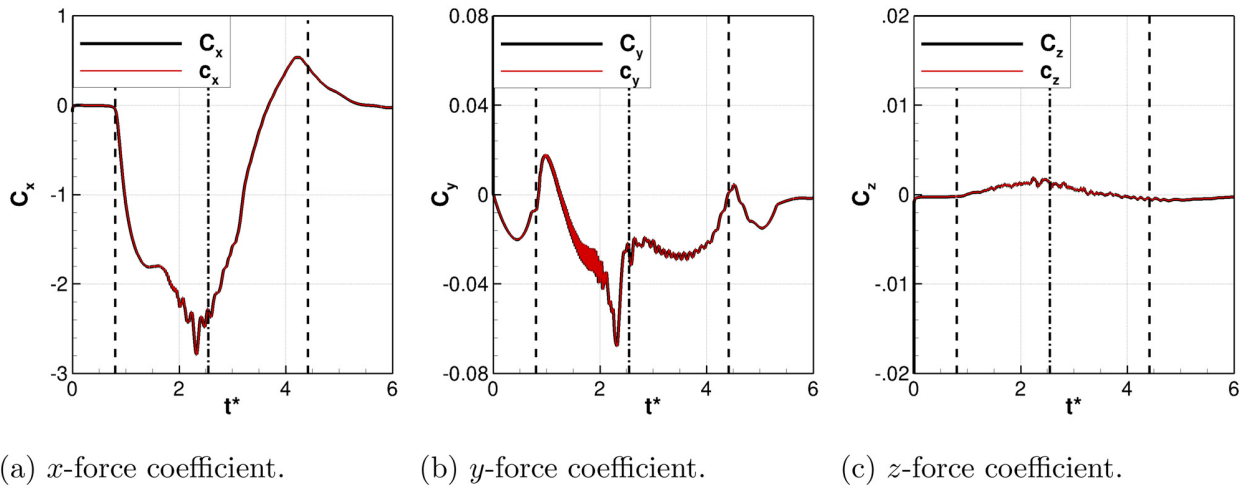


FIG. 23. Temporal evolution of the ensemble-averaged Lagrangian force coefficients evaluated over all Lagrangian markers [i.e., Fig. 22(a)] and their Eulerian counterpart. Dashed lines mark Δt_{gust} , the dashed-dotted line marks the maximum blocking.

paddle's resistance to the flow in the streamwise direction. This counteracting force continues to increase until it records its maximum magnitude slightly before the maximum blocked area is reached. As the paddle starts returning back to its idle position, the opposing force starts to steadily decrease and even exhibit positive values as the paddle completely leaves the nozzle's outlet. This is attributed to the negative streamwise velocity in the vicinity of the paddle during this phase [e.g., see instant 6 in Fig. 10(a)] and the associated low pressure [see instant 6 in Fig. 11(a)]. Furthermore, by considering Fig. 24(a) which depicts the force coefficient related to the blocked region only, it becomes evident that the zone above the nozzle's outlet is the source for the temporal asymmetry and the fluctuations in the x -component of the force coefficient. Due to the inherent geometric symmetry of the problem,

the equal x -forces exerted by the left and right halves of the paddle within the direct interaction region indicate the symmetry of the x -velocity component.

The y -component of the force coefficient on the entire paddle [see Fig. 23(b)] exhibits a more complex behavior compared to the x -component. Note, however, that the values of the force in y -direction are about two orders of magnitude smaller than for the x -direction. It initially takes a negative value directing the stationary fluid downward when the paddle is moving from its idle position toward the nozzle. At this stage, the y -force on the blocked region is still zero as visible in Fig. 24(b). However, as the paddle starts blocking the nozzle's outlet, positive y -force values emerge to counteract the strong negative vertical velocity generated by the counterclockwise rotating vortical

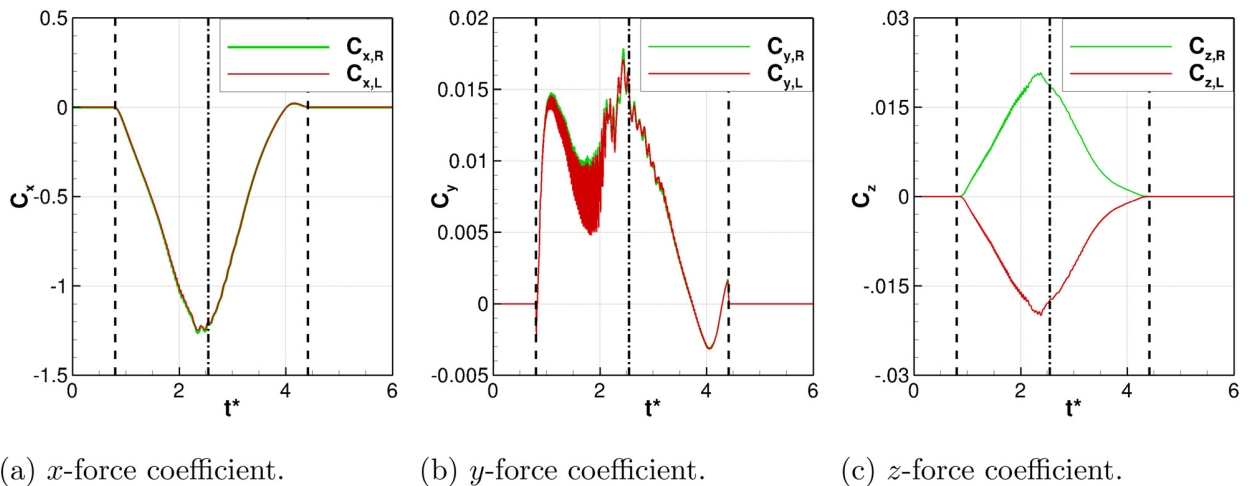


FIG. 24. Temporal evolution of the ensemble-averaged Lagrangian force coefficients focusing exclusively on the Lagrangian markers located in the region of the direct flow-paddle interaction [i.e., Fig. 22(b)]. Dashed lines mark Δt_{gust} , the dashed-dotted line marks the maximum blocking.

structure originating from the paddle's bottom edge [see the region behind the paddle at instant 2 in Fig. 10(b)]. Shortly thereafter, while the Lagrangian markers within the interaction region persist in directing the fluid upward [see Fig. 24(b)], an overall negative y -force component [see Fig. 23(b)] returns to oppose the high positive vertical velocity present in the gap between the paddle and the nozzle's outlet [e.g., see the fluid escaping in front of the paddle at instant 3 in Fig. 10(b)]. This means that two different effects work against each other in this phase of the paddle movement leading to this complex behavior of the y -component of the force. Even when the prescribed vertical velocity of the paddle becomes positive during the upstroke [see instant 5 in Fig. 10(b)], the paddle maintains its decelerating effect (obvious by an overall negative y -force component), indicating that the positive vertical velocity in the gap remains relatively high.

The z -component of the force coefficient on the entire paddle consistently maintains a value close to zero throughout the simulation time as depicted in Fig. 23(c). This behavior aligns with expectations due to the symmetric w -velocity pattern observed in Fig. 13(c). This is further supported by Fig. 24(c) where the right and left halves of Lagrangian markers within the interaction region exhibit a canceling effect.

VII. CONCLUSIONS

As demonstrated in the experimental study by Wood, Breuer, and Neumann,¹ a wind gust can be artificially generated in a wind tunnel by dipping a paddle into the free-stream leaving the nozzle. The paddle blocks the cross-section that leads to a rising velocity. The kinematics of the paddle determines the characteristics of the gust. In order to get a deeper insight into the paddle-flow interaction and the arising flow field, large-eddy simulations of the process are carried out. The motion of the paddle is described by the immersed boundary method relying on a direct forcing approach. Two sets of conclusions can be drawn from the study, one concerning the numerical method and the other related to the characteristics of the generated gusts.

A. Numerical method

- The original implementation of the immersed boundary method relying on an equidistant distribution of the Lagrangian markers was found to be very intensive concerning the required computational time since the spacing between the markers is dictated by the minimum local Eulerian grid. Furthermore, the markers are very unevenly distributed across the parallel processors, which also deteriorates the computational speed significantly.
- Introducing a non-uniform distribution of the markers in the spanwise direction with one marker assigned to each cell center strongly reduces the computational effort by about one order of magnitude, whereas the results obtained are nearly identical.
- Analyzing the results and especially the pressure distribution, it has to be mentioned that the IB method guarantees a smooth pressure field without any numerically induced oscillations when the paddle is moving in the fluid stream.
- The IB method employed assures the rigorous conservation of the force and moment coefficients. This underscores the fidelity of the numerical approach to the underlying physical principles delivering physically meaningful results.

B. Characteristics of the gust

- When the paddle enters the stream, the streamwise velocity rises and the flow is directed toward the ground plate. A strong vortex is generated at the bottom edge of the paddle and a new shear layer develops behind the paddle.
- At the maximum blocking ratio, the region above the ground plate reaches high streamwise velocities characterizing the resulting horizontal gust.
- When the paddle starts traveling back to its idle position, the large vortical structure reaches the ground plate and the gust strength already decreases again.
- An analysis of the spanwise distribution the generated gust shows symmetric velocity patterns for all velocity components. The homogeneity of the streamwise component of the predictions along the z axis is found to be in agreement with the experimental work of Wood, Breuer, and Neumann.¹
- As speculated in Wood, Breuer, and Neumann¹ and now fully confirmed by the predictions, the undisturbed gust region limited by the neighboring shear layer is not very large, which has to be taken into account when using the gust generator for FSI studies in the wind tunnel.
- Due to the free shear layer and the high Reynolds number considered, the flow induced by the paddle's motion is dependent on the initial flow state at which the paddle starts to move. Thus, strong cycle-to-cycle variations are visible in both the experimental and numerical instantaneous data. In order to still be able to compare the results, an ensemble-averaging was conducted over several realizations.
- The agreement between measurements and predictions at specific locations in the flow field is found to be very good which confirms the quality of the simulations.

ACKNOWLEDGMENTS

The initial part of the project was financially supported by the *German Research Foundation (Deutsche Forschungsgemeinschaft)* under the Contract No. BR 1847/17-1, which ended already. Computational resources (HPC cluster HSUper) have been provided by the project hpc.bw, funded by dtec.bw—Digitalization and Technology Research Center of the Bundeswehr. dtec.bw is funded by the European Union—NextGenerationEU. Last but not least, we would like to thank Dr. J. N. Wood (HSU Hamburg) for providing the additional non-published measurement data³⁵ and fruitful discussions on the topic.

AUTHOR DECLARATIONS

Conflict of Interest

The authors have no conflicts to disclose.

Author Contributions

Khaled Boulbrachene: Conceptualization (equal); Investigation (lead); Methodology (equal); Software (lead); Validation (lead); Visualization (lead); Writing – original draft (equal); Writing – review & editing (equal). **Michael Breuer:** Conceptualization (equal); Funding acquisition (lead); Investigation (supporting); Methodology (equal); Project administration (lead); Resources (lead); Software (supporting);

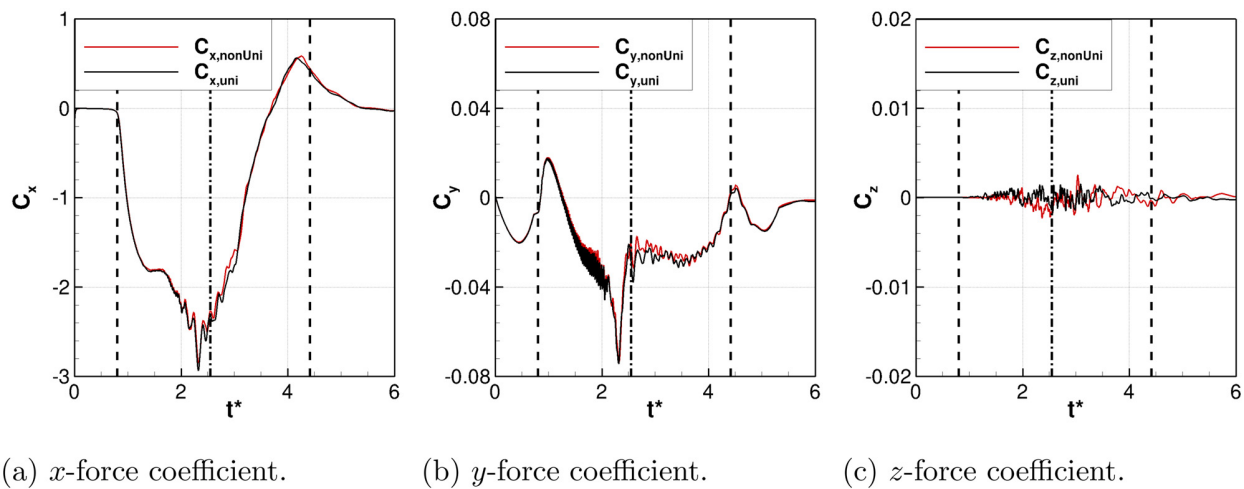


FIG. 25. Comparison of the instantaneous Lagrangian force coefficients between the simulation with the one-dimensional non-uniform distribution of Lagrangian markers (red) and its uniform counterpart (black). Dashed lines mark Δt_{gust} , the dashed-dotted line marks the maximum blocking.

Supervision (lead); Writing – original draft (equal); Writing – review & editing (equal).

DATA AVAILABILITY

The data that support the findings of this study are available within the article.

APPENDIX: COMPARISON OF FORCES FOR BOTH LAGRANGIAN MARKER DISTRIBUTIONS

In this section, the results obtained when the two different distributions of Lagrangian markers mentioned in Sec. V are used, is presented. Figure 25 depicts the instantaneous Lagrangian force coefficients of the simulation involving the one-dimensional non-uniform distribution of Lagrangian markers and its uniform counterpart. A detailed description of the computation of force coefficients is provided in Sec. VID. The initial flow state, at which the paddle starts its motion, remains consistent for both simulations (here $t_{init} = 10$ s). It can be observed that the deviations between the force coefficients between the different distributions are barely perceptible. This is particularly apparent in the x -force coefficient, which exhibits the highest order of magnitude. Despite discrepancies observed in the y - and z -force coefficients, these exhibit similar trends and are two orders of magnitude lower than the x -component, rendering them negligible. Since the forces applied by both distributions of Lagrangian markers on the flow exhibit a high degree of similarity, it is reasonable to anticipate a similar reaction from the flow for both distributions. Given the substantial difference in the computational time, the one-dimensional non-uniform distribution is preferred and consequently adopted throughout the results presented in this study.

REFERENCES

- ¹J. N. Wood, M. Breuer, and T. Neumann, "A novel approach for artificially generating horizontal wind gusts based on a movable plate: The paddle," *J. Wind Eng. Ind. Aerodyn.* **230**, 105170 (2022).
- ²I. Neunaber and C. Braud, "First characterization of a new perturbation system for gust generation: The chopper," *Wind Energy Sci.* **5**, 759–773 (2020).
- ³H. H. Hu, N. Patankar, and M. Zhu, "Direct numerical simulations of fluid-solid systems using the arbitrary Lagrangian–Eulerian technique," *J. Comput. Phys.* **169**, 427–462 (2001).
- ⁴C. Peskin, "Flow patterns around heart valves: A digital computer method for solving the equations of motion," Ph.D. thesis (Albert Einstein College of Medicine of Yeshiva University, NY, USA, 1972).
- ⁵R. Mittal and G. Iaccarino, "Immersed boundary methods," *Annu. Rev. Fluid Mech.* **37**, 239–261 (2005).
- ⁶S. Majumdar, G. Iaccarino, P. Durbin *et al.*, "RANS solvers with adaptive structured boundary non-conforming grids," in *Proceedings of Annual Research Briefs* (Center for Turbulence Research, 2001), pp. 353–364.
- ⁷D. K. Clarke, M. D. Salas, and H. A. Hassan, "Euler calculations for multielement airfoils using Cartesian grids," *AIAA J.* **24**, 353–358 (1986).
- ⁸H. S. Udaykumar, R. Mittal, and W. Shyy, "Computation of solid-liquid phase fronts in the sharp interface limit on fixed grids," *J. Comput. Phys.* **153**, 535–574 (1999).
- ⁹H. S. Udaykumar, R. Mittal, P. Rampungoon, and A. Khanna, "A sharp interface Cartesian grid method for simulating flows with complex moving boundaries," *J. Comput. Phys.* **174**, 345–380 (2001).
- ¹⁰C. S. Peskin, "The immersed boundary method," *Acta Numer.* **11**, 479–517 (2002).
- ¹¹C. S. Peskin, "Numerical analysis of blood flow in the heart," *J. Comput. Phys.* **25**, 220–252 (1977).
- ¹²R. P. Beyer and R. J. LeVeque, "Analysis of a one-dimensional model for the immersed boundary method," *SIAM J. Numer. Anal.* **29**, 332–364 (1992).
- ¹³M.-C. Lai and C. S. Peskin, "An immersed boundary method with formal second-order accuracy and reduced numerical viscosity," *J. Comput. Phys.* **160**, 705–719 (2000).
- ¹⁴D. Goldstein, R. Handler, and L. Sirovich, "Modeling a no-slip flow boundary with an external force field," *J. Comput. Phys.* **105**, 354–366 (1993).
- ¹⁵E. Saiki and S. Biringen, "Numerical simulation of a cylinder in uniform flow: Application of a virtual boundary method," *J. Comput. Phys.* **123**, 450–465 (1996).
- ¹⁶C. Lee, "Stability characteristics of the virtual boundary method in three-dimensional applications," *J. Comput. Phys.* **184**, 559–591 (2003).
- ¹⁷P. Angot, C.-H. Bruneau, and P. Fabrie, "A penalization method to take into account obstacles in incompressible viscous flows," *Numerische Math.* **81**, 497–520 (1999).

- ¹⁸K. Khadra, P. Angot, S. Parneix, and J.-P. Caltagirone, "Fictitious domain approach for numerical modelling of Navier–Stokes equations," *Int. J. Numer. Methods Fluids* **34**, 651–684 (2000).
- ¹⁹E. Fadlun, R. Verzicco, and P. Orlandi, "Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations," *J. Comput. Phys.* **161**, 35–60 (2000).
- ²⁰M. Uhlmann, "An immersed boundary method with direct forcing for the simulation of particulate flows," *J. Comput. Phys.* **209**, 448–476 (2005).
- ²¹A. Pinelli, I. Naqavi, U. Piomelli, and J. Favier, "Immersed-boundary methods for general finite-difference and finite-volume Navier–Stokes solvers," *J. Comput. Phys.* **229**, 9073–9091 (2010).
- ²²J. N. Wood and M. Breuer, "The paddle: A novel procedure for artificial gust generation in a wind tunnel," in *Proceedings of Gala–Fachtagung 'Experimentelle Strömungsmechanik'*, München, 5–7 September 2023, edited by C. J. Kähler, T. Fuchs, R. Hain, S. Scharnowski, B. Ruck, and A. Leder (2023).
- ²³IEC-Standard, "Measurement and assessment of power quality of grid connected wind turbines," Technical Report No. IEC-61400–21, 2002.
- ²⁴A. M. Roma, C. S. Peskin, and M. J. Berger, "An adaptive version of the immersed boundary method," *J. Comput. Phys.* **153**, 509–534 (1999).
- ²⁵E. Anderson, Z. Bai, C. Bischof, S. Blackford, J. Demmel, J. Dongarra, J. Du Croz, A. Greenbaum, S. Hammarling, A. McKenney, and D. Sorensen, *LAPACK Users' Guide*, 3rd ed. (Society for Industrial and Applied Mathematics, Philadelphia, PA, 1999).
- ²⁶F. Durst and M. Schäfer, "A parallel block-structured multigrid method for the prediction of incompressible flows," *Int. J. Numer. Methods Fluids* **22**, 549–565 (1996).
- ²⁷P. K. Khosla and S. G. Rubin, "A diagonally dominant second-order accurate implicit scheme," *Comput. Fluids* **2**, 207–209 (1974).
- ²⁸J. H. Ferziger and M. Perić, *Computational Methods for Fluid Dynamics*, 3rd ed. (Springer Berlin, 2002).
- ²⁹C. M. Rhie and W. L. Chow, "Numerical study of the turbulent flow past an air-foil with trailing-edge separation," *AIAA J.* **21**, 1525–1532 (1983).
- ³⁰M. Breuer, G. De Nayer, M. Münsch, T. Gallinger, and R. Wüchner, "Fluid-structure interaction using a partitioned semi-implicit predictor-corrector coupling scheme for the application of large-eddy simulation," *J. Fluids Struct.* **29**, 107–130 (2012).
- ³¹H. L. Stone, "Iterative solution of implicit approximations of multidimensional partial differential equations," *SIAM J. Numer. Anal.* **5**, 530–558 (1968).
- ³²J. Smagorinsky, "General circulation experiments with the primitive equations I: The basic experiment," *Mon. Weather Rev.* **91**, 99–165 (1963).
- ³³F. Schäfer and M. Breuer, "Comparison of c-space and p-space particle tracing schemes on high-performance computers: Accuracy and performance," *Int. J. Numer. Methods Fluids* **39**, 277–299 (2002).
- ³⁴L. F. Moody, "An approximate formula for pipe friction factors," *Trans. ASME* **69**, 1005–1011 (1947).
- ³⁵J. N. Wood, "CTA measurements of the paddle case," private communication (2023).

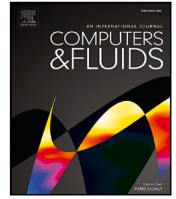
Publication V: Combination of two FSI methods and their validation based on artificial wind gusts impacting a flexible T-structure

Reference

M. Breuer, K. Boulbrachene and G. De Nayer. Combination of two FSI methods and their validation based on artificial wind gusts impacting a flexible T-structure. *Computers & Fluids* 294 (2025), 106621. DOI: [10.1016/j.compfluid.2025.106621](https://doi.org/10.1016/j.compfluid.2025.106621)

Copyright

© 2025 The Authors. Published by Elsevier Ltd. This is an open access article distributed under the terms of the Creative Commons CC-BY license (<http://creativecommons.org/licenses/by/4.0/>), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.



Combination of two FSI methods and their validation based on artificial wind gusts impacting a flexible T-structure

M. Breuer^{1,*}, K. Boulbrachene¹, G. De Nayer¹

¹ Professur für Strömungsmechanik, Helmut-Schmidt-Universität Hamburg, D-22043 Hamburg, Germany

ARTICLE INFO

Keywords:

Artificial (horizontal) wind gust
Fluid–structure interaction (FSI)
Coupled numerical FSI simulations
Immersed boundary method
Wind tunnel experiments
Validation

ABSTRACT

The study focuses on the combination of two numerical approaches that are typically not used together in this manner. The first is a well-established partitioned fluid–structure interaction (FSI) simulation methodology relying on a finite-volume fluid solver for curvilinear, block-structured, body-fitted grids written in the Arbitrary Lagrangian–Eulerian (ALE) formulation, and a finite-element solver for the structural analysis. The second approach is an immersed boundary (IB) method employing a continuous and direct forcing strategy. The IB method, often applied to Cartesian grids, is also referred to as an approach to simulate fluid–structure interactions. In this study, both methods are combined to exploit their respective advantages in simulating a complex flow problem. The coupled FSI problem involves the interaction of a thin, flexible structure deforming under the dynamic load of a wind gust (task 1). The gust itself is generated by an artificial wind gust generator, which includes a *paddle* that partially obstructs the wind tunnel's outlet, thereby defining an FSI problem of its own (task 2). For task 1, the classical partitioned ALE approach is employed, while the IB method is more appropriate for task 2. Using available experimental measurement data for both the fluid flow and the structural deformation, the combined simulation framework is first validated for the case without gust. In a second step, the more challenging FSI problem of discrete gusts impacting the T-structure is thoroughly analyzed and the predicted data are compared with the available measurement data. For both cases without and with gusts, a very good agreement between simulation and experiment is achieved, which justifies the chosen approach.

1. Introduction

Extreme weather phenomena characterized by high wind velocities and strong wind gusts have been observed more and more frequently in recent years. Structures exposed to such wind conditions are not only subjected to high static forces. Dynamic loads caused by the short-term wind gusts are also particularly relevant for stability and fatigue strength. This is especially true for constructions made of thin structures and/or lightweight materials, which are vulnerable to wind events.

To better understand the impact of strong wind gusts on such constructions, conducting measurements on real structure may be seen like the most straightforward approach for investigating fluid–structure interactions. However, simultaneously measuring the fluid flow and the structure deformation at real scales is extremely challenging and costly. Additionally, the difficulty exists that the wind speed and gust strength cannot be controlled in nature. Instead, one must set up the measuring equipment and wait for the right weather conditions to occur. To conduct these investigations in a laboratory setting, it is necessary to develop methods to artificially generate wind gusts in a wind tunnel.

Recently, Wood et al. [1] conducted a comprehensive literature review on methods for generating artificial wind gusts in a laboratory environment. Generally speaking, wind gusts can be divided into two classes, vertical and horizontal gusts. Vertical gusts are directed perpendicular to the main flow direction and are particularly relevant in aeronautics leading to rapid changes in lift forces when an airplane is subjected to such a gust, c.f., [2–5]. Contrarily, in the horizontal case the gust is directed in the main flow direction. This gust type is especially of relevance in civil engineering applications, i.e., for the stability of bridges, high-rise buildings or thin-walled constructions. Moreover, energy harvesting wind turbines are also vulnerable to such gusts [6] inducing a variety of studies, c.f., [7–10].

Building on their literature review and concentrating on horizontal wind gusts, Wood et al. [1] developed a novel artificial wind gust generator (WGG) known as the *paddle*. By partially obstructing the outlet of the wind tunnel nozzle with the *paddle* within a short time interval, the flow is significantly accelerated leading to a horizontal gust. In Wood et al. [1] and Wood and Breuer [11] it is demonstrated that the shape and the intensity of the generated wind gust can be

* Corresponding author.

E-mail address: breuer@hsu-hh.de (M. Breuer).

<https://doi.org/10.1016/j.compfluid.2025.106621>

Received 28 November 2024; Received in revised form 21 January 2025; Accepted 24 March 2025

Available online 3 April 2025

0045-7930/© 2025 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

effectively controlled achieving high reproducibility and allowing the superposition of these wind gusts onto the free-stream flow.

The possibility to synchronize the wind gust generator with the measurement equipment for both the fluid flow and the structural deformation using a trigger signal was first used in a follow-up study by Breuer and Neumann [12]. The aim of this study was to develop a measurement strategy capable of simultaneously capturing the fluid flow and the structural deformation during the impact of the gust with high temporal resolution. For this purpose, a synchronized setup was utilized, comprising the previously described WGG and two distinct high-speed optical measurement techniques, i.e., particle-image velocimetry (PIV) for the fluid flow analysis and digital-image correlation (DIC) for monitoring the structural deformations. More than 100 gust events with consistent characteristics were recorded, enabling the comparison of different gust realizations and facilitating phase averaging. Designed as a benchmark case, a geometrically simple T-shaped thin structure was selected as the test object. The structure was mounted onto the bottom wall of the test section and subjected to the fluid load of the impacting gusts. The data generated in this study are employed in this paper for comparative analysis.

In addition to experimental investigations, engineers are also interested in describing such coupled FSI problems using advanced numerical simulations. This naturally raises the question of how to accurately reproduce the experimental generation of wind gusts observed in the wind tunnel within a numerical framework. The study by Boulbrachene and Breuer [13] addressed this question in detail. To describe the motion of the *paddle* that partially blocks the nozzle's outlet and is responsible for the gust generation [1], the immersed boundary (IB) method with a continuous and direct forcing approach was implemented into a finite-volume flow solver for large-eddy simulations. The IB method ensures smooth pressure and velocity fields without any numerically induced oscillations when the *paddle* moves through the fluid stream. Compared to alternative approaches for generating gusts, which will be discussed in more detail in Section 2.2, the IB method offers the most realistic option for reproducing the gusts generated by the experimental device (*numerical wind tunnel*). The simulation results in Boulbrachene and Breuer [13] demonstrated excellent agreement with the measurements conducted by Wood et al. [1] at specific points in the flow field, validating the robustness and accuracy of the proposed simulation methodology.

The objective of this study is to create a numerical counterpart of the experimental investigations on the T-structure under wind gust load conducted by Breuer and Neumann [12]. To achieve this goal, the IB gust generation methodology described above is combined with the possibilities of coupled high-fidelity simulations for flexible structures in complex turbulent flows. This combination is a novelty. In contrast to the experiment, all field quantities, including velocities, pressure and deformation, will be provided in the entire domain with high spatial and temporal resolution. This measure shall facilitate a deeper analysis of the complex fluid–structure interaction of the thin structure and the gust-induced flow. The T-shaped thin structure mounted in the wind tunnel can be considered as a first step to prove and validate the newly combined methodology. As an initial step, the test case offers several advantages: the structure is geometrically simple, it consists of a thin brass sheet with easily determinable material properties, and, finally, comprehensive experimental measurements are available for validation and comparison.

The paper is organized as follows: Section 2 outlines the entire numerical methodology used for simulating both the fluid flow and the structural deformation, including the coupling procedure between the two. Section 3 elaborates on the definition of the test case, covering the physical properties as well as the numerical setup employed in the simulations. Section 4 validates and analyzes the results obtained in two main substeps: As a first step, Section 4.1 serves as an additional validation of the already established ALE FSI computational framework. The IB method is then activated within the ALE procedure and the entire framework is validated against the experimental results in Section 4.2.

2. Numerical methods

Numerically simulating the coupled FSI problem, which involves the turbulent flow including wind gusts and the flexible structure, can be broken down into four key subtasks: Calculating the turbulent flow field, accurately representing the generation of gusts in the wind tunnel, determining the structural deformation, and finally coupling the fluid and structural simulations. For each individual subtask, a general description, the relevant equations, and previous publications thoroughly addressing these topics are provided below. The interactions among these subtasks are illustrated by the flowchart depicted in Fig. 1.

2.1. Numerical description of the fluid flow

The complex turbulent flow is predicted using the large-eddy simulation (LES) technique. The underlying computational fluid dynamics (CFD) solver, FASTEST-3D [14], solves the filtered Navier–Stokes equations in an Arbitrary Lagrangian–Eulerian (ALE) formulation [15]. These equations are discretized using the finite-volume technique on a curvilinear, block-structured, body-fitted grid with a collocated variable arrangement. The integral form of the governing equations formulated for a temporally varying domain read:

$$\int_{S(t)} \rho_f (\mathbf{u} - \mathbf{u}^{\text{grid}}) \cdot \mathbf{n} dS = 0, \quad (1)$$

$$\frac{d}{dt} \int_{V(t)} \rho_f \mathbf{u} dV + \int_{S(t)} \rho_f (\mathbf{u} - \mathbf{u}^{\text{grid}}) \mathbf{u} \cdot \mathbf{n} dS = - \int_{S(t)} \underline{\underline{\tau}} \cdot \mathbf{n} dS - \int_{S(t)} p \mathbf{n} dS, \quad (2)$$

where $\mathbf{u}$ is the flow velocity, ρ_f is the density of the fluid, $\underline{\underline{\tau}}$ is the stress tensor, and $\mathbf{n}$ is the normal unit vector of the surface $S(t)$ belonging to the control volume $V(t)$. Since the grid is deformable, $\mathbf{u}^{\text{grid}}$ denotes the grid velocity with which the surface of a control volume is moving.

Surface and volume integrals appearing in Eqs. (1) and (2) are approximated using the midpoint rule. The pressure and the viscosity are linearly interpolated to the cell faces, resulting in a second-order accurate central scheme. To avoid wiggles in the simulation data, the velocities used for the prediction of the convective fluxes on the cell faces are approximated using flux blending [16], which combines 3 % of a first-order accurate upwind scheme with 97 % of a second-order accurate central scheme. Furthermore, the momentum interpolation technique of Rhie and Chow [17] is applied to predict the mass fluxes through the cell faces preventing unwanted oscillations on non-staggered grids. To couple the pressure and velocity fields, a semi-implicit predictor–corrector scheme is employed for solving the incompressible Navier–Stokes equations [15]. First, the momentum equations are time-marched using a low-storage multi-stage Runge–Kutta method to derive an intermediate velocity $\mathbf{u}^*$. Then, the corrector step ensures mass conservation, yielding a divergence-free velocity field by solving a Poisson equation for the pressure correction. In the LES technique, the large scales of turbulence are resolved while the small scales are modeled with a subgrid-scale (SGS) model [18]. This study uses the classical Smagorinsky [19] model with a standard parameter $C_s = 0.1$ and the Van-Driest damping function for the near-wall region. Given that the near-wall grid resolution is sufficient to resolve the viscous sublayer, as detailed in Section 3, wall functions are not required, enabling a fully wall-resolved LES approach.

In the FSI simulation framework, the structural deformations resulting from fluid forces f_{ALE} , which are computed based on the fluid pressure and shear stresses acting on the structure, are taken into account at the interface between the fluid and the solid. This necessitates that the ALE formulation accounts for the fluid motion within a temporally varying domain, requiring a body-fitted grid that must be adapted at each time step. Several techniques are available for grid adaptation depending on the magnitude of the structural deformations.

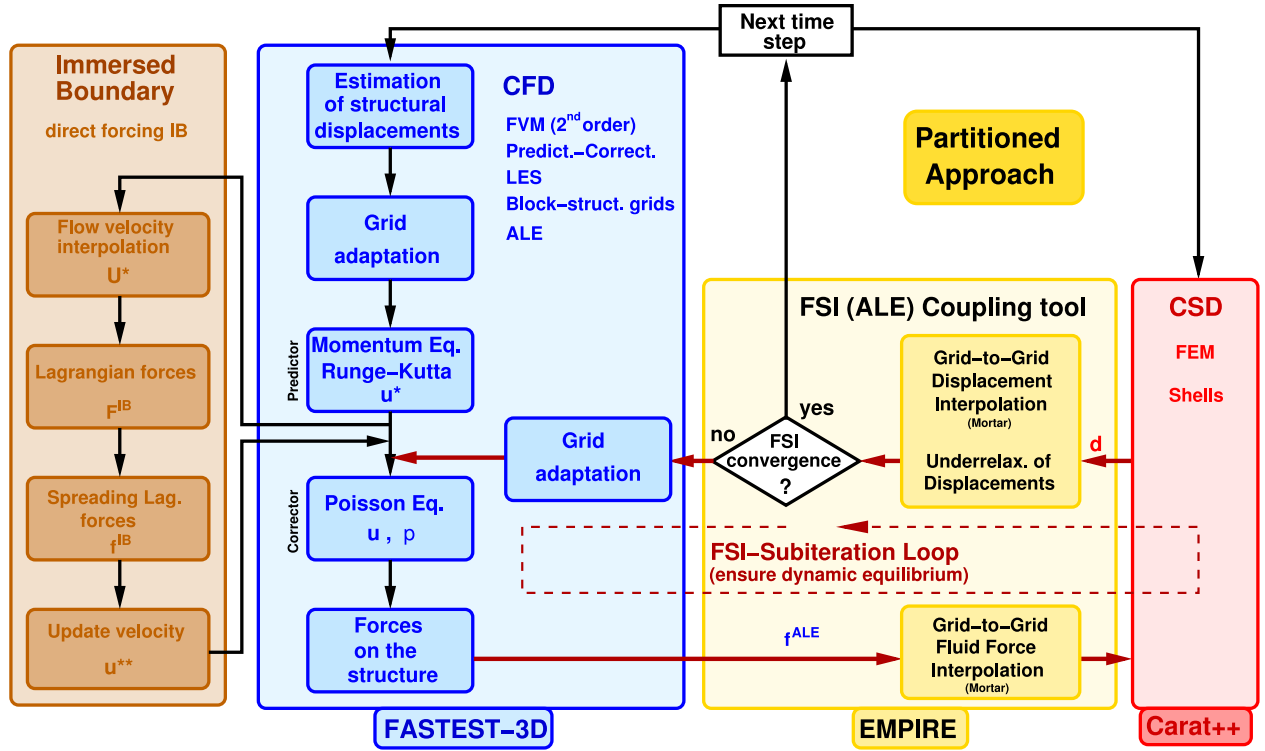


Fig. 1. Flowchart of the computational framework.

For small deformations as in the present test case, a very fast algebraic procedure is employed, utilizing a combination of linear and transfinite interpolations (TFI) [20]. This method is efficient, with its computational effort scaling proportionally to the number of volume grid points, and is well-suited for scenarios where the structural displacements do not significantly alter the fluid domain. A measure for the grid quality was introduced by Knupp [21] as the skew quality metric. For LES predictions, this value should not fall below 0.9 during the grid adaptation to guarantee a high grid quality. For alternatives in case of substantial deformations, refer to Sen et al. [22].

2.2. Numerical description of the gust generation

2.2.1. Overview

In the literature, continuous and discrete gust models are distinguished [23,24]. Continuous gust models, which describe wind conditions in form of multi-correlated wind-velocity time series, are often applied in the context of fatigue evaluation. However, for the determination of extreme loads they are rather rarely employed. For this purpose, discrete models, describing the gust in a simplistic manner via mathematical functions representing the spatial and temporal velocity distributions [6], are more appropriate. Therefore, the brief overview is limited to this category.

A discrete wind gust can be described within a simulation environment using various approaches. One classical method for introducing wind gusts into the computational domain is to superimpose them at the inlet boundary, allowing the gusts to freely propagate through the domain. This technique is commonly known as the far-field boundary condition method. To maintain the global mass balance, especially in simulations involving incompressible fluid flows, this method requires dynamic adjustments of the inlet velocity in regions away from the gust. This measure ensures a constant total mass flow through the inlet [25,26]. Additionally, a highly resolved grid is essential to prevent significant numerical damping of the gust while being convected through the flow domain toward the region of interest.

To address the need for fine grids throughout the computational domain, several alternative approaches have been proposed. One such approach is the pure source-term formulation method, as suggested by De Nayer and Breuer [27]. In this method, the wind gust is injected at any position in the computational domain by adding a source term to the momentum conservation equation. This approach offers the advantage of maintaining the global mass conservation while also accounting for the feedback effect of the flow field on the gust itself. This method circumvents the need for globally fine grids, thereby reducing computational costs.

Another class of approaches, often used in aeronautical applications for evaluating indicial responses of airplanes, is referred to as the prescribed velocity method. Unlike the previously mentioned methods, which allow gusts to propagate freely after the injection phase, the prescribed velocity methods define the position of the imposed field at each instant in time. Within this category, two distinct approaches are recognized: the field velocity method (FVM) and the split velocity method (SVM). The field velocity method was initially proposed by Parameswaran and Baeder [28] and Singh and Baeder [29]. The key idea behind FVM is to modify the convective fluxes of the mass and momentum conservation equations by superimposing a pre-defined additional velocity that varies in space and time according to a gust profile. To account for the missing feedback effect of the surrounding flow on the gust itself, the split velocity method was introduced, as originally proposed by Wales et al. [4]. In SVM, the velocity field is decomposed into a prescribed gust velocity and the remaining background velocity, as denoted by Huntley et al. [30]. The momentum equations are effectively rearranged to solve for the background velocity, with additional source terms included to account for the feedback effect. Typical applications of this method are found in the works of Biler et al. [31], Badrya and Baeder [32], Badrya et al. [5] and Boulbrachene et al. [33].

All discrete gust approaches described so far rely on the assumption that the velocity distribution within the gust is both known and can be explicitly prescribed. Typically, deterministic functions are employed

for the spatial and temporal distributions, such as the *Extreme Coherent Gust* (ECG) or the *Extreme Operating Gust* (EOG) as defined by IEC-Standard [6]. However, in the current study, where the gust is generated within the wind tunnel using the WGG, these velocity distributions are not explicitly known. Although the experimental investigations conducted by Wood et al. [1] aimed to replicate the 1-cosine gust shape characteristic of ECG gusts, this does not mean that the artificial gust generated by the paddle precisely matches the corresponding velocity profile. To realistically reproduce the gust generated in the wind tunnel, the generation process of the gust involving the moving paddle must be taken into account. Unfortunately, none of the previously mentioned methods is capable of accurately capturing this process. Consequently, an alternative approach, as suggested by Boulbrachene and Breuer [13], must be applied to achieve a more realistic simulation. This alternative approach will be explained next.

2.2.2. Immersed boundary gust generator

The working principle of the wind gust generator installed in a Göttingen-type wind tunnel with an open test section is straightforward (see Fig. 2): Initially, the nozzle of the wind tunnel is fully open. The fluid flows through the measurement section between the nozzle outlet and the receiver. The gust is generated by partially blocking the outlet of the nozzle for a short time period. This closure operation is achieved by a rigid plate (referred to as the paddle) mounted on a movable saddle of a tooth belt axis. All key kinematic parameters of the tooth belt axis – such as stroke length, velocity and acceleration – can be fully controlled by the software allowing to generate various gust shapes, as detailed in Wood et al. [1].

The finite-volume method described in Section 2.1 in the Eulerian (E) frame of reference forms the basis for calculating the flow in the cell centers $\mathbf{x}_{i,j,k}$ of the CFD grid g_h . The paddle is modeled as a moving boundary employing the immersed boundary (IB) method with a direct forcing approach, as proposed by Uhlmann [34]. The effect of the immersed boundary is incorporated into the governing equations in the ALE formulation by adding a forcing term to the right-hand side of the momentum equation:

$$\underbrace{\frac{d}{dt} \int_{V(t)} \rho_f \mathbf{u} dV + \int_{S(t)} \rho_f (\mathbf{u} - \mathbf{u}^{\text{grid}}) \cdot \mathbf{n} dS = - \int_{S(t)} \underline{\underline{\tau}} \cdot \mathbf{n} dS - \int_{S(t)} p \mathbf{n} dS}_{\text{N.-S. in ALE formulation}} + \underbrace{\int_{V(t)} \mathbf{f}^{\text{IB}} dV}_{\text{IB forcing term}} \quad (3)$$

This volume force $\mathbf{f}^{\text{IB}}$, applied at the cell centers of the Eulerian grid, represents the influence of the solid paddle on the surrounding fluid. The immersed boundary Γ^{IB} is represented by a set of Lagrangian (L) markers with coordinates denoted as $\mathbf{X}_l$ (i.e., $\mathbf{X}_l \in \Gamma \forall 1 \leq l \leq N_L$, where N_L represents the total number of Lagrangian markers). In the direct forcing scheme [34], the boundary force is first evaluated at the Lagrangian force points as:

$$\mathbf{F}_l^{\text{IB}} = \rho_f \frac{\mathbf{U}_l^d - \mathbf{U}_l^*}{\Delta t} \quad \forall \mathbf{X}_l, \quad (4)$$

where $\mathbf{U}_l^d$ is the desired velocity at the fluid–solid interface and $\mathbf{U}_l^*$ is the intermediate velocity after the predictor step. Due to the movement of the paddle and the resulting time-dependent immersed boundary, the fluid–solid interface represented by the Lagrangian markers is not aligned with the cell centers of the Eulerian fluid grid. To effectively couple the fluid and the solid phases, it is necessary to transfer quantities between the Eulerian and the Lagrangian frames of reference. Therefore, the original IB method introduced by Peskin [35] depending on the use of a regularized delta function δ_h [36] for the transfer of quantities is employed. The coupling is achieved through the interpolation operation $\mathcal{I}(\mathbf{u}^*)$ for the velocity (E $\rightarrow$ L) and the spreading operation $\mathcal{S}(\mathbf{F}^{\text{IB}})$ for the forces (L $\rightarrow$ E) defined as:

$$\mathbf{U}^*(\mathbf{X}_l) = \sum_{i,j,k \in g_h} \mathbf{u}^*(\mathbf{x}_{i,j,k}) \delta_h(\mathbf{x}_{i,j,k} - \mathbf{X}_l) \Delta v_{i,j,k} \quad \forall 1 \leq l \leq N_L, \quad (5)$$

$$\mathbf{f}^{\text{IB}}(\mathbf{x}_{i,j,k}) = \sum_{l=1}^{N_L} \mathbf{F}_l^{\text{IB}}(\mathbf{X}_l) \delta_h(\mathbf{x}_{i,j,k} - \mathbf{X}_l) \Delta V_l \quad \forall \mathbf{x}_{i,j,k} \in g_h. \quad (6)$$

Hence, the steps of the predictor–corrector method accounting for the immersed boundary can be summarized by the following equations:

$$1. \mathbf{u}^* = \mathbf{u}^n + \frac{\Delta t}{\rho_f} \mathbf{r}hs^n \quad \forall \mathbf{x}_{i,j,k} \in g_h, \quad (7)$$

$$2. \mathbf{U}^* = \mathcal{I}(\mathbf{u}^*) \quad \forall \mathbf{X}_l \in \Gamma, \quad (8)$$

$$3. \mathbf{F}^{\text{IB}} = \rho_f \frac{\mathbf{U}^d - \mathbf{U}^*}{\Delta t} \quad \forall \mathbf{X}_l \in \Gamma, \quad (9)$$

$$4. \mathbf{f}^{\text{IB}} = \mathcal{S}(\mathbf{F}^{\text{IB}}) \quad \forall \mathbf{x}_{i,j,k} \in g_h, \quad (10)$$

$$5. \mathbf{u}^{**} = \mathbf{u}^* + \frac{\Delta t}{\rho_f} \mathbf{f}^{\text{IB}} \quad \forall \mathbf{x}_{i,j,k} \in g_h, \quad (11)$$

$$6. \Delta p' = \frac{\rho_f}{\Delta t} \nabla \cdot \mathbf{u}^{**} \quad \forall \mathbf{x}_{i,j,k} \in g_h, \quad (12)$$

$$7. \mathbf{u}^{n+1} = \mathbf{u}^n - \frac{\Delta t}{\rho_f} \nabla p' \quad \forall \mathbf{x}_{i,j,k} \in g_h. \quad (13)$$

The flow field around the T-structure in the test section simulated in this work necessitates a stretched grid that clusters grid points near the walls to accurately capture the boundary layers. As a result, a general curvilinear non-uniform grid is employed. To account for the non-uniformity of the grid, a modified window function is defined according to Pinelli et al. [37]. This window function includes 10 unknown polynomial coefficients, which are determined by imposing a set of discrete modified moment conditions to ensure the integral conservation of both force and moment. The number of polynomial coefficients arises from the three-dimensionality of the problem and the second-order accurate spatial discretization scheme used [37]. The resulting linear system of equations is solved using the linear algebra package LAPACK [38]. For further details, readers are referred to Boulbrachene and Breuer [13].

The tracking of Lagrangian markers is performed in the computational space (c-space), which is derived by transforming the structured grid connecting the cell centers to an orthonormal grid, as described by Schäfer and Breuer [39]. By using this transformation, the time-consuming search process in the physical space (p-space) associated with finding the new computational cell containing the Lagrangian marker is avoided [40]. In c-space, the location of the Lagrangian marker is defined as a combination of an index triple, representing the cell containing the marker and a fractional offset within that cell. This approach not only facilitates efficient searching of the nearest cell center, necessary for determining the bounds of the regularized delta function used in the IB method, but also allows for the conversion of the marker's position back to p-space. This transformation from c-space to p-space is accomplished using a trilinear interpolation based on the cell-centered coordinates [39]. In addition to its role in evaluating the precise location of the Lagrangian markers in physical coordinates, this interpolation is also crucial for visualizing the immersed boundary in p-space.

The immersed boundary (i.e., the paddle) is set into motion according to a prescribed velocity defined in the physical space. To accurately update the marker's location in c-space, it is first necessary to transform the prescribed velocity components from p-space to c-space. This transformation is accomplished using the spatial metric coefficients followed by a temporal integration. The metric coefficients are the entries of the Jacobian matrix of the p-space to c-space coordinate transformation and are evaluated at the Lagrangian location. For this purpose, the CZ+ tracking scheme proposed by Schäfer and Breuer [39] is employed. Further details can be found in Boulbrachene and Breuer [13].

2.3. Numerical description of the structural deformation

The deformation of the T-shaped structure due to fluid loads at the FSI interface is predicted using the Carat++ solver. This software specializes in the finite-element and isogeometric analysis of structures, with focus on predicting the geometrically non-linear behavior of thin-walled components, such as shells and membranes [41,42]. In the

current study, the structure is assumed to behave in a linear-elastic and isotropic manner and a St. Venant-Kirchhoff material law is employed to link the second Piola–Kirchhoff stress tensor $\mathbf{S}$ with the Green–Lagrange strain tensor $\mathbf{E}$ [43]. The dynamic equilibrium of the structure is described by the momentum equation (Cauchy's first equation of motion) written in a Lagrangian frame of reference and expressed with the second Piola–Kirchhoff stress tensor as:

$$\rho_s \frac{\partial^2 \mathbf{d}}{\partial t^2} = \nabla \cdot (\mathbf{F} \cdot \mathbf{S}) + \rho_s \mathbf{b}. \quad (14)$$

Here, ρ_s represents the density of the structure, $\mathbf{d}$ denotes the displacement of the material point in space, $\mathbf{F}$ is the material deformation gradient, and $\mathbf{b}$ represents the specific volume forces. To solve the problem in the finite-element context numerically, the weak form of Eq. (14) is formulated based on the principle of the virtual work. To derive the principle of virtual work, Eq. (14) is multiplied by an arbitrary test function (chosen here as the variation of the displacement field $\delta \mathbf{d}$) and integrated over the volume of the structural domain Ω_s . In a state of equilibrium, the resulting virtual work δW that is performed by the inertial, internal, and external forces of the system due to the virtual displacement field $\delta \mathbf{d}$ shall be zero. The weak form then reads:

$$\begin{aligned} \delta W = & \underbrace{\int_{\Omega_s} \rho_s \frac{\partial^2 \mathbf{d}}{\partial t^2} \cdot \delta \mathbf{d} d\Omega_s}_{\delta W_{\text{dyn}}} + \underbrace{\int_{\Omega_s} \delta \mathbf{E} : \mathbf{S} d\Omega_s}_{\delta W_{\text{int}}} \\ & - \underbrace{\left(\int_{\Gamma_s} \hat{\mathbf{T}} \cdot \delta \mathbf{d} d\Gamma_s + \int_{\Omega_s} \rho_s \mathbf{b}_s \cdot \delta \mathbf{d} d\Omega_s \right)}_{\delta W_{\text{ext}}} = 0, \end{aligned} \quad (15)$$

where $\delta \mathbf{E}$ is the virtual strain and $\hat{\mathbf{T}} = \mathbf{n} \cdot (\mathbf{F} \cdot \mathbf{S})$ is the traction on the boundary Γ_s . Given the weak form, a spatial and a temporal discretization is conducted to construct a non-linear system of equations that can be solved in an iterative manner. The continuous displacement field $\mathbf{d}(\mathbf{x})$ is approximated by interpolating the discrete nodal displacements $\bar{\mathbf{d}}$ with the help of shape functions $N(\theta^1, \theta^2, \theta^3)$ (where θ is the curvilinear convective coordinate):

$$\mathbf{d}(\theta^1, \theta^2, \theta^3) \approx \sum_{i=1}^{n_{\text{nod}}} N_i(\theta^1, \theta^2, \theta^3) \bar{\mathbf{d}}_i. \quad (16)$$

Here, n_{nod} is the number of nodes within the finite element. Inserting Eq. (16) in Eq. (15) yields the semi-discrete form:

$$\mathbf{M} \ddot{\bar{\mathbf{d}}} + \mathbf{f}^{\text{int}}(\bar{\mathbf{d}}) = \mathbf{f}^{\text{ext}}, \quad (17)$$

where $\mathbf{M}$ is the mass matrix, $\mathbf{f}^{\text{int}}$ is the internal force (non-linear function of $\bar{\mathbf{d}}$), and $\mathbf{f}^{\text{ext}}$ is the external force. The damping effect in the structure is added by adding a damping matrix $\mathbf{C}$ in Eq. (17) as:

$$\mathbf{M} \ddot{\bar{\mathbf{d}}} + \mathbf{C} \dot{\bar{\mathbf{d}}} + \mathbf{f}^{\text{int}}(\bar{\mathbf{d}}) = \mathbf{f}^{\text{ext}}. \quad (18)$$

Given the significance of internal structural damping in such scenarios, the well-established Rayleigh damping method is applied. The constant mass- and stiffness-proportional parameters (α_r , β_r) of the Rayleigh damping are evaluated based on additional experimental structural tests, as detailed in Appendix A.

For time integration, the structural response is discretized using the standard second-order Newmark scheme. This method ensures accurate prediction of dynamic structural responses under fluid-induced loads.

2.4. Coupling procedure

The simulation framework for addressing the FSI problem is based on a partitioned two-way coupling procedure [15], as sketched in Fig. 1. This method integrates the fluid solver (FASTEST-3D, discussed in Section 2.1) with the computational structure dynamics (CSD) solver (Carat++, Section 2.3). The coupling between these two solvers, as well as the mapping of data between the non-matching CFD and CSD surface grids at the FSI interface, is managed by the open-source software

EMPIRE (Enhanced Multi Physics Interface Research Engine, [44]). The mapping process, critical for accurate data transfer between the CFD and CSD solvers, utilizes a mortar mapping technique suitable for the FEM discretizations. Comprehensive details on the mortar mapping approach can be found in the works of Wang et al. [45] and Apostolatos et al. [46]. The framework supports both standard loose and strong coupling schemes, ensuring flexibility in handling various FSI scenarios. All data exchange between the fluid and structural solvers relies on message passing interface (MPI) communications, which ensures efficient parallel processing and synchronization of the coupled simulations.

This FSI simulation framework without IBM has been validated using a variety of test cases, including experimental FSI benchmarks, i.e., De Nayer et al. [47], De Nayer et al. [48], and De Nayer and Breuer [49] to cite just the latest studies. These studies provide a robust basis for the reliability and accuracy of the FSI framework. However, it is important to note that the IB technique, integrated to the methodology in this study, was validated separately in Boulbrachene and Breuer [13] and Boulbrachene and Breuer [50]. What makes the current study distinctive is the integration of the FSI simulation framework with the IB technique – a combination that has not been previously applied. This novel integration is a special feature of this research, allowing for the accurate simulation of complex fluid–structure interactions where additional moving boundaries are involved, as in the case of the WGG discussed earlier. The successful application of this combined methodology (see Fig. 1) demonstrates its potential for advancing the simulation of dynamic FSI problems.

3. Test case configuration

The current numerical study is based on two aforementioned experimental investigations: the implementation and testing of the wind gust generator in the wind tunnel conducted by Wood et al. [1], and the synchronized measurements of the T-structure under wind gust load in the wind tunnel by Breuer and Neumann [12] (see Fig. 2). These studies serve as the experimental basis for validating the numerical simulations in this work. The relevant physical parameters¹ will be described next.

3.1. Physical parameters

The subsonic wind tunnel with an open test section of the length l has a rectangular nozzle outlet (width $w \times$ height h). The parameters relevant to the fluid flow and the gust generation are summarized in Table 1. The wind tunnel blower is operated at a low speed, resulting in a low free-stream velocity u_∞ . To generate a horizontal gust, the paddle of the WGG performs a rapid downward and upward movement, thus obstructing the nozzle outlet for a short period of time Δt_{gust} , denoted the gust generation time. When the paddle reaches its maximum displacement Δy_{max} , the maximum blocking ratio BR of the nozzle is attained, which approximately coincides with the instant at which the maximum velocity of the artificially generated horizontal gust is achieved. Wood et al. [1] investigated five different motion patterns of the paddle and analyzed their effect on the gusts generated. The present study focuses on gust type 1, which features a fast and symmetric motion pattern designed to mimic the 1-cosine shape of the ECG gust as defined by IEC-Standard [6]. The characteristic parameters of the gust generated by this motion pattern are outlined in Table 1. It is important to note that the chosen gust case leads to a strong variation of the velocity at the outlet of the nozzle with a maximum gust velocity, which is about 63 % higher than the free-stream velocity [1]. For detailed

¹ Note that the experimental study by Breuer and Neumann [12] and the numerical study by Boulbrachene and Breuer [13] use different coordinate systems. In this study, we follow the latter, where the direction normal to the ground plate is defined as the y -axis.

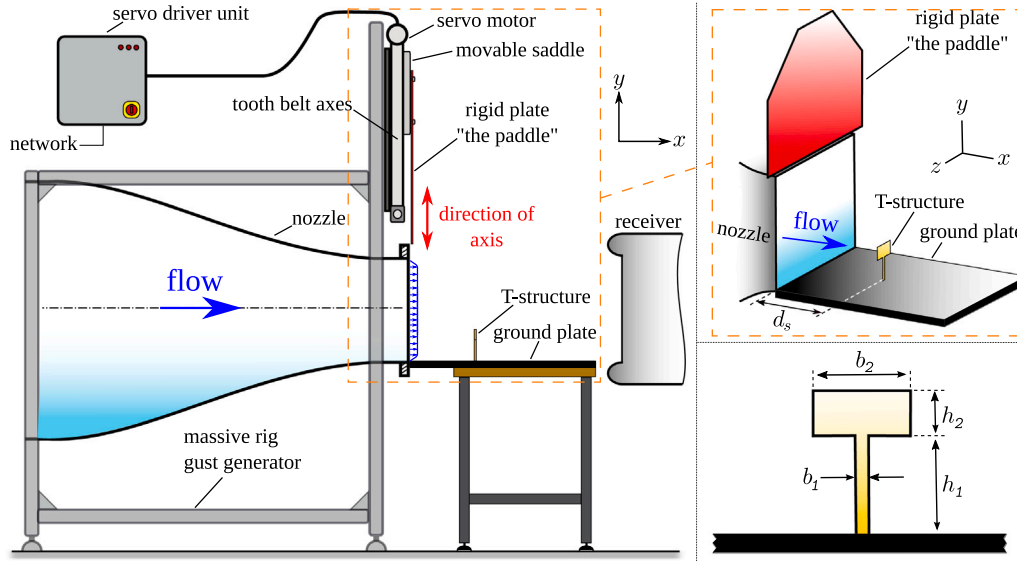


Fig. 2. Sketch of the wind tunnel with the open test section, the wind gust generator, and the T-structure mounted on the ground plate.

Table 1
Fluid flow and gust generation parameters.

Free-stream velocity	u_{∞}	5.14	m/s
Density ($\theta = 20^\circ\text{C}$)	$\rho_f = \rho_{\text{air}}$	1.225	kg/m ³
Dynamic viscosity	μ_{air}	18.27×10^{-6}	kg/(m s)
Width of the nozzle outlet	w	0.5	m
Height of the nozzle outlet	h	0.375	m
Test section length in wind tunnel	l	0.8	m
Maximum displacement of paddle in flow	Δy_{max}	0.1437 (0.145) ^a	m
Maximum blocking ratio	$BR = \Delta y_{\text{max}}/h$	0.387	–
Duration of paddle in flow	Δt_{gust}	0.352 (0.356) ^a	s
Extremal downward velocity	v_{WGG}^d	–1.05	m/s
Extremal upward velocity	v_{WGG}^u	+0.95	m/s
Extremal downward acceleration	a_{WGG}^d	–20	m/s ²
Extremal upward acceleration	a_{WGG}^u	+15	m/s ²

^a The values in brackets are the original values provided in Wood et al. [1] based on the prescribed values of the tooth belt axis in the experimental setup. They slightly deviate from the values adjusted in Boulbrachene and Breuer [13] and employed in this study to better match the real measured values in the experiment.

time histories of the paddle's kinematics – displacement, velocity and acceleration – please refer to Fig. 4 in Wood et al. [1].

Designed as a benchmark case, a geometrically simple T-shaped structure is mounted perpendicularly on the flat and smooth ground plate, which serves as the bottom boundary of the wind tunnel's test section, as sketched in Fig. 2. The structure consists of a thin sheet of brass (alloy CuZn37) with the thickness t_s . The dimensions of the rectangular thin rod (b_1 , h_1) and the shield-shaped part at the top (b_2 , h_2) are given in Table 2. The structure is mounted in the symmetry plane of the test section at a distance of d_s from the nozzle's outlet. Gravitational acceleration with $|g| = 9.81 \text{ m/s}^2$ in negative y -direction is considered. The material properties such as the density ρ_s , the Young's modulus E_s and the Poisson's ratio ν_s given in Breuer and Neumann [12] were estimations based on literature values. Thorough investigations on the material used in the experiment are detailed in Appendix A. These tests showed that the value of the Young's modulus assumed in Breuer and Neumann [12] was too high. The revised value is also provided in Table 2. It is worth noting that this initial false estimation of the Young's modulus has no influence on the measurement data [12] of the flow field and the structural deformation used for comparison purpose in the present study.

The Reynolds number for the undisturbed free-stream flow around the T-structure, calculated based on the entire height $h_s = h_1 + h_2$ and the properties provided in Table 1, is approximately $\text{Re} = 41,360$.

Table 2
Properties of the T-structure.

Width of rectangular thin rod	b_1	9.4×10^{-3}	m
Height of rectangular thin rod	h_1	80×10^{-3}	m
Width of shield-shaped part	b_2	80×10^{-3}	m
Height of shield-shaped part	h_2	40×10^{-3}	m
Total height of the structure	$h_s = h_1 + h_2$	0.12	m
Thickness of structure	t_s	0.5×10^{-3}	m
Distance between structure and nozzle	d_s	0.25	m
Density ($\theta = 20^\circ\text{C}$)	ρ_s	8.435×10^3	kg/m ³
Young's modulus	E_s	93.7×10^9	N/m ²
Poisson's ratio	ν_s	0.34	–

3.2. Numerical parameters

3.2.1. Computational domain and boundary conditions

The complex wind tunnel geometry depicted in Fig. 2 is simplified to model the nozzle, the ground plate and the moving paddle as shown in Fig. 3. For this purpose, the geometry of the nozzle is reduced to a straight duct of length 1.5 m neglecting the nozzle's contraction ratio. The rectangular cross-section of the duct matches the nozzle's outlet in the experimental setup. A uniform velocity distribution with $u_{\text{inlet}}^{\text{steady}} = 4.914 \text{ m/s}$ is applied at the inlet patch. As the boundary layers develop along the walls of the duct, the flow reaches the desired free-stream

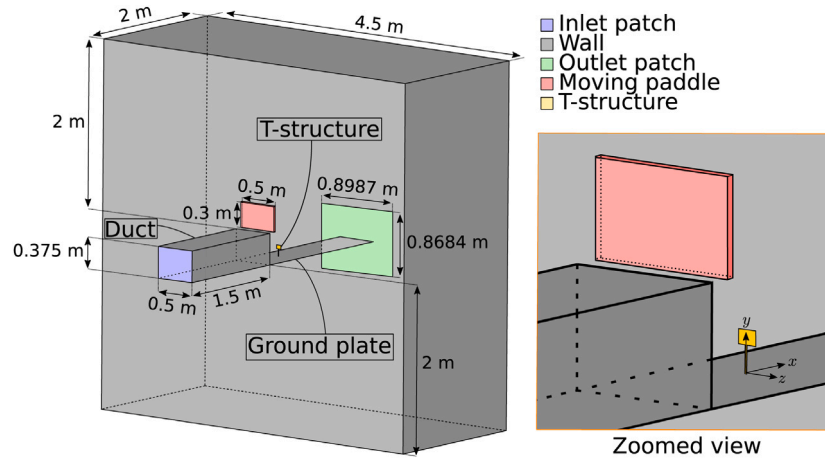


Fig. 3. Geometry of the test case and definition of the boundary conditions.
Source: Schematic sketch adapted from Boulbrachene and Breuer [13].

velocity $u_\infty = 5.14$ m/s at the outlet, consistent with the experimental conditions.

This description of the inlet velocity applies to the scenario where the wind tunnel is operated without WGG. However, Wood et al. [1] observed an undershoot of the streamwise velocity below the free-stream velocity at the nozzle's outlet during and after the paddle encounter. This drop in the inlet velocity is attributed to the paddle acting as a bluff body, obstructing the fluid flow and thus causing significant pressure losses. These increased pressure losses cannot be compensated by the blower in the short term. Consequently, less air is blown through the wind tunnel manifesting as an undershoot below the free-stream velocity. Further experimental tests in Wood et al. [1] demonstrated that the recovery time for the free-stream velocity to be reestablished is much longer than Δt_{gust} . To account for this inlet velocity drop, Constant Temperature Anemometry (CTA) measurements recorded inside the nozzle during the gust generation process were utilized in Boulbrachene and Breuer [13] to define a time-dependent inlet boundary condition. The specifics of the inlet velocity function including the CTA measurements on which it is based are detailed in Boulbrachene and Breuer [13]. Additionally, no-slip and impermeability boundary conditions are imposed on the walls of the duct, the ground plate, and the T-structure.

The duct is connected to a large container that extends 2 m in all directions from the duct's outlet. This container, measuring $2 \text{ m} \times 4.375 \text{ m} \times 4.5 \text{ m}$, includes the test section, which contains the ground plate and the mounted T-structure, as well as the immediate surrounding area of the open measuring section. To simulate the real wind tunnel environment, no-slip and impermeability boundary conditions are applied at the far-field walls of this container, except at a small outlet patch that models the receiver of the wind tunnel. At this outlet, a convective boundary condition is employed, with the convective velocity set to $u_{\text{conv}} = u_{\text{inlet}}^{\text{steady}}$, ensuring the proper outflow of air from the domain.

3.2.2. Computational grids

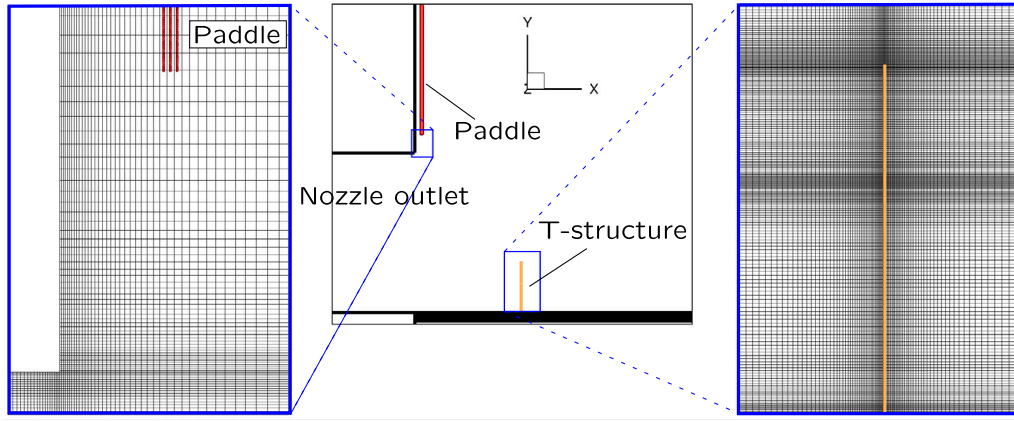
Given the geometric simplification of the wind tunnel's nozzle and the specific T-structure under consideration, the computational domain can be represented using a Cartesian grid, as long as the flexible structure remains undeformed by the flow field. This allows for the use of an initial Eulerian grid, which is a block-structured Cartesian grid consisting of approximately 117.5×10^6 control volumes (CVs). Compared to the preliminary validation study [13] of the flow in the empty test section without the T-structure, which showed good agreement between the data predicted by the immersed boundary method and the experimental data [1], the grid in the current study

comprises a factor of about 3 more CVs to account for an appropriate resolution of the flow around the T-structure.

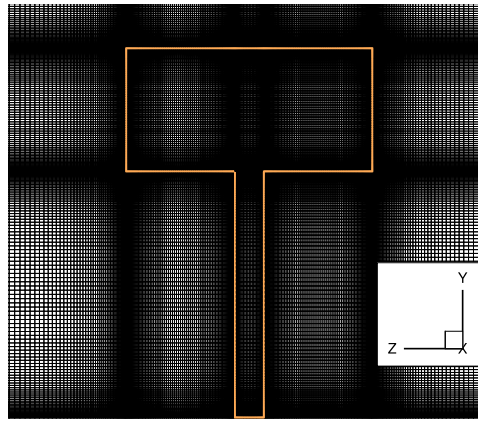
The grid design and distribution of grid points across the computational domain in the current case are motivated by the following considerations. The geometric block-structure comprises 120 blocks, which are further subdivided into a parallel block-structure consisting of 942 blocks. This subdivision enables parallel execution across 942 computational cores. Grid refinement near relevant rigid walls (i.e., duct walls, ground plate and T-structure) is essential to capture the significant viscous effects leading to the formation of thin boundary layers. The goal is to position at least some nodes (i.e., the centers of the cells) within the viscous sublayer ($y^+ \leq 5$). This careful refinement allows to accurately resolve the near-wall turbulence and boundary layer effects, which is critical for the fidelity of the simulation results. To ensure this, the following considerations were made.

The hydraulic diameter of the rectangular duct, calculated as $D_h = 4wh/(2(w+h)) = 0.4286$ m, serves as the characteristic length for determining the Reynolds number $\text{Re}_{D_h} = \rho_{\text{air}} D_h u_\infty / \mu_{\text{air}} = 147,701$. Assuming a smooth pipe, the Darcy friction factor can be estimated based on the relation provided by Moody [51], yielding a skin friction coefficient c_f of approximately 3.97×10^{-3} . To accurately resolve the near-wall region, the height of the first grid cell off the wall (Δy) is crucial. The grid spacing must ensure that the dimensionless wall distance y_{cc}^+ at the cell center is approximately 2, a value suitable for capturing near-wall effects. This can be calculated using the relation $\Delta y/D_h = (4/\text{Re}_{D_h}) \cdot \sqrt{2/c_f} \approx 6 \times 10^{-4}$, which results in a required grid spacing of $\Delta y \approx 2.6 \times 10^{-4}$ m. This grid spacing is applied along the walls of the duct, the ground plate, and in the vicinity of the T-structure (see Figs. 4(a) and (b)). This careful design of the grid allows the simulation to accurately capture the complex interactions between the fluid and the structure, particularly in regions of high shear stress and strong velocity gradients near solid boundaries.

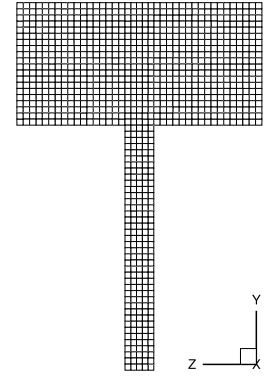
To reduce the computational costs, the grid is mildly stretched in the off-wall directions with an expansion factor of about 1.05. Nearly the same expansion factor is also applied in the streamwise direction. Starting with the finest grid resolution at the outlet of the nozzle, where the paddle is located, the grid is mildly coarsened in upstream and downstream direction. Furthermore, in the vicinity of the flexible structure the same grid refinement is applied in all directions. Note that the current grid was already used in the preliminary study by Boulbrachene and Breuer [50] and the predicted results for the fluid flow induced by the gust generator but without considering the T-structure were validated against the experimental data by Wood et al. [1]. Beside the validation study [13], this is a further evidence that the presently applied grid is appropriate.



(a) Zoom of the nozzle outlet with the Eulerian grid and the uniformly distributed Lagrangian markers sketched in red (left). Zoom of the Eulerian grid (x - y -plane) surrounding the T-structure sketched in orange (right).



(b) Eulerian fluid grid on and surrounding the T-structure (y - z -plane).



(c) T-structure FE grid.

Fig. 4. Details on the fluid and structure grids.

Based on the free-stream velocity, the stability of the explicit time integration scheme is ensured by choosing a time step size of $\Delta t = 10^{-5}$ s leading to a Courant–Friedrichs–Lewy number of less than 0.43. The same time step is set for the CSD solver.

For the structural dynamics simulation, the thin, flexible T-structure is discretized by 980 quadrilateral four-node shell elements, resulting in a total of 1080 nodes for the standard grid, as visible in Fig. 4(c). This choice is based on a grid sensitivity study detailed in Appendix B considering a static and a dynamic test case. Taking two finer grid levels into account, this investigation confirms that the standard grid leads to accurate results making further refinement and the associated increase of the already high computational effort superfluous. The elements used are special seven-parameters shell elements [52]. To avoid locking phenomena, the Enhanced Assumed Strain (EAS) method as proposed by Bischoff et al. [53] is employed. Specifically, the EAS parameters are set to $(M, B, T, Q, S) = (0, 0, 4, 4, 0)$. The Rayleigh damping values mentioned in Section 2.3 are set to $\alpha_r = 0.47383$ Hz and $\beta_r = 2.1602 \times 10^{-8}$ s based on preliminary studies detailed in Appendix A.

3.2.3. Lagrangian markers

The geometry of the thin moving paddle, with a thickness of 3×10^{-3} m, is simplified to a parallelepiped with right angles, measuring $0.3 \text{ m} \times 0.5 \text{ m}$. This rectangular cross-section penetrates the flow exiting the nozzle and is positioned 0.015 m away from the nozzle outlet in accordance with the experiment. The spacing between the Lagrangian markers that describe the paddle is carefully chosen based on the minimum local Eulerian grid spacing encountered by the paddle

during its closing and opening stroke (see Fig. 4(a)). This strategy is crucial to prevent any fluid from penetration through the surface of the paddle due to insufficient Eulerian cells coverage by the Lagrangian markers' window function. In each coordinate direction, the minimum local Eulerian grid spacing is determined leading to the required number of uniformly distributed Lagrangian markers. As shown in Boulbrachene and Breuer [13], this procedure leads to a very high number of required Lagrangian markers, significantly slowing down the computations. This is because only a limited number of blocks contain Lagrangian markers and have to conduct the interpolation, spreading and tracking processes. A one-dimensional non-uniform distribution in spanwise direction allows to reduce the number of Lagrangian points from 6.7 million (uniform distribution) to about 1.57 million.

In this enhanced variant, the Lagrangian markers maintain a uniform distribution along the x - and y -axes. However, along the z -axis, the distribution is defined in such a way that for each cell center a Lagrangian marker is assigned, following the grid stretching employed along the z -axis. This strategy is chosen because, during the paddle's opening and closing stroke, the distribution of Eulerian points along the z -axis does not change. By matching the Lagrangian marker distribution with the Eulerian grid stretching, the simulation ensures accurate interaction between the fluid and the paddle while minimizing unnecessary computational costs. This tailored distribution of markers optimizes the computational efficiency without compromising the fidelity of the simulation.

In addition to specifying the initial positions of the markers, the vertical velocity of the paddle as a function of time must also be

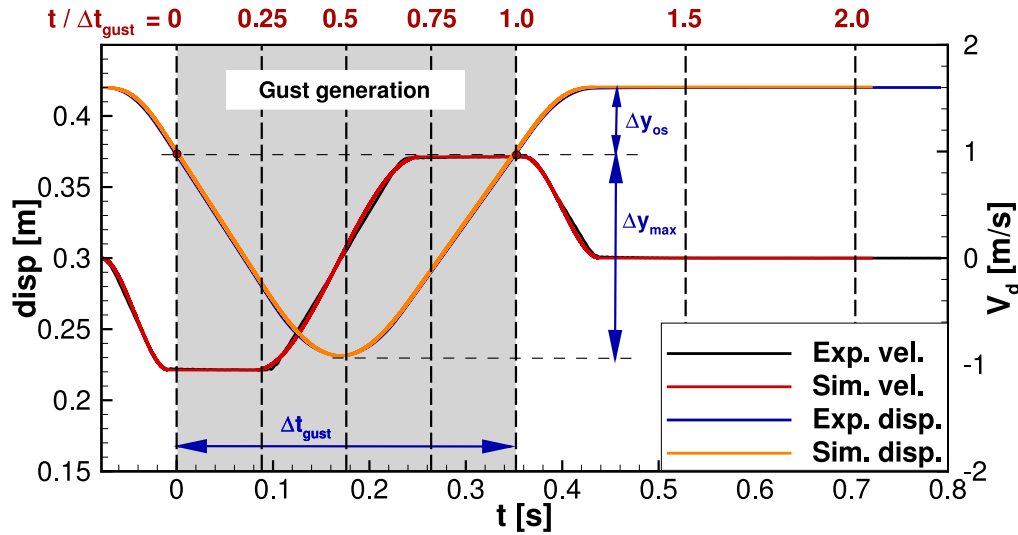


Fig. 5. Kinematic characteristics of the fitted gust type 1 paddle movement pattern.
Source: Adapted from Boulbrachene and Breuer [13].

defined to describe its movement. For the gust *type 1* considered in this study, Boulbrachene and Breuer [13] fitted a smooth (differentiable) analytical function to the vertical velocity curve derived from the experimental data, as shown in Fig. 5. In accordance with the experiment, an offset displacement of $\Delta y_{os} = 0.045$ m is included to account for the distance the paddle must travel from its idle position to start blocking the nozzle's outlet. This measure prevents unwanted initial interactions between the paddle and the undisturbed free-stream flow. Additionally, it facilitates a gradual buildup of the paddle's velocity prior to the interaction. A bell-shaped displacement pattern is prescribed, allowing the paddle to block a maximum of $\Delta y_{max} = 0.1437$ m of the nozzle's outlet, with a total interaction time of $\Delta t_{gust} = 0.352$ s. Following this, the paddle returns to its idle position. The present values of Δy_{max} and Δt_{gust} differ slightly from those reported in Wood et al. [1]. The reason is that the latter correspond to the prescribed values for the tooth belt axis in the experimental setup, whereas the present values more accurately reflect the actual measured motion pattern [13]. As described in Section 2.1, the displacement of the paddle is a derived quantity. Consequently, in the simulation, it is sufficient to prescribe the vertical velocity of the paddle.

4. Results and discussion

For the discussion of the results, the Cartesian coordinate system introduced in the zoomed view of Fig. 3 is used. The origin is located in the symmetry plane at the clamping point of the T-structure on the ground plate. The coordinates are normalized by the height h_s of the entire T-structure, and the velocities by the free-stream velocity u_∞ . The time t is made dimensionless by the gust generation time Δt_{gust} , as detailed in Section 3.1. The time instant, at which the paddle starts to obstruct the outlet of the nozzle initiating the gust generation process, serves as the reference point in time defining $t = 0$ for the subsequent analysis.

The discussion of the results is organized as follows: To understand the flow around the T-structure, Section 4.1 begins with an examination of the flow field and the structural deflections in the case without a gust as an additional validation of the FSI framework. The analysis of the wind gust and its impact on the structure throughout the entire process is presented in Section 4.2. For both scenarios, a comparison with the available experimental data is carried out.

4.1. Flow field and structural displacements without gust

In Breuer and Neumann [12], time-resolved measurements were repeated 104 times under identical operational conditions of the wind tunnel and wind gust generator, enabling phase-averaging of both the flow and the structural data. The data recorded prior to the gust injection, i.e., $t/\Delta t_{gust} \leq 0$, are used to analyze the flow field without the influence of the gust. This part of the study can also be considered as an additional validation study for the ALE framework complementing the previously mentioned studies [13,50] for the case including the flexible T-structure.

As described in detail below, the deflection of the T-structure, and particularly the amplitude of its oscillations, is minimal in the absence of a gust. This allows the flow field to be time-averaged in the numerical simulation, facilitating comparison with the phase-averaged experimental data. For this purpose, the flow field was averaged over 1 million time steps, corresponding to a time interval of 10 s. Assuming that the T-structure approximately oscillates at its first dominant eigenfrequency $f_1^{num} = 6.826$ Hz, as determined for the bending mode in Appendix A, this time interval corresponds to approximately 68 oscillations, which is sufficient to ensure statistical convergence.

Fig. 6 presents contours of the averaged streamwise and vertical velocity fields in the geometric symmetry plane ($z = 0$). Additionally, streamlines calculated for the averaged flow in this plane are depicted for both the measured and the predicted data. The flow decelerates in front of the structure and deflects both in upward and downward direction, as evident in the contours of the vertical velocity component. The resulting stagnation region is clearly visible in the streamwise velocity contours. Both the rectangular thin rod and the shield-shaped section act as bluff bodies, disrupting the flow. Flow separation occurs at the sharp corners of the T-structure, generating a large recirculation region, which is especially prominent behind the shield-shaped part. A black contour line with the streamwise velocity $\bar{u} = 0$ marks the recirculation region in the symmetry plane, further highlighted by the streamlines of the time-averaged flow. The length of this backflow region is approximately 1.25 times the total height h_s of the structure. Comparing the numerical and the experimental data depicted in Fig. 6, the predicted streamwise and vertical velocity components generally align well with the measured data. However, due to reflections of the laser light used for the particle-image velocimetry, velocities in the immediate vicinity of the ground wall could not be determined. Furthermore, the comparison clearly reveals that the statistics of the

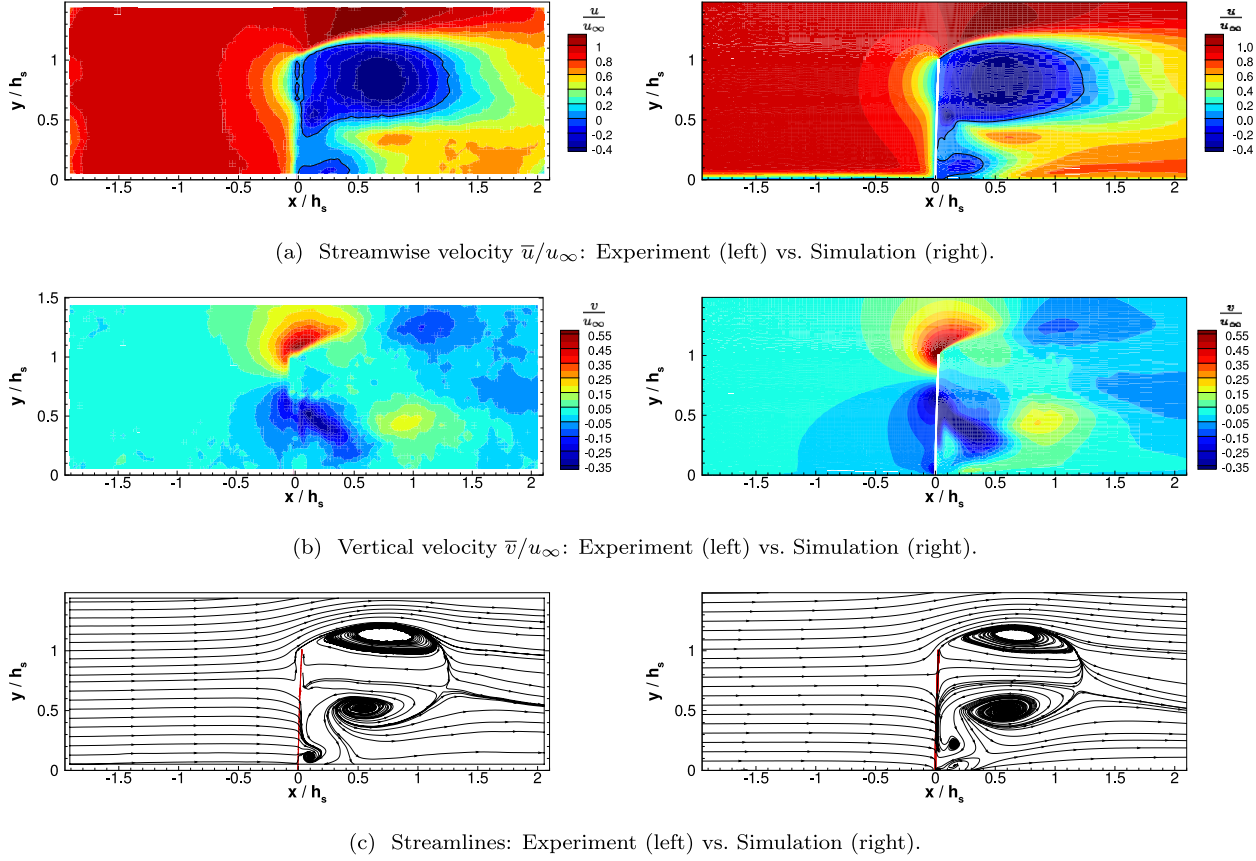


Fig. 6. Comparison between measured and predicted data of the averaged flow field in the symmetry plane: streamwise velocity, vertical velocity, and streamlines. Source: Experimental data taken from Breuer and Neumann [12].

simulation data have converged better than the measurements. Considering that the experimentally determined mean values are based on only 104 independent measurements, while in the simulations the flow field was continuously averaged over a time interval of 10 s including 10^5 samples, it is obvious why the predicted distributions are smoother than the measured data. This minor discrepancy arises due to the fact that the experiment was not designed to determine time-averaged quantities, but rather aimed to capture the instantaneous and statistically unsteady processes of wind gusts. Thus, small differences in the streamlines based on the averaged data can be expected. Despite this issue, the shape and size of the recirculation region in both data sets show good agreement.

The three-dimensional extension of the recirculation region can be clearly seen in Fig. 7(a), which shows an iso-surface of the predicted flow field based on $\bar{u} = 0$. Additionally, Figs. 7(b) and 7(c) depict contours of the streamwise velocity and the pressure at three y - z -slices in the wake ($x/h_s = 1/6, 2/3$, and $4/3$). At the slice $x/h_s = 2/3$, located approximately in the middle of the backflow region, both the velocity and pressure contours display a nearly circular shape, and the region extends well beyond the width of the T-structure in the spanwise z -direction. Also notable are the nearly circular wall streamlines around the T-structure on the ground plate visible in Fig. 7, which form due to the blockage of the flow field. A backflow area near the ground plate, directly in front of the rectangular section, can also be observed. This is attributed to the horseshoe vortex system forming in front of bluff bodies, a phenomenon commonly observed in flows around wall-mounted obstacles such as cubes [e.g., 18] or hemispheres [e.g., 54]. Surprisingly, the horseshoe vortex system forms relatively far upstream of the T-structure, despite the thin profile of the rod and the minimal blockage of the flow field near the bottom wall.

The good agreement between experiment and simulation is also evident in the velocity profiles extracted along a vertical line in front of the T-structure ($x/h_s = -1/6$) and three vertical lines in the wake ($x/h_s = 1/6, 2/3$, and $4/3$). Fig. 8 shows the time-averaged streamwise and vertical velocity components in the symmetry plane. Four dotted lines denote the four positions in the flow field where the velocities are compared. For reference, the T-structure is drawn as a red line. Each profile is plotted at its respective position which explains the shift $+x/h_s$ used in the caption of Fig. 8. Due to the different ranges of the two velocity components, the profiles are scaled with different factors for improved visibility, e.g., 0.3 for the streamwise component and 0.8 for the vertical component. Minor deviations are observed, especially for the vertical component, which is significantly smaller than the streamwise velocity.

The predicted Reynolds stresses in the symmetry plane are depicted in Fig. 9, showing all three normal components and the most significant shear stress component $\overline{u'v'}$. The strong shear layers forming in the wake of the shield-shaped part of the T-structure are clearly visible, particularly in the normal Reynolds stress component $\overline{u'u'}$ and the Reynolds shear stress $\overline{u'v'}$. The normal vertical Reynolds stress $\overline{v'v'}$ is notably large in the region where the end of the recirculation area is located. In Breuer and Neumann [12], the Reynolds stresses presented were limited to the two normal components ($\overline{u'u'}$, $\overline{v'v'}$) and the shear stress component ($\overline{u'v'}$), as the 2D PIV measurements could only capture velocities within the plane of the laser light sheet.² Furthermore, the distributions were not very smooth due to the 104 measurements available being insufficient for fully converged higher-order moments.

² Note the different coordinate system in Breuer and Neumann [12].

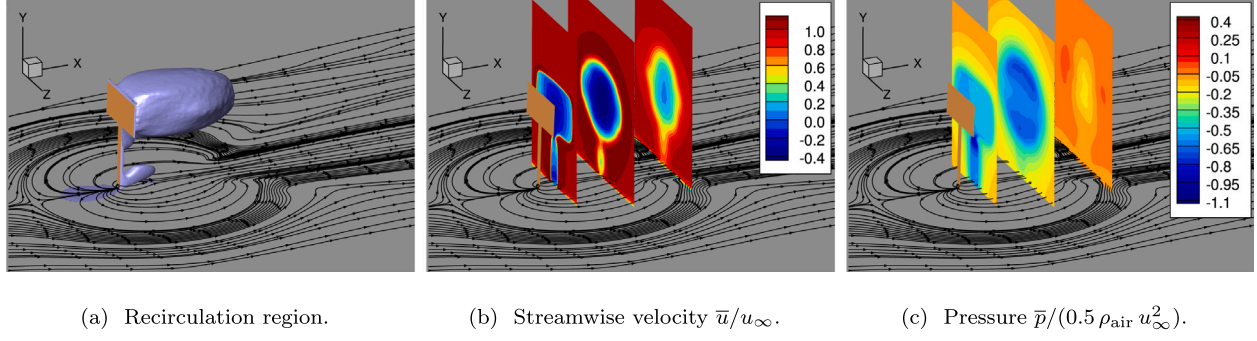


Fig. 7. Predicted time-averaged three-dimensional flow field in the wake of the T-structure: Slices at $x/h_s = 1/6, 2/3, \text{ and } 4/3$. Wall streamlines at the ground plate.

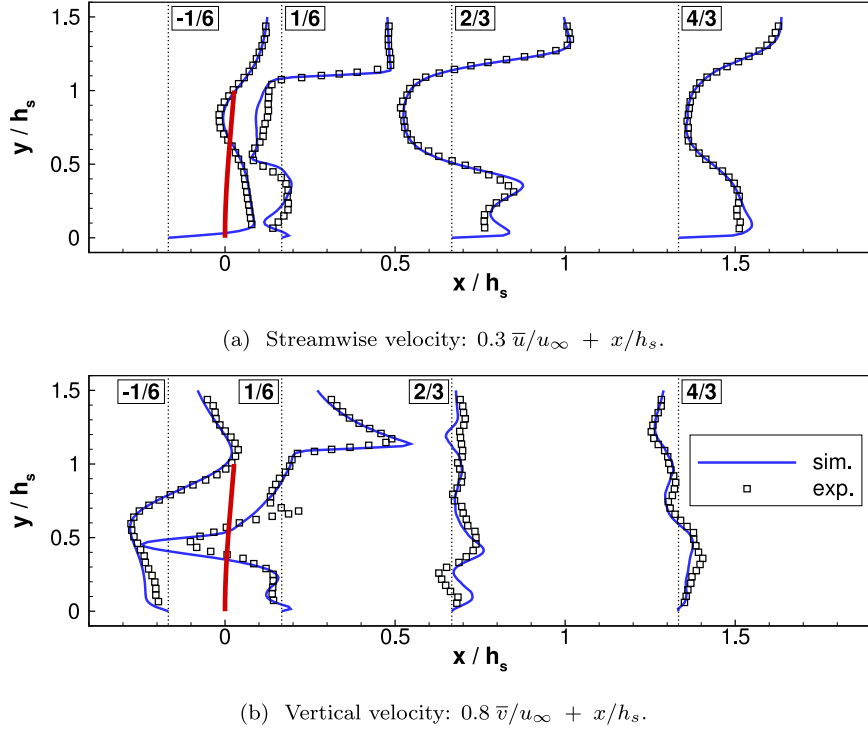


Fig. 8. Comparison of predicted and measured time-averaged velocity profiles at four positions in the flow field in front of and behind the T-structure depicted as a red line: $x/h_s = -1/6, 1/6, 2/3, \text{ and } 4/3$.

Source: Experimental data by Breuer and Neumann [12].

Consequently, a direct comparison between the experimental and simulated results is not meaningful. However, qualitatively, there is a high level of agreement between the simulation and the measurement data for the Reynolds stresses.

Finally, a comparison of the mean deformation between the experiment and the simulation is carried out. Fig. 10(a) shows the averaged deformation of the T-structure in the symmetry plane. The bending of the structure under the fluid load is clearly visible. Due to the small thickness of the T-structure, shear forces play a marginal role for the resulting fluid force in streamwise direction. In contrast, the stagnation pressure with a pressure coefficient c_p close to one in the majority of the front face (Fig. 10(b)) and the low-pressure recirculation region with negative c_p values in the back (Fig. 10(c)) lead to the dominance of the pressure forces. Nevertheless, the deflections remain overall small. For instance, the center of the shield-shaped part of the T-structure, denoted as point 1 in [12], deflects by only about 2.45 % of h_s , and for the upper edge of the structure, the deflection is approximately 3.2 %. As visible in Fig. 10(a) the predicted data agree well with the measurements. Note the different ranges of values assigned to the axes, which makes the

minor deviations appear much larger than they actually are, i.e., in the order of a few micrometers.

To analyze the deflection behavior of the T-structure for the flow case without gust, Fig. 11(a) presents selected excerpts from the time histories of the structural displacements of point 1 for both the measurement and the simulation. The experimental data clearly reveal two oscillation modes between which the coupled system switches back and forth. In the time period shown, a phase of about 4 s is initially visible, characterized by relatively large oscillation amplitudes. This is followed by a phase of about 6 s, in which the amplitudes have noticeably decreased. This phenomenon is succeeded by another and even longer phase with large amplitudes. Such an irregular switching between the modes with distinctly different amplitudes is also observed in the simulations. A thorough investigation of the deflection behavior, using animations of the predicted results, shows very clearly that there are two different vibration modes. Evidence for this statement is provided by Fast Fourier transformations of the deflections discussed below. If predominantly large vibration amplitudes are observed, the structure is in the bending-dominated mode. If, on the other hand, the amplitudes become small, the twisting mode becomes dominant,

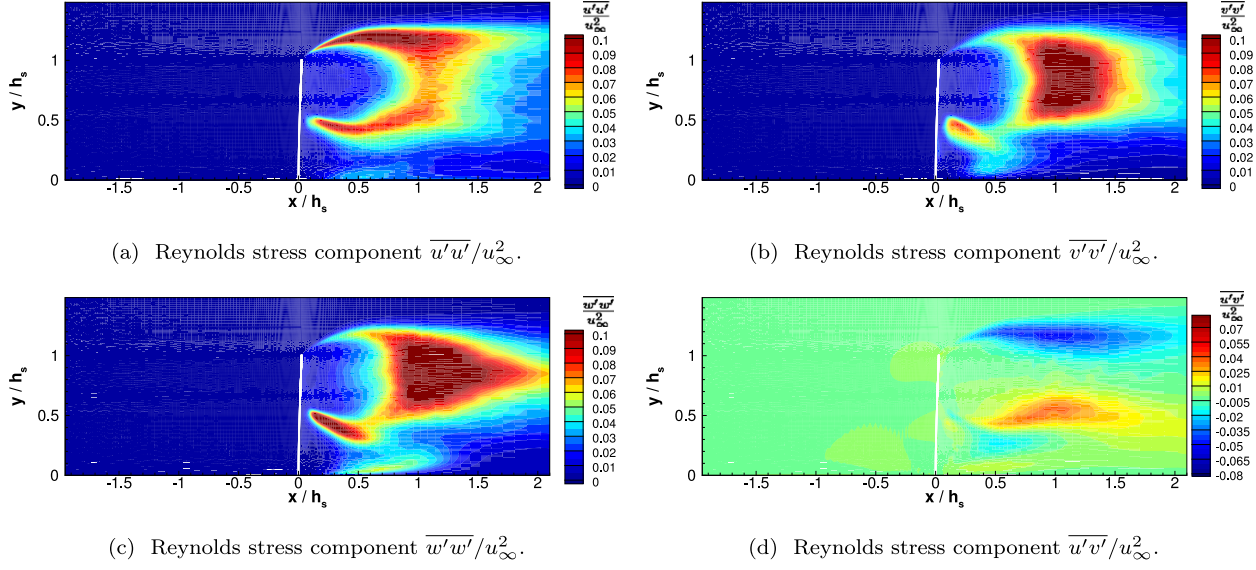


Fig. 9. Predicted Reynolds stress components in the symmetry plane.

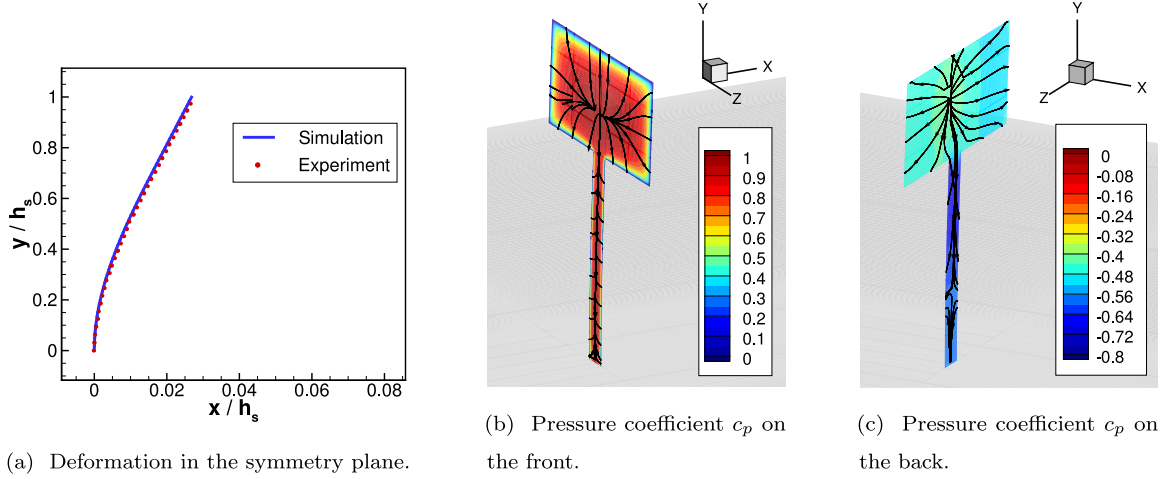


Fig. 10. (a) Comparison of predicted and measured time-averaged deformation of the T-structure in the symmetry plane; Time-averaged distribution of the pressure coefficient $c_p = (p - p_\infty)/(0.5 \rho_{\text{air}} u_\infty^2)$ on the front side (b) and the back side (c) of the structure including wall streamlines. Source: Experimental data from Breuer and Neumann [12].

though bending does not disappear completely. The oscillation mode changes in an unpredictable sequence, with both modes occasionally occurring simultaneously, as evident in the animations of the structure.

Fast Fourier transformations of the deflections for both simulated and measured data are depicted in Fig. 11(b). Two dominant frequencies are found ($f = 6.34$ Hz and 6.68 Hz) in both data sets. These frequencies are close to the natural bending frequency of approximately 6.82 Hz, as determined in Appendix A. A detailed analysis of the predicted data reveals that the stronger peak at $f = 6.68$ Hz corresponds to the oscillation mode dominated by the bending of the structure. The slight reduction in frequency compared to the pure bending frequency in vacuum is attributed to the additional damping caused by the surrounding fluid. The second peak in the FFT at $f = 6.34$ Hz is associated to the state when the amplitudes of the oscillations are found to be significantly smaller and the twisting mode plays a prominent role in the kinematics of the T-structure. In this case, a FFT of the deflections of a point on the T-structure outside the symmetry plane depicted in Fig. 12 also shows a perceptible peak at the natural frequency of the second mode (see f_2^{num} in Appendix A), i.e., the twisting mode.

It can be observed that the amplitudes of the measured oscillations are larger than those achieved in the simulations, likely due to the absence of inflow turbulence in the simulations. In principle, this could have been incorporated in the coupled predictions by using a synthetic turbulence generator, as done in previous studies such as Breuer [55] and De Nayer et al. [47,48,56]. However, implementing this option would require the knowledge of the characteristic properties of the incoming turbulence, such as the relevant length scales, which were not available for this setup. Consequently, this aspect was excluded in the initial approach.

4.2. Flow field and structural displacements with gust

The analysis of the flow field and structural displacements for the case without gust carried out in the previous section provides a description of the initial state before the gust impacts the structure. A large recirculation region is found behind the shield-shaped part, extending approximately $1.25 h_s$, and the maximum deflections remain quite small.

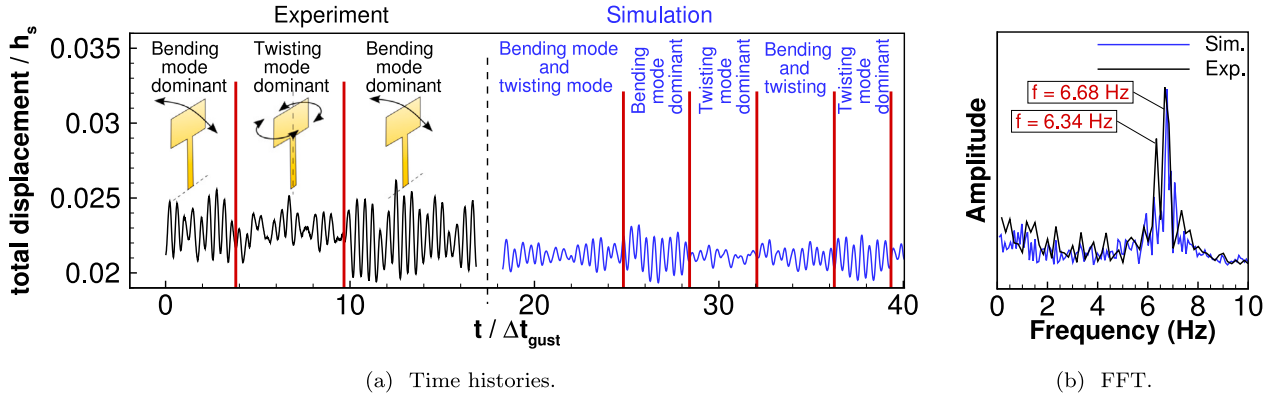


Fig. 11. Time histories and fast Fourier transformation of the total displacements of the center of the upper rectangular part of the T-structure. Predicted data and experimental data by Breuer and Neumann [12] for the case without gust.

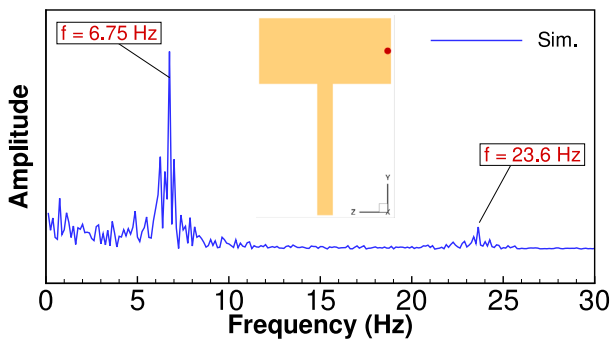


Fig. 12. Fast Fourier transformation of the total displacements of a point (red circle) on the T-structure outside the symmetry plane.

In the subsequent section, the more interesting case of the gust impact is analyzed. This scenario significantly differs from the previous one. In the former case, the incoming flow was steady, except for the very low turbulence intensity of approximately 0.2 % found in the wind tunnel by Wood et al. [54] and assumed to be valid in Breuer and Neumann [12]. This facilitated the analysis of the data based on time averaging. However, this is no longer possible in case of an active gust generator tackled in this section. The time-averaging approach is deemed invalid as the flow is statistically unsteady. Therefore, a phase-averaging procedure is required to obtain comparable numerical and experimental data.

In the experiment [12], this phase-averaging procedure relied on the synchronization of the measurement devices (PIV and DIC cameras) and the gust generator. The trigger signal for the PIV cameras was used to also trigger the paddle and the DIC cameras. Consequently, each recorded time step was comparable to the respective time step of subsequent gusts. The start of the paddle's motion was defined as $t = 0$ for both PIV and DIC recordings. The same definition is applied in the numerical simulation. Thus, the temporal reference for phase averaging is the position of the paddle during the gust generation.

As mentioned in Section 4.1, the phase averaging in the experiments relied on 104 realizations. Indeed, cycle-to-cycle variations with standard deviations of the deflections of about 5 to 10 % of the phase-averaged total deflections were observed across different repetitions of the wind gust. Besides the (low) turbulence level of the flow within the wind tunnel, the timing of the gust initiation relative to the respective oscillation state is mainly made responsible for these variations. Due to the very high numerical effort involved in the coupled simulations, solely 32 predictions are used to determine the phase averages. For this purpose, the gust generation process is initiated starting from different,

randomly chosen time instants of the flow and structural deformation predicted by the simulations without gust.

At $t = 0$, the paddle begins to obstruct the outlet of the nozzle, initiating the gust generation process. In the following, the effect of the gust on the flow field and the structural deformation is analyzed at different instants in time. A comparison between the simulation results and the measured data is presented for $t/\Delta t_{\text{gust}} = 0, 0.25, 0.5, 0.75, 1.0, 1.5, 2.0$, and 2.5 (see Fig. 5 for the definition of the time instants). This time interval encompasses the gust generation time, the direct interaction time between the gust and the T-structure, and the phase when the gust exits the domain of interest.

Fig. 13 depicts a comparison between the measured data [12] and predicted data based on the phase-averaged streamwise velocity in the geometric symmetry plane. The flow field at $t/\Delta t_{\text{gust}} = 0$ is presented as a reference, using the same contour levels as in the subsequent figures. Furthermore, Fig. 14(a) shows the corresponding displacements at this time instant. At $t/\Delta t_{\text{gust}} = 0.25$, first alterations of the flow field due to the growing wind gust become visible in the region upstream and above the T-structure, whereas the region behind the obstacle is only marginally affected. The structure is also only minimally deflected at this stage (see Fig. 14(b)). At the third instant ($t/\Delta t_{\text{gust}} = 0.5$) when the paddle reaches its reversal point, the peak gust velocity is found in the region beneath the paddle. Consequently, significant changes occur for both the flow field (Fig. 13c) and the deflection of the structure (Fig. 14(c)). The size of the recirculation region behind the shield-shaped part and the structural deflections of the entire structure increase noticeably compared to the previous snapshot. The maximum deformation is, however, reached slightly later at $t/\Delta t_{\text{gust}} \approx 0.6$, when the upper edge of the T-structure deflects by approximately $0.1 h_s$. At the time instants when the paddle gradually unblocks the nozzle outlet, i.e., $t/\Delta t_{\text{gust}} = 0.75$ and 1.0 depicted in Figs. 13d/13e and 14(d)/14(e), the previously described trends reverse. The recirculation region starts to shrink and the flow velocities gradually recover as the structure swings back. At approximately $t/\Delta t_{\text{gust}} = 1.0$ the recirculation region has the smallest size with a length of less than $1 h_s$. Afterwards, it increases again and slowly reaches its pre-gust level again. The same applies to the deflections, which aside from some weak oscillations, return to their pre-gust level. At about $t/\Delta t_{\text{gust}} = 2.5$, the original flow conditions and structural deflections are almost completely restored as visible in Figs. 13h and 14(h).

To further analyze the structural deformation during the gust impact, Fig. 15(a) presents time histories of the deflection of the center of the shield-shaped part of the T-structure (point 1). On the one hand, the instantaneous deformations predicted for ten individual gusts are shown. On the other hand, the corresponding phase-averaged data based on all predicted realizations (i.e., 32) are determined. Again $t = 0$ marks the initialization of the gust generation, and the time

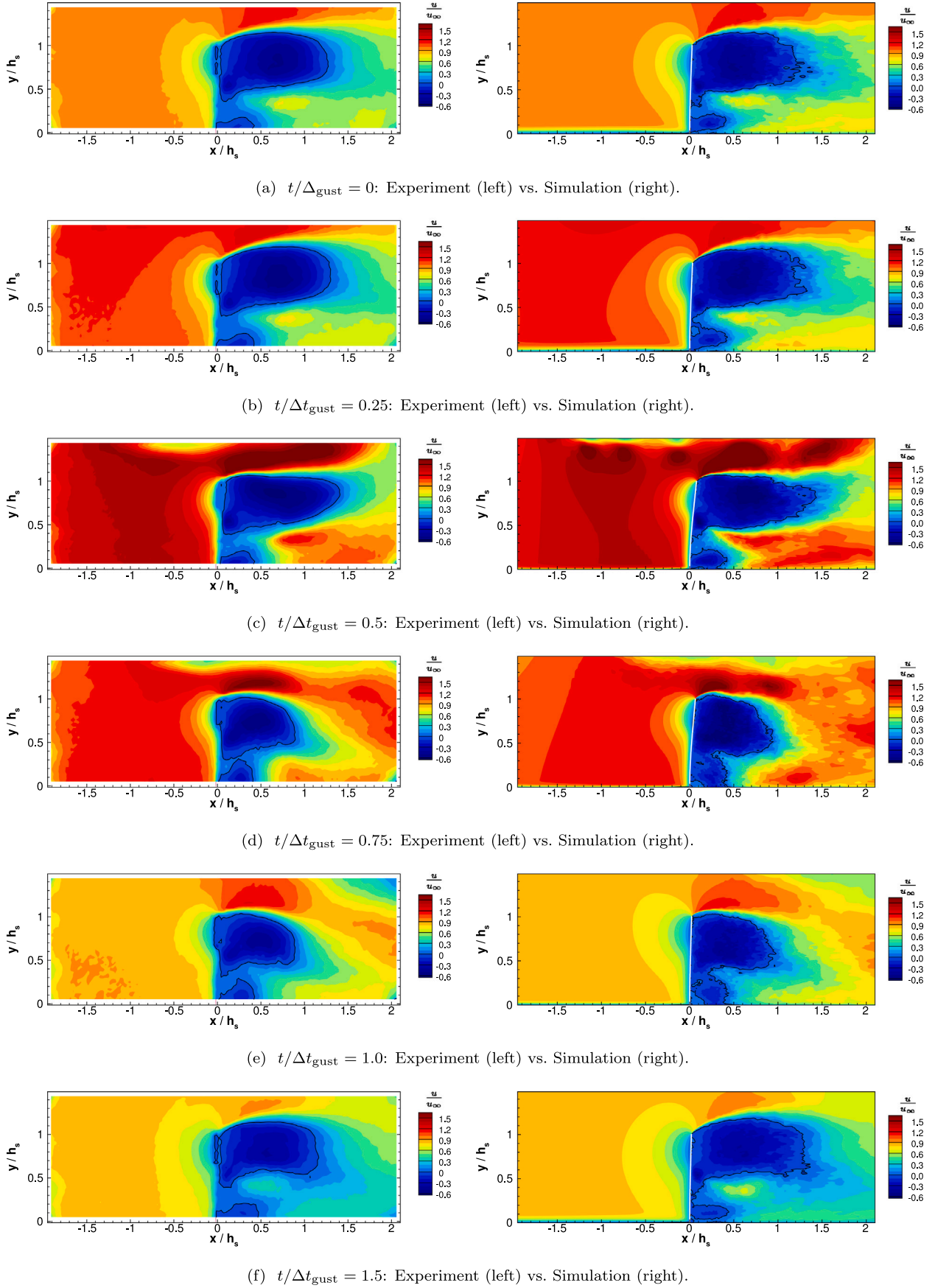


Fig. 13. Comparison between measured and predicted data of the phase-averaged streamwise velocity in the symmetry plane at different time instants of the gust encounter. *Source:* Experimental data taken from Breuer and Neumann [12].

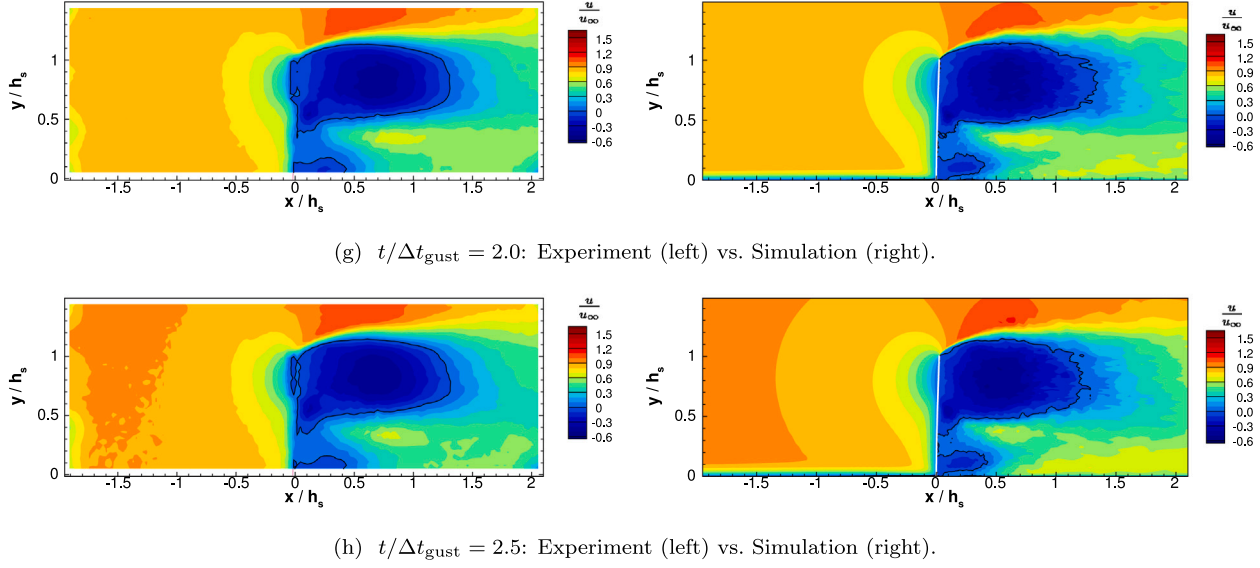


Fig. 13. (continued).

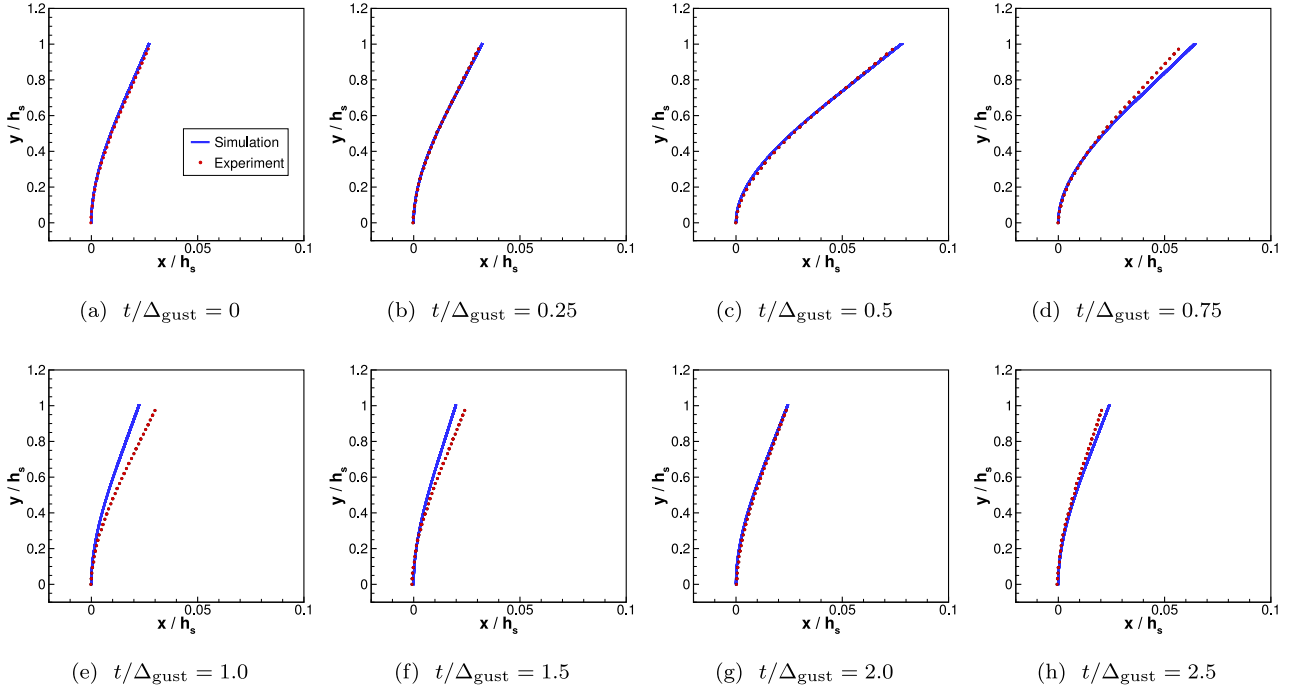


Fig. 14. Comparison between measured and predicted data of the phase-averaged displacements in the symmetry plane at different time instants of the gust encounter. Source: Experimental data taken from Breuer and Neumann [12].

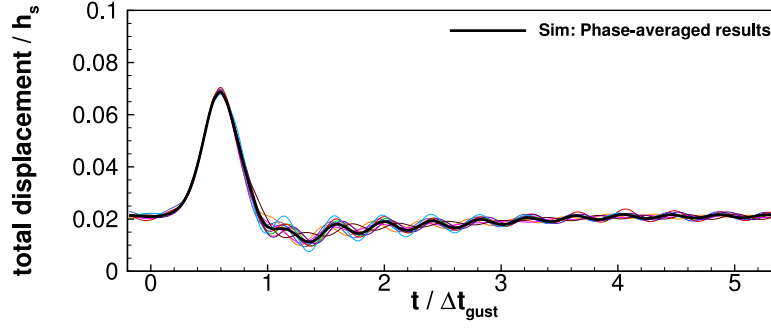
axis is normalized by the gust generation time Δt_{gust} . For comparison purposes, Fig. 15(b) depicts the corresponding experimental data. Note that the phase-averaged data in this case are based on a larger sample of individual gusts (i.e., 104).

Before the paddle enters the fluid stream ($t < 0$), minor flow-induced oscillations of the structure are visible. As discussed in Section 4.1 for the case without gust, the amplitudes of the measured oscillations are slightly larger than those observed in the simulations. The most likely cause for this deviation is the lack of inflow turbulence in the simulation as explained above.

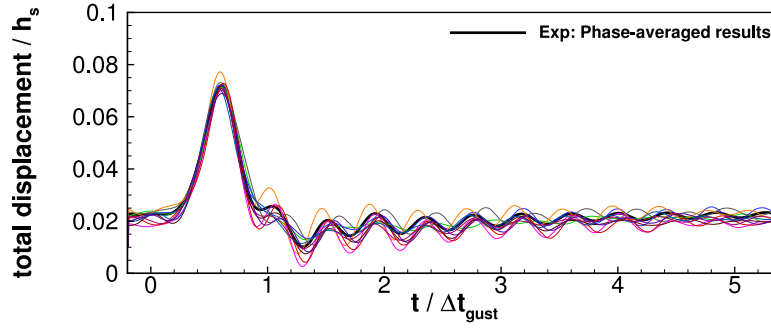
Between the initiation of the gust and its first noticeable effect on the structure, a time delay of approximately $0.2 \Delta t_{\text{gust}}$ is evident in the predicted data. This phenomenon was also noted in the measurements by Breuer and Neumann [12] and is attributed to the distance between

the nozzle's outlet and the T-structure, as well as the gust development time. Following this, the deflection increases rapidly, reaching its maximum value of about $0.07 h_s$ at approximately $t/\Delta t_{\text{gust}} = 0.6$. Obviously, cycle-to-cycle variations are evident affecting the maximum deflection reached in each gust. In the experiment, slightly larger maximum deflections of approximately $40 \mu\text{m}$ are observed, leading to a marginally higher peak deflection in the phase-averaged data.

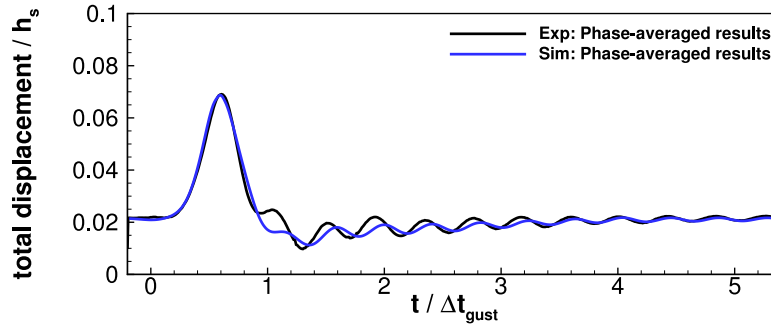
After reaching its maximum deflection, the structure swings back in the opposite direction. At $t/\Delta t_{\text{gust}} \approx 0.95$, the primary effect of the gust is largely alleviated, and the monitoring point returns to a deflection level similar to its pre-gust state. Due to the structure's elasticity and inertia, it oscillates beyond this state, reaching a minimum deformation of approximately $0.01 h_s$ at about $t/\Delta t_{\text{gust}} = 1.3$. An interesting observation is that most gusts pass through a local maximum



(a) Simulations.



(b) Experimental data by Breuer and Neumann [12].



(c) Direct comparison: Experiment vs. simulation.

Fig. 15. Time histories of the total displacements of the center of the upper rectangular part of the T-structure: Phase-averaged data (black thick line) and 10 singular gusts (colored lines).

before reaching this minimum, with considerable variations from gust to gust. In the corresponding time interval of $0.9 \leq t/\Delta t_{\text{gust}} \leq 1.3$, the largest deviations are visible between the phase-averaged predicted displacement and its measured counterpart. Overall, during the initial rebound smaller deflections are seen in the simulation compared to the experiment. This discrepancy can be attributed to the absence of turbulence intensity in the inflow of the current computational framework. Additional preliminary predictions taking a certain level of background turbulence into account, confirm its influence on the amplitude of the T-structure's response. Nonetheless, the finally achieved minimum deflections in the simulation and the experiment are very similar.

Afterwards, at $t/\Delta t_{\text{gust}} > 1.5$, the oscillations gradually return to the status before the gust impact. The oscillation frequencies found in this time interval are similar to the dominant bending frequency of approximately $f = 6.68$ Hz observed in Section 4.1 for the case without gust. Therefore, the gust can also be seen as a kind of an external excitation.

Overall, the predicted time histories exhibit significant cycle-to-cycle variations, similar to those observed in the measurements. These

variations are primarily attributed to the pre-deflected position of the structure at the moment of the gust impact.

4.3. Three-dimensional flow field induced by the gust

To visualize the three-dimensional representation of the gust phenomenon, Fig. 16 illustrates pressure differences obtained by subtracting the pressure of the time-averaged flow results without gust from the phase-averaged pressure of the gust case. A negative iso-surface of this pressure difference highlights additional vortex cores induced by the gust.

During the paddle's downstroke a large horizontal vortex forms directly behind its base, where the gust is generated (see Fig. 16(a)). Due to the velocity gradient in the shear layer (high velocity at the nozzle's outlet and low velocity behind the paddle), the central region of this vortex rotates around the positive z -axis. A similar velocity gradient on either side of the nozzle outlet leads to the formation of vertical vortices, which later evolve into the 'feet' of a large arc-shaped vortex. Over time, this arc-type structure is convected downstream as it grows in size. At $t/\Delta t_{\text{gust}} = 0.5$, when the paddle reaches its maximum

blocking ratio, the arc-shaped vortex, marking the gust front, passes beyond the T-structure (see Fig. 16(b)). The feet of this structure, symmetrically positioned on both sides of the bottom plate, rotate around the y -axis. This large arc-like structure is connected to the paddle's sides through two prominent side vortices formed at the paddle's lower corners and rotating around the x -axis.

As expected, the shear layer behind the paddle and the gust front contains a series of Kelvin–Helmholtz vortices. Furthermore, the symmetric vortices at the top of the strong front vortex adopt a horn-like shape. These vortices are likely resulting from the interaction between the y -axis, the x -axis and the z -axis rotating vortices (see Fig. 16(b)). Fluid particles flowing close to the nozzle's sides are captured by the y -axis rotating vortices and redirected toward the center of the test section. Some of these fluid particles, located near the x -axis rotating vortices, are accelerated upward and drawn into the counter-rotating flow generated by the large z -axis rotating vortex. The clockwise rotation of the large front vortex pulls the fluid downward toward the floor, and, in combination with the main flow, funnels it toward the center of the test section, giving rise to the elongated horn-like vortices.

As the paddle moves upward, returning to its original position, the arc-shaped vortical structure descends into the wake while remaining connected to the paddle via the side vortices, which gradually weaken. The large z -rotating vortex defining the gust front is initially quasi-2D (see Fig. 16(a)). At later stages (see Figs. 16(b) and 16(c)), the two-dimensionality is disrupted. This is attributed to the deceleration effect that the stationary fluid surrounding the ground plate has on the generated gust. Therefore, the center of the z -rotating vortex (undergoing less resistance) progresses faster. Moreover, the horn-like vortices above the arc-shaped vortex begin to dissipate around $t/\Delta t_{\text{gust}} = 0.75$. Overall, Fig. 16 illustrates the complexity of the flow dynamics induced by the wind gust.

5. Conclusions and outlook

Combining different numerical methods in an appropriate way, which at first glance do not seem to belong together, can be useful for simulating complex coupled fluid–structure interactions. In this case, the focus is on investigating a flexible structure in a wind tunnel, subjected to highly dynamic loads generated by a custom-designed wind gust generator. The simulation of the turbulent flow around the T-structure and its dynamic deformation is carried out using a partitioned method based on the ALE approach on general, curvilinear, body-fitted grids, a technique that has been widely applied to similar problems. The gust generation process, which involves a rigid paddle moving through the flow field, is modeled using the IB method with direct forcing. Despite the paddle causing significant disruption to the free-stream flow in the wind tunnel and inducing a rapid, short-term increase in the main flow velocity, the IB method ensures smooth pressure and velocity fields without any numerically induced oscillations. This enables a precise reproduction of the real gust generator in the wind tunnel and a thorough analysis of the actual FSI problem including three-dimensional visualizations of the entire flow field not available in the experiments. Furthermore, it allows for a meaningful comparison of the simulation results with the experimental data reported by Breuer and Neumann [12]. The main findings can be summarized as follows:

- In the case without gust, the deformation of the T-structure remains small. Furthermore, marginal bending and even weaker torsional oscillations of the structure were observed, which allow averaging of the fluid flow and structure in time. The predicted flow field is found to be in close agreement with the PIV measurements of Breuer and Neumann [12]. The same applies to the predicted structural deformations and the DIC measurements.
- Both simulation and experiment exhibit a large recirculation region behind the shield-shaped section and strong shear layers

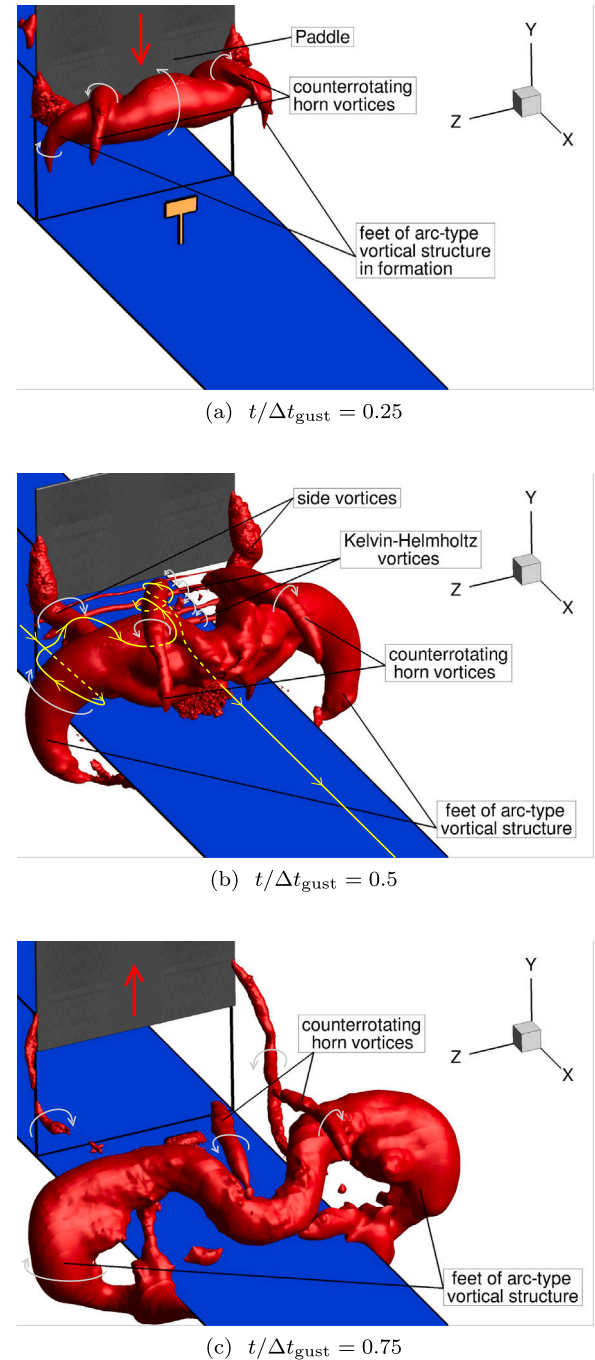


Fig. 16. Three-dimensional visualization of the gust effect on the flow field: Iso-surfaces of the pressure differences between the time-averaged pressure without the gust and the phase-averaged pressure in case of the gust.

occurring around this backflow area. The maximal structural deformations are in the range of 3 to 4 mm.

- Both measurements and simulations exhibit two distinct oscillation modes of the structure, between which the coupled system alternates irregularly. Relatively large amplitudes are observed in the bending-dominated mode, while noticeably smaller amplitudes occur in the mode where twisting also plays a role. Both corresponding bending frequencies are close to the natural bending frequency in vacuum.
- Wind gusts significantly change the flow field around the structure. In close agreement between simulation and experiment,

the recirculation region behind the shield-shaped part increases noticeably to about $1.45 h_s$ when the velocity of the approaching flow rises due to the wind gust. A maximum deformation of the upper edge of the T-structure of about $0.1 h_s$ is approximately reached at $t/\Delta t_{\text{gust}} = 0.6$.

- When the paddle unblocks the nozzle's outlet and the gust starts to diminish, the recirculation region shrinks significantly reaching a minimum size of about $0.9 h_s$ at approximately $t/\Delta t_{\text{gust}} = 1.0$. Simultaneously, the structure moves against the main flow direction. After a few bending oscillations the flow field as well as the structural behavior of the pre-gust levels are recovered.
- Cycle-to-cycle variations play a role for both cases, with and without wind gusts. For the gust case, these variations are mainly attributed to the pre-deflected position of the structure at the moment of the gust impact. A second factor relevant to the gust and the no-gust case is the turbulence intensity of the flow exiting the wind tunnel nozzle. Since the inflow turbulence was not taken into account in the simulations for simplification, the oscillation amplitudes are found to be slightly smaller than those in the experiments. Additional preliminary simulations taking the inflow turbulence into account confirm that the amplitudes of the structural oscillations rise with an increasing turbulence intensity level. Further studies in the future shall quantify this effect.

The study demonstrates that the simulation methodology developed works well, successfully replicating both the gust generation process in the wind tunnel induced by the moving paddle and the coupled FSI problem of the thin structure interacting with the strongly varying turbulent fluid flow with maximum inlet velocities rising by more than 60 %. In future studies, this methodology will enable comprehensive investigations of complex fluid–structure interaction problems, especially those involving membrane structures subjected to predefined wind gust loads.

CRedit authorship contribution statement

M. Breuer: Writing – review & editing, Writing – original draft, Validation, Supervision, Resources, Project administration, Investigation, Funding acquisition, Conceptualization. **K. Boulbrachene:** Validation, Software, Methodology, Investigation. **G. De Nayer:** Visualization, Validation, Software, Methodology, Investigation, Formal analysis, Data curation.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgments

Computational resources on the HPC cluster HSUper have been provided by the project hpc.bw, funded by dtec.bw – Digitalization and Technology Research Center of the Bundeswehr. dtec.bw is funded by the European Union – NextGenerationEU. Furthermore, we would like to thank Dr. J.N. Wood (HSU Hamburg) for providing the displacement measurements by the laser vibrometer mentioned in [Appendix A](#).

Appendix A. Determination of the Young's modulus and material damping parameters

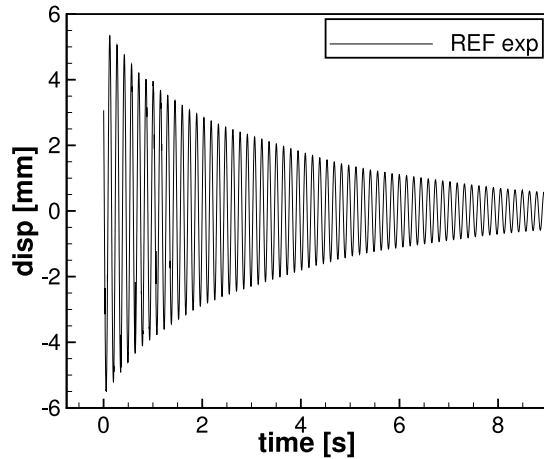
To determine the parameters for the material model used in this study, two simple structural test cases were conducted in the laboratory and numerically simulated using the Carat++ solver (see [Fig. A.17](#)). The first test involved bending the T-shaped structure and then releasing it in a vacuum environment. The second considered the same

but for the twisting mode. The displacements at point 1, as defined in [\[12\]](#), were measured using a laser vibrometer (see [Fig. A.17\(a\)](#)). This measurement point is located at the center of the upper rectangular part of the T-structure.

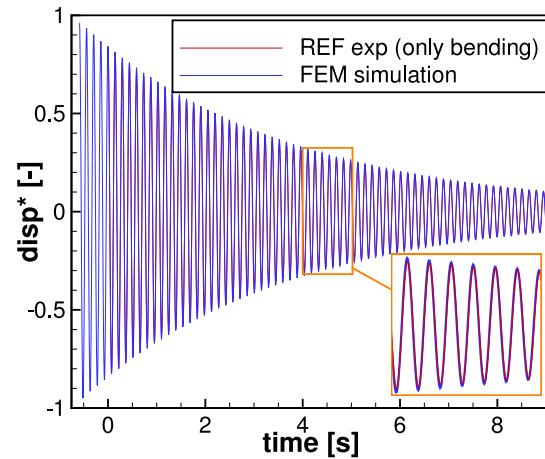
The structural simulation employed the finite-element (FE) mesh described in [Section 3.2.2](#), with the dimensions of the T-shaped structure corresponding to those used in the experimental study by Breuer and Neumann [\[12\]](#). The material density ρ_s was set to $\rho_s = 8435 \text{ kg/m}^3$, as provided in [\[12\]](#). The brass material (alloy CuZn37) was modeled using the St. Venant-Kirchhoff law, with a typical Poisson's ratio of 0.34 for brass. Using a Young's modulus of $E_s = 110 \text{ GPa}$, as initially suggested by Breuer and Neumann [\[12\]](#), led to an overestimation of the first dominant eigenfrequency associated with the bending mode of the T-shaped structure. To achieve a more accurate match, a reduced Young's modulus of $E_s = 93.7 \text{ GPa}$ was determined through a trial-and-error process based on the dynamic bending test. In these tests the gravitational acceleration was taken into account but the structural damping was first of all ignored. The adjustment of E_s resulted in a natural bending frequency of $f_1^{\text{num}} = 6.814 \text{ Hz}$, closely aligning with the experimental bending frequency of $f_1^{\text{exp}} = 6.82 \text{ Hz}$. The frequency of the second mode, determined by the pure twisting case around the vertical y-axis, also agrees reasonably well between the simulation ($f_2^{\text{num}} = 23.75 \text{ Hz}$) and the experiment ($f_2^{\text{exp}} = 22.45 \text{ Hz}$).

As visible in the experimental data in [Fig. A.17\(a\)](#), the structure exhibits structural damping. To account for the damping, the Rayleigh damping model is employed in this study. The mass- and stiffness-related Rayleigh damping coefficients, α_r and β_r , were determined using an automated search process with the surrogate-based MATLAB optimization function *surrogateopt*. The objective was to minimize the L^2 -norm of the error between the logarithmic decrement of the experimental and numerical deflection data at point 1. To isolate the pure bending mode in the experimental data, a discrete Fourier transformation was performed on the temporal response of the measured data. A rectangular window function was then applied to isolate the frequency response associated with the bending frequency. Finally, a clean signal in the time domain (depicted by the red curve in [Fig. A.17\(b\)](#)) was obtained via an inverse Fourier transformation. The numerical optimization process was configured to perform a maximum of 200 evaluations, with 12 evaluations running in parallel on a computing cluster. The bounds for the optimization were set to $0.4 \text{ Hz} \leq \alpha_r \leq 0.5 \text{ Hz}$ and $10^{-8} \text{ s} \leq \beta_r \leq 10^{-6} \text{ s}$, respectively. Each evaluation involved setting up the simulation, running the structural solver, and post-processing the results to evaluate the error in the logarithmic decrement. The procedure yielded several sets of α_r and β_r values, that resulted in a very good agreement between the predicted and the measured data of the bending deflections. For example, [Fig. A.17\(b\)](#) shows the results achieved with $\alpha_r = 0.47383 \text{ Hz}$ and $\beta_r = 2.1602 \times 10^{-8} \text{ s}$. The structural damping in these cases has only a marginal influence on the bending frequency of the dynamic simulation. Thus, the simulated value of $f_1^{\text{num}} = 6.826 \text{ Hz}$ remains quasi unchanged and in close agreement with the experimental value of $f_1^{\text{exp}} = 6.82 \text{ Hz}$.

To obtain the final values from the possible parameter combinations (α_r , β_r) resulting from the bending test, the twisting behavior was also taken into account. Although twisting plays a less important role than bending in the present case, it should still be considered in order to represent the physics as good as possible. In the experiment and in the simulation, this second vibration mode was therefore also excited and its decay behavior was analyzed. For parameter combinations that had shown almost identical results for the first mode, there were noticeable differences in the decay behavior of the second mode. The final selection from this pool of best-fit parameter combinations for bending was made based on the best fit with the experimental twisting decay curve resulting in $\alpha_r = 0.47383 \text{ Hz}$ and $\beta_r = 2.1602 \times 10^{-8} \text{ s}$.



(a) Experimental displacements measured by the laser vibrometer.



(b) Displacements predicted by the FEM simulation normalized by the initial displacement and comparison with the experimental signal limited to the bending mode.

Fig. A.17. Free-oscillation test in vacuum: Determination of the material parameters of the model.

Table B.3

Static grid sensitivity study: Total displacements d of the center of the shield-shaped part of the T-structure for different grids under increasing wind load.

Load		Grid		Standard		Fine		Reference
Wind velocity [m/s]	Wind load [N/m ²]	d [m]	Error [%]	d [m]	Error [%]	d [m]	Error [%]	d [m]
5.14	16	2.0593×10^{-3}	0.42	2.0608×10^{-3}	0.35	2.0680×10^{-3}	0.35	2.0680×10^{-3}
8.40	43	5.5212×10^{-3}	0.42	5.5252×10^{-3}	0.35	5.5446×10^{-3}	0.35	5.5446×10^{-3}
12.8	100	12.6886×10^{-3}	0.42	12.6984×10^{-3}	0.34	12.7422×10^{-3}	0.34	12.7422×10^{-3}

Appendix B. Grid sensitivity study for the structure model

A special seven-parameters shell element is used for the finite-element discretization of the T-structure. The material parameters determined in Appendix A are applied. Three levels of refinement are investigated under uniformly applied wind loads of 16, 43 and 100 N/m², resulting from wind velocities of $u_{\text{inflow}} = u_{\infty} = 5.14$ m/s, $1.6 u_{\infty} = 8.40$ m/s and $2.5 u_{\infty} = 12.8$ m/s. These exemplary loads are calculated based on the assumption that the static pressure at the front of the body is approximately $0.5 \rho_{\text{air}} u_{\text{inflow}}^2$. The resulting total displacements d at point 1, i.e., the center of the shield-shaped part of the T-structure, are given in Table B.3. The load-controlled geometrically non-linear analysis yields very similar results for the standard, fine, and reference grids, consisting of 980, 4000, and 16,000 quadrilateral shell elements, respectively. This table also includes the relative errors with respect to the results obtained on the reference grid. It is evident that spatial convergence has already been achieved with the standard grid.

To assess the influence of the grid on the dynamic response of the flexible T-structure, the structure is first deformed and then released in a vacuum. In the first configuration, the structure oscillates at a frequency of f_1^{num} in its first dominant mode, the bending mode. In the second configuration, it oscillates at a frequency of f_2^{num} in its second mode, the twisting mode. Table B.4 summarizes these frequencies, which were determined by fast Fourier transformations of the total displacement at the center of the shield-shaped part of the T-structure. The convergence of both frequencies is achieved with the standard grid.

Data availability

Data will be made available on request.

Table B.4

Dynamic grid sensitivity study: First and second dominant frequencies associated with the total displacement of the center of the shield-shaped part of the T-structure.

Frequencies	Grid		Fine		Reference
	f [Hz]	Error [%]	f [Hz]	Error [%]	f [Hz]
f_1^{num}	6.826	0.18	6.814	0.0	6.814
f_2^{num}	23.78	0.93	23.67	0.47	23.56

References

- [1] Wood JN, Breuer M, Neumann T. A novel approach for artificially generating horizontal wind gusts based on a movable plate: The paddle. *J Wind Eng Ind Aerodyn* 2022;230:105170. <http://dx.doi.org/10.1016/j.jweia.2022.105170>.
- [2] Sitaraman J. CFD based unsteady aerodynamic modeling for rotor aeroelastic analysis (Ph.D. thesis), University of Maryland, College Park, USA; 2003.
- [3] Zaide A, Raveh DE. Numerical simulation and reduced-order modeling of airfoil gust response. *AIAA J* 2006;44(8):1826–34. <http://dx.doi.org/10.2514/1.16995>.
- [4] Wales C, Jones D, Gaitonde A. Prescribed velocity method for simulation of aerofoil gust responses. *J Aircr* 2014;52(1):64–76. <http://dx.doi.org/10.2514/1.C032597>.
- [5] Badrya C, Biler H, Jones AR, Baeder JD. Effect of gust width on flat-plate response in large transverse gust. *AIAA J* 2021;59(1):49–64. <http://dx.doi.org/10.2514/1.J059678>.
- [6] IEC-Standard. Measurement and assessment of power quality of grid connected wind turbines. Technical report 61400-21. Tech. rep., 2002.
- [7] Bos R. Extreme gusts and their role in wind turbine design (Ph.D. thesis), Delft University of Technology, The Netherlands; 2017. <http://dx.doi.org/10.4233/uuid:d6097e3a-1cdd-4845-a71c-90f469d28b7a>.
- [8] Storey RC, Norris SE, Cater JE. Modelling turbine loads during an extreme coherent gust using large eddy simulation. *J Phys: Conf Ser* 2014;524:012177. <http://dx.doi.org/10.1088/1742-6596/524/1/012177>.
- [9] Onol AO, Yesilyurt S. Effects of wind gusts on a vertical axis wind turbine with high solidity. *J Wind Eng Ind Aerodyn* 2017;162:1–11. <http://dx.doi.org/10.1016/j.jweia.2017.01.003>.

- [10] Menegozzo L, Dal Monte A, Benini E, Benato A. Small wind turbines: A numerical study for aerodynamic performance assessment under gust conditions. *Renew Energy* 2018;121:123–32. <http://dx.doi.org/10.1016/j.renene.2017.12.086>.
- [11] Wood JN, Breuer M. The Paddle: A novel procedure for artificial gust generation in a wind tunnel. In: Kähler CJ, Fuchs T, Hain R, Scharnowski S, Ruck B, Leder A, editors. *Proc. der 30. Gala-Fachtagung 'Experimentelle Strömungsmechanik'*, Sept. 7–9, 2023. München; 2023, 37–1–37–8.
- [12] Breuer M, Neumann T. Synchronous high-speed measurements of a flexible structure under wind gust load. *Phys Fluids* 2024;36(7):075147. <http://dx.doi.org/10.1063/5.0215724>.
- [13] Boulbrachene K, Breuer M. A numerical method to mimic an experimental wind gust generator: The immersed boundary gust generator. *Phys Fluids* 2024;36(1):015146. <http://dx.doi.org/10.1063/5.0187787>.
- [14] Durst F, Schäfer M. A parallel block-structured multigrid method for the prediction of incompressible flows. *Int J Numer Methods Fluids* 1996;22(6):549–65. [http://dx.doi.org/10.1002/\(SICI\)1097-0363\(19960330\)22:6<549::AID-FLD366>3.0.CO;2-7](http://dx.doi.org/10.1002/(SICI)1097-0363(19960330)22:6<549::AID-FLD366>3.0.CO;2-7).
- [15] Breuer M, De Nayer G, Münch M, Gallinger T, Wüchner R. Fluid-structure interaction using a partitioned semi-implicit predictor-corrector coupling scheme for the application of large-eddy simulation. *J Fluids Struct* 2012;29:107–30. <http://dx.doi.org/10.1016/j.jfluidstruct.2011.09.003>.
- [16] Ferziger JH, Perić M. *Computational methods for fluid dynamics*. 3rd ed. Springer Berlin; 2002.
- [17] Rhie CM, Chow WL. Numerical study of the turbulent flow past an airfoil with trailing-edge separation. *AIAA J* 1983;21(11):1525–32. <http://dx.doi.org/10.2514/3.8284>.
- [18] Breuer M. *Direkte Numerische Simulation und Large-eddy Simulation turbulenter Strömungen auf Hochleistungsrechnern*. Habilitationsschrift, Universität Erlangen-Nürnberg, Berichte aus der Strömungstechnik, Aachen: Shaker Verlag; 2002.
- [19] Smagorinsky J. General circulation experiments with the primitive equations I: The basic experiment. *Mon Weather Rev* 1963;91(3):99–165. [http://dx.doi.org/10.1175/1520-0493\(1963\)091<0099:GCEWTP>2.3.CO;2](http://dx.doi.org/10.1175/1520-0493(1963)091<0099:GCEWTP>2.3.CO;2).
- [20] Thompson JF, Warsi ZUA, Wayne Mastin C. *Numerical grid generation: Foundations and applications*, vol. 45, North-Holland; 1985.
- [21] Knupp PM. Algebraic mesh quality metrics for unstructured initial meshes. *Finite Elem Anal Des* 2003;39(3):217–41.
- [22] Sen S, De Nayer G, Breuer M. A fast and robust hybrid method for block-structured mesh deformation with emphasis on FSI-LES applications. *Int J Numer Methods Eng* 2017;111(3):273–300. <http://dx.doi.org/10.1002/nme.5465>.
- [23] Hoblit FM. Gust loads on aircraft: Concepts and applications. *AIAA Education Series*, Washington D.C.; 1988, <http://dx.doi.org/10.2514/4.861888>.
- [24] Wu Z, Cao Y, Ismail M. Gust loads on aircraft. *Aeronaut J* 2019;123(1266):1216–74. <http://dx.doi.org/10.1017/aer.2019.48>.
- [25] Norris SE, Cater JE, Stol KA, Unsworth CP. Wind turbine wake modelling using large-eddy simulation. In: *Proceedings of the 17th Australian Fluid Mechanics Conference*. University of Auckland, Australia: Curran Associates, Inc.; 2010.
- [26] De Nayer G, Breuer M, Perali P, Grollmann K. Modeling of wind gusts for large-eddy simulations related to fluid-structure interactions. In: Salvetti MV, Armenio V, Fröhlich J, Geurts BJ, Kuerten H, editors. *ERCOFTAC Series, Direct and Large-eddy Simulation XI*, 11th Int. ERCOFTAC Workshop on Direct and Large-eddy Simulation: DLES-11, Pisa, Italy, May 29–31, 2017. 25, Springer Nature Switzerland AG; 2019, p. 453–9. http://dx.doi.org/10.1007/978-3-030-04915-7_60.
- [27] De Nayer G, Breuer M. A source-term formulation for injecting wind gusts in CFD simulations. *J Wind Eng Ind Aerodyn* 2020;207:104405. <http://dx.doi.org/10.1016/j.jweia.2020.104405>.
- [28] Parameswaran V, Baeder JD. Indicial aerodynamics in compressible flow—Direct computational fluid dynamic calculations. *J Aircr* 1997;34(1):131–3. <http://dx.doi.org/10.2514/2.2146>.
- [29] Singh R, Baeder JD. Direct calculation of three-dimensional indicial lift response using computational fluid dynamics. *J Aircr* 1997;34(4):465–71. <http://dx.doi.org/10.2514/2.2214>.
- [30] Huntley SJ, Jones D, Gaitonde A. Aeroelastic gust response of an aircraft using a prescribed velocity method in viscous flows. In: *23rd AIAA Computational Fluid Dynamics Conference*. Denver, Colorado; 2017.
- [31] Biler H, Badrya C, Jones AR. Experimental and computational investigation of transverse gust encounters. *AIAA J* 2019;57(11):4608–22. <http://dx.doi.org/10.2514/1.J057646>.
- [32] Badrya C, Baeder JD. Numerical study of a flat plate wing response to large transverse gusts at low Reynolds number. San Diego, California: AIAA Scitech 2019 Forum; 2019, p. 0638, <https://arc.aiaa.org/doi/pdf/10.2514/6.2019-0638> URL <https://arc.aiaa.org/doi/abs/10.2514/6.2019-0638>.
- [33] Boulbrachene K, De Nayer G, Breuer M. Assessment of two wind gust injection methods: Field velocity vs. split velocity method. *J Wind Eng Ind Aerodyn* 2021;218:104790. <http://dx.doi.org/10.1016/j.jweia.2021.104790>.
- [34] Uhlmann M. An immersed boundary method with direct forcing for the simulation of particulate flows. *J Comput Phys* 2005;209(2):448–76. <http://dx.doi.org/10.1016/j.jcp.2005.03.017>.
- [35] Peskin CS. The immersed boundary method. *Acta Numer* 2002;11:479–517. <http://dx.doi.org/10.1017/S0962492902000077>.
- [36] Roma AM, Peskin CS, Berger MJ. An adaptive version of the immersed boundary method. *J Comput Phys* 1999;153(2):509–34. <http://dx.doi.org/10.1006/jcph.1999.6293>.
- [37] Pinelli A, Naqavi I, Piomelli U, Favier J. Immersed-boundary methods for general finite-difference and finite-volume Navier-Stokes solvers. *J Comput Phys* 2010;229(24):9073–91. <http://dx.doi.org/10.1016/j.jcp.2010.08.021>.
- [38] Anderson E, Bai Z, Bischof C, Blackford S, Demmel J, Dongarra J, Du Croz J, Greenbaum A, Hammarling S, McKenney A, Sorensen D. *LAPACK users' guide*. 3rd ed. Philadelphia, PA: Society for Industrial and Applied Mathematics; 1999.
- [39] Schäfer F, Breuer M. Comparison of c-space and p-space particle tracing schemes on high-performance computers: Accuracy and performance. *Int J Numer Methods Fluids* 2002;39(4):277–99. <http://dx.doi.org/10.1002/flid.324>.
- [40] Breuer M, Baytekin HT, Matida EA. Prediction of aerosol deposition in 90 degrees bends using LES and an efficient Lagrangian tracking method. *J Aerosol Sci* 2006;37(11):1407–28. <http://dx.doi.org/10.1016/j.jaerosci.2006.01.013>.
- [41] Breitenberger M, Apostolatos A, Philipp B, Wüchner R, Bletzinger K-U. Analysis in computer aided design: Nonlinear isogeometric B-Rep analysis of shell structures. *Comput Methods Appl Mech Engrg* 2015;284:401–57. <http://dx.doi.org/10.1016/j.cma.2014.09.033>.
- [42] Philipp B, Wüchner R, Bletzinger K-U. Advances in the form-finding of structural membranes. *Procedia Eng* 2016;155:332–41. <http://dx.doi.org/10.1016/j.proeng.2016.08.036>.
- [43] Basar Y, Weichert D. *Nonlinear continuum mechanics of solids: Fundamental mathematical and physical concepts*. Springer Science & Business Media; 2013.
- [44] Sicklinger S, Belsky V, Engelmann B, Elmqvist H, Olsson H, Wüchner R, Bletzinger K-U. Interface Jacobian-based co-simulation. *Int J Numer Methods Eng* 2014;98:418–44. <http://dx.doi.org/10.1002/nme.4637>.
- [45] Wang T, Wüchner R, Sicklinger S, Bletzinger K-U. Assessment and improvement of mapping algorithms for non-matching meshes and geometries in computational FSI. *Comput Mech* 2016;57(5):793–816. <http://dx.doi.org/10.1007/s00466-016-1262-6>.
- [46] Apostolatos A, De Nayer G, Bletzinger K-U, Breuer M, Wüchner R. Systematic evaluation of the interface description for fluid-structure interaction simulations using the isogeometric mortar-based mapping. *J Fluids Struct* 2019;86:368–99. <http://dx.doi.org/10.1016/j.jfluidstruct.2019.02.012>.
- [47] De Nayer G, Apostolatos A, Wood JN, Bletzinger KU, Wüchner R, Breuer M. Numerical studies on the instantaneous fluid-structure interaction of an air-inflated flexible membrane in turbulent flow. *J Fluids Struct* 2018;82:577–609. <http://dx.doi.org/10.1016/j.jfluidstruct.2018.08.005>.
- [48] De Nayer G, Breuer M, Boulbrachene K. FSI simulations of wind gusts impacting an air-inflated flexible membrane at Re=100,000. *J Fluids Struct* 2022;109:103462. <http://dx.doi.org/10.1016/j.jfluidstruct.2021.103462>.
- [49] De Nayer G, Breuer M. Assessment of discrete wind gust parameters: Towards the worst-case scenario of a FSI application in form of an inflated hemisphere. *J Wind Eng Ind Aerodyn* 2022;231:105207. <http://dx.doi.org/10.1016/j.jweia.2022.105207>.
- [50] Boulbrachene K, Breuer M. Application of an immersed boundary method to predict the flow in an experimental wind gust generator: An LES study. In: *14th Int. ERCOFTAC Workshop on DNS and LES: DLES-14*, Erlangen, Germany, April 10–12. 2024.
- [51] Moody LF. An approximate formula for pipe friction factors. *Trans ASME* 1947;69(12):1005–11.
- [52] Bischoff M. *Theorie und Numerik einer dreidimensionalen Schalenformulierung* (Ph.D. thesis), Institut für Baustatik, Universität Stuttgart, Germany; 1999.
- [53] Bischoff M, Wall WA, Bletzinger K-U, Ramm E. Models and finite elements for thin-walled structures. In: Stein E, De Borst R, Hughes TJB, editors. *Encyclopedia of computational mechanics*. Vol. 2, Chichester: John Wiley & Sons Ltd; 2004, p. 59–138.
- [54] Wood JN, De Nayer G, Schmidt S, Breuer M. Experimental investigation and Large-Eddy simulation of the turbulent flow past a smooth and rigid hemisphere. *J Flow Turbul Combust* 2016;97(1):79–119. <http://dx.doi.org/10.1007/s10494-015-9690-5>.
- [55] Breuer M. Effect of inflow turbulence on an airfoil flow with laminar separation bubble: An LES study. *J Flow Turbul Combust* 2018;101(2):433–56. <http://dx.doi.org/10.1007/s10494-017-9890-2>.
- [56] De Nayer G, Schmidt S, Wood JN, Breuer M. Enhanced injection method for synthetically generated turbulence within the flow domain of eddy-resolving simulations. *Comput Math Appl* 2018;75(7):2338–55. <http://dx.doi.org/10.1016/j.camwa.2017.12.012>.

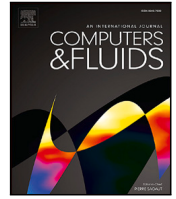
Publication VI: A dynamic immersed boundary method for simulating an adaptive nozzle generating discrete wind gusts

Reference

K. Boulbrachene and M. Breuer. A dynamic immersed boundary method for simulating an adaptive nozzle generating discrete wind gusts. *Computers & Fluids* 301 (2025), 106775. DOI: [10.1016/j.compfluid.2025.106775](https://doi.org/10.1016/j.compfluid.2025.106775)

Copyright

© 2025 The Authors. Published by Elsevier Ltd. This is an open access article distributed under the terms of the Creative Commons CC-BY license (<http://creativecommons.org/licenses/by/4.0/>), which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.



A dynamic immersed boundary method for simulating an adaptive nozzle generating discrete wind gusts

K. Boulbrachene¹, M. Breuer^{1,*}

¹ Professur für Strömungsmechanik, Helmut-Schmidt-Universität Hamburg, D-22043 Hamburg, Germany

ARTICLE INFO

Keywords:

Horizontal wind gusts
Numerical wind tunnel
Immersed boundary method
Moving-least-squares
Large-eddy simulation
Fluid–structure interaction

ABSTRACT

The recently developed wind gust generator, the *adaptive nozzle* (Wood and Breuer, 2025), features a nozzle with a fully rotatable upper contour, enabling a smooth gust generation with low unwanted flow disturbances. While preserving the underlying gust-generation principle of its predecessor, the new design significantly reduces pressure losses caused by flow blockage and preserves the original horizontal trajectory of the flow along the streamwise direction. While experiments have validated the improved design, a comprehensive numerical analysis is crucial to resolve the three-dimensional flow fields across the entire computational domain. This shall also facilitate capturing the resulting transient aerodynamic loads on a wind tunnel specimen — quantities difficult to measure experimentally. To accurately capture the complex flow dynamics, high-fidelity large-eddy simulations are conducted, modeling the nozzle's upper contour as a dynamic immersed boundary (IB). A curvilinear Eulerian grid is employed to ensure both efficient and precise spatial resolution of the problem. The moving least-squares (MLS) version of the direct forcing IB approach (Vanella and Balaras, 2009) is used to construct an IB kernel for each Lagrangian marker. Additionally, the MLS approach is also applied to construct one-sided kernel functions for Lagrangian points near the boundaries of the computational domain. Challenges related to the efficient IB simulation on curvilinear grids are addressed, and a solution is proposed within the MLS framework. The predicted results are analyzed in detail and validated against the experimental data by Wood and Breuer (2025), providing insights into the effectiveness of the new design in generating controlled wind gusts.

1. Introduction

Wind gusts, a common yet powerful meteorological phenomenon, pose a significant challenge to engineers due to their potentially devastating impact on structures. In particular, lightweight constructions made of thin membranes such as large umbrellas, tents, and stadium roofs are highly susceptible to sudden wind loads. Short-term gusts can critically affect their stability and fatigue strength. To gain deeper insight into the interaction between strong wind gusts and such constructions, direct measurements on real structures may seem like the most straightforward approach (see, e.g., [1]). However, conducting large-scale experiments presents considerable challenges: simultaneous measurements of fluid flow and structural deformation at real scales are not only complex and costly but also depend on unpredictable natural wind conditions. One must set up the measurement equipment and wait for suitable weather conditions, making controlled investigations nearly impossible.

As an alternative, wind tunnel experiments at reduced scales provide a viable and practical solution. However, this requires the ability

to generate wind gusts artificially under controlled conditions. Developing reliable gust-generation methods in laboratory settings is therefore essential for advancing the understanding of fluid–structure interactions (FSI) and improving the resilience of structures subjected to extreme wind events. Li et al. [2] conducted a comprehensive review on experimental methods for investigating the impact of downburst winds on civil infrastructures. Gust fronts generated by downbursts, characterized by strong horizontal velocities, are closely related to horizontal wind gusts. The study categorized the experimental methods into three types: (i) standalone simulators (gravity-current-based; see [3] or impinging-jet-based; see [4]), (ii) modified boundary layer wind tunnels with active elements (e.g., rotatable plates or movable gates; see [5,6]), and (iii) multi-fan wind tunnels with either active [7] or hybrid active–passive [8] turbulence control.

Among these, modified boundary layer wind tunnels provide a cost-effective approach for studying the effect of wind gusts on structures. Recently, Wood et al. [9] developed a novel artificial wind gust generator (WGG) falling within this category, known as the *paddle*. By

* Corresponding author.

E-mail address: breuer@hsu-hh.de (M. Breuer).

<https://doi.org/10.1016/j.compfluid.2025.106775>

Received 21 May 2025; Received in revised form 8 July 2025; Accepted 27 July 2025

Available online 6 August 2025

0045-7930/© 2025 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY license (<http://creativecommons.org/licenses/by/4.0/>).

rapidly and partially obstructing the outlet of the wind tunnel nozzle, the paddle induces a significant acceleration of the flow, generating a controlled horizontal gust. This gust generator has already been successfully applied in a follow-up experimental study by Breuer and Neumann [10], where synchronous high-speed measurements were conducted on a flexible T-structure subjected to wind gust loading.

Both experimental studies have been numerically reproduced in Boulbrachene and Breuer [11] and Breuer et al. [12], respectively. In Boulbrachene and Breuer [11], the immersed boundary (IB) method with a direct forcing approach was implemented into a finite-volume flow solver to model the translation motion of the paddle. The numerical results demonstrated excellent agreement with the measurements of Wood et al. [9] at key monitoring points within the flow field, validating the robustness and accuracy of the simulation approach. Similarly, in Breuer et al. [12], a numerical counterpart of the experimental investigation on the T-structure under wind gust loading was developed. This was achieved by combining the IB-based gust generation methodology with a partitioned FSI approach using the Arbitrary Lagrangian–Eulerian (ALE) framework. The study successfully replicated both the gust generation process in the wind tunnel and the complex FSI problem of the thin flexible structure interacting with the highly unsteady turbulent flow.

In the meantime, Wood and Breuer [13] proposed an enhanced WGG design, known as the *adaptive nozzle*. The improved design addresses the key drawbacks of the paddle setup, particularly the high pressure loss in the nozzle due to flow blockage and the pronounced detached shear layer caused by flow separation at the bottom edge of the plate. Compared to the paddle design, the adaptive nozzle significantly reduced velocity fluctuations on the falling flank of the generated gust. Additionally, it achieved a shorter restitution time – defined as the period during which the wake flow recovers from the velocity undershoot to its initial state – enhancing the overall efficiency and stability of the generated gust. A detailed description of the setup and the measuring techniques employed is provided in Section 2.

While wind tunnel experiments offer valuable insights, they are often constrained by the spatial and temporal resolution of the measurement techniques applied, limiting the ability to capture complex three-dimensional flow structures and transient aerodynamic loads across the entire test section. A numerical study can provide a more detailed understanding of the fundamental mechanisms governing the adaptive nozzle's performance. Simulations enable a comprehensive analysis of the entire flow field, including regions that are challenging to access experimentally, such as the interior of the nozzle. By leveraging high-fidelity numerical methods, it becomes possible to quantify velocity fluctuations, pressure distributions, and turbulence characteristics in great detail.

Since the experimental gust generator relies on a nozzle with a movable upper contour, accurately capturing this feature in numerical flow simulations is essential. A viable approach is the ALE method, which allows the fluid mesh to adapt to the contour's motion while enforcing the no-slip boundary condition at the FSI interface at each time step. While the ALE method effectively resolves flow dynamics along moving interfaces, it leads to significant challenges in FSI problems involving large deformations. One major drawback is the computational expense associated with frequent grid adaptations [14], which can substantially increase the simulation time. Moreover, excessive grid adaptation can degrade the grid quality – compromising properties such as orthogonality and smoothness – which may ultimately affect the accuracy and stability of the numerical results, especially when these are based on large-eddy simulations.

An alternative approach is the immersed boundary method (IBM), originally introduced by Peskin [15], which employs overlapping discretizations for the fluid and structural domains. The no-slip boundary condition at the fluid–solid interface is enforced by introducing an appropriate boundary forcing term. Among the various IBM formulations available in the literature (see, e.g., Mittal and Iaccarino [16]), the

present study adopts the continuous forcing IBM (diffuse-interface IB) with the direct forcing approach of Uhlmann [17], which is particularly well-suited for handling moving immersed boundaries. In this method, boundary forces are computed on Lagrangian markers rather than Eulerian grid points. These forces are then distributed over a few neighboring Eulerian cells using a regularized delta function kernel (IB kernel), ensuring smoother hydrodynamic force distributions.

In his foundational work, Peskin [15] analytically derived a four-point IB kernel based on a set of postulates designed to ensure several essential characteristics. Others [18–20] adopted a similar approach to derive IB kernels with different support sizes (e.g., three-, five-, and six-point kernels). The postulates considered include polynomial reproduction conditions (also known as discrete moment conditions), grid translational invariance, and compact support. However, a key limitation of this analytical approach is its reliance on a uniform Cartesian Eulerian grid. In FSI problems, it is often beneficial to employ stretched or even curvilinear Eulerian grids to enhance computational efficiency and avoid unnecessary high spatial resolution in regions far from the object of interest. For instance, in this study it is preferable to cluster grid points in the vicinity of the upper contour and around the region corresponding to the motion range of it. This necessitates embedding the Lagrangian markers within a non-uniform curvilinear structured Eulerian grid, which renders the direct application of IB kernels designed for uniform grids unsuitable, as it would violate the integral conservation of force and moment.

To address this issue, the current study employs a numerical approach to derive the IB kernel, specifically the moving least-squares (MLS) method [21]. The MLS version of the direct forcing IB method was first introduced by Vanella and Balaras [22]. This approach constructs an interpolation kernel (also referred to as the MLS generating function) for each Lagrangian marker by solving a weighted least-squares problem defined over a set of neighboring Eulerian points. The method supports any non-uniform distribution of Eulerian points, and the resulting interpolation kernels satisfy the polynomial reproduction constraints. This, in turn, ensures the integral conservation of force and moment while preserving numerical accuracy and stability. Although Vanella and Balaras [22] applied the IB-MLS method to a uniform Cartesian grid, the approach is general and has been extended to unstructured grids (see, e.g., [23,24]).

In previous IB-MLS studies [22,23,25], a cubic spline weight function is transformed into a regularized kernel function that satisfies the zeroth- and the first-order discrete moment conditions. However, it is important to note that if the conditions imposed by the MLS construction are already satisfied by the weight function (e.g., when an IB kernel is used as a weight function on a uniform grid), the generating function obtained through the MLS procedure remains identical to the weight function [26,27]. However, Bale et al. [27] have highlighted that when the MLS construction is used to reduce the support of the weighting function, the resulting generating function generally differs from the original kernel. The authors leveraged the MLS construction in uniform Cartesian grids to generate one-sided (asymmetric) IB kernels by modifying the support of a standard two-sided (symmetric) weight function. The primary motivation for introducing one-sided IB kernels was to avoid spurious flows inside closed geometries in the diffuse-interface IB method and to address interactions between immersed structures and the boundaries of the computational domain. The latter objective holds particular significance for the current study, as the nozzle's upper contour is designed to traverse between two side walls, thereby restricting the applicability of symmetric kernels in the near-wall regions. While this study builds upon previous MLS work, it highlights the challenges associated with its direct application and instead proposes a new method for efficient and robust IB simulations on a curvilinear block-structured grid, as described in Section 3.3.2.

The structure of this paper is as follows: Section 2 presents the experimental setup of the second-generation wind gust generator. Section 3 details the numerical methodologies used to develop the computational framework, beginning with the fluid flow simulation approach

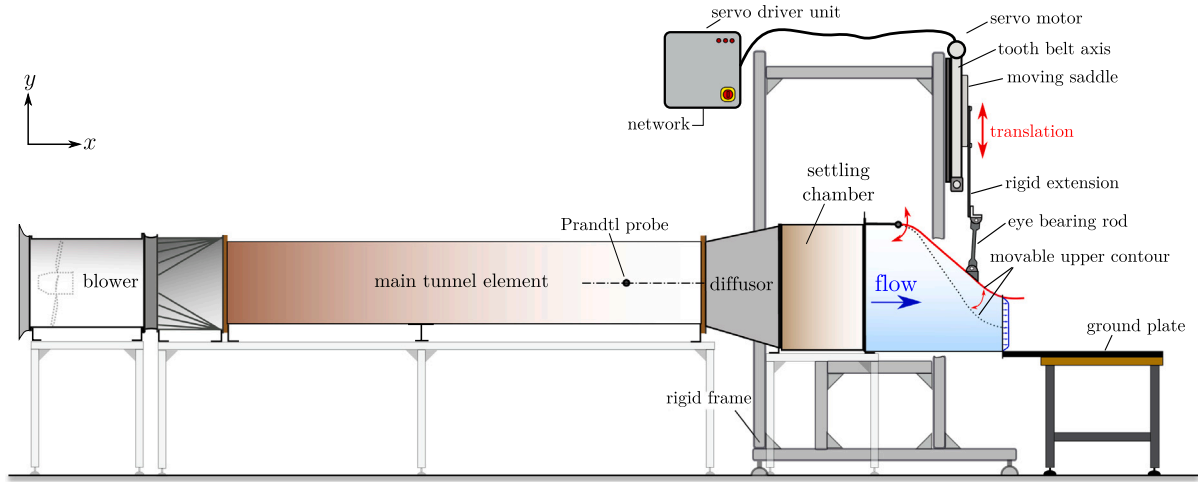


Fig. 1. Sketch of the Eiffel-type wind tunnel with the adaptive nozzle contour in red (Two sketches from Wood and Breuer [13] merged).

in Section 3.1, followed by the scheme for efficient tracking of Lagrangian markers in Section 3.2. The immersed boundary method, specifically adapted for simulating the moving nozzle on a curvilinear block-structured grid, is introduced in Section 3.3. Section 4 provides a comprehensive overview of the computational setup, including the kinematics of the nozzle's upper contour. Finally, Section 5 presents an analysis of the instantaneous flow field induced by the adaptive nozzle, along with a detailed comparison of phase-averaged experimental and numerical results for velocity fields and surface pressure signals.

2. Test case configuration and measuring techniques

The current numerical study builds upon the aforementioned experimental investigations conducted by Wood and Breuer [13,28], which serve as the foundational basis for validating the numerical simulations presented here. Therefore, it is essential to briefly outline the experimental setup and highlight the relevant physical parameters.

Fig. 1 shows a sketch of the Eiffel-type wind tunnel designed to study the adaptive nozzle concept. The key components of the wind tunnel are the blower, the main tunnel element, the diffuser, the settling chamber equipped with four honeycomb flow straighteners, and the adaptive nozzle with its movable upper contour. The nozzle outlet, in its initial position, features a cross-section of $0.25 \text{ m} \times 0.5 \text{ m}$, with a nozzle contraction ratio of 2:1. The flow exiting the nozzle enters the open laboratory space. A ground plate is attached to the underside of the nozzle's outlet, allowing the generated horizontal wind gusts to flow across its surface. By adjusting the rotation rate of the blower, the free-stream velocity in the open test section can range from 1.5 m/s to 20 m/s . The streamwise turbulence intensity at the nozzle's outlet has been determined to be 1.5% [13].

Fig. 1 also depicts the operating principle of the gust generator, which relies on dynamically adjusting the outlet area of the wind tunnel nozzle. According to the principle of mass conservation, a reduction of the cross-section leads to an increase of the velocity at the nozzle's outlet. If this reduction of the cross-section occurs within a short time interval, a sharp rise of the velocity, resembling a wind gust, has to be expected. The general working principle was previously proved by Wood et al. [9] using the first-generation wind gust generator (the *paddle*). However, the paddle-based WGG presented notable drawbacks, including significant pressure losses inside the nozzle and a pronounced detached shear layer caused by flow separation at the paddle. Consequently, improved variants warrant consideration.

The key ingredients of the second-generation WGG are the movable upper contour attached to a rotatable bearing and the tooth belt axis driven by a servo motor. The saddle of the tooth belt axis moves

vertically, both downward and upward, with a stroke length of l_s . All kinematic quantities (length, velocity, acceleration) are controlled using the FESTO automation software tool. An eye bearing rod, coupled with a rigid extension of the saddle, connects the tooth belt axis to the upper contour of the nozzle. Thus, this setup converts the saddle's vertical translational motion into the rotational motion of the upper contour around the bearing. By combining a downward stroke and an upward stroke of the saddle, the nozzle contour is dynamically adapted within a short time interval, leading to a temporary reduction followed by a re-enlargement of the outlet area. In Wood and Breuer [13], an outlet area reduction ratio of 37.2% was chosen, offering a balanced compromise between the gust velocity achieved and the size of the effective area within the test section of the wind tunnel where the gust remains reproducible.

To comprehensively analyze the flow field during the gust event, Wood and Breuer [13] employed three different measurement techniques.

Particle-image velocimetry (PIV) was employed for synchronized measurements of the flow field in the x - y -symmetry plane of the test section. The synchronization procedure relied on a master trigger signal to align the operation of the gust generator with the flow field measurements, ensuring a common time frame. The measurement system utilized a classical mono-PIV configuration, comprising a Litron Nano L PIV laser (532 nm , 200 mJ) and a TSI Powerview CCD camera (6400×4400 pixels, 12-bit). DEHS tracer droplets, with a most penetrating particle size of $0.3 \mu\text{m}$, were used for seeding on the suction side of the wind tunnel. To enable phase-averaging of the flow field, a total of 100 individual gust generations were recorded. Each gust event was resolved by 33 instants in time, ensuring an adequate temporal resolution of the phase-averaged data. With a temporal resolution of 25 ms , the total time span covered was 800 ms , corresponding to a dimensionless time of $t/\Delta t_{\text{gust}} = 1.6$, where $\Delta t_{\text{gust}} = 0.5 \text{ s}$ is the gust generation time defined in Section 4.3.

The second measuring device utilized a constant-temperature anemometer (CTA). This technique was applied to capture time-resolved velocity data for the generated gusts at 11 specific monitoring points within the wind tunnel depicted in Fig. 2. These monitoring locations were distributed along the streamwise direction at normalized coordinates $x/l = 0., 0.1, 0.2 \dots 1.0$, at a fixed height of $y/l = 0.15$ above the ground plate. Note that the origin of the coordinate system is defined on the surface of the ground plate at a distance of $\Delta x = 0.05 \text{ m}$ from the nozzle outlet. The reference length was set to $l = 0.5 \text{ m}$, corresponding to the width of the nozzle. Here phase-averaging of the velocity magnitude was conducted based on 46 gust events. The data were recorded to capture the characteristic development of the gust

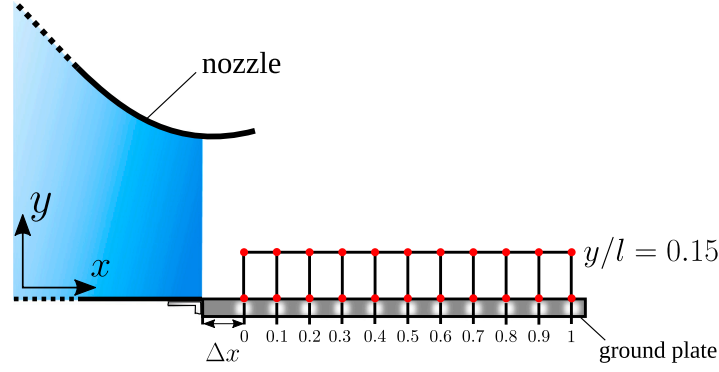


Fig. 2. Definition of the monitoring points on and above the ground plate.

at the defined monitoring points and to validate the conducted PIV measurements.

In addition to PIV and CTA, a pressure transducer (Scanivalve MSP4232 pressure scanner, 32 channels) was employed to record time histories of the surface pressure signals. The positions of the 11 monitoring points correspond to those of the CTA measurements but were now located directly on the ground plate ($y/l = 0$). Surface pressures at all monitoring points were recorded synchronously at a sampling rate of 1000 Hz. Similar to the CTA measurements, phase-averaging was performed using data from 46 wind gust events per measurement point.

3. Numerical framework

3.1. Flow solver

In this study, the incompressible Navier–Stokes equations are solved using the in-house flow solver FASTEST-3D [29]. The equations are spatially discretized via the finite-volume method on a curvilinear, block-structured, body-fitted mesh using a collocated cell-centered variable arrangement. Parallel computation is enabled through domain decomposition and explicit message passing with MPI. Surface and volume integrals are approximated using the midpoint rule, while face-centered flow variables are linearly interpolated from adjacent cell-centered values, yielding second-order accuracy in space. Convective fluxes are computed using the flux-blending technique [30,31], combining 3 % first-order upwind and 97 % second-order central schemes. To ensure a stable pressure-velocity coupling on non-staggered grids and avoid pressure oscillations, the momentum interpolation scheme of Rhie and Chow [32] is used.

Time integration is performed using a second-order accurate semi-implicit predictor–corrector scheme [33]. In the predictor step, a low-storage multi-stage Runge–Kutta method is used to advance the momentum equations and compute an intermediate velocity field, which is not divergence-free. To account for the presence of an immersed boundary, the direct forcing IB method (detailed in Section 3.3) is applied. In this approach, the intermediate velocity is first interpolated to the discrete Lagrangian locations. Based on the difference between the interpolated velocity and the prescribed desired velocity, the fluid–solid boundary force is computed. This force is then spread as a volume force to the surrounding Eulerian grid cells, yielding a modified intermediate velocity that (nearly) satisfies the no-slip condition at the fluid–solid interface. The corrector step then solves a Poisson equation for the pressure correction using an incomplete LU factorization [34], projecting the velocity field onto a solenoidal space and ensuring mass conservation. This approach achieves second-order accuracy in both space and time.

The large-eddy simulation (LES) technique is employed where the large-scale turbulent structures are directly resolved and the effects of the unresolved smaller scales are modeled using a subgrid-scale (SGS) model. Specifically, the standard Smagorinsky model [35] is

applied with a commonly used model constant $C_S = 0.1$ and van-Driest damping in the vicinity of solid walls. With sufficient near-wall grid resolution (as demonstrated in Section 4.1), wall-resolved LES is performed and no wall functions are required. The flow solver was validated based on a variety of test cases, see, e.g., [36–42]. The ingredients, which are required for the incorporation of the IB method, are described next.

3.2. Lagrangian marker tracking

Lagrangian markers are tracked in the computational space (c-space), which is obtained by mapping the structured grid connecting the cell centers (i.e., the z-grid shown in Fig. 3(a)) to an orthonormal grid [43]. This approach circumvents the need for the time-intensive search in the physical space (p-space) that would otherwise be required to locate the new computational cell containing each Lagrangian marker. In c-space, the position of each Lagrangian marker, denoted as ξ_l (see Fig. 3(b)), is expressed as the sum of the cell’s index triple and a fractional offset:

$$\xi_l = [\xi_l, \eta_l, \zeta_l] = [i_l, j_l, k_l] + [\Delta\xi_l^+, \Delta\eta_l^+, \Delta\zeta_l^+]. \quad (1)$$

Tracking the Lagrangian markers in this manner also serves in efficiently identifying the nearest cell center to each Lagrangian marker (defined here as $\mathbf{x}_{i,j,k}$), necessary for setting the bounds of the regularized delta function used in the IB method. The index triples $[i_l, j_l, k_l]$ and $[\hat{i}_l, \hat{j}_l, \hat{k}_l]$ can be directly obtained by applying “int” and “nint” operators (available in programming languages) to the computational coordinates $[\xi_l, \eta_l, \zeta_l]$. The fractional offset can then be evaluated using Eq. (1), and its complement then reads:

$$\Delta\xi_l^- = 1 - \Delta\xi_l^+, \quad \Delta\eta_l^- = 1 - \Delta\eta_l^+, \quad \Delta\zeta_l^- = 1 - \Delta\zeta_l^+. \quad (2)$$

The fractional offset and its complement can be employed to write the transformation T of the marker’s position ξ_l to the p-space by means of a trilinear interpolation using the cell centered values $\mathbf{x}_{i,j,k}$ [43]:

$$\begin{aligned} \mathbf{X}_l = T(\xi_l) = & \mathbf{x}_{i,j,k} \left[\Delta\xi_l^- \Delta\eta_l^- \Delta\zeta_l^- \right] + \mathbf{x}_{i+1,j,k} \left[\Delta\xi_l^+ \Delta\eta_l^- \Delta\zeta_l^- \right] + \\ & \mathbf{x}_{i,j+1,k} \left[\Delta\xi_l^- \Delta\eta_l^+ \Delta\zeta_l^- \right] + \mathbf{x}_{i,j,k+1} \left[\Delta\xi_l^- \Delta\eta_l^- \Delta\zeta_l^+ \right] + \\ & \mathbf{x}_{i+1,j+1,k} \left[\Delta\xi_l^+ \Delta\eta_l^+ \Delta\zeta_l^- \right] + \mathbf{x}_{i+1,j,k+1} \left[\Delta\xi_l^+ \Delta\eta_l^- \Delta\zeta_l^+ \right] + \\ & \mathbf{x}_{i,j+1,k+1} \left[\Delta\xi_l^- \Delta\eta_l^+ \Delta\zeta_l^+ \right] + \mathbf{x}_{i+1,j+1,k+1} \left[\Delta\xi_l^+ \Delta\eta_l^+ \Delta\zeta_l^+ \right]. \end{aligned} \quad (3)$$

This interpolation is used to determine the physical coordinates of the Lagrangian markers, which are required by the immersed boundary solver (as discussed in Section 3.3) and to facilitate visualization of the immersed boundary during the post-processing stage.

The nozzle’s upper contour numerically modeled as an immersed boundary is set into motion by prescribing the desired velocity $\mathbf{U}_l^d(t) = [U_l^d, V_l^d, W_l^d]$ in the physical space for each Lagrangian marker. To update each marker’s position within the c-space, the prescribed velocity components must first be transformed into the c-space as follows:

$$\frac{d\xi_l}{dt} = \left(\frac{\partial \xi}{\partial x} \right)_l U_l^d + \left(\frac{\partial \xi}{\partial y} \right)_l V_l^d + \left(\frac{\partial \xi}{\partial z} \right)_l W_l^d, \quad (4)$$

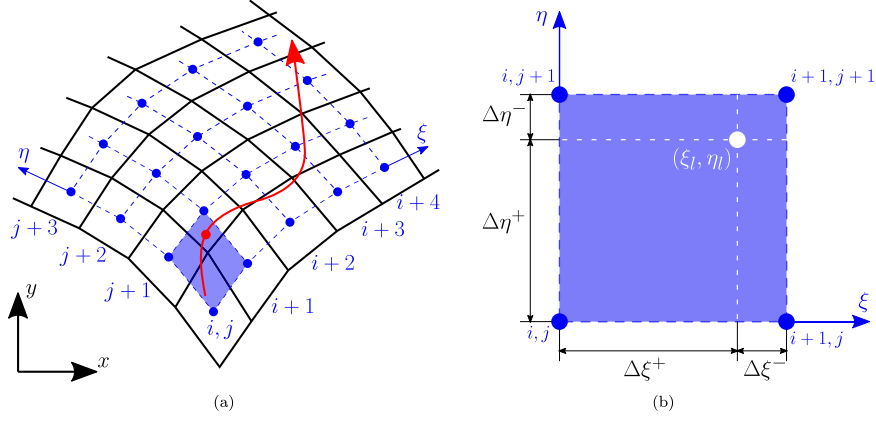


Fig. 3. (a) Two-dimensional p-space schematic sketch showing a computational grid (black solid lines), cell centers (blue dots), z-grid (blue dashed lines), and a Lagrangian marker's trajectory (red solid line). (b) Corresponding c-space representation showing the coefficients of the trilinear interpolation.

$$\frac{d\eta_l}{dt} = \left(\frac{\partial \eta}{\partial x} \right)_l U_l^d + \left(\frac{\partial \eta}{\partial y} \right)_l V_l^d + \left(\frac{\partial \eta}{\partial z} \right)_l W_l^d, \quad (5)$$

$$\frac{d\xi_l}{dt} = \left(\frac{\partial \xi}{\partial x} \right)_l U_l^d + \left(\frac{\partial \xi}{\partial y} \right)_l V_l^d + \left(\frac{\partial \xi}{\partial z} \right)_l W_l^d. \quad (6)$$

Hence, the location of a Lagrangian marker is updated by carrying out the temporal integration:

$$\xi_l^{n+1} = \xi_l^n + \left(\frac{\partial \xi}{\partial x} \right)_l \int_{t^n}^{t^{n+1}} U_l^d dt + \left(\frac{\partial \xi}{\partial y} \right)_l \int_{t^n}^{t^{n+1}} V_l^d dt + \left(\frac{\partial \xi}{\partial z} \right)_l \int_{t^n}^{t^{n+1}} W_l^d dt, \quad (7)$$

$$\eta_l^{n+1} = \eta_l^n + \left(\frac{\partial \eta}{\partial x} \right)_l \int_{t^n}^{t^{n+1}} U_l^d dt + \left(\frac{\partial \eta}{\partial y} \right)_l \int_{t^n}^{t^{n+1}} V_l^d dt + \left(\frac{\partial \eta}{\partial z} \right)_l \int_{t^n}^{t^{n+1}} W_l^d dt, \quad (8)$$

$$\zeta_l^{n+1} = \zeta_l^n + \left(\frac{\partial \zeta}{\partial x} \right)_l \int_{t^n}^{t^{n+1}} U_l^d dt + \left(\frac{\partial \zeta}{\partial y} \right)_l \int_{t^n}^{t^{n+1}} V_l^d dt + \left(\frac{\partial \zeta}{\partial z} \right)_l \int_{t^n}^{t^{n+1}} W_l^d dt. \quad (9)$$

For a fixed grid as used in the present case, the spatial metric coefficients appearing in Eqs. (7) to (9) are time independent. They represent the entries of the Jacobian of the p-space to the c-space coordinate transformation $\mathbf{J}$ and are evaluated at the Lagrangian location. Employing the CZ+ tracking scheme proposed by Schäfer and Breuer [43], the Jacobian of the inverse coordinate transformation $\mathbf{J}^{-1}$ (i.e., c-space to p-space) is analytically evaluated by differentiating Eq. (3). The first column of $\mathbf{J}^{-1}$ at a position ξ_l reads:

$$\begin{aligned} \mathbf{J}_1^{-1}(\xi_l) &= \frac{\partial \mathbf{X}_l}{\partial \xi_1} = \frac{\partial T(\xi_l)}{\partial \xi_1} = \Delta \eta_l^- \Delta \zeta_l^- (\mathbf{x}_{i+1,j,k} - \mathbf{x}_{i,j,k}) + \\ &\quad \Delta \eta_l^+ \Delta \zeta_l^- (\mathbf{x}_{i+1,j+1,k} - \mathbf{x}_{i,j+1,k}) + \\ &\quad \Delta \eta_l^- \Delta \zeta_l^+ (\mathbf{x}_{i+1,j,k+1} - \mathbf{x}_{i,j,k+1}) + \\ &\quad \Delta \eta_l^+ \Delta \zeta_l^+ (\mathbf{x}_{i+1,j+1,k+1} - \mathbf{x}_{i,j+1,k+1}). \end{aligned} \quad (10)$$

Following the evaluation of $\mathbf{J}_m^{-1}(\xi_l)$ where $m = 1, 2, 3$ and inverting it, the spatial metrics in Eqs. (7) to (9) are obtained.

3.3. Immersed boundary method with direct forcing approach

The nozzle's upper contour is modeled as a rigid boundary moving in an incompressible viscous fluid by the immersed boundary (IB) method with the direct forcing approach proposed by Uhlmann [17]. At the continuous level, the impact of the immersed boundary (Γ) on the surrounding fluid is incorporated into the governing equations by adding a forcing term $\mathbf{f}$ to their right-hand side. The Navier–Stokes equations for an incompressible fluid on the domain Ω then read:

$$\left. \begin{aligned} \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \nabla \cdot (\mathbf{u}\mathbf{u}) &= \mu \Delta \mathbf{u} - \nabla p + \mathbf{f} \\ \nabla \cdot \mathbf{u} &= 0 \end{aligned} \right\} \quad \mathbf{x} \in \Omega, \quad (11)$$

where $\mathbf{f}$ is a volume force arising from the presence of the immersed boundary. At the discrete level, variables are differentiated by their locations, with lower-case letters indicating Eulerian (fluid grid) quantities and upper-case letters referring to Lagrangian (solid boundary) quantities. In the collocated grid arrangement employed by the finite-volume solver described in Section 3.1, Eulerian locations correspond to the cell centers $\mathbf{x}_e$ (i.e., $\mathbf{x}_e \in \Omega \forall 1 \leq e \leq N_E$, where N_E denotes the total number of cell centers). The Lagrangian locations $\mathbf{X}_l$ represent the distributed Lagrangian force points defining the immersed boundary Γ (i.e., $\mathbf{X}_l \in \Gamma \forall 1 \leq l \leq N_L$, where N_L denotes the total number of Lagrangian markers).

The methodology of the direct forcing method involves evaluating the force at the fluid–solid interface (i.e., within the Lagrangian frame of reference). The force is formulated in such a manner that it enforces (to a certain extent) the no-slip boundary condition at each Lagrangian marker. In the context of the predictor–corrector method, the Lagrangian volume force at a time step t^n reads [17]:

$$\mathbf{F}_l^n = \rho \frac{\mathbf{U}_l^d - \mathbf{U}_l^*}{\Delta t}. \quad (12)$$

Here, $\mathbf{U}_l^d$ and $\mathbf{U}_l^*$ are the desired and the intermediate velocity after the predictor step at a Lagrangian location $\mathbf{X}_l$, respectively. The desired velocity $\mathbf{U}_l^d$ is explicitly prescribed by the rigid-body motion of the nozzle's upper contour defined in Section 4.3. In the context of time-dependent immersed boundaries, the Lagrangian markers are typically not coinciding with the Eulerian cell centers. This scenario is illustrated in Fig. 4, which shows a representative case of the spatial relationship between the Lagrangian markers and the Eulerian grid. Since the fluid solver predicts the intermediate velocity only at the Eulerian locations $\mathbf{x}_e$, $\mathbf{U}_l^*$ has to be evaluated by interpolating the Eulerian flow variables to the corresponding Lagrangian locations $\mathbf{X}_l$. This process of transferring information between the two discrete frames of reference is referred to as the interpolation operation. On a uniform one-dimensional Eulerian grid of grid spacing h , this operation is performed by employing a regularized Dirac delta function δ_h [15]:

$$\delta_d(x_e - X_l) = \frac{1}{d} \varphi(r), \quad (13)$$

where d is the dilation parameter ($d = h$ for uniform grids) and φ is a continuous kernel function of symmetric compact support. The dilation parameter defines the extension of the kernel's support Δ_l , and the parameter r is the normalized distance between an Eulerian and a Lagrangian coordinate ($r = (x - X)/d$). On a three-dimensional uniform Cartesian grid g_h of grid size h , a three-dimensional regularized delta function is formed as the tensor product of three one-dimensional functions (one for each coordinate direction) and employed to perform

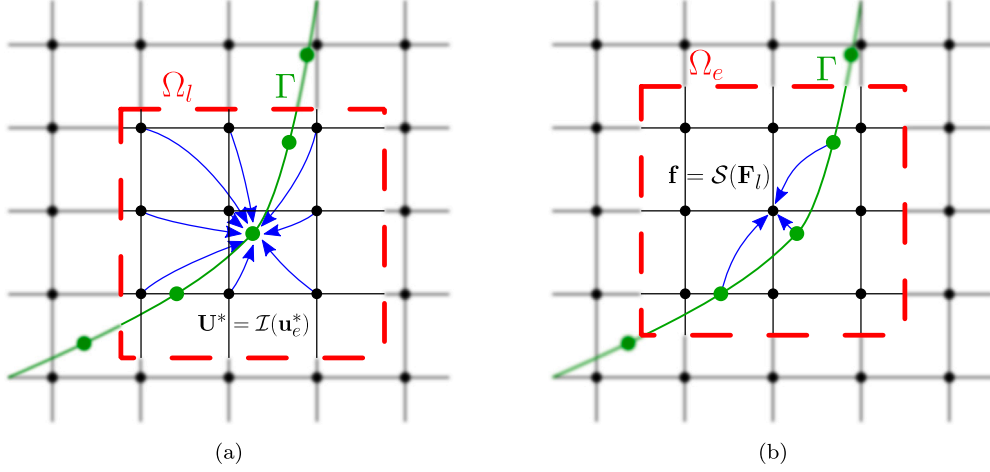


Fig. 4. (a) Interpolation of Eulerian velocities to a Lagrangian location. (b) Spreading of Lagrangian forces to an Eulerian location.

the interpolation operation $\mathcal{I}(\mathbf{u}_e^*)$:

$$\mathbf{U}_l^* = \sum_{e \in N_{E,l}} \mathbf{u}_e^* \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e, \quad (14)$$

where $N_{E,l}$ denotes the set of Eulerian cells whose centroids $\mathbf{x}_e$ lie within the support of the kernel associated with the l th Lagrangian marker, and Δv_e is the volume of the e th Eulerian cell. An illustration of the interpolation operation, which evaluates a discrete Lagrangian quantity as a weighted average of neighboring discrete Eulerian quantities, is depicted in Fig. 4(a).

Following the interpolation step, the Lagrangian boundary force $\mathbf{F}_l$ is directly computed at each Lagrangian force point using Eq. (12). The presence of the immersed boundary is then accounted for in the momentum equations governing the fluid by spreading the Lagrangian volume forces onto the Eulerian grid and incorporating the spread force as a volumetric source term on the right-hand side. The spreading operation $\mathcal{S}(\mathbf{F}_l)$, illustrated in Fig. 4(b), utilizes the same regularized delta function as the interpolation operation to evaluate the Eulerian volume force:

$$\mathbf{f}_e = \sum_{l \in N_{L,e}} \mathbf{F}_l \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta V_l, \quad (15)$$

where $N_{L,e}$ denotes the set of Lagrangian markers whose kernel support encompasses the e th Eulerian cell, and ΔV_l is the volume associated with the l th Lagrangian marker. The Lagrangian volume can be expressed as $\Delta V_l = \Delta S_l \varepsilon_l$, where ΔS_l represents the surface area associated with the l th marker and ε_l denotes its characteristic thickness. In uniform Eulerian grids, this volume is assumed to be equal to the volume of the fluid cell, provided that the Lagrangian markers are uniformly distributed such that $\Delta S_l \approx \Delta v_e^{2/3}$ (i.e., $\varepsilon_l \approx h$ [17]). An alternative approach for determining the characteristic thickness is to set it such that when the force is spread from the Lagrangian marker and interpolated back to it, the exact force is recovered [44]. This method necessitates the solution of a linear system of equations that enforces the duality of the interpolation and the spreading operators. Zhou and Balachandar [45], however, have shown that the Lagrangian volume serves as a relaxation factor governing how fast the no-slip condition is satisfied. The authors have also found that the optimal choice of the Lagrangian volume corresponds to the largest value permitted by a stability condition. Eventually, the choice of the Lagrangian volume is somewhat arbitrary, as it primarily influences the magnitude of the feedback forcing [46]. In this work, the real thickness of the nozzle's upper contour is specifically used to define the third dimension for the Lagrangian grid as discussed in Section 4.2.

The steps of the predictor–corrector method, modified to account for the immersed boundary, can now be summarized by the following

equations:

$$1. \quad \mathbf{u}_e^* = \mathbf{u}_e^n + \frac{\Delta t}{\rho} \mathbf{r} \mathbf{h} \mathbf{s}^n \quad \forall \mathbf{x}_e \in g_h, \quad (16)$$

$$2. \quad \mathbf{U}_l^* = \mathcal{I}(\mathbf{u}_e^*) \quad \forall \mathbf{X}_l \in \Gamma, \quad (17)$$

$$3. \quad \mathbf{F}_l = \rho \frac{\mathbf{U}_l^d - \mathbf{U}_l^*}{\Delta t} \quad \forall \mathbf{X}_l \in \Gamma, \quad (18)$$

$$4. \quad \mathbf{f}_e = \mathcal{S}(\mathbf{F}_l) \quad \forall \mathbf{x}_e \in g_h, \quad (19)$$

$$5. \quad \mathbf{u}_e^{**} = \mathbf{u}_e^* + \frac{\Delta t}{\rho} \mathbf{f}_e \quad \forall \mathbf{x}_e \in g_h, \quad (20)$$

$$6. \quad \Delta p'_e = \frac{\rho}{\Delta t} \nabla \cdot \mathbf{u}_e^{**} \quad \forall \mathbf{x}_e \in g_h, \quad (21)$$

$$7. \quad \mathbf{u}_e^{n+1} = \mathbf{u}_e^n - \frac{\Delta t}{\rho} \nabla p'_e \quad \forall \mathbf{x}_e \in g_h. \quad (22)$$

Compared to the standard predictor–corrector scheme, steps 2 to 5 have been added and lead to a second intermediate velocity $\mathbf{u}_e^{**}$, that satisfies (with a decreasing error at each time step) the no-slip boundary condition at the fluid–solid interface. This velocity serves as the foundation for constructing the right-hand side of the Poisson Eq. (21) used to compute the pressure correction, p'_e (step 6), and subsequently determine the updated velocity, $\mathbf{u}_e^{n+1}$ (step 7).

Several regularized delta function kernels, effectively smearing the boundary forces over Eulerian grid cells within their compact support, have been derived in the literature. The selection of a particular kernel often depends on the required level of accuracy, the smoothing properties, and the desired numerical efficiency. Nevertheless, all smoothing kernels are designed to satisfy, at a minimum, the zeroth- and the first-order discrete moment conditions on uniform grids:

$$\sum_{e \in N_{E,l}} \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e = 1 \quad \forall l, \quad (23)$$

$$\sum_{e \in N_{E,l}} (\mathbf{x}_e - \mathbf{X}_l) \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e = 0 \quad \forall l. \quad (24)$$

These discrete analogues of the kernel's fundamental properties ensure the integral conservation of the force and torque in the spreading operation:

$$\begin{aligned} \sum_{e \in N_E} \mathbf{f}_e \Delta v_e &= \sum_{e \in N_E} \sum_{l \in N_{L,e}} \mathbf{F}_l \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta V_l \Delta v_e \\ &= \sum_{l \in N_L} \mathbf{F}_l \left(\sum_{e \in N_{E,l}} \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e \right) \Delta V_l = \sum_{l \in N_L} \mathbf{F}_l \Delta V_l, \end{aligned} \quad (25)$$

$$\sum_{e \in N_E} \mathbf{x}_e \times \mathbf{f}_e \Delta v_e = \sum_{e \in N_E} \mathbf{x}_e \times \left(\sum_{l \in N_{L,e}} \mathbf{F}_l \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta V_l \right) \Delta v_e$$

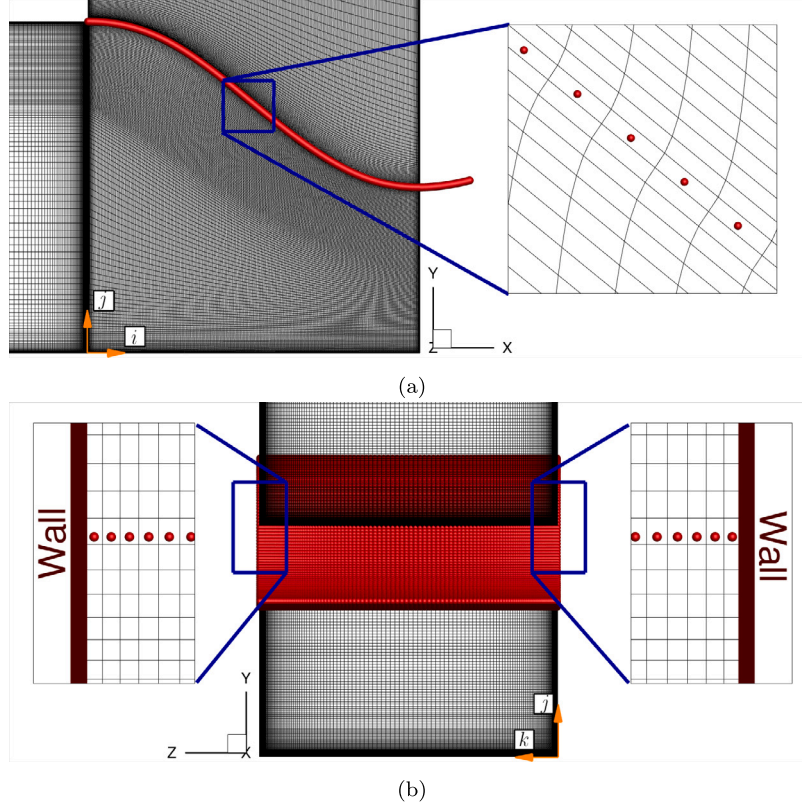


Fig. 5. Schematic showing the distribution of Lagrangian markers (red spheres) (a) in the z -plane of the curvilinear grid. (b) Near-wall boundaries in an i -plane of the grid within the nozzle region.

$$= \sum_{l \in N_L} \left(\sum_{e \in N_{E,l}} \mathbf{x}_e \delta_h(\mathbf{x}_e - \mathbf{X}_l) \Delta v_e \right) \times \mathbf{F}_l \Delta V_l = \sum_{l \in N_L} \mathbf{X}_l \times \mathbf{F}_l \Delta V_l. \quad (26)$$

Unlike most IBM applications that rely on a Cartesian Eulerian grid, this work employs a curvilinear stretched grid to discretize the nozzle region (as shown in Fig. 5(a) and detailed in Section 4.1). This approach clusters grid points near the walls, enabling an accurate resolution of the boundary layer. Furthermore, employing a curvilinear grid avoids extending the refinement to flow regions where it is not necessary, leading to significant computational savings.

Since the nozzle's upper contour moves between two side walls (see Fig. 5(b)), Lagrangian points must inevitably be positioned near the boundaries of the computational domain to ensure a perfect seal at the corners. This proximity limits the applicability of kernels with symmetric support for these specific Lagrangian markers, necessitating the use of non-symmetric kernel's support that accounts for the boundaries of the computational domain.

Remedies to the two aforementioned deviations from the classical IBM (i.e., markers on curvilinear stretched grids and markers with one-sided kernels) have been previously proposed in the literature. This work incorporates the moving least squares version of the direct forcing immersed boundary method (MLS-IBM) introduced by Vanella and Balaras [22], beside its recent one-sided extension proposed by Bale et al. [27]. A key advantage of the MLS method is that it does not impose a specific arrangement of Eulerian interpolation points around a Lagrangian marker (e.g., uniform distribution). This characteristic makes the MLS approach effective for handling immersed boundaries on curvilinear grids.

The work of Bale et al. [27] extends the MLS technique to construct one-sided (non-symmetric) IB kernels from standard two-sided (symmetric) ones. This advancement enables velocity interpolation and force spreading exclusively on one side of the fluid–solid interface. This

specific ability is particularly beneficial for avoiding spurious flows within enclosed areas and for facilitating the handling of immersed structures near the boundaries of the computational domain.

The following provides a summary of the MLS method, along with a new proposition for its application to curvilinear block-structured grids. A detailed explanation of how the general framework is applied to different Lagrangian markers is also presented.

3.3.1. Standard MLS formulation

The basic idea of the moving least squares version of the direct forcing immersed boundary method is to compute interpolation weights that transfer a quantity to a Lagrangian marker from arbitrarily distributed Eulerian cell centers in its vicinity by solving a weighted least squares optimization problem. The MLS method approximates the value of the Eulerian variable g at the discrete location of the l th Lagrangian marker by:

$$g_l = \sum_{i=1}^m p_i(\mathbf{X}_l) c_i^g(\mathbf{X}_l) = \mathbf{p}(\mathbf{X}_l)^T \mathbf{c}^g(\mathbf{X}_l). \quad (27)$$

Here, $\mathbf{p}(\mathbf{X}_l) \in \mathbb{R}^m$ is the basis function vector evaluated at the Lagrangian location $\mathbf{X}_l$ and $\mathbf{c}^g(\mathbf{X}_l) \in \mathbb{R}^m$ is the vector of unknown coefficients depending on the Eulerian quantity g_e . For a second-order accurate three-dimensional spatial discretization scheme, it is sufficient to define $\mathbf{p}$ as a linear basis function vector (i.e., $m = 4$ and $\mathbf{p}(\mathbf{x}) = [1 \ x \ y \ z]$). The unknown coefficient vector $\mathbf{c}^g(\mathbf{X}_l)$ is found by minimizing the weighted L^2 -norm of the error function:

$$\min_{\mathbf{c}^g(\mathbf{X}_l)} J_l = \frac{1}{2} \sum_{e \in N_{E,l}} \phi_{e,l} (\mathbf{p}(\mathbf{x}_e)^T \mathbf{c}^g(\mathbf{X}_l) - g_e)^2, \quad (28)$$

where ϕ denotes the MLS weighting function and $\phi_{e,l} = \phi(\mathbf{x}_e - \mathbf{X}_l)$ represents the discrete interpolation weight for the e th Eulerian cell. This discrete weight is equivalent to the product of the kernel function normalized by the dilation coefficient and the volume of the fluid cell

Δv_e (i.e., $\delta_h(\mathbf{x}_e - \mathbf{X}_I) \Delta v_e$ appearing in Eq. (14)). Eq. (28) can also be written in a matrix form as:

$$\min_{\mathbf{c}^g(\mathbf{X}_I)} J_I = \frac{1}{2} (\mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_I) - \mathbf{g})^\top \text{diag}(\phi_I) (\mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_I) - \mathbf{g}), \quad (29)$$

where the entries of the polynomial matrix $\mathcal{A} \in \mathbb{R}^{m \times N_{E,I}}$ are the values of the basis functions at the Eulerian interpolation stencil (i.e., $\mathcal{A}_{ij} = p_i(\mathbf{x}_j)$, with $i = 1, \dots, m$, $j = 1, \dots, N_{E,I}$). The minimization of the error J_I with respect to the vector of unknown coefficients (i.e., $\frac{\partial J_I}{\partial \mathbf{c}^g(\mathbf{X}_I)} = 0$) yields (see Appendix A.1 for details):

$$\mathbf{c}^g(\mathbf{X}_I) = \mathcal{G}^{-1} B \mathbf{g}, \quad (30)$$

where $\mathcal{G} = \mathcal{A} \text{diag}(\phi_I) \mathcal{A}^\top$ is the positive-definite Gram matrix and $B = \mathcal{A} \text{diag}(\phi_I)$. Substituting the solution of $\mathbf{c}^g(\mathbf{X}_I)$ in Eq. (27) yields:

$$\mathbf{g}_I = \mathbf{p}(\mathbf{X}_I)^\top \mathcal{G}^{-1} B \mathbf{g} = \tilde{\phi}_I^\top \mathbf{g} = \sum_{e \in N_{E,I}} \tilde{\phi}_{e,I} g_e, \quad (31)$$

where $\tilde{\phi}_I = B^\top \mathcal{G}^{-1} \mathbf{p}(\mathbf{X}_I) \in \mathbb{R}^{N_{E,I}}$ is the MLS kernel function vector (also known as the *generating function*). Unlike the expression given in Eq. (27), where the coefficient vector (i.e., $\mathbf{c}^g(\mathbf{X}_I)$) is dependent on both the Lagrangian marker's position and the Eulerian variable being interpolated, the form in Eq. (31) depends only on the coordinates of the Lagrangian marker. In the context of the immersed boundary method, it is more practical to directly use the latter form as this enables the use of a single kernel function for the interpolation of velocity components and the spreading of the forces.

It is worth mentioning that, although not explicitly imposed, the generating functions evaluated by the MLS method satisfy the discrete moment conditions (also known as the polynomial reproduction constraints):

$$\sum_{e \in N_{E,I}} p_i(\mathbf{x}_e) \tilde{\phi}_{e,I} = p_i(\mathbf{X}_I), \quad \forall i = 1, \dots, m, \quad (32)$$

with the matrix form written as:

$$\mathcal{A} \tilde{\phi}_I = \mathbf{p}(\mathbf{X}_I). \quad (33)$$

A proof can be found in Appendix A.2.

3.3.2. New proposition for curvilinear block-structured grids

In diffuse-interface IB methods, the dilation parameter (appearing in Eq. (13)) defines the interpolation window Ω_I and, consequently, the number of Eulerian cells included in the stencil of the Lagrangian marker ($N_{E,I}$). In addition, this parameter plays a role in computing the initial interpolation weights ($\phi_{e,I}$) required for the MLS procedure.

In most classical IBM applications, the structure of interest is immersed in a Cartesian uniform grid. In such cases, the dilation parameter for each coordinate direction is straightforward, and it is equal to the uniform grid spacing along that direction. In non-uniform grids, however, local dilation parameters are estimated in the literature for each Lagrangian marker using a criterion that considers the nearest Eulerian cell ($\mathbf{x}_\hat{e}$) and its immediate neighbors (see, e.g., [23,44,47]). Then, based on the evaluated parameters, the set of Eulerian cell centers within the interpolation window is sought. However, some Eulerian centroids may lie at the boundary of the interpolation window, resulting in near-zero weight values, which can render the moment matrix ($\mathcal{G}$ in Eq. (30)) singular. In addition, this approach introduces an extra computational overhead due to the need for a larger search radius around the Lagrangian marker. That would also result in systems of varying sizes for different Lagrangian markers, requiring dynamic memory allocation at each time step (for moving immersed boundaries) for the vectors and matrices needed for the MLS method. The issue becomes more pronounced when solving the problem in parallel, particularly when the interpolation window of a Lagrangian marker, located on a connectivity patch between two parallel blocks of a block-structured grid, extends to cover multiple ghost cells. In second-order

accurate finite-volume codes, however, only one ghost cell is typically employed.

The proposition made here makes the determination of the interpolation weights independent of the dilation parameter. Consequently, there is no need to estimate the dilation parameter, and the risk of obtaining zero interpolation weights is eliminated. The interpolation window is also fixed to encompass the exact number of Eulerian cells that the kernel φ is intended to cover. In other words, a n -point kernel incorporates n Eulerian cell centers. This is achieved by evaluating the interpolation weights with respect to the computational space (explained in Section 3.2) instead of the physical space as:

$$\phi_{e,I}^c = \frac{1}{\Delta v_{\hat{e}}} \varphi(\xi_e - \xi_I) \varphi(\eta_e - \eta_I) \varphi(\zeta_e - \zeta_I) \Delta v_e, \quad (34)$$

where the superscript c indicates that the value is computed with respect to the computational space and $\Delta v_{\hat{e}}$ is the volume of the closest Eulerian cell. Note that the normalized parameter r has been replaced by the difference between the Eulerian and the Lagrangian computational coordinates, which are already normalized. For a uniform grid, $\phi_{e,I}^c$ will immediately satisfy the moment conditions in Eqs. (23) and (24) provided that the full support of the kernel φ , which satisfies these conditions, is used. For a non-uniform distribution of Eulerian cells around a Lagrangian marker, the kernel's weights are evaluated without accounting for the non-uniformity of the grid. However, they are scaled by the ratio $\frac{\Delta v_e}{\Delta v_{\hat{e}}}$ for each cell center. This scaling causes $\phi_{e,I}^c$ to deviate from the moment conditions as $\frac{\Delta v_e}{\Delta v_{\hat{e}}} \neq 1$ for $e \neq \hat{e}$.

A thorough validation study of the proposed immersed boundary approach on a block-structured curvilinear grid is performed in Appendix B based on the flow past a stationary and an oscillating cylinder.

3.3.3. General framework

The MLS procedure detailed in Section 3.3.1 is used in conjunction with the approach proposed in Section 3.3.2. If a Lagrangian marker is far from the boundaries of the computational domain, the 3-point kernel proposed by Roma et al. [18] is applied in all local directions leading in total to 27 points. Contrarily, when a Lagrangian marker is near the domain boundary, the 5-point kernel by Bao et al. [20] is employed only along the constrained directions. Fig. 6 depicts the selection of kernels based on the marker's proximity to the boundaries of the computational domain. This selection is motivated by the findings of Bale et al. [27], which demonstrate that using a wider kernel helps to prevent the Gram matrix in the one-sided MLS from becoming ill-conditioned due to insufficient support.

The general procedure for each Lagrangian marker can be summarized as follows:

1. The nearest Eulerian cell center in the computational space and its direct neighbors in each local block direction (i.e., i , j and k) are identified.
2. A check is carried out whether any of the Eulerian neighbors belong to the boundaries of the computational domain and the affected local directions are constrained accordingly to exclude those neighbors (e.g., wall). If a local direction is constrained, it is extended from its other side to include one extra Eulerian interpolation point.
3. Based on the absolute difference between the Lagrangian computational coordinates and its Eulerian counterparts along each local block direction, a kernel function is assigned. For instance, if $\max(|\xi_e - \xi_I|) \leq 1.5 \forall e \in N_{E,I}$, the 3-point kernel is employed. This will be the case (in all local block directions) for all Lagrangian markers far from the boundaries of the computational domain. However, if $\max(|\xi_e - \xi_I|) > 1.5$, this indicates that the interpolation window was extended in step 2 and the 5-point kernel is assigned to the i th direction.

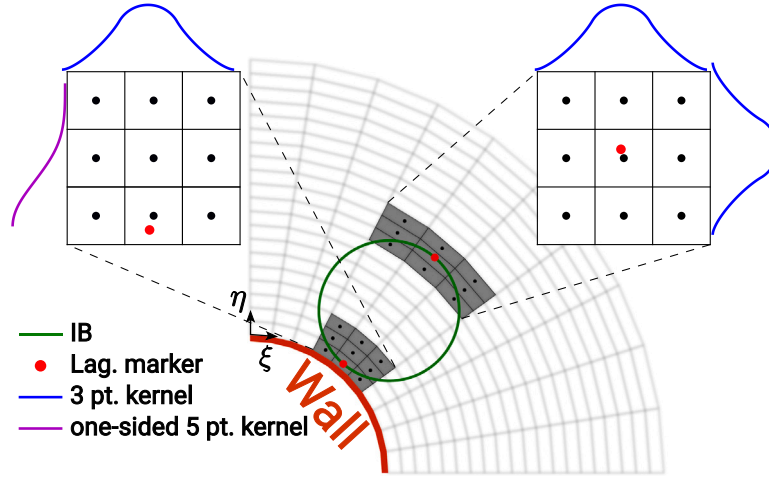


Fig. 6. Schematic sketch showing two Lagrangian markers within a curvilinear block-structured grid. The sketch highlights the interpolation kernels employed along each local coordinate direction for markers near and far from the boundary of the computational domain.

4. The MLS weights are evaluated based on Eq. (34). These weights do not satisfy the discrete moment conditions due to the curvilinear Eulerian grid, the reduced (one-sided) kernel support, or both.
5. The MLS kernel function vector is found using Eq. (A.15). These modified weights satisfy the polynomial reproduction constraints needed for the conservation of force and torque.

4. Description of the setup

4.1. Computational domain, boundary conditions and numerical grid

The purpose of the current setup is to develop a numerical model of the experimental wind gust generator described by Wood and Breuer [13,28], i.e., the adaptive nozzle. This approach enables a more detailed analysis of the complex flow generated by the nozzle's moving upper contour, as all flow field variables, including velocities and pressure, are computed across the entire computational domain. Fig. 7 illustrates the numerical setup designed to replicate its experimental counterpart shown in Fig. 1. A uniform horizontal velocity field of $u_{inlet}^{steady} = 2.53$ m/s is defined at the inlet patch, i.e., the left cross-section of the square duct. Within the duct and the nozzle, the boundary layers develop along all walls, reaching a free-stream velocity of $u_{\infty} = 5.14$ m/s at the nozzle's outlet, thus matching the conditions of the experimental setup.

Note that the inflow turbulence intensity found in the experiment (see Section 2) has been neglected in the present predictions due to the following two reasons. First, the reported value of 1.5 % refers only to the bulk flow in the wind tunnel. However, the turbulence intensity is expected to vary across the nozzle outlet cross-section, particularly near the walls. Second, the knowledge about the turbulence intensity alone is not sufficient to artificially generate inflow turbulence. At the very least, additional information regarding the characteristic length scales of the approaching inflow turbulence is required. Since none of these quantities is currently available, it is more reasonable to neglect the low-level inflow turbulence entirely rather than to represent it based on potentially incorrect assumptions.

No-slip and impermeability boundary conditions are imposed on the walls of the duct, the nozzle, the far-field walls and the ground plate. A convective boundary condition is applied at the outlet patch with a convective velocity of $u_{conv} = u_{\infty}$.

As depicted in Fig. 7, the real geometry in front of the nozzle is simplified, retaining only the square duct directly ahead of the nozzle with a newly defined length of $l_{duct} = 1$ m and a cross-sectional side length of $w = 0.5$ m. This duct replaces the flow straightener, the

diffusor, and the square duct of the wind tunnel. The nozzle's geometry with a contraction ratio A_{inlet}/A_{outlet} of 2:1 is exactly replicated to ensure that identical boundary conditions as in the experiment are applied. The computer-aided design (CAD) sketch used to define the nozzle's upper contour, which is fabricated and used in the experiments of Wood and Breuer [13,28] is depicted in Fig. 8. A precise definition of this curve is crucial for generating the curvilinear Eulerian grid and for providing an accurate distribution of Lagrangian markers along the curve.

Based on the coordinates and boundary conditions depicted in Fig. 8, the geometry of the upper contour is defined by the following piecewise cubic spline:

$$y_{noz} = \begin{cases} -1.662x^3 - 1.884x^2 + 0.5, & 0 \leq x < 0.12 \\ 6.855(x - 0.12)^3 - 2.482(x - 0.12)^2 - 0.524(x - 0.12) + 0.47, & 0.12 \leq x < 0.25 \\ 4.581(x - 0.25)^3 + 0.191(x - 0.25)^2 - 0.821(x - 0.25) + 0.375, & 0.25 \leq x < 0.43 \\ -4.461(x - 0.43)^3 + 2.665(x - 0.43)^2 - 0.307(x - 0.43) + 0.26, & 0.43 \leq x < 0.5 \\ -21.045(x - 0.5)^3 + 2.926(x - 0.5)^2 + 0.25, & 0.5 \leq x < 0.525 \\ 2.63(x - 0.525)^3 + 1.347(x - 0.525)^2 + 0.106(x - 0.525) + 0.251, & 0.525 \leq x < 0.575. \end{cases} \quad (35)$$

Note that for the sake of brevity the unit [m] is left out here. The spline in the range $0 \text{ m} \leq x \leq 0.5 \text{ m}$ is used to generate a structured curvilinear grid around the nozzle. However, the entire spline including the tail of the nozzle compromising the range $0.5 \text{ m} \leq x \leq 0.575 \text{ m}$ is employed to define the coordinates of the Lagrangian markers. The Eulerian grid depicted as a zoomed excerpt in Fig. 9 is a block-structured curvilinear grid composed of 105.6×10^6 control volumes. Note that this number of control volumes and their distribution in the computational domain are the outcome of an optimization process carried out prior to the final simulations. This preliminary study is not described here for the sake of brevity.

The resulting geometric structure, made up of 12 blocks, is subdivided into 786 parallel blocks. This setup enables the parallel execution across 786 cores, achieving a load balancing efficiency according to the pure CFD prediction without Lagrangian markers of 100 %.

To accurately capture the significant viscous effects that give rise to thin boundary layers, grid refinement near rigid walls including duct walls, nozzle walls and the ground plate is necessary. This refinement ensures that the first cell centers are positioned within the viscous sublayer ($y^+ < 5$). The hydraulic diameter at the nozzle's outlet, defined as $D_h = 4wh/(2(w+h)) = 0.333$ m (with $h = 0.25$ m), is used as the characteristic length for determining the Reynolds number Re

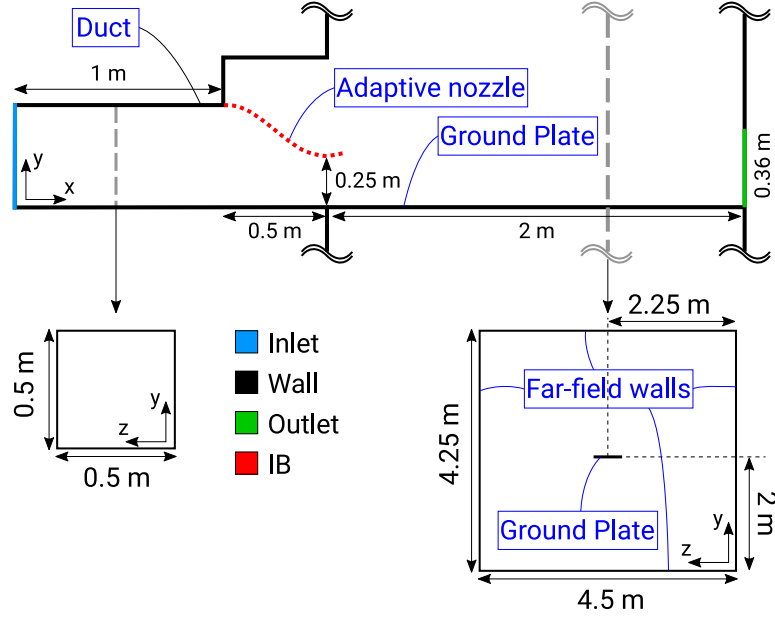


Fig. 7. Geometry of the test case and definition of the boundary conditions.

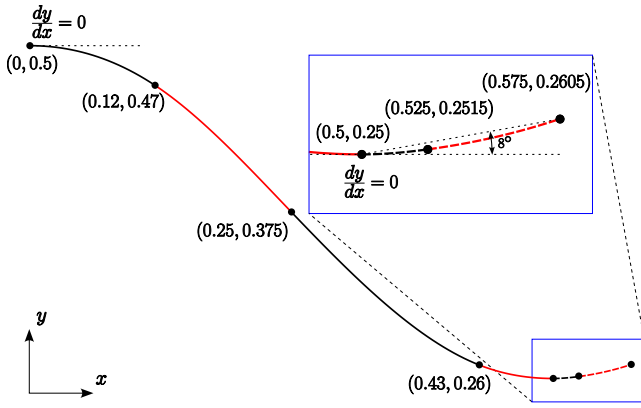


Fig. 8. Nozzle's upper contour spline. All coordinates are given in meter.

$= \rho D_h u_\infty / \mu = 114,878$. Based on this Reynolds number, the Darcy friction factor as reported by Moody [48] is:

$$f = 0.0055 \left[1 + \left(2 \times 10^{-4} \frac{k}{D_h} + \frac{10^6}{\text{Re}} \right)^{\frac{1}{3}} \right], \quad (36)$$

where k is the pipe's effective roughness height. Assuming a smooth pipe (i.e., $k = 0$) the skin friction coefficient is evaluated by $c_f = f/4 = 4.2 \times 10^{-3}$.

To accurately resolve the near-wall region, the maximum height of the first grid cell off the wall, denoted as Δy , can be estimated such that a dimensionless wall distance at the cell center of $y_{cc}^+ = 2$ is achieved. This results in the relationship $\Delta y/D_h = (4/\text{Re}) \cdot \sqrt{2/c_f} \approx 7.6 \times 10^{-4}$, yielding $\Delta y \approx 2.5 \times 10^{-4}$ m. To optimize computational efficiency, the grid is mildly stretched in the off-wall directions, with a maximum expansion factor of 1.04. A maximum grid spacing of 10^{-3} m is applied in the x - y plane within the region that will be passed by the nozzle's contour during its movement. This refinement ensures an accurate fluid-IB interaction and a precise representation of the moving shear layer induced by the nozzle's upper contour. Note that Van-Driest damping of the eddy viscosity near the immersed boundary is presently not considered. This omission stems from the fact that implementing

a wall-distance-based damping function within a diffuse-interface immersed boundary framework is non-trivial and particularly challenging on curvilinear grids. Alternatively, subgrid-scale models that do not rely on such damping functions, such as the WALE model [49] or the dynamic SGS model [50], can be used in future studies.

On the ground plate, where the numerical results are to be compared with experimental measurements, the grid is uniformly spaced in the streamwise direction with a spacing of 5×10^{-3} m. This grid definition is applied to avoid excessive coarseness due to stretching, ensuring that the generated gust is properly resolved.

Outside the comparison region, the grid is stretched with a stretching factor of 1.04 in the downstream direction extending to the far right boundary of the computational domain. In the spanwise direction, the same grid stretching factor is applied without any imposed maximum grid spacing constraints. To maintain the stability of the explicit time integration scheme and to appropriately resolve the turbulent flow in time, a time step size $\Delta t = 2 \times 10^{-5}$ s is chosen, ensuring a CFL number of less than 0.4.

4.2. Distribution of Lagrangian markers describing the adaptive nozzle

The immersed boundary given by the nozzle's upper contour is defined by a set of Lagrangian markers distributed along the continuous spline defined by Eq. (35). These markers are positioned such that their spacing in each local coordinate direction is set by the minimum local Eulerian grid spacing (i.e., $\Delta x_{i,\min}$ for $i = 1, 2, 3$) encountered by the immersed boundary during the nozzle's closing and opening motion.

Within the nozzle's block (i.e., $0 \text{ m} \leq x \leq 0.5 \text{ m}$) the distribution of Lagrangian markers is defined such that one marker is assigned to each cell center. At the contour's free end (i.e., the tail of the nozzle in the range $0.5 \text{ m} \leq x \leq 0.575 \text{ m}$), a uniform distribution of Lagrangian markers with a spacing equal to the minimum Eulerian grid spacing is employed. This distribution is motivated by the fact that, during the downstroke, these Lagrangian points overlap with the region where the finest Eulerian grid spacing is defined. In this way, any potential fluid penetration through the immersed boundary surface caused by insufficient coverage of Eulerian cells by the Lagrangian markers window function is prevented. In the spanwise direction (i.e., along the z -axis), the distribution is defined such that one marker is assigned to each cell center. Consequently, the markers are set to follow the grid stretching applied along the z -axis. This decision is driven by the fact that, during

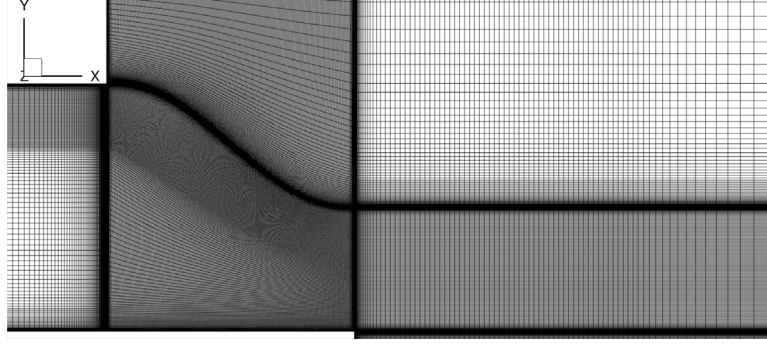


Fig. 9. Zoom of the computational grid around the nozzle's contour in the x - y plane. Only every second grid line in each direction is shown.

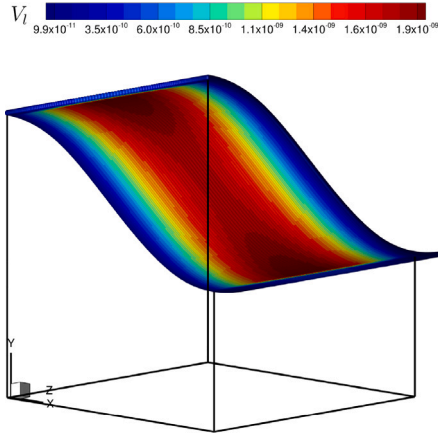


Fig. 10. Volumes associated with the non-uniformly distributed Lagrangian markers.

the nozzle's closing and opening motion, the distribution of Eulerian points along the z -axis remains fixed. A similar approach was used by Boulbrachene and Breuer [11], resulting in a significant reduction in the number of Lagrangian markers compared to a uniform distribution without compromising the accuracy of the results. The fabricated metal sheet used as the nozzle's upper contour in the wind tunnel has a thickness of 2×10^{-3} m. This is accounted for numerically by assigning the same thickness to the immersed boundary. A uniform Lagrangian grid spacing equal to the minimum Eulerian cell spacing of 2.5×10^{-4} m is set in the direction normal to the IB surface. As a result, 8 layers of Lagrangian markers are defined along this direction.

Based on the description above, the number of Lagrangian markers in each IB local direction is $N_1 = 892$, $N_2 = 160$ and $N_3 = 8$, i.e., $N_L = N_1 N_2 N_3 = 1,141,760$. Such a non-uniform distribution of Lagrangian markers within a non-uniform curvilinear Eulerian grid is efficient, as it places markers more densely only where necessary. Fewer markers help to reduce the additional computational load associated with the interpolation, spreading and tracking processes. It is also worth mentioning that due to the non-uniform distribution of Lagrangian markers, the Lagrangian volumes are also spatially non-uniform as depicted in Fig. 10.

4.3. Definition of the rotational velocity of the adaptive nozzle

While the saddle of the artificial wind gust generator follows a prescribed vertical translational motion, the adaptive nozzle undergoes a rotational motion as depicted in Fig. 11. This conversion from translational to rotational motion is achieved through an eye-bearing rod of length $b = 0.15$ m. Since the velocity of each Lagrangian marker is required to compute the Lagrangian volume force in Eq. (12),

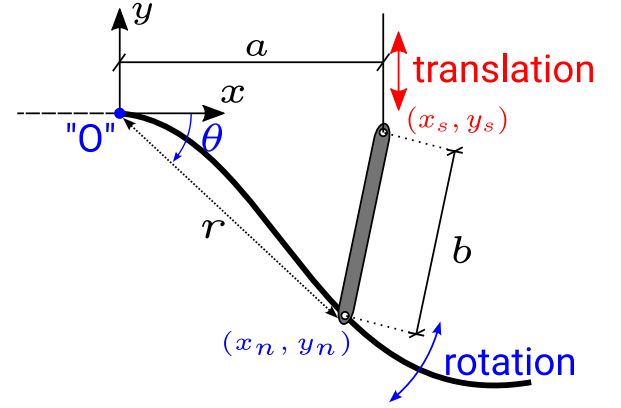


Fig. 11. Schematic sketch of the nozzle's upper contour connected to the saddle via an eye-bearing rod.

it is essential to deduce the corresponding rotational velocity of the adaptive nozzle. Based on the pure rotational motion around the origin of the coordinate system (point "O" in Fig. 11), the velocity components of the Lagrangian markers can be determined.

A detailed description of the transformation is provided in Appendix C. The following two geometric conditions hold throughout the nozzle's motion:

$$b^2 = (x_s - x_n)^2 + (y_s - y_n)^2, \quad (37)$$

$$r^2 = x_n^2 + y_n^2, \quad \text{where } r = 0.336 \text{ m}. \quad (38)$$

By substituting $x_s = a = 0.295$ m, $x_n = r \cos \theta$ and $y_n = r \sin \theta$ in Eq. (37), one can find the relation between the coordinate y_s and the angle of rotation θ :

$$y_s = r \sin \theta + \sqrt{b^2 - (a - r \cos \theta)^2} \quad (39)$$

Differentiating Eq. (39) with respect to time and solving for ω_z yields:

$$\omega_z = \frac{v_s}{x_n - \frac{(a - x_n)y_n}{\sqrt{b^2 - (a - x_n)^2}}}. \quad (40)$$

The unknown coordinates of the point on the nozzle (i.e., $x_n(t)$ and $y_n(t)$) are determined by expanding Eq. (37) and replacing $x_n^2 + y_n^2$ by r^2 according to Eq. (38), leading to the following system of equations to be solved:

$$\begin{aligned} x_n a + y_n y_s &= k, \quad \text{where: } k = \frac{r^2 + a^2 + y_s^2 - b^2}{2} \\ x_n^2 + y_n^2 &= r^2, \end{aligned}$$

whose solutions are provided in Appendix C.

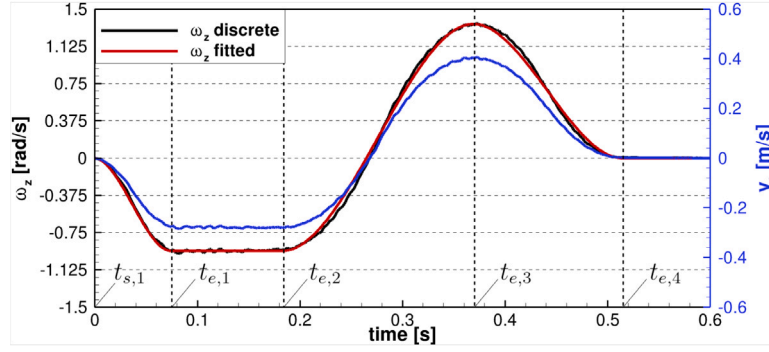


Fig. 12. Kinematic characteristics of the nozzle's rotational velocity and the saddle's vertical velocity.

The time-dependent vertical displacement $y_s(t)$ defined in the experiment by the saddle's vertical translational motion is used to evaluate the coordinates of the point on the nozzle (x_n, y_n) . These coordinates, along with the vertical velocity curve of the saddle (i.e., $v_s(t)$), are used to determine a discrete $\omega_z(t)$ curve as depicted in Fig. 12.

A smooth (differentiable) analytical function is fitted to the discrete one as shown in Fig. 12. The fitted rotational velocity function found reads:

$$\omega_{z,\text{fit}}(t) = \begin{cases} -A_1 (3\phi_1^2 - 2\phi_1^3), & \text{if } t_{s,1} = 0 \leq t < t_{e,1} = 0.075 \text{ s} \\ -A_1, & \text{if } t_{s,2} = t_{e,1} \leq t < t_{e,2} = 0.184 \text{ s} \\ A_2 (3\phi_3^2 - 2\phi_3^3) - A_1, & \text{if } t_{s,3} = t_{e,2} \leq t < t_{e,3} = 0.37 \text{ s} \\ (A_2 - A_1) - (A_2 - A_1) (3\phi_4^2 - 2\phi_4^3), & \text{if } t_{s,4} = t_{e,3} \leq t < t_{e,4} = 0.515 \text{ s} \\ 0, & \text{otherwise,} \end{cases} \quad (41)$$

where $\phi_i = (t - t_{\text{mid},i}) / t_{w,i}$, $t_{\text{mid},i} = (t_{e,i} + t_{s,i}) / 2$, $t_{w,i} = t_{e,i} - t_{s,i}$, $A_1 = 0.934$, and $A_2 = 2.284$.

The translational velocities of each Lagrangian force point are then evaluated by:

$$\mathbf{U}_i^d(\mathbf{X}_i) = \boldsymbol{\omega}_{\text{fit}} \times (\mathbf{X}_i - \mathbf{x}_c), \quad (42)$$

where $\mathbf{x}_c$ is the center of rotation of the immersed boundary and $\boldsymbol{\omega}_{\text{fit}} = [0, 0, \omega_{z,\text{fit}}]$. As visible in Fig. 12, the entire gust generation time is presently set to $\Delta t_{\text{gust}} = 0.5$ s. This characteristic time scale is also used in Section 5 to normalize the time axis.

As shown in Wood and Breuer [13], the presently applied motion pattern of the saddle and the resulting rotation of the upper contour are set to resemble the so-called extreme coherent gust (ECG) with a $(1 - \cos)$ -type signal as defined in the IEC-Standard [51].

4.4. Definition of the inlet velocity variation

In Wood and Breuer [13], an undershooting of the streamwise velocity below the free-stream velocity was observed in the wake of the generated gust. A similar behavior was observed in the first-generation gust generator [9], though it was much more pronounced and suffered from a significantly longer restitution phase. In both gust generators, the authors attributed this phenomenon to pressure losses caused by the sudden partial closure of the nozzle's outlet. This effect combined with the fact that the blower is not operated in a mode that compensates for these increased pressure losses, results in less air being blown through the wind tunnel leading to an undershoot below the free-stream velocity. This was further confirmed in the experiments by placing a Prandtl probe inside the main tunnel element (see Fig. 1) during the gust generation and calculating the dynamic pressure as the difference between the measured total and static pressures. Based on this dynamic pressure, the reduction of the inlet velocity was determined as shown in Fig. 13(a). However, the amplitudes of the velocity drop applied in

the simulation are not directly based on the measured velocity drop, as the relevant parts of the geometry (i.e., the main tunnel element, the diffuser, and the settling chamber) are not included in the simulation. Instead, they are determined numerically by analyzing the difference between the experimental and simulated streamwise velocity signal at the first monitoring point $\Delta u_{\text{mon},1} = u_{\text{mon},1}^{\text{num}} - u_{\text{mon},1}^{\text{exp}}$ for the case that the preliminary simulation is carried out with a constant inlet velocity at the inlet patch (i.e., $u_{\text{inlet}} = u_{\text{inlet}}^{\text{steady}} = 2.53$ m/s). The continuity equation applied to the velocity difference is used to determine the transient inlet velocity:

$$\begin{aligned} \rho \Delta u_{\text{inlet}}(t) A_{\text{inlet}} &= \rho \Delta u_{\text{mon},1} A_{\text{noz,out}}(t), \\ \Delta u_{\text{inlet}}(t) &= \Delta u_{\text{mon},1} \frac{A_{\text{noz,out}}(t)}{A_{\text{inlet}}} = \Delta u_{\text{mon},1} \frac{A_{\text{outlet}}(1 - BR)}{A_{\text{inlet}}}, \\ u_{\text{inlet}}^{\text{trans}}(t) &= u_{\text{inlet}}^{\text{steady}} - \Delta u_{\text{mon},1} \frac{(1 - BR)}{2}. \end{aligned} \quad (43)$$

Since the motion of the nozzle's upper contour is prescribed, the blocking ratio can be determined as a function of time, given by $BR(t) = (A_{\text{outlet}} - A_{\text{noz,out}}(t)) / A_{\text{outlet}}$. The inlet velocity function found is depicted in Fig. 13(b) and reads:

$$u_{\text{inlet}}^{\text{trans}}(t) = \begin{cases} u_{\text{inlet}}^{\text{steady}}, & \text{if } 0 \leq t < t_{s,1}^{\text{in}} = 0.168 \text{ s} \\ u_{\text{inlet}}^{\text{steady}} - A_3 (3\phi_1^2 - 2\phi_1^3), & \text{if } t_{s,1}^{\text{in}} \leq t < t_{e,1}^{\text{in}} = 0.221 \text{ s} \\ u_{\text{inlet}}^{\text{steady}} - A_3, & \text{if } t_{e,1}^{\text{in}} \leq t < t_{e,2}^{\text{in}} = 0.243 \text{ s} \\ u_{\text{inlet}}^{\text{steady}} - A_3 - A_4 (3\phi_3^2 - 2\phi_3^3), & \text{if } t_{s,3}^{\text{in}} = t_{e,2}^{\text{in}} \leq t < t_{e,3}^{\text{in}} = 0.319 \text{ s} \\ (u_{\text{inlet}}^{\text{steady}} - A_3 - A_4) + (A_3 + A_4) (3\phi_4^2 - 2\phi_4^3), & \text{if } t_{s,4}^{\text{in}} = t_{e,3}^{\text{in}} \leq t < t_{e,4}^{\text{in}} = 0.57 \text{ s} \\ u_{\text{inlet}}^{\text{steady}}, & \text{otherwise,} \end{cases} \quad (44)$$

where ϕ_i is defined in a similar manner as in Eq. (41) with $A_3 = 0.073$, and $A_4 = 0.233$. Interestingly, the trend of the numerically obtained transient inlet velocity function closely matches that measured by the Prandtl probe. However, a direct comparison of the amplitudes between the prescribed inlet velocity function and the measured data is not meaningful, as the latter is obtained within the main tunnel element.

5. Results

5.1. Instantaneous flow field induced by the wind gust generator

The initial analysis focuses on demonstrating the time-dependent flow characteristics of the gust induced by the rotational motion of the nozzle's upper contour. Fig. 14 annotates five distinct time instants, at which the streamwise velocity component and the pressure fields in the x - y symmetry plane are illustrated in Fig. 15. The monitoring points, labeled 1 to 11, are those defined in Fig. 2 and denote specific positions within the wind tunnel's test section selected for comparison with

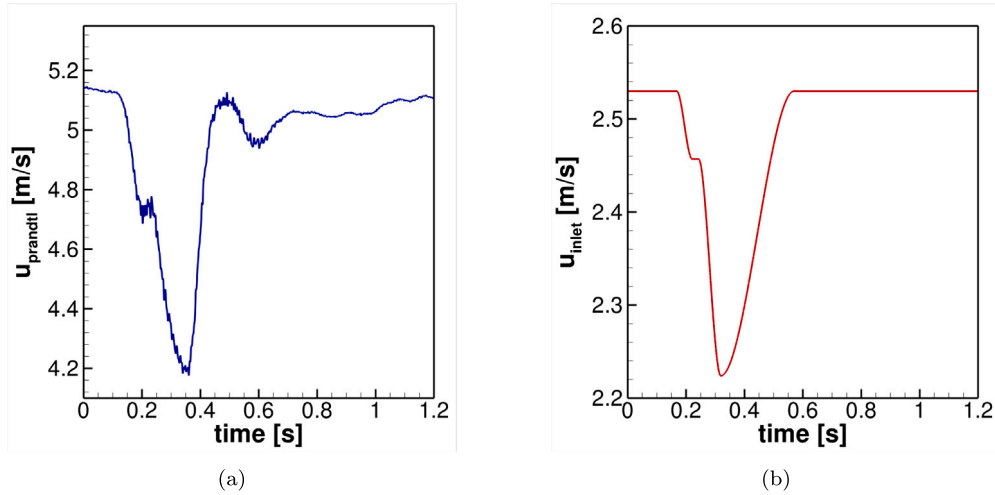


Fig. 13. Time-dependent inlet boundary condition: (a) Measured inlet velocity based on Prandtl probe [13] and (b) the prescribed transient inlet velocity applied in the simulation.

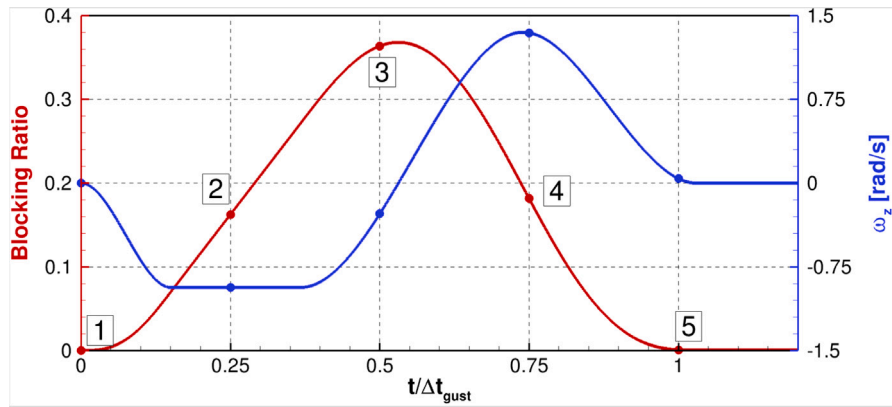


Fig. 14. Nozzle's blocking ratio annotated with five characteristic time instants at which the flow is visualized in Fig. 15; rotational velocity of the upper contour during the downward and upward strokes.

the experimental data by Wood and Breuer [13,28]. These positions correspond to the typical area, in which a wind tunnel model is placed.

At instant 1 (i.e., at $t/\Delta t_{\text{gust}} = 0$), the nozzle's upper contour is in its idle position, and the flow fields define the initial conditions for the numerical gust generator. Prior to this instant, the flow is allowed to fully develop while the immersed boundary remains stationary, ensuring a stable reference condition before any dynamic motion begins. Two distinct flow phenomena can be observed at this stage. First, boundary layers form along the top and bottom walls of the nozzle, with the latter continuing to develop along the ground plate. Second, a free shear layer emerges, separating the high-velocity core flow from the surrounding slower-moving fluid. Obviously, this detached shear layer results from the boundary layer separating from the nozzle's upper contour slightly before its end.

At instant 2 (i.e., at $t/\Delta t_{\text{gust}} = 0.25$), the upper contour of the nozzle moves downward from its idle position, partially obstructing approximately 16 % of the nozzle's original outlet area. This reduction in area accelerates the streamwise velocity above the ground plate. However, unlike the flow dynamics generated by the paddle gust generator, the flow exiting the nozzle maintains its original trajectory pointing in the main flow direction, highlighting a key advantage of the new design. Additionally, the clockwise rotation of the upper contour results in an increase in the static pressure within the nozzle, as the

contour pushes against the flow direction, causing a pressure buildup upstream. A small counter-clockwise rotating vortex forms near the tip of the nozzle's upper contour due to the local flow separation caused by the sudden downward movement.

At instant 3 (i.e., at $t/\Delta t_{\text{gust}} = 0.5$), the upper contour's downstroke is nearly complete, reaching a blocking ratio of 36 %. The recorded high streamwise velocity values along the ground plate characterize the strong horizontal wind gust generated. The initially thick free shear layer observed near the tip of the upper contour at instant 1 has evolved into a thin sharp shear layer, adjusting its altitude dynamically in response to the contour's motion. Moreover, higher pressure values are recorded inside the nozzle compared to instant 2, resulting from a stronger negative pressure gradient developing near the nozzle's outlet and further accelerating the flow beneath the nozzle's upper contour. The vortical structure, formed by the interaction between the moving contour's tip and the flow, has grown in size and been convected further downstream.

Instant 4 (i.e., $t/\Delta t_{\text{gust}} = 0.75$) captures a moment during the upward return of the contour toward its idle position. In this phase, the strength of the generated gust decreases as the flow expands within the outlet. Simultaneously, the free shear layer destabilizes, shedding small vortices from the upper contour's tip, resulting in a streamwise velocity field with strong fluctuations. Moreover, the free shear layer tilts and

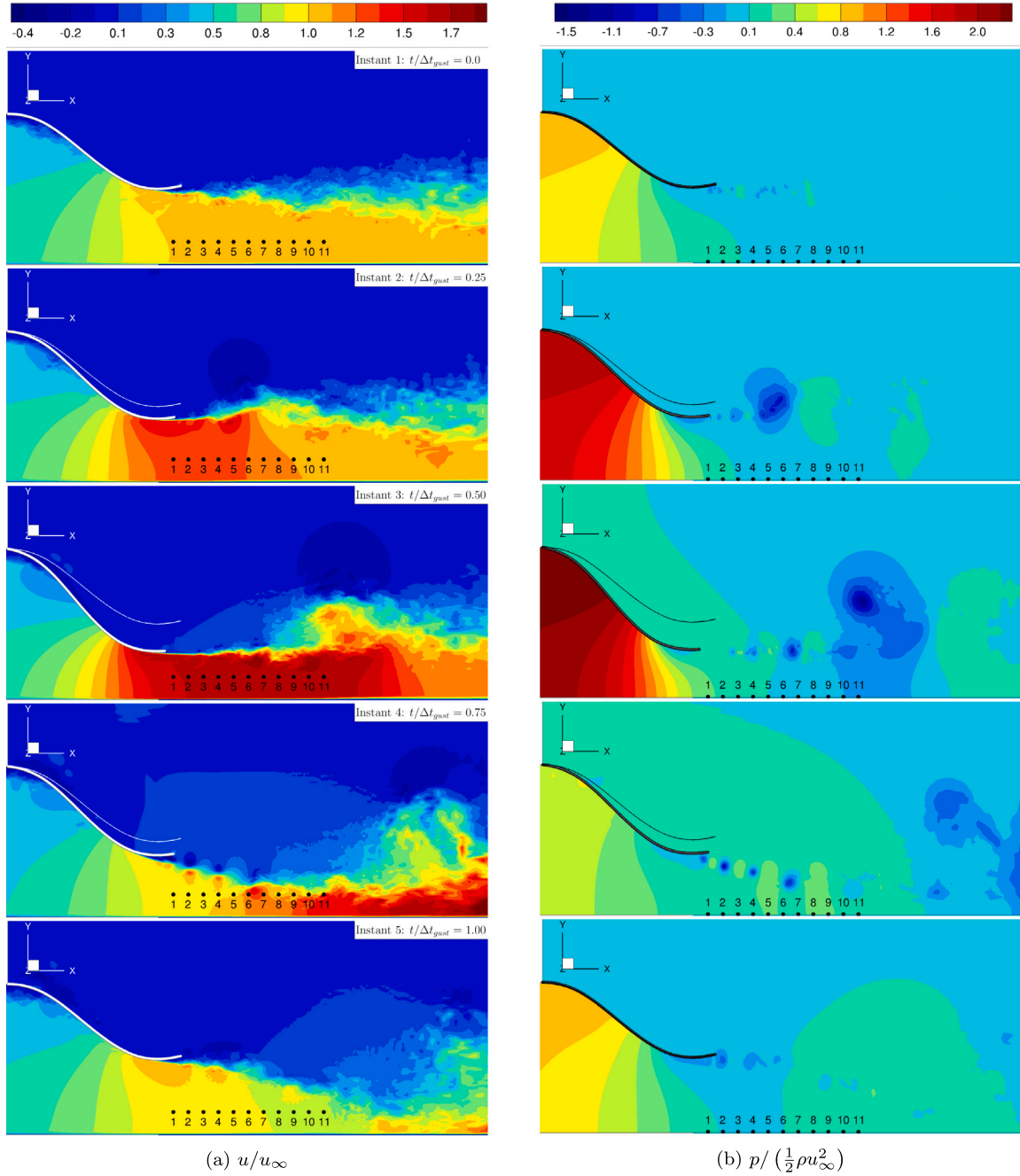


Fig. 15. Snapshots of the streamwise velocity (a) and the pressure (b) fields during the nozzle's downward and upward strokes. Instants correspond to those depicted in Fig. 14. For the instants 2 to 4, a line indicating the upper contour's idle position is depicted for reference.

no longer remains horizontal, as it fails to adjust quickly enough to the rapid upward movement of the nozzle's contour. The lowest pressure values inside the nozzle, compared to all previous instants, are recorded. This is attributed to the counter-clockwise rotation of the nozzle's contour, which moves in favor of the main flow direction, reducing the flow redirection and consequently lowering the pressure.

At instant 5 (i.e., at $t/\Delta t_{\text{gust}} = 1$), the nozzle's upper contour has returned to its idle position, marking the end of the dynamic motion. The flow gradually recovers, returning to its initial state.

5.2. Comparison of numerical and experimental results

The proposed numerical framework is validated through a direct comparison of the numerical predictions with the experimental

measurements of Wood and Breuer [13]. In the experiments, phase-averaging over 46 gusts was performed on the measured quantities to capture the characteristic development of the gust. This was crucial for two primary reasons. First, a free shear layer exists at the top surface of the jet exiting the nozzle, where small vortical structures are irregularly generated and convected downstream. This makes the flow generated by the upper contour's motion dependent on the flow state at the onset of movement. Second, phase averaging is required due to the presence of a streamwise turbulence intensity of about 1.5 % measured at the nozzle's outlet of the Eiffel-type wind tunnel. Although no inflow turbulence is considered in the simulation for simplicity, the free shear layer and its subsequent effects are present in the simulations as shown at instant 1 in Fig. 15. Therefore, simulation results are phase-averaged in the same manner as in the experiment to enable a

consistent comparison. The phase-averaged numerical results are based on 46 initial condition states separated by an interval of 0.2 s. The choice is motivated by a detailed examination of the flow variables at the shear layer, which revealed that for the chosen time interval the flow states exhibit statistical independence.

5.2.1. Pointwise and field velocity

The comparison is based on the time histories of the velocity magnitude at the aforementioned 11 streamwise positions shown in Fig. 15(a) falling within the range $0 \leq x/l \leq 1.0$. The phase-averaged CTA measurements of the velocity magnitude and their numerical counterpart are depicted in Fig. 16.

The time histories at individual monitoring points generally exhibit a bell-shaped pattern, reflecting the evolution of the nozzle's blocking ratio. In accordance with the principle of mass conservation, the flow beneath the upper contour accelerates as the nozzle's outlet contracts further, reaching its peak velocity at the maximum blocking ratio. This phenomenon can also be clearly seen in Fig. 17, which shows a comparison between the measured and predicted phase-averaged streamwise velocity contours in the symmetry plane at different instants of the gust generation. Both experiment and simulation depict the highest streamwise velocity at $t/\Delta t_{gust} = 0.5$. As the nozzle's contour returns to its idle position, the streamwise velocity decreases accordingly. The subsequent drop below the free-stream velocity (see Fig. 16, Fig. 17(f) and Fig. 17(g)) is at least partially attributed to the prescribed transient inlet velocity defined in Eq. (44), which accounts for the reduced airflow through the wind tunnel's blower. This velocity undershoot becomes more pronounced as the gust propagates downstream.

The velocity magnitude curves for the monitoring points within $0 \leq x/l \leq 0.2$ (i.e., Figs. 16(a) to 16(c)) closely align with those observed in the experimental data. However, for the monitoring points within $0.3 \leq x/l \leq 1$ (i.e., Figs. 16(d) to 16(k)), the descending slope of these curves includes an additional peak, becoming more pronounced as one moves downstream. This fluctuation stems from the flow separation near the tip of the nozzle's upper contour, triggered by its rapid upward motion. This separation generates a series of small vortices in the detached, inclined shear layer, which destabilize and ultimately disrupt the flow at the downstream monitoring points, as illustrated at instant 4 in Fig. 15(a). This series of small vortices can even be found in the phase-averaged numerical data in Figs 17(e) and 17(f). While this additional peak is absent in the phase-averaged experimental data, the instantaneous data of Wood and Breuer [13,28] (not shown here) reveal similar velocity oscillations to those predicted by LES. The persistence of this velocity peak in the phase-averaged numerical results is attributed to the absence of inflow turbulence. When turbulence intensity is accounted for, flow separation occurs at varying times during the nozzle's upstroke. Consequently, vortex shedding becomes desynchronized across different gust realizations, causing the peak to be averaged out. That explains the differences in the shear layer observed between the measurements and the predictions.

A notable feature in the velocity signals is the presence of two distinct local maxima of the velocity. In addition to the maximum velocity observed during the gust generation period at $t/\Delta t_{gust} \approx 0.5$, another maximum emerges after the gust has formed (i.e., at $t/\Delta t_{gust} > 1$), particularly at the monitoring points 5 to 9. Obviously, a second, smaller local velocity maximum is formed in the far wake of the generated gust. This overshoot, much like the preceding undershoot, becomes more pronounced further downstream. Overall, this quantitative comparison demonstrates that the simulation methodology suggested effectively captures the gust generation process in the wind tunnel. With the exception of the free shear layer, the predicted velocity distributions in Fig. 17 are in close agreement with the PIV measurements by Wood and Breuer [13].

Note that Fig. D.24 in Appendix D also depicts a comparison between the measured and predicted phase-averaged vertical velocity contours at different instants of the gust generation process. Overall,

the agreement is again good, but the deviations in the free shear layer are also present there, as expected.

5.2.2. Surface pressure at monitoring points

For a more detailed analysis of the numerical predictions, a surface pressure based comparison is carried out. The time history of the predicted phase-averaged pressure coefficient (c_p) = $2 \langle p \rangle / (\rho u_\infty^2)$ and its measured counterpart at the 11 streamwise monitoring points shown in Fig. 15(b) is depicted in Fig. 18. It is worth mentioning that the predicted phase-averaged pressure coefficients are uniformly shifted upward by a constant factor of 0.0383. This shift is applied because, for the simulation of incompressible fluids, a pressure reference point must be defined, anchoring the entire pressure field. As the simulation progresses over a long period of time, the flow not exiting the computational domain through the outlet eventually reaches the pressure reference point. Consequently, the pressure at the monitoring point decreases, necessitating the upward shift to align the numerical predictions with the experimental measurements.

For all monitoring points, the pressure coefficient is initially greater than zero, with its value decreasing downstream. This trend is clearly depicted in Fig. 18(l) showing the recorded pressure coefficients at the monitoring points for the case without gust. The higher values of the pressure coefficient at the upstream points are attributed to the elevated static pressure inside the nozzle, which exceeds the atmospheric pressure p_∞ (i.e., here the pressure at the pressure reference point). This negative pressure gradient is also visible at $t/\Delta t_{gust} = 0$ in Fig. 19, showing the phase-averaged pressure coefficient within the domain of interest.

The temporal evolution of $\langle c_p \rangle$ follows a similar pattern across all monitoring points. During the initial phase of the nozzle's downward stroke, the surface pressure increases due to the compression effect caused by the contraction of the nozzle's outlet as shown at $t/\Delta t_{gust} = 0.1$ in Fig. 19. This pressure rise is observed at all monitoring points, though its magnitude decreases with increasing downstream distance.

Following the initial rise, the pressure coefficient undergoes a sharp drop, reaching negative values for $x/l > 0.4$ (see Figs. 18(f) to 18(k)). The contour plots at $0.2 \leq t/\Delta t_{gust} \leq 0.5$ in Fig. 19 clearly indicate that this drop is caused by the large vortical structure forming at the tip of the nozzle's upper contour due to the sudden downward motion. While this vortex influences the upstream points only slightly, its impact becomes more pronounced downstream as it expands and gains intensity.

As the large vortical structure is convected downstream, positive pressure values are recovered due to the continued pressure buildup inside the nozzle as depicted at $0.3 \leq t/\Delta t_{gust} \leq 0.5$ in Fig. 19. Obviously, points near the nozzle's outlet (i.e., $0 \leq x/l \leq 0.1$) experience this pressure rise first, reaching a peak at the end of the downstroke, significantly earlier than those further downstream. For points within $0.2 \leq x/l \leq 1.0$, the majority of the pressure rise leading to a peak occurs during the upstroke phase, aligning with the velocity drop caused by the expansion of the nozzle's outlet (see $0.7 \leq t/\Delta t_{gust} \leq 0.8$ in Fig. 19). Unlike the nearly constant velocity peaks across the monitoring points, the pressure coefficient peaks increase significantly in downstream direction, eventually reaching a nearly constant maximum for $x/l > 0.7$.

Subsequently, a sharp pressure drop occurs during the upward stroke of the nozzle as the velocity begins to rise again, recovering from its undershoot.

Overall, the comparison between the numerical and the experimental surface pressure data demonstrates strong agreement in terms of both trend and magnitude. The observed discrepancies, particularly at the first two points, can be attributed to geometric inaccuracies associated with the fabricated nozzle. This surface pressure analysis reinforces the validity of the proposed numerical framework in capturing the fundamental mechanisms of the gust generation.

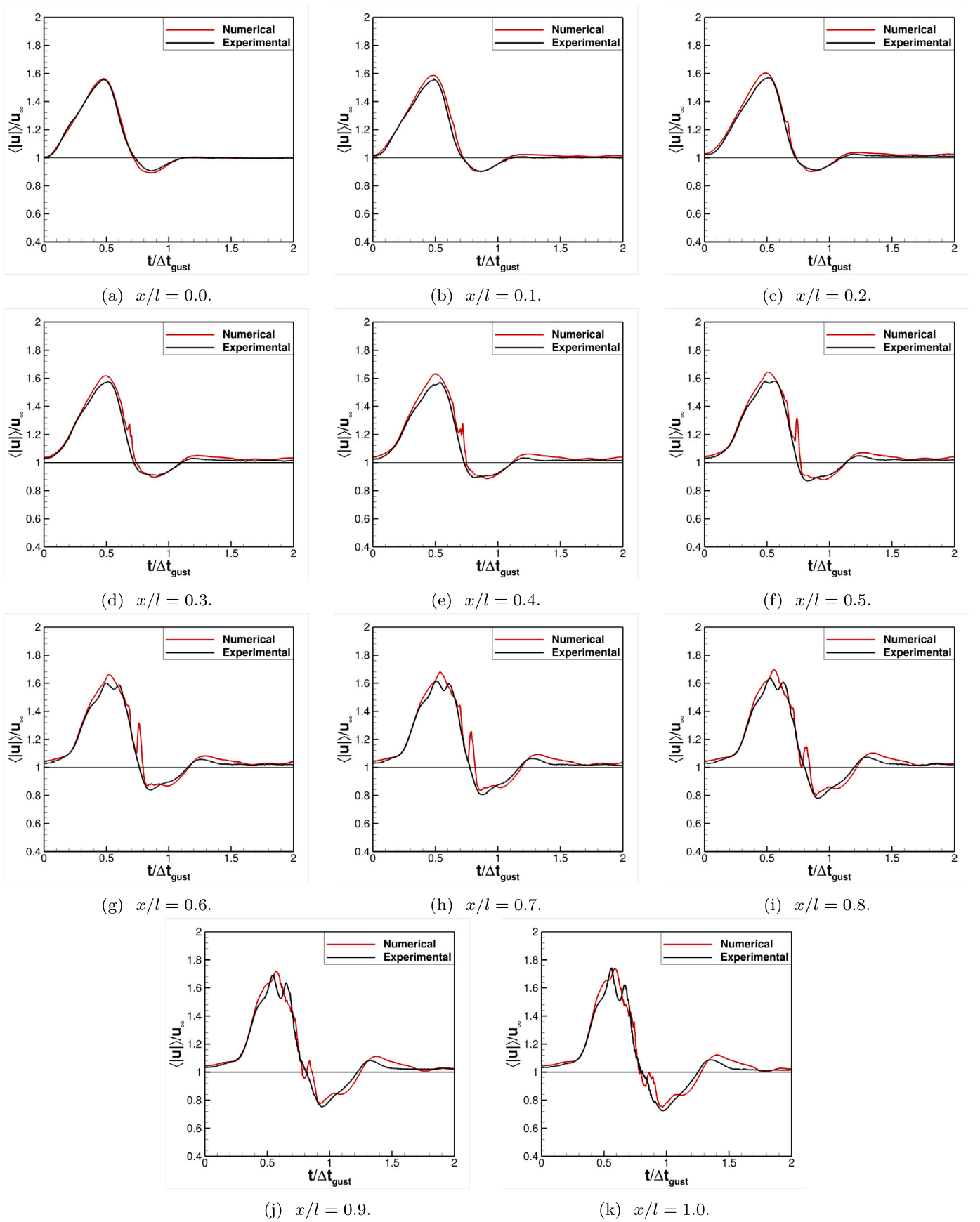


Fig. 16. Time histories of the phase-averaged experimental velocity magnitude [13,28] and its numerical counterpart at the 11 monitoring points depicted in Fig. 15(a).

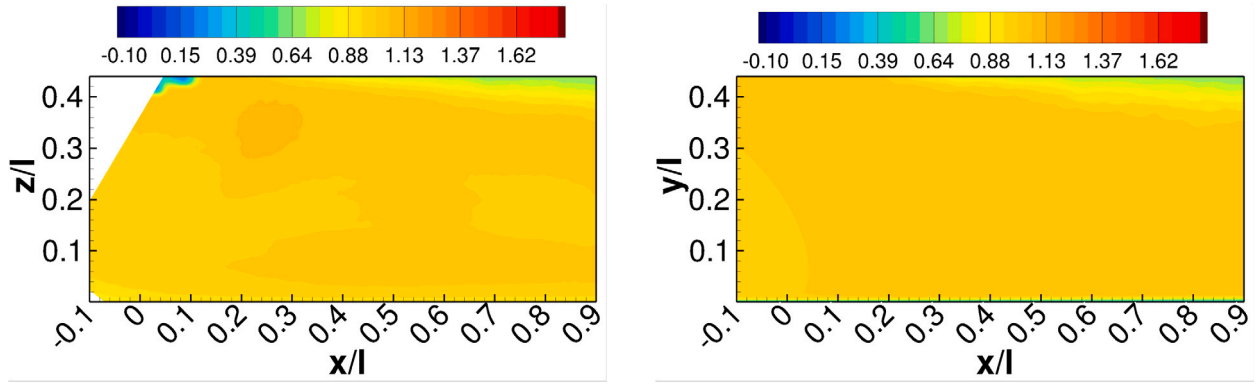
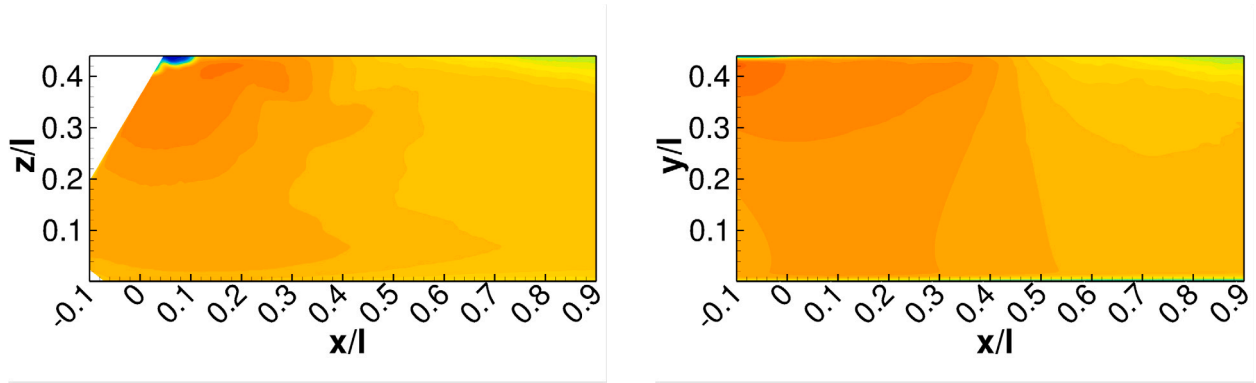
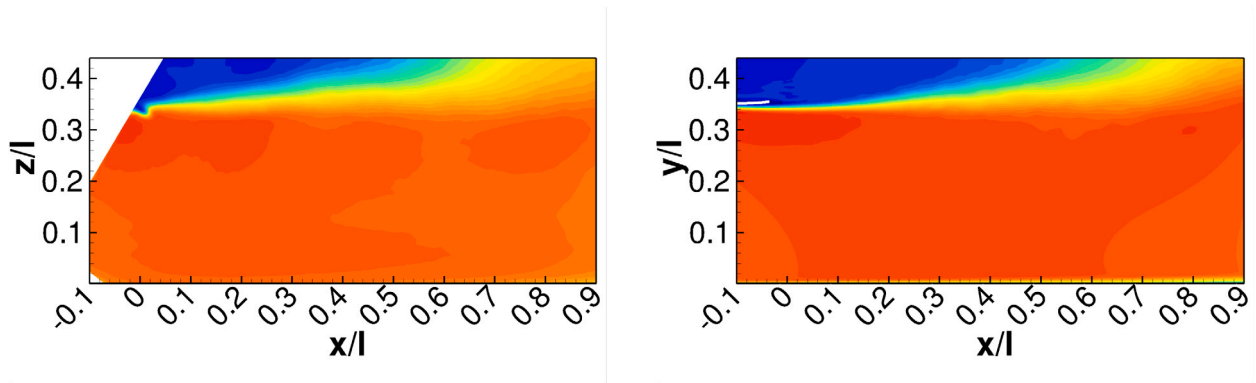
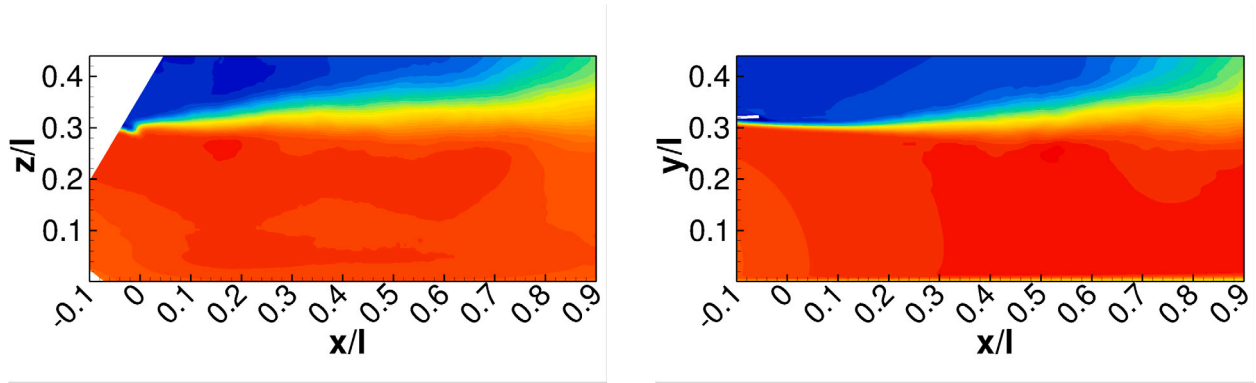
(a) $t/\Delta_{\text{gust}} = 0$: Experiment (left) vs. Simulation (right).(b) $t/\Delta_{\text{gust}} = 0.2$: Experiment (left) vs. Simulation (right).(c) $t/\Delta_{\text{gust}} = 0.4$: Experiment (left) vs. Simulation (right).(d) $t/\Delta_{\text{gust}} = 0.5$: Experiment (left) vs. Simulation (right).

Fig. 17. Comparison between measured and predicted data of the phase-averaged streamwise velocity in the symmetry plane at different time instants of the gust generation. Experimental data taken from Wood and Breuer [13].

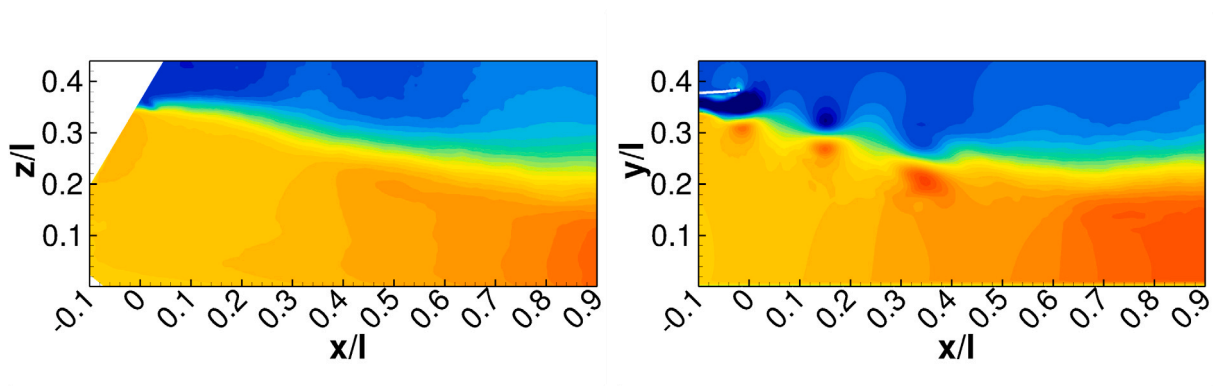
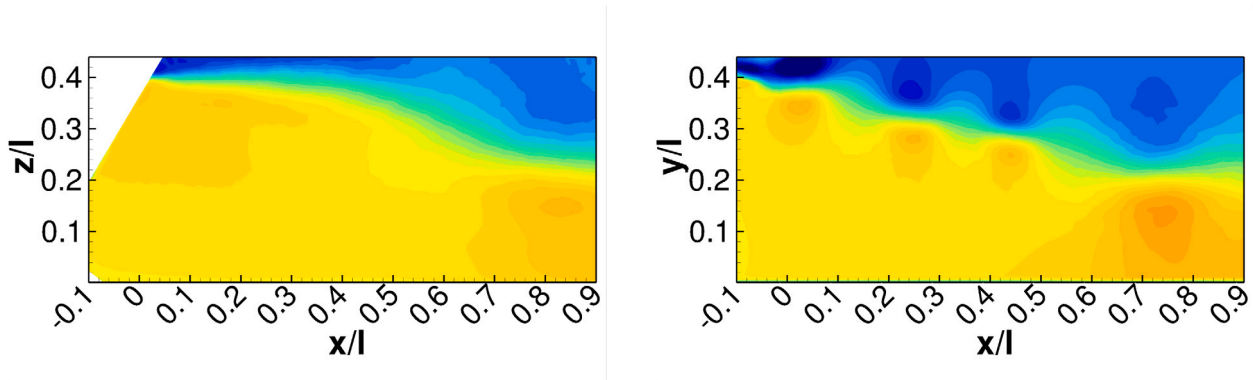
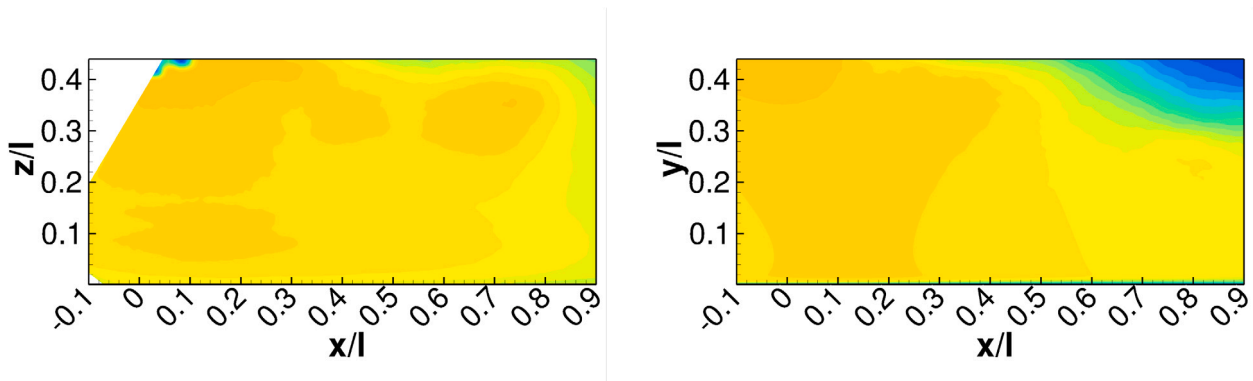
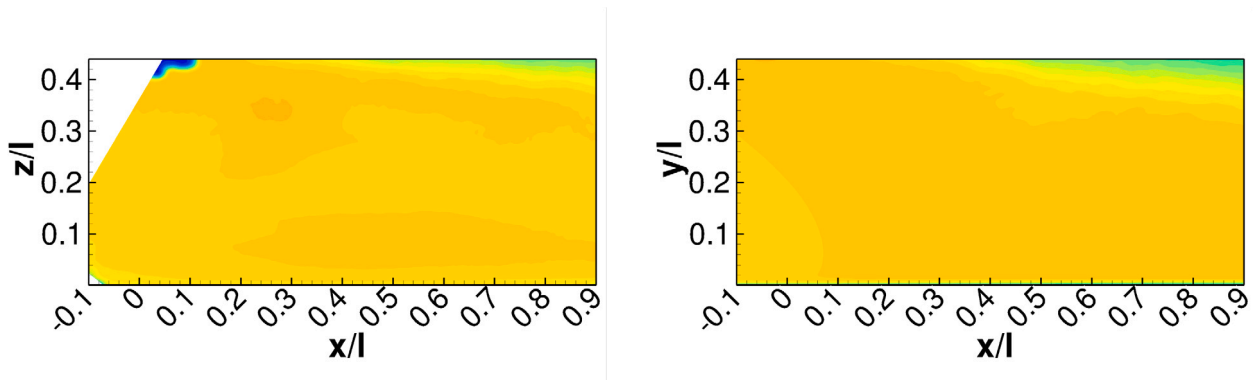
(e) $t/\Delta_{\text{gust}} = 0.7$: Experiment (left) vs. Simulation (right).(f) $t/\Delta_{\text{gust}} = 0.8$: Experiment (left) vs. Simulation (right).(g) $t/\Delta_{\text{gust}} = 1.0$: Experiment (left) vs. Simulation (right).(h) $t/\Delta_{\text{gust}} = 1.6$: Experiment (left) vs. Simulation (right).

Fig. 17. (continued).

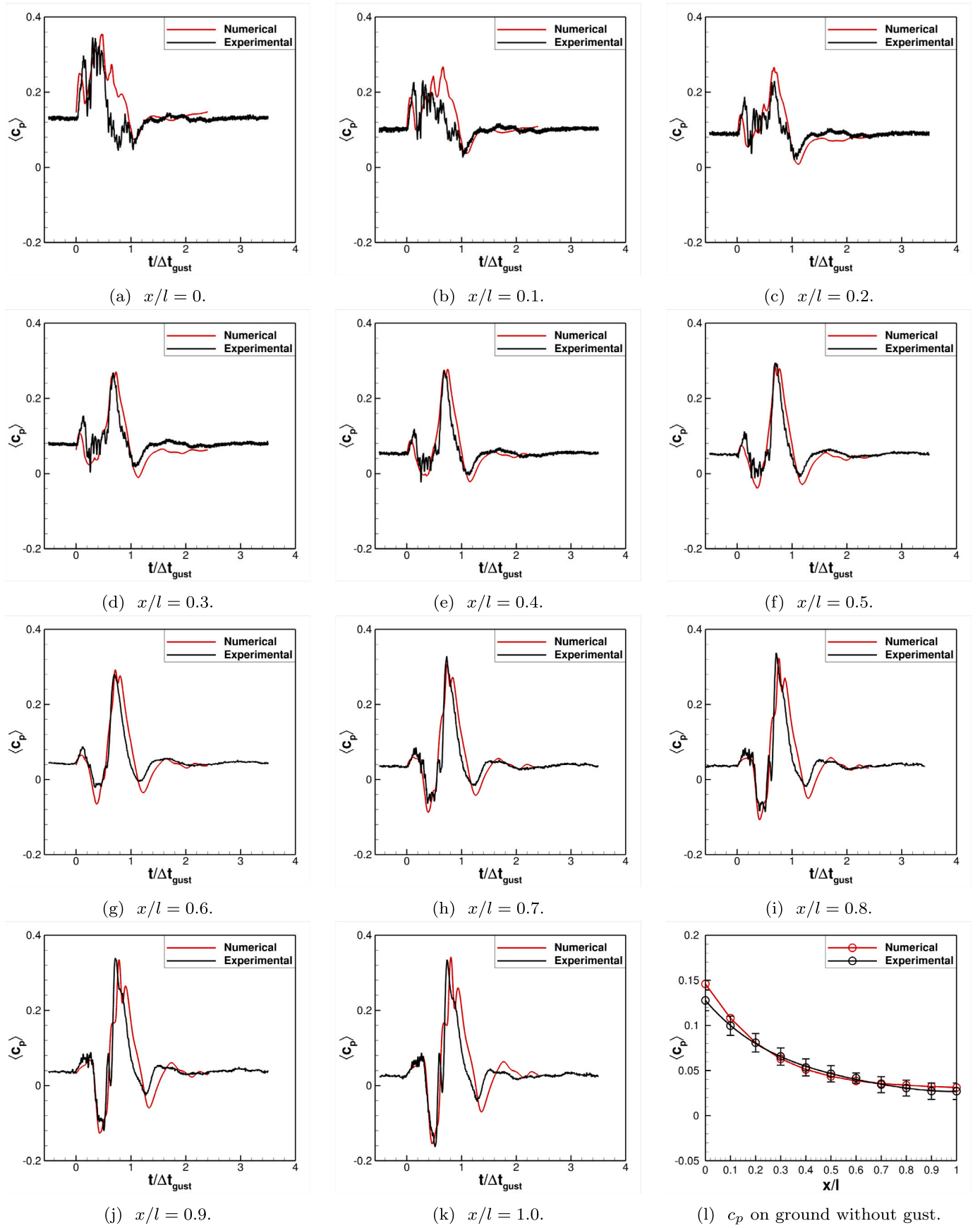


Fig. 18. Time histories of the phase-averaged experimental surface pressure coefficient [13] and its numerical counterpart at the 11 monitoring points depicted in Fig. 15(b).

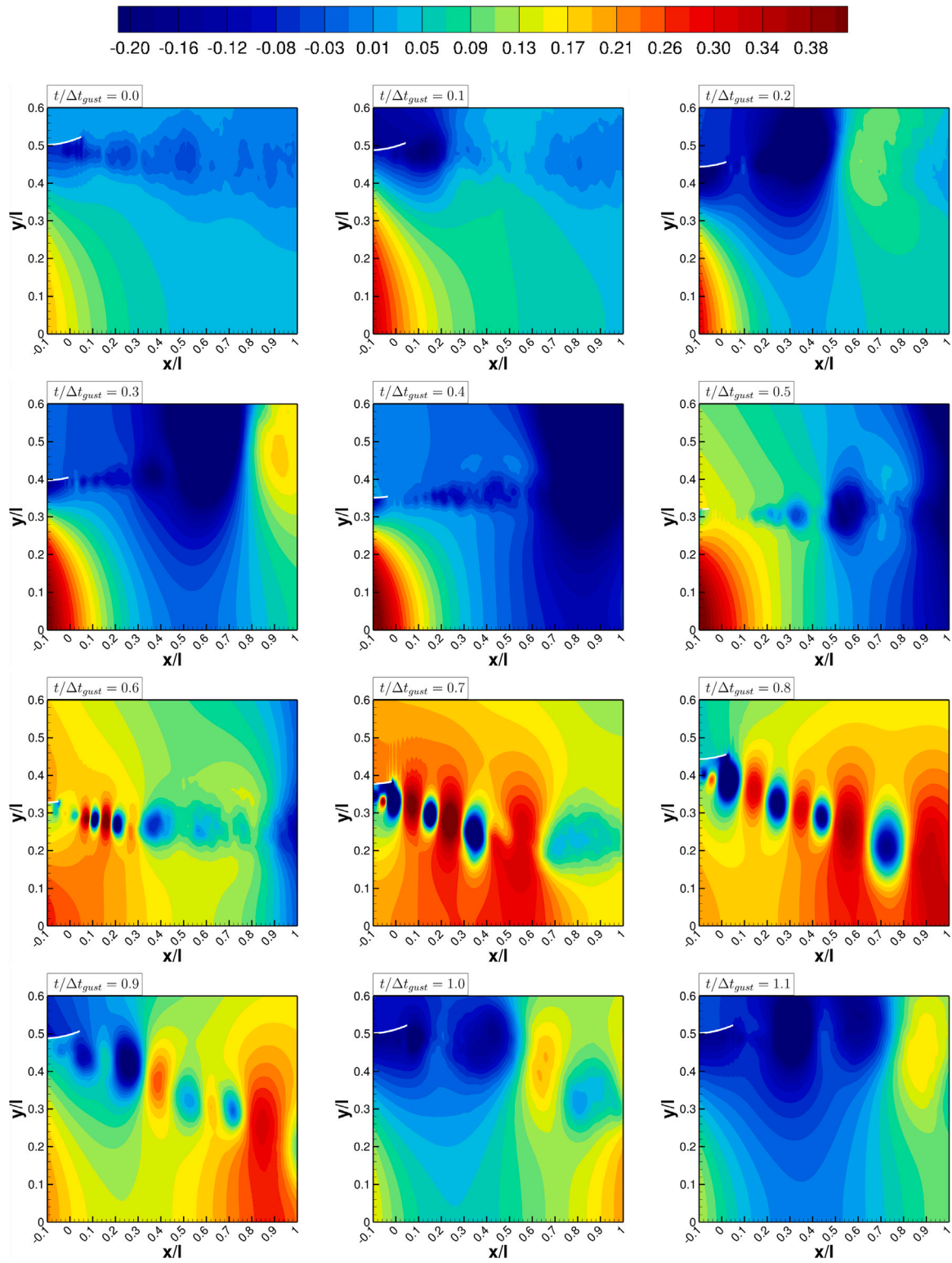


Fig. 19. Predicted phase-averaged pressure coefficient in the symmetry plane at different time instants of the gust generation.

6. Conclusions

This contribution presents a dynamic immersed boundary method designed for the application on curvilinear block-structured grids, specifically intended for high-fidelity large-eddy simulations of complex flow dynamics. Building on the direct forcing diffuse-interface IB approach of Uhlmann [17], the method employs the moving least square approach of Vanella and Balaras [22] to construct an IB kernel for the Lagrangian markers. In the present study, a new variant tailored for curvilinear block-structured grids is proposed, enabling the interpolation weights for transferring Eulerian flow variables to Lagrangian locations to be determined independently of the dilation parameter. By utilizing a fixed interpolation window, the interpolation weights are evaluated with respect to the computational space rather than the physical space. To construct one-sided kernel functions for Lagrangian points near boundaries, the MLS approach is also applied. This is particularly relevant in the present case, where the nozzle's upper contour traverses between two side walls, preventing the use of symmetric kernels.

The newly devised IB methodology is applied to numerically replicate the highly instantaneous turbulent flow field induced by a recently developed wind gust generator [13]. This generator utilizes a wind tunnel nozzle with a rotatable upper contour, enabling a rapid reduction of the nozzle's outlet area within a short time interval to produce a smooth horizontal wind gust. The motion of the nozzle's upper contour is captured by approximately 1.1 million non-uniformly distributed Lagrangian markers, which move through the flow field following the motion pattern of the experimental setup. Currently, a (1 - cos)-type signal is considered to generate extreme coherent gusts (ECG) as defined in the IEC-Standard [51].

First, the instantaneous flow field induced by the gust generator is analyzed based on the velocity and pressure fields. Unlike the measurements, the predicted data provide all flow quantities across the entire computational domain – both inside the nozzle and within the test section – at a high temporal resolution, allowing for deeper insight into the gust generation process. For a direct comparison with the measurements, the flow data are averaged over 46 gust cycles. The predictions of the phase-averaged data align well with the measurements, except for a series of small vortices in the detached shear layer at a later stage of the gust generation. These vortices appear only in the simulations, likely due to the absence of inflow turbulence, which is currently not accounted for in the predictions. In the experiments, a streamwise turbulence intensity of approximately 1.5 % was observed, which is assumed to desynchronize the flow separation during the nozzle's upstroke and the subsequent vortex formation. As a result, these vortical structures are averaged out in the phase-averaged experimental data, whereas they persist in the simulations due to the lack of inflow turbulence. Nevertheless, comparisons with the PIV and CTA data of Wood and Breuer [13,28] reveal that the proposed simulation methodology effectively captures the gust generation process in the wind tunnel. In addition to velocity field comparisons, the analysis of the surface pressure at multiple monitoring points further validated the numerical approach. The predicted phase-averaged pressure coefficients showed good agreement with the measured values in both trend and magnitude, confirming the model's capability to reproduce the unsteady pressure dynamics induced by the gust. Minor deviations observed near the nozzle's outlet were attributed to geometric inaccuracies in the physical setup. Thus, an efficient IB-based method for the application on curvilinear grids has been developed, providing a valuable tool for future studies on the effect of wind gusts on flexible structures and supporting complementary experimental investigations. The IB method proposed can also be combined with the ALE method, as recently demonstrated in [12], offering unprecedented possibilities in the simulation of FSI problems and enabling the selective use of the respective strengths of both approaches. Furthermore, extending the

present study beyond the discrete gust scenario to include periodic gust cycles or continuously modulated velocity disturbances offers a natural avenue for future investigations.

CRediT authorship contribution statement

K. Boulbrachene: Writing – review & editing, Writing – original draft, Visualization, Validation, Software, Methodology, Investigation, Formal analysis, Conceptualization. **M. Breuer:** Writing – review & editing, Writing – original draft, Supervision, Resources, Project administration, Methodology, Funding acquisition, Conceptualization.

Declaration of competing interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgments

The initial part of the project was financially supported by the German Research Foundation (Deutsche Forschungsgemeinschaft) under the contract number BR 1847/17-1, which ended already. Presently, funding computational resources (HPC cluster HSUper) have been provided by the project hpc.bw, funded by dtec.bw – Digitalization and Technology Research Center of the Bundeswehr. dtec.bw is funded by the European Union – NextGenerationEU. Last but not least we would like to thank Dr. J.N. Wood (HSU Hamburg) for the fruitful discussions on the topic.

Appendix A. Minimizing error functions

This section derives the solutions of the minimization problem for the standard MLS method and the Backus–Gilbert formulation of the MLS method.

A.1. Standard MLS

The standard MLS formulation seeks the vector of unknown coefficients $\mathbf{c}^g(\mathbf{X}_i)$ by minimizing the error function, which is expressed in a matrix form as:

$$\min_{\mathbf{c}^g(\mathbf{X}_i)} J_i = \frac{1}{2} (\mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_i) - \mathbf{g})^\top \text{diag}(\boldsymbol{\phi}_i) (\mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_i) - \mathbf{g}). \quad (\text{A.1})$$

Expanding the terms yields:

$$\begin{aligned} J_i &= \frac{1}{2} ((\mathbf{c}^g(\mathbf{X}_i))^\top \mathcal{A} - \mathbf{g}^\top) \text{diag}(\boldsymbol{\phi}_i) (\mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_i) - \mathbf{g}), \\ &= \frac{1}{2} [(\mathbf{c}^g(\mathbf{X}_i))^\top \mathcal{A} \text{diag}(\boldsymbol{\phi}_i) \mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_i) - (\mathbf{c}^g(\mathbf{X}_i))^\top \mathcal{A} \text{diag}(\boldsymbol{\phi}_i) \mathbf{g} \\ &\quad - \mathbf{g}^\top \text{diag}(\boldsymbol{\phi}_i) \mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_i) + \mathbf{g}^\top \text{diag}(\boldsymbol{\phi}_i) \mathbf{g}]. \end{aligned} \quad (\text{A.2})$$

Rewriting the third term in Eq. (A.2) leads to:

$$\begin{aligned} \mathbf{g}^\top \text{diag}(\boldsymbol{\phi}_i) \mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_i) &= [(\mathbf{c}^g(\mathbf{X}_i))^\top (\mathbf{g}^\top \text{diag}(\boldsymbol{\phi}_i) \mathcal{A}^\top)^\top]^\top \\ &= (\mathbf{c}^g(\mathbf{X}_i))^\top (\mathbf{g}^\top \text{diag}(\boldsymbol{\phi}_i) \mathcal{A}^\top)^\top \\ &= (\mathbf{c}^g(\mathbf{X}_i))^\top (\text{diag}(\boldsymbol{\phi}_i) \mathcal{A}^\top)^\top \mathbf{g} \\ &= (\mathbf{c}^g(\mathbf{X}_i))^\top \mathcal{A} \text{diag}(\boldsymbol{\phi}_i) \mathbf{g}. \end{aligned} \quad (\text{A.3})$$

Hence Eq. (A.2) simplifies to:

$$J_i = \frac{1}{2} [(\mathbf{c}^g(\mathbf{X}_i))^\top \mathcal{A} \text{diag}(\boldsymbol{\phi}_i) \mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_i) - 2(\mathbf{c}^g(\mathbf{X}_i))^\top \mathcal{A} \text{diag}(\boldsymbol{\phi}_i) \mathbf{g} + \mathbf{g}^\top \text{diag}(\boldsymbol{\phi}_i) \mathbf{g}]. \quad (\text{A.4})$$

Differentiating J_i with respect to the vector of unknown coefficients $\mathbf{c}^g(\mathbf{X}_i)$ yields:

$$\frac{\partial J_i}{\partial \mathbf{c}^g(\mathbf{X}_i)} = \mathcal{A} \text{diag}(\boldsymbol{\phi}_i) \mathcal{A}^\top \mathbf{c}^g(\mathbf{X}_i) - \mathcal{A} \text{diag}(\boldsymbol{\phi}_i) \mathbf{g} \quad (\text{A.5})$$

Setting $\frac{\partial J_I}{\partial \mathbf{c}^g(\mathbf{X}_I)} = 0$ and solving $\mathbf{c}^g(\mathbf{X}_I)$ leads to:

$$\mathbf{c}^g(\mathbf{X}_I) = \mathcal{G}^{-1} B \mathbf{g}, \quad (\text{A.6})$$

where:

$$\mathcal{G} = \mathcal{A} \text{diag}(\phi_I) \mathcal{A}^T, \quad B = \mathcal{A} \text{diag}(\phi_I). \quad (\text{A.7})$$

A.2. Proof of the MLS method's satisfaction of discrete moment conditions

The proof relies on considering the Backus–Gilbert formulation of the moving least squares method [26,27], which incorporates the polynomial reproduction conditions into the definition of the quadratic minimization problem:

$$\min_{\tilde{\phi}_I} J_I = \frac{1}{2} \tilde{\phi}_I^T \text{diag}(\phi_I)^{-1} \tilde{\phi}_I,$$

$$\text{subject to: } \mathcal{A} \tilde{\phi}_I = \mathbf{p}(\mathbf{X}_I). \quad (\text{A.8})$$

By the use of the Lagrange multiplier $\lambda \in \mathbb{R}^m$, the constrained problem in Eq. (A.8) is reformulated to an unconstrained minimization as:

$$\min_{\tilde{\phi}_I, \lambda_I} \mathcal{L}_I = \frac{1}{2} \tilde{\phi}_I^T \text{diag}(\phi_I)^{-1} \tilde{\phi}_I - \lambda_I^T (\mathcal{A} \tilde{\phi}_I - \mathbf{p}(\mathbf{X}_I)). \quad (\text{A.9})$$

The solution to the problem defined in Eq. (A.8) is obtained by minimizing the Lagrangian $\mathcal{L}_I$ by imposing the conditions $\frac{\partial \mathcal{L}_I}{\partial \tilde{\phi}_I} = 0$ and $\frac{\partial \mathcal{L}_I}{\partial \lambda_I} = 0$. First, expanding Eq. (A.9) yields:

$$\min_{\tilde{\phi}_I, \lambda_I} \mathcal{L}_I = \frac{1}{2} \tilde{\phi}_I^T \text{diag}(\phi_I)^{-1} \tilde{\phi}_I - \lambda_I^T \mathcal{A} \tilde{\phi}_I + \lambda_I^T \mathbf{p}(\mathbf{X}_I). \quad (\text{A.10})$$

The second term appearing in Eq. (A.10) can be written as:

$$\lambda_I^T \mathcal{A} \tilde{\phi}_I = [\tilde{\phi}_I^T (\lambda_I^T \mathcal{A})^T]^T = \tilde{\phi}_I^T (\lambda_I^T \mathcal{A})^T = \tilde{\phi}_I^T \mathcal{A}^T \lambda_I. \quad (\text{A.11})$$

Substituting Eq. (A.11) in Eq. (A.10) and minimizing the Lagrangian with respect to the vector unknown weights (i.e., $\frac{\partial \mathcal{L}_I}{\partial \tilde{\phi}_I} = 0$) yields the first equation:

$$\text{diag}(\phi_I)^{-1} \tilde{\phi}_I - \mathcal{A}^T \lambda_I = 0. \quad (\text{A.12})$$

Similarly, the Lagrangian in Eq. (A.10) is minimized with respect to the Lagrange multiplier (i.e., $\frac{\partial \mathcal{L}_I}{\partial \lambda_I} = 0$) to deduce the second equation:

$$\mathcal{A} \tilde{\phi}_I - \mathbf{p}(\mathbf{X}_I) = 0. \quad (\text{A.13})$$

The solution to the two equations above is determined by first solving for $\tilde{\phi}_I$ in Eq. (A.12) and substituting the result into Eq. (A.13). This yields the Lagrange multiplier:

$$\lambda_I = (\mathcal{A} \text{diag}(\phi_I) \mathcal{A}^T)^{-1} \mathbf{p}(\mathbf{X}_I) = \mathcal{G}^{-1} \mathbf{p}(\mathbf{X}_I), \quad (\text{A.14})$$

where $\mathcal{G}$ is the positive-definite Gram matrix. Finally, the vector of unknown weights reads:

$$\tilde{\phi}_I = \text{diag}(\phi_I) \mathcal{A}^T \lambda_I = \text{diag}(\phi_I) \mathcal{A}^T \mathcal{G}^{-1} \mathbf{p}(\mathbf{X}_I) = B^T \mathcal{G}^{-1} \mathbf{p}(\mathbf{X}_I). \quad (\text{A.15})$$

By examining Eqs. (31) and (A.15), it becomes evident that both MLS approaches yield identical weight vectors. Due to the fact that the polynomial reproduction constraints are inherently embedded into the Backus–Gilbert formulation, the equivalence of the two methods indicates that the standard moving least squares method also reproduces the polynomial basis functions.

Appendix B. Validation

B.1. Flow past a stationary cylinder

In this test case, the accuracy of the proposed IBM is assessed based on the prediction of the flow past a stationary cylinder. This benchmark is aligned to previous experimental [52,53] and numerical studies [17, 44,54]. A circular computational domain of a diameter $30D$ is adopted, where D corresponds to the diameter of the cylinder. Assuming that the flow is two-dimensional, the domain size in z -direction is set to 1 control volume. The center of the cylinder (x_{cyl} , y_{cyl}) is located at the center of the computational domain. The domain is discretized using an O-grid with an angular spacing of $\Delta\theta = 2\pi/240 = 0.0262$. In the vicinity of the cylinder, i.e., for $0.25D < r < 0.75D$, two different resolutions with uniform radial spacings of $\Delta r = 0.0025D$ (200 points) and $\Delta r = 0.005D$ (400 points) are used. This ensures a similar spatial resolution for the subsequent oscillating cylinder case. Within the inner region (i.e., for $r < 0.25D$), the domain is described by a specific O-grid with a square block filling the center, while in the outer region (i.e., for $r > 0.75D$), the grid is stretched geometrically with a stretching factor of 1.025. A total of 240 Lagrangian markers are employed, distributed uniformly along the cylinder circumference according to the angular resolution of the Eulerian grid. The thickness of the immersed boundary is set to the radial grid spacing of each respective grid resolution. The computational domain along with a zoomed view of the grid near the cylinder are shown in Fig. B.20.

A uniform inlet velocity u_∞ is set at the front part of the domain, while a convective outflow boundary condition is used at the rear part. Due to the assumption of two-dimensionality, the application of periodic boundary conditions in z -direction is permitted. To validate the IBM framework for flow regimes dominated by either viscous or inertial effects, simulations are carried out at Reynolds numbers $Re = u_\infty D/\nu = 20, 100$, and 185 . A constant time step is used, corresponding to a CFL number of 0.2.

Fig. B.21 shows snapshots of the z -vorticity field of the flow for the different Reynolds numbers along with the corresponding temporal evolution of the drag and lift coefficients for both grid resolutions (i.e., $D/\Delta r = 400$ and $D/\Delta r = 200$) mentioned above. It is worth mentioning that the angular resolution and the number of Lagrangian markers is identical on both grids. At the lowest Re number the prediction converges to a steady-state solution, while the two higher Re numbers show the development of the classical von Kármán vortex street in the wake leading to periodic oscillations of C_L and C_D . The computed drag coefficients and Strouhal numbers obtained using the present IB formulation, as well as those from boundary-fitted simulations, are compared in Table B.1 with reference values from the literature. The IB method yields slightly higher drag coefficients than found in experimental data and boundary-fitted results. This is typical for diffuse-interface IB methods, where the fluid–solid interface is smeared over several grid cells, effectively increasing the apparent body size and, consequently, the computed drag. A similar effect explains the slightly higher drag observed for the coarse grid compared to the fine one (see Fig. B.21(b)). If the Lagrangian volume is not explicitly prescribed and the thickness of the immersed boundary is numerically computed, this would generally lead to lower drag coefficients, bringing the results closer to those obtained with boundary-fitted methods [17,47]. Nonetheless, the results obtained with both resolutions are nearly identical, confirming the grid independence of the solution. Overall, good agreement with the reference data is found. To complement the quantitative validation and to provide a direct visual assessment of the boundary condition enforcement, Fig. B.22 shows streamline plots around the stationary cylinder at the three Reynolds numbers. In all cases, the streamlines remain tangential to the surface, clearly demonstrating that the proposed immersed boundary

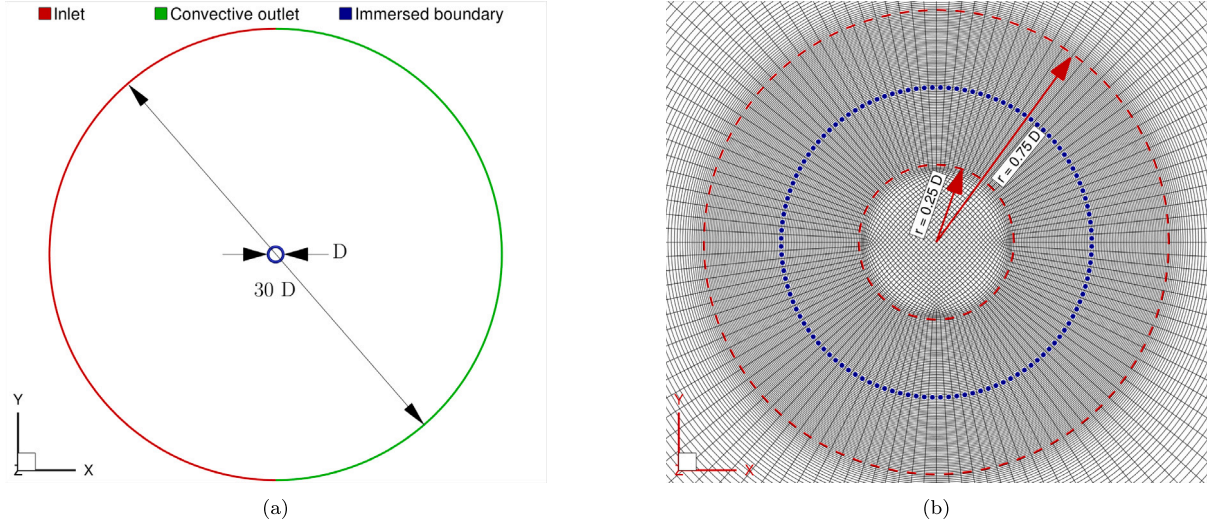


Fig. B.20. Test case setup: (a) Computational domain and boundary conditions. (b) Zoom of the computational grid around the cylinder in the x - y plane. Only every second grid line in each direction is displayed and the Lagrangian markers are indicated by blue circles.

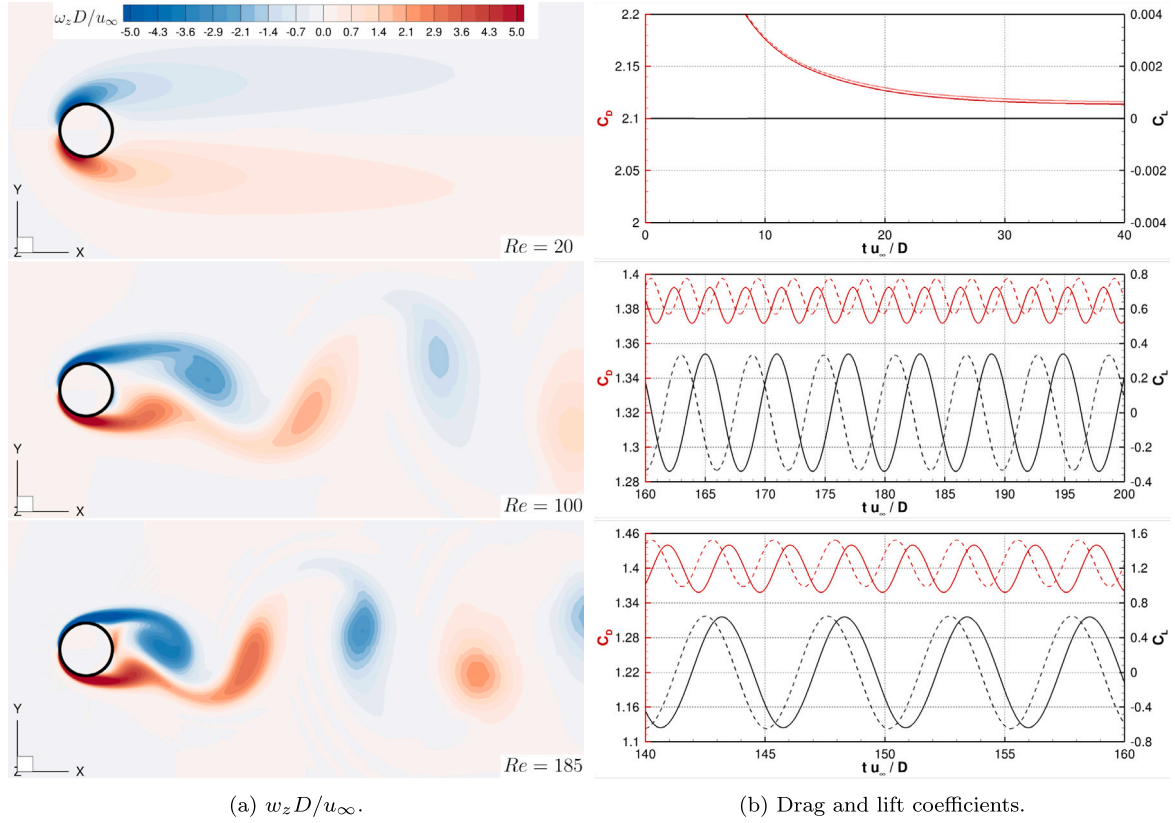


Fig. B.21. Snapshot of (a) the z -vorticity field and (b) the time history of the drag and lift coefficients of the fine grid (continuous lines) and the coarse grid (dashed lines).

formulation accurately enforces the no-slip condition at the fluid–solid interface.

B.2. Flow past an oscillating cylinder

In this section, the case of a transversely oscillating cylinder in a cross-flow is considered as a benchmark case to assess the accuracy of the IBM in scenarios involving forced motion of the immersed boundary. The computational domain, boundary conditions, and grid setup used here are the same as those described in [Appendix B.1](#). The

cylinder undergoes a time-periodic motion perpendicular to the main flow direction, with its vertical position defined by:

$$y_{cyl}(t) = A_e \sin(2\pi f_e t) \quad \text{with} \quad \frac{A_e}{D} = 0.2, \quad \frac{f_e}{f_0} = 0.8. \quad (\text{B.1})$$

Here, f_0 represents the natural shedding frequency of the flow past a stationary cylinder determined in [Appendix B.1](#). The Reynolds number is set to $Re = 185$, resulting in the same flow conditions as those considered in Uhlmann [17]. This configuration yields a vortex shedding frequency that synchronizes with the imposed oscillation

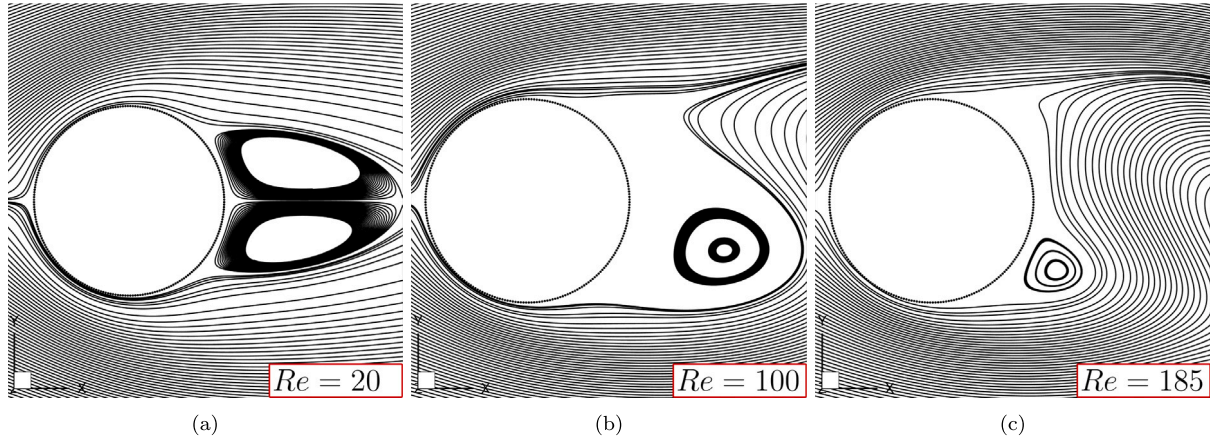


Fig. B.22. Snapshots of the streamlines for the flow past a stationary cylinder at (a) $Re = 20$, (b) $Re = 100$, and (c) $Re = 185$.

Table B.1

Mean drag coefficient $\bar{C}_D$ and Strouhal number St for the flow past a stationary cylinder at different Reynolds numbers. The corresponding lateral domain sizes H_y/D and grid resolutions $D/\Delta x$ used in the simulations are also listed.

	Method	H_y/D	$D/\Delta x$	$\bar{C}_D$	St
Re = 20					
Tritton [52]	Experiment	–	–	2.09	–
Taira and Colonius [55]	Diffuse-interface IBM	60	50	2.06	–
Yi et al. [56]	Diffuse-interface IBM	16	120	2.066	–
Present	Diffuse-interface IBM	30	200	2.115	–
Present	Diffuse-interface IBM	30	400	2.113	–
Present	Boundary-fitted	30	400	2.111	–
Re = 100					
Liu et al. [57]	Boundary-fitted	–	–	1.350	0.165
Mittal et al. [58]	Sharp-interface IBM	40	66.7	1.350	0.166
Uhlmann [17]	Diffuse-interface IBM	40	38.4	1.453	0.169
Yi et al. [56]	Diffuse-interface IBM	16	120	1.354	0.165
Present	Diffuse-interface IBM	30	200	1.387	0.168
Present	Diffuse-interface IBM	30	400	1.382	0.167
Present	Boundary-fitted	30	400	1.364	0.168
Re = 185					
Lu and Dalton [53]	Experiment	–	–	1.28	0.195
Guilmineau and Queutey [54]	Boundary-fitted	51	100	1.287	0.195
Yang and Balaras [59]	Sharp-interface IBM	30	–	1.366	0.197
Pinelli et al. [44]	Diffuse-interface IBM	34	200	1.430	0.196
Yi et al. [56]	Diffuse-interface IBM	16	120	1.352	0.193
Present	Diffuse-interface IBM	30	200	1.408	0.196
Present	Diffuse-interface IBM	30	400	1.399	0.195
Present	Boundary-fitted	30	400	1.347	0.195

frequency of the cylinder, indicating a lock-in condition. Fig. B.23 shows the drag coefficient as a function of the instantaneous vertical position of the cylinder once a quasi-steady state is reached. The results obtained with the present IB method at the two different grid resolutions are compared against the boundary-fitted ALE simulation and previous reference studies [17,56,60]. Overall, the IB simulations reproduce the general trend of the drag coefficient with respect to the cylinder's vertical displacement and the computed curves fall within the range bounded by the reference data. The minor spurious oscillations observed in the coarse grid simulation are significantly reduced upon grid refinement, reflecting the improvement in interface representation. These oscillations are attributed to the use of a 3-point kernel, which is set by the MLS construction to satisfy only up to the first moment condition and has a weak translational invariance. Table B.2 summarizes the time-averaged drag coefficients ($\bar{C}_D$), the mean oscillation amplitudes of both drag (C'_D) and lift (C'_L) coefficients, and the root-mean-square

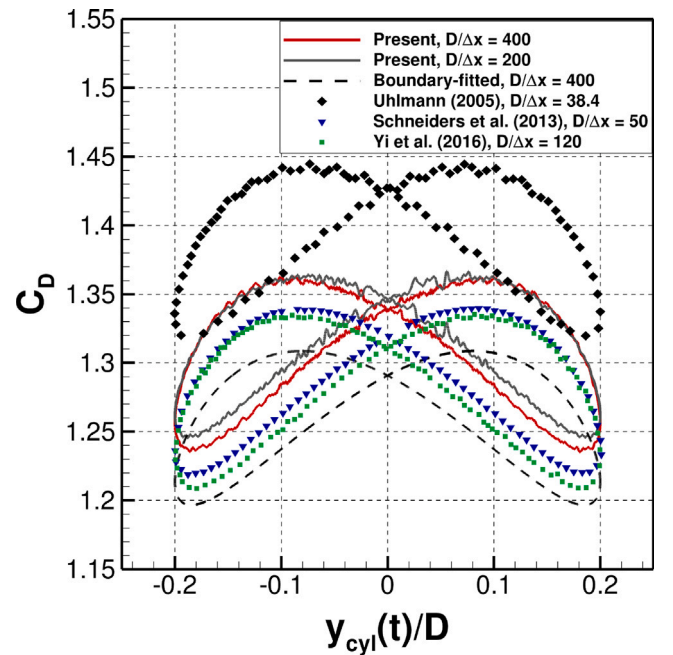


Fig. B.23. Variation of the drag coefficient for a transversely oscillating circular cylinder in a uniform cross-flow at $Re = 185$ as a function of the cylinder's vertical position.

Table B.2

Drag and lift coefficients and associated quantities for the flow past an oscillating cylinder at $Re = 185$.

	$\bar{C}_D$	C'_D	C'_L	$C_{L,RMS}$
Lu and Dalton [53]	1.25	–	–	0.18
Uhlmann [17]	1.354	± 0.065	–	0.166
Schneiders et al. [60]	1.279	–	–	0.082 ~ 0.086
Yi et al. [56]	1.273	± 0.062	± 0.124	0.083
Present ($D/\Delta x = 200$)	1.303	± 0.060	± 0.132	0.089
Present ($D/\Delta x = 400$)	1.297	± 0.063	± 0.127	0.085
Present boundary-fitted ($D/\Delta x = 400$)	1.253	± 0.056	± 0.110	0.076

value of the lift coefficient ($C_{L,RMS}$). To accurately evaluate the lift force, the inertial contribution of the “pseudo-fluid” enclosed by the cylinder is subtracted from the total Lagrangian force summation [17]. The obtained results show very good agreement with the reference data confirming reproducibility and reliability of the predictions.

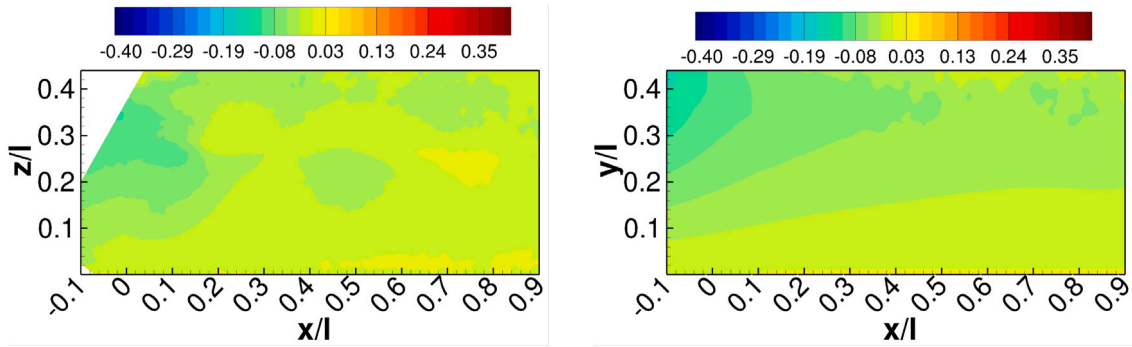
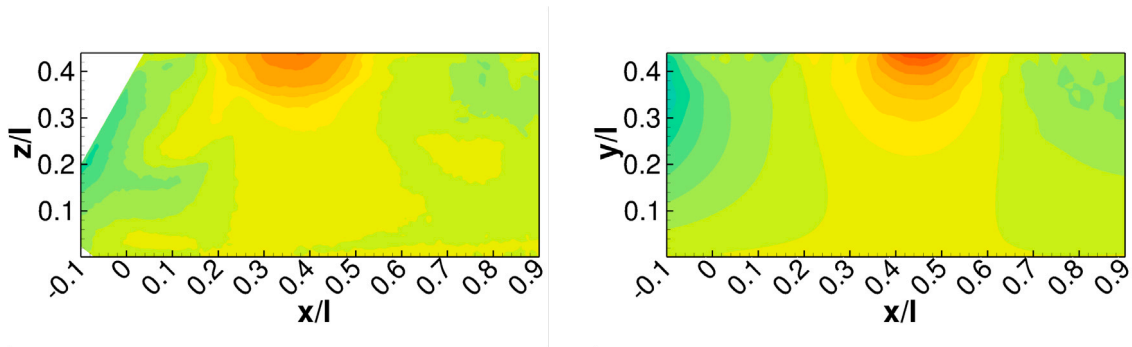
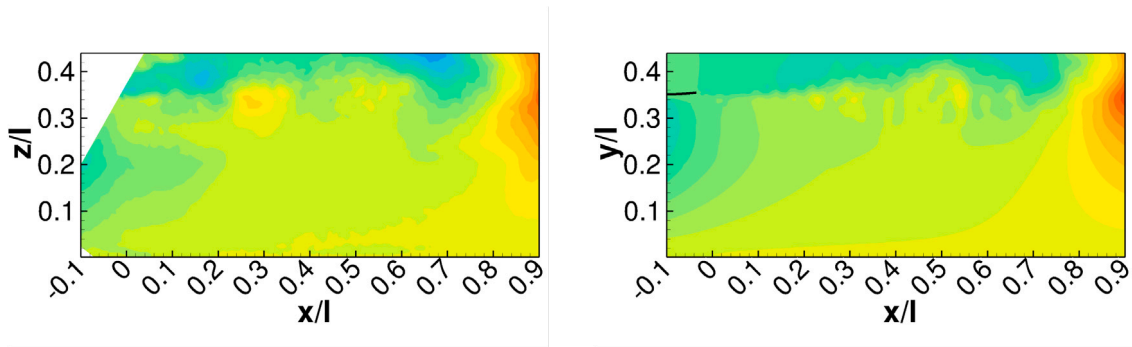
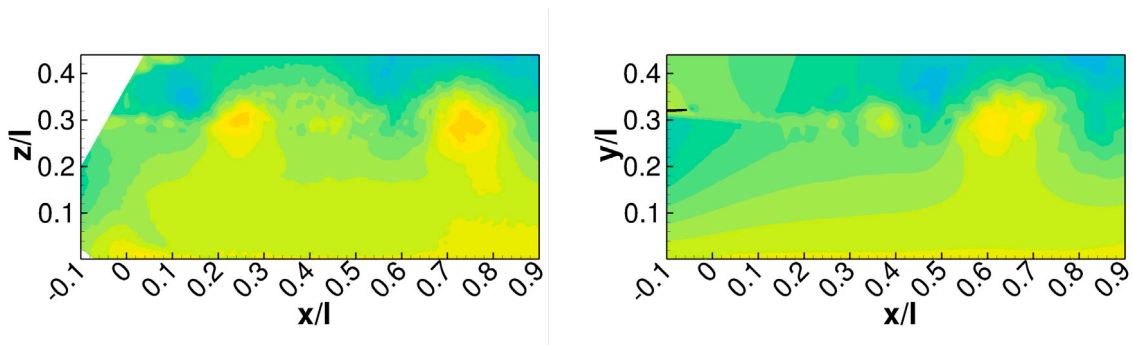
(a) $t/\Delta_{\text{gust}} = 0$: Experiment (left) vs. Simulation (right).(b) $t/\Delta_{\text{gust}} = 0.2$: Experiment (left) vs. Simulation (right).(c) $t/\Delta_{\text{gust}} = 0.4$: Experiment (left) vs. Simulation (right).(d) $t/\Delta_{\text{gust}} = 0.5$: Experiment (left) vs. Simulation (right).

Fig. D.24. Comparison between measured and predicted data of the phase-averaged vertical velocity in the symmetry plane at different time instants of the gust generation. Experimental data taken from Wood and Breuer [13].

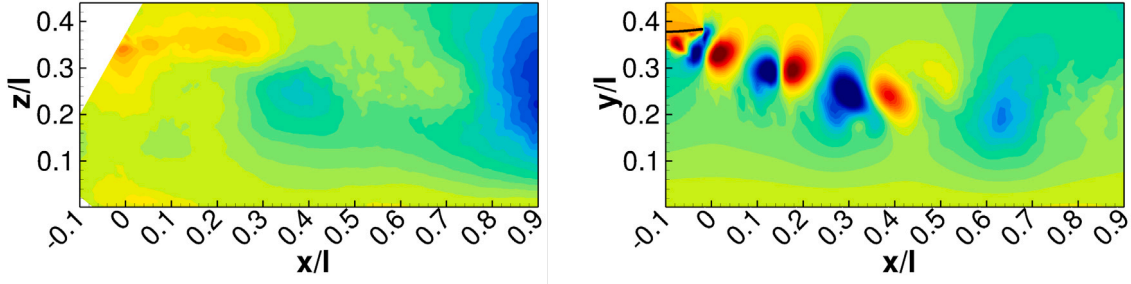
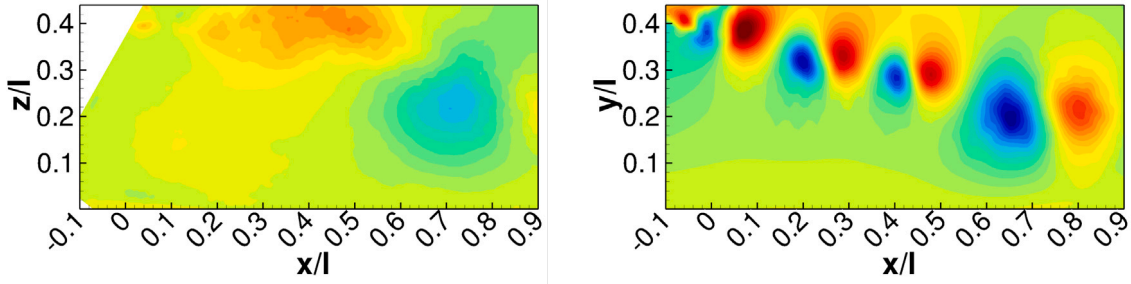
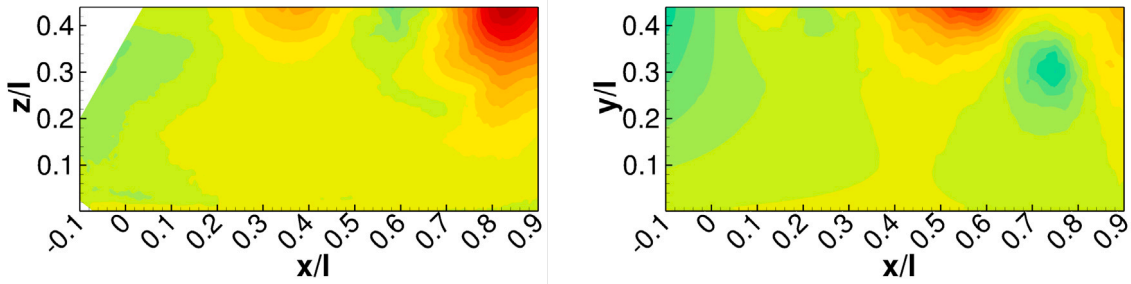
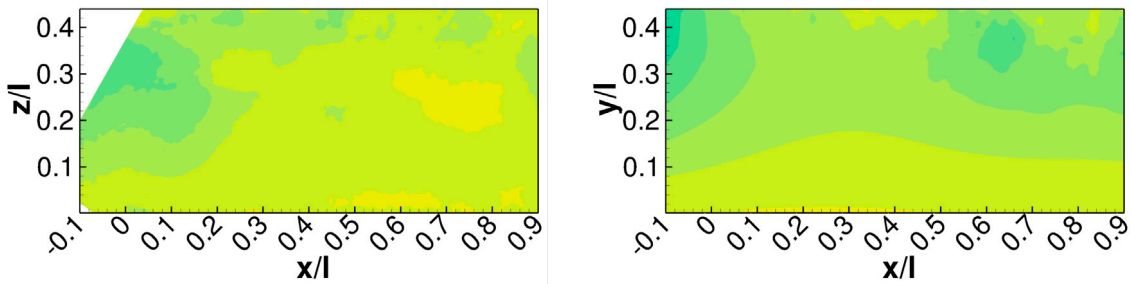
(e) $t/\Delta_{\text{gust}} = 0.7$: Experiment (left) vs. Simulation (right).(f) $t/\Delta_{\text{gust}} = 0.8$: Experiment (left) vs. Simulation (right).(g) $t/\Delta_{\text{gust}} = 1.0$: Experiment (left) vs. Simulation (right).(h) $t/\Delta_{\text{gust}} = 1.6$: Experiment (left) vs. Simulation (right).

Fig. D.24. (continued).

Appendix C. Transforming the saddle's translational motion to the nozzle's rotational motion

In this section, a detailed description on the transformation of the vertical motion of the WGG's saddle to the corresponding rotational motion of the nozzle is presented. The following two geometric

conditions hold throughout the nozzle's motion (see Fig. 11):

$$b^2 = (x_s - x_n)^2 + (y_s - y_n)^2, \quad (\text{C.1})$$

$$r^2 = x_n^2 + y_n^2, \quad \text{where } r = 0.336 \text{ m}. \quad (\text{C.2})$$

By substituting $x_s = a = 0.295 \text{ m}$, $x_n = r \cos \theta$ and $y_n = r \sin \theta$ in Eq. (C.1), one can find the relation between the coordinate y_s and the

angle of rotation θ :

$$\begin{aligned} b^2 &= (a - r \cos \theta)^2 + (y_s - r \sin \theta)^2, \\ (y_s - r \sin \theta)^2 &= b^2 - (a - r \cos \theta)^2, \\ (y_s - r \sin \theta) &= \sqrt{b^2 - (a - r \cos \theta)^2}, \\ y_s &= r \sin \theta + \sqrt{b^2 - (a - r \cos \theta)^2} \end{aligned} \quad (\text{C.3})$$

Differentiating Eq. (C.3) with respect to time and solving for ω_z yields:

$$\begin{aligned} v_s &= \omega_z r \cos \theta - \frac{(a - r \cos \theta) \omega_z r \sin \theta}{\sqrt{b^2 - (a - r \cos \theta)^2}}, \\ \omega_z &= \frac{v_s}{x_n - \frac{(a - x_n) y_n}{\sqrt{b^2 - (a - x_n)^2}}}. \end{aligned} \quad (\text{C.4})$$

The unknown coordinates of the point on the nozzle (i.e., $x_n(t)$ and $y_n(t)$) are determined by expanding Eq. (C.1) as:

$$\begin{aligned} (a - x_n)^2 + (y_s - y_n)^2 &= b^2, \\ a^2 - 2x_n a + x_n^2 + y_s^2 - 2y_n y_s + y_n^2 &= b^2. \end{aligned} \quad (\text{C.5})$$

Eq. (C.5) can be simplified replacing $x_n^2 + y_n^2$ by r^2 according to Eq. (38) leading to:

$$x_n a + y_n y_s = \underbrace{\frac{r^2 + a^2 + y_s^2 - b^2}{2}}_k.$$

This yields the following system of equations to be solved:

$$x_n a + y_n y_s = k, \quad x_n^2 + y_n^2 = r^2,$$

whose solutions are:

$$y_n = \frac{-B - \sqrt{B^2 - 4AC}}{2A}, \quad x_n = \frac{k - y_n y_s}{a},$$

where:

$$A = y_s^2 + a^2, \quad B = -2k y_s, \quad C = k^2 - r^2 a^2.$$

Note that in the solution for $y_n(t)$, the negative root is chosen to ensure $y_n(t) < y_s(t)$ and to maintain the consistency with the coordinate system defined in Fig. 11.

Appendix D. Vertical velocity field

In Section 5.2.1 a comparison between the measured and the predicted phase-averaged streamwise velocity contours in the symmetry plane was depicted in Fig. 17. Fig. D.24 complements this comparison by providing the vertical velocity component of the phase-averaged velocity fields at different instants of the gust generation process.

Data availability

Data will be made available on request.

References

- [1] Michalski A, Gawnat B, Gellenne P, Haug E. Computational wind engineering of large umbrella structures. *J Wind Eng Ind Aerodyn* 2015;144:96–107. <http://dx.doi.org/10.1016/j.jweia.2015.05.002>.
- [2] Li S, Catarelli RA, Phillips BM, Bridge JA, Gurley KR. Physical simulation of downburst winds for civil structures: A review. *J Wind Eng Ind Aerodyn* 2024;254:105900. <http://dx.doi.org/10.1016/j.jweia.2024.105900>.
- [3] Babaei R, Graat K, Chan C, Savory E. Experimental simulation of stationary and traveling density-driven thunderstorm downbursts using the two-fluid model. *J Wind Eng Ind Aerodyn* 2021;211:104553. <http://dx.doi.org/10.1016/j.jweia.2021.104553>.
- [4] Junayed C, Jubayer C, Parvu D, Romanic D, Hangan H. Flow field dynamics of large-scale experimentally produced downburst flows. *J Wind Eng Ind Aerodyn* 2019;188:61–79. <http://dx.doi.org/10.1016/j.jweia.2019.02.008>.
- [5] Aboutabikh M, Ghazal T, Chen J, Elgamel S, Aboshosha H. Designing a blade-system to generate downburst outflows at boundary layer wind tunnel. *J Wind Eng Ind Aerodyn* 2019;186:169–91. <http://dx.doi.org/10.1016/j.jweia.2019.01.005>.
- [6] Alawode KJ, Azzi Z, Elawady A, Chowdhury AG. Dynamic properties of an aeroelastic transmission tower subjected to synoptic and downburst-like outflows. *J Wind Eng Ind Aerodyn* 2023;242:105557. <http://dx.doi.org/10.1016/j.jweia.2023.105557>.
- [7] Butler K, Cao S, Kareem A, Tamura Y, Ozono S. Surface pressure and wind load characteristics on prisms immersed in a simulated transient gust front flow field. *J Wind Eng Ind Aerodyn* 2010;98(6):299–316. <http://dx.doi.org/10.1016/j.jweia.2009.11.003>.
- [8] Catarelli RA, Fernández-Cabán PL, Phillips BM, Bridge JA, Masters FJ, Gurley KR, et al. Automation and new capabilities in the university of florida NHERI boundary layer wind tunnel. *Front Built Environ* 2020;6:558151. <http://dx.doi.org/10.3389/fbuil.2020.558151>.
- [9] Wood JN, Breuer M, Neumann T. A novel approach for artificially generating horizontal wind gusts based on a movable plate: The paddle. *J Wind Eng Ind Aerodyn* 2022;230:105170. <http://dx.doi.org/10.1016/j.jweia.2022.105170>.
- [10] Breuer M, Neumann T. Synchronous high-speed measurements of a flexible structure under wind gust load. *Phys Fluids* 2024;36(7):075147. <http://dx.doi.org/10.1063/5.0215724>.
- [11] Boulbrachene K, Breuer M. A numerical method to mimic an experimental wind gust generator: The immersed boundary gust generator. *Phys Fluids* 2024;36(1):015146. <http://dx.doi.org/10.1063/5.0187787>.
- [12] Breuer M, Boulbrachene K, De Nayer G. Combination of two FSI methods and their validation based on artificial wind gusts impacting a flexible T-structure. *Comput & Fluids* 2025;294:106621. <http://dx.doi.org/10.1016/j.compfluid.2025.106621>.
- [13] Wood JN, Breuer M. Experimental study on a novel wind gust generator based on an adaptive nozzle design. *J Wind Eng Ind Aerodyn* 2025;261:106080. <http://dx.doi.org/10.1016/j.jweia.2025.106080>.
- [14] Sen S, De Nayer G, Breuer M. A fast and robust hybrid method for block-structured mesh deformation with emphasis on FSI–LES applications. *Int J Numer Meth. Engng* 2017;111(3):273–300. <http://dx.doi.org/10.1002/nme.5465>.
- [15] Peskin CS. The immersed boundary method. *Acta Numer* 2002;11:479–517. <http://dx.doi.org/10.1017/S0962492902000077>.
- [16] Mittal R, Iaccarino G. Immersed boundary methods. *Annu Rev Fluid Mech* 2005;37(1):239–61. <http://dx.doi.org/10.1146/annurev.fluid.37.061903.175743>.
- [17] Uhlmann M. An immersed boundary method with direct forcing for the simulation of particulate flows. *J Comput Phys* 2005;209(2):448–76. <http://dx.doi.org/10.1016/j.jcp.2005.03.017>.
- [18] Roma AM, Peskin CS, Berger MJ. An adaptive version of the immersed boundary method. *J Comput Phys* 1999;153(2):509–34. <http://dx.doi.org/10.1006/jcph.1999.6293>.
- [19] Stockie JM. Analysis and computation of immersed boundaries, with application to pulp fibres (Ph.D. thesis), Retrospective theses and dissertations, 1919–2007, University of British Columbia, Canada; 1997. <http://dx.doi.org/10.14288/1.0080530>.
- [20] Bao Y, Kaye J, Peskin CS. A Gaussian-like immersed-boundary kernel with three continuous derivatives and improved translational invariance. *J Comput Phys* 2016;316:139–44. <http://dx.doi.org/10.1016/j.jcp.2016.04.024>.
- [21] Wendland H. Local polynomial reproduction and moving least squares approximation. *IMA J Numer Anal* 2001;21(1):285–300. <http://dx.doi.org/10.1093/imanum/21.1.285>.
- [22] Vanella M, Balaras E. A moving-least-squares reconstruction for embedded-boundary formulations. *J Comput Phys* 2009;228(18):6617–28. <http://dx.doi.org/10.1016/j.jcp.2009.06.003>.
- [23] Krishnan S, Shaqfeh ES, Iaccarino G. Fully resolved viscoelastic particulate simulations using unstructured grids. *J Comput Phys* 2017;338:313–38. <http://dx.doi.org/10.1016/j.jcp.2017.02.068>.
- [24] Saadat A, Guido CJ, Iaccarino G, Shaqfeh ESG. Immersed-finite-element method for deformable particle suspensions in viscous and viscoelastic media. *Phys Rev E* 2018;98:063316. <http://dx.doi.org/10.1103/PhysRevE.98.063316>.
- [25] Li D, Wei A, Luo K, Fan J. An improved moving-least-squares reconstruction for immersed boundary method. *Int J Numer Methods Eng* 2015;104(8):789–804. <http://dx.doi.org/10.1002/nme.4949>.
- [26] Bhalla APS. On immersed boundary kernel functions: A constrained quadratic minimization perspective. 2021. arXiv preprint [arXiv:2111.11025](https://arxiv.org/abs/2111.11025).
- [27] Bale R, Bhalla APS, Griffith BE, Tsubokura M. A one-sided direct forcing immersed boundary method using moving least squares. *J Comput Phys* 2021;440:110359. <http://dx.doi.org/10.1016/j.jcp.2021.110359>.
- [28] Wood JN, Breuer M. Artificial wind gust generation based on an adaptive nozzle design. In: Rauh C, Ruck B, Leder A, editors. *Proc. der 31. Gala-Fachtagung 'Experimentelle Strömungsmechanik'*, Sept. 3–5, 2024. Berlin; 2024. 9–1–9–8.
- [29] Durst F, Schäfer M. A parallel block-structured multigrid method for the prediction of incompressible flows. *Int J Numer Methods Fluids* 1996;22(6):549–65. [http://dx.doi.org/10.1002/\(SICI\)1097-0363\(19960330\)22:6<549::AID-FLD366>3.0.CO;2-7](http://dx.doi.org/10.1002/(SICI)1097-0363(19960330)22:6<549::AID-FLD366>3.0.CO;2-7).

- [30] Khosla PK, Rubin SG. A diagonally dominant second-order accurate implicit scheme. *Comput & Fluids* 1974;2(2):207–9. [http://dx.doi.org/10.1016/0045-7930\(74\)90014-0](http://dx.doi.org/10.1016/0045-7930(74)90014-0).
- [31] Ferziger JH, Perić M. Computational methods for fluid dynamics. 3rd ed.. Springer Berlin; 2002. <http://dx.doi.org/10.1007/978-3-319-99693-6>.
- [32] Rhie CM, Chow WL. Numerical study of the turbulent flow past an airfoil with trailing-edge separation. *AIAA J* 1983;21(11):1525–32. <http://dx.doi.org/10.2514/3.8284>.
- [33] Breuer M, De Nayer G, Münsch M, Gallinger T, Wüchner R. Fluid–structure interaction using a partitioned semi-implicit predictor-corrector coupling scheme for the application of large-eddy simulation. *J Fluids Struct* 2012;29:107–30. <http://dx.doi.org/10.1016/j.jfluidstructs.2011.09.003>.
- [34] Stone HL. Iterative solution of implicit approximations of multidimensional partial differential equations. *SIAM J Numer Anal* 1968;5(3):530–58. <http://dx.doi.org/10.1137/0705044>.
- [35] Smagorinsky J. General circulation experiments with the primitive equations I: The basic experiment. *Mon Weather Rev* 1963;91(3):99–165. [http://dx.doi.org/10.1175/1520-0493\(1963\)091<0099:GCEWTP>2.3.CO;2](http://dx.doi.org/10.1175/1520-0493(1963)091<0099:GCEWTP>2.3.CO;2).
- [36] De Nayer G, Kalmbach A, Breuer M, Sicklinger S, Wüchner R. Flow past a cylinder with a flexible splitter plate: A complementary experimental–numerical investigation and a new FSI test case (FSI-PfS-1a). *Comput & Fluids* 2014;99:18–43. <http://dx.doi.org/10.1016/j.compfluid.2014.04.020>.
- [37] De Nayer G, Breuer M. Numerical FSI investigation based on LES: Flow past a cylinder with a flexible splitter plate involving large deformations (FSI-PfS-2a). *Int J Heat Fluid Flow* 2014;50:300–15. <http://dx.doi.org/10.1016/j.ijheatfluidflow.2014.08.013>.
- [38] De Nayer G, Apostolatos A, Wood JN, Bletzinger KU, Wüchner R, Breuer M. Numerical studies on the instantaneous fluid–structure interaction of an air-Inflated flexible membrane in turbulent flow. *J Fluids Struct* 2018;82:577–609. <http://dx.doi.org/10.1016/j.jfluidstructs.2018.08.005>.
- [39] De Nayer G, Breuer M, Wood JN. Numerical investigations on the dynamic behavior of a 2-DOF airfoil in the transitional Re number regime based on fully coupled simulations relying on an eddy-resolving technique. *Int J Heat Fluid Flow* 2020;85:108631. <http://dx.doi.org/10.1016/j.ijheatfluidflow.2020.108631>.
- [40] De Nayer G, Breuer M, Boulbrachene K. FSI simulations of wind gusts impacting an air-inflated flexible membrane at $Re=100,000$. *J Fluids Struct* 2022;109:103462. <http://dx.doi.org/10.1016/j.jfluidstructs.2021.103462>.
- [41] De Nayer G, Breuer M, Goldbach A-K. Fluid-structure interaction simulations of wind gusts impacting a hyperbolic paraboloid tensile structure. *Phys Fluids* 2024;36(10):105152. <http://dx.doi.org/10.1063/5.0232369>.
- [42] Wood JN, De Nayer G, Schmidt S, Breuer M. Experimental investigation and large-eddy simulation of the turbulent flow past a smooth and rigid hemisphere. *J Flow, Turbul Combust* 2016;97(1):79–119. <http://dx.doi.org/10.1007/s10494-015-9690-5>.
- [43] Schäfer F, Breuer M. Comparison of c-space and p-space particle tracing schemes on high-performance computers: Accuracy and performance. *Int J Numer Methods Fluids* 2002;39(4):277–99. <http://dx.doi.org/10.1002/fld.324>.
- [44] Pinelli A, Naqavi I, Piomelli U, Favier J. Immersed-boundary methods for general finite-difference and finite-volume Navier-Stokes solvers. *J Comput Phys* 2010;229(24):9073–91. <http://dx.doi.org/10.1016/j.jcp.2010.08.021>.
- [45] Zhou K, Balachandrar S. An analysis of the spatio-temporal resolution of the immersed boundary method with direct forcing. *J Comput Phys* 2021;424:109862. <http://dx.doi.org/10.1016/j.jcp.2020.109862>.
- [46] Zhou K, Ding Z, Sun K. Is Lagrangian weight crucial in the direct forcing immersed boundary method? *J Phys: Conf Ser* 2019;1324:012081. <http://dx.doi.org/10.1088/1742-6596/1324/1/012081>.
- [47] Abdol Aziz MH, Evrard F, van Wachem B. An immersed boundary method for incompressible flows in complex domains. *J Comput Phys* 2019;378:770–95. <http://dx.doi.org/10.1016/j.jcp.2018.10.048>.
- [48] Moody LF. An approximate formula for pipe friction factors. *Trans ASME* 1947;69(12):1005–11.
- [49] Ducros F, Nicoud F, Poinot T. Wall-adapting local eddy-viscosity models for simulations in complex geometries. In: Baines M, editor. *Numerical methods for fluid dynamics VI*. 1998, p. 293–9.
- [50] Germano M, Piomelli U, Moin P, Cabot WH. A dynamic subgrid-scale eddy viscosity model. *Phys Fluids A* 1991;3:1760–5. <http://dx.doi.org/10.1063/1.857955>.
- [51] IEC-Standard. 61400-21. Tech. Rep., measurement and assessment of power quality of grid connected wind turbines. 2002.
- [52] Tritton DJ. Experiments on the flow past a circular cylinder at low Reynolds numbers. *J Fluid Mech* 1959;6(4):547–67. <http://dx.doi.org/10.1017/S0022112059000829>.
- [53] Lu X-Y, Dalton C. Calculation of the timing of vortex formation from an oscillating cylinder. *J Fluids Struct* 1996;10(5):527–41. <http://dx.doi.org/10.1006/jfls.1996.0035>.
- [54] Guilmineau E, Queutey P. A numerical simulation of vortex shedding from an oscillating circular cylinder. *J Fluids Struct* 2002;16(6):773–94. <http://dx.doi.org/10.1006/jfls.2002.0449>.
- [55] Taira K, Colonius T. The immersed boundary method: A projection approach. *J Comput Phys* 2007;225(2):2118–37. <http://dx.doi.org/10.1016/j.jcp.2007.03.005>.
- [56] Yi W, Corbett D, Yuan X-F. An improved Rhie–Chow interpolation scheme for the smoothed-interface immersed boundary method. *Internat J Numer Methods Fluids* 2016;82(11):770–95. <http://dx.doi.org/10.1002/fld.4240>.
- [57] Liu C, Zheng X, Sung C. Preconditioned multigrid methods for unsteady incompressible flows. *J Comput Phys* 1998;139(1):35–57. <http://dx.doi.org/10.1006/jcph.1997.5859>.
- [58] Mittal R, Dong H, Bozkurtas M, Najjar F, Vargas A, von Loebbecke A. A versatile sharp interface immersed boundary method for incompressible flows with complex boundaries. *J Comput Phys* 2008;227(10):4825–52. <http://dx.doi.org/10.1016/j.jcp.2008.01.028>.
- [59] Yang J, Balaras E. An embedded-boundary formulation for large-eddy simulation of turbulent flows interacting with moving boundaries. *J Comput Phys* 2006;215(1):12–40. <http://dx.doi.org/10.1016/j.jcp.2005.10.035>.
- [60] Schneiders L, Hartmann D, Meinke M, Schröder W. An accurate moving boundary formulation in cut-cell methods. *J Comput Phys* 2013;235(C):786–809. <http://dx.doi.org/10.1016/j.jcp.2012.09.038>.

