



A methodology for predicting the hydroelastic response of appendages of high-speed boats using STAR-CCM+

Bernardo Gomes Ribeiro dos Santos Rocha

Dissertation submitted to
Faculdade de Engenharia da Universidade do Porto
for the degree of:

Mestre em Engenharia Mecânica

Advisor:
Pedro Manuel Leal Ribeiro
(Associate Professor)

Co-Advisor:
Gonzalo Redondo
(Managing Director at D3 Applied Tech)

Departamento de Engenharia Mecânica
Faculdade de Engenharia da Universidade do Porto

Porto, 2022

The work presented in this dissertation was performed at
D3 Applied Technologies
Santiago de Compostela, Spain.

Bernardo Gomes Ribeiro dos Santos Rocha
E-mail: up201705027@edu.fe.up.pt

Faculdade de Engenharia da Universidade do Porto
Departamento de Engenharia Mecânica
Rua Dr. Roberto Frias s/n
4200-465 Porto
Portugal

jdr@feup_2022

Abstract

Due to technological advances, hydrofoils have grown popular within the yacht racing world. In such a competitive context, the designs reflect the focus on hydrodynamic performance and weight-saving considerations that lead to more slender and flexible appendages. This, combined with the high speeds attained, accentuates fluid-structure interaction effects and, consequently, adds to the importance of hydroelastic analysis.

The under-development of hydroelasticity on a theoretical and experimental level is reflected on the lack of computational tools purposefully designed for hydroelastic analysis. Thus, there is a need to resort to more general numerical methods such as Finite Element (FEM) and Finite Volume (FVM) Methods, combined through a thought-out methodology. In this dissertation, the two methods are joined in a methodology specifically designed for application to appendages of high-speed boats. Its basis are (i) an Arbitrary Lagrangean-Eulerian (ALE) formulation of the Reynolds-Averaged Navier-Stokes (RANS) equations, for the fluid domain, (ii) strong coupling achieved through an iterative approach and (iii) a geometrically nonlinear momentum balance that describes the structural domain. For implementation, the commercial software Siemens Simcenter STAR-CCM+ is used.

The proposed methodology is tested and validated through comparison with experimental data in the literature. Subsequently, it is applied to the study of a rudder of a Moth International Class boat in typical racing conditions, demonstrating its applicability for design in a competitive sailing context. The results show, depending on the adopted configurations, good predictions of the hydroelastic response, in both transient and steady state. By predicting the behaviour of structure and flow, the developed methodology can be used for structural and hydrodynamic optimisation, fatigue studies and hydroelastic stability analysis.

Keywords: Fluid-Structure Interaction, Hydroelasticity, FEM, FVM

Resumo

Devido a avanços tecnológicos, os hidrofólios vêm ganhando popularidade, com particular expressão dentro do desporto da vela. No contexto competitivo em que se inserem as regatas, o projeto das embarcações coloca o foco no desempenho hidrodinâmico e na poupança de peso, resultando em apêndices mais esguios e flexíveis. Estas características, juntamente com as altas velocidades atingidas, intensificam os efeitos decorrentes da interação entre fluido e estrutura e, conseqüentemente, aumentam a importância da análise hidroelástica.

O subdesenvolvimento da hidroelasticidade ao nível teórico e experimental reflecte-se na falta de ferramentas computacionais propositadamente concebidas para a análise hidroelástica. Assim, há necessidade de recorrer a métodos numéricos mais gerais como os métodos dos Elementos Finitos (MEF) e dos Volumes Finitos (MVF), combinados através de uma metodologia previamente definida. Nesta dissertação, propõe-se tal metodologia, especificamente concebida para o estudo hidroelástico de apêndices de embarcações de alta velocidade. A proposta assenta (*i*) numa formulação Lagrangiana-Euleriana arbitrária das equações de Navier-Stokes com decomposição de Reynolds, para o domínio dos fluidos, (*ii*) num forte acoplamento conseguido através de uma abordagem iterativa e (*iii*) para modelar o domínio estrutural, num equilíbrio dinâmico que contabiliza não-linearidades geométricas. Para a sua implementação, é utilizado o software comercial Siemens Simcenter STAR-CCM+.

A metodologia proposta é testada e validada através da comparação com dados experimentais na literatura. Posteriormente, é aplicada ao estudo de um leme de um barco da Classe Internacional Moth em condições típicas de regata, com vista a demonstrar a sua aplicabilidade no contexto do projeto de barcos de competição. Os resultados mostram, dependendo das configurações adotadas, boas previsões da resposta hidroelástica, tanto transiente como em regime permanente. Ao prever o comportamento da estrutura e do escoamento, a metodologia desenvolvida pode ser usada para otimização estrutural e hidrodinâmica, estudos de fadiga e análise de estabilidade hidroelástica.

Keywords: Interação Fluido-Estrutura, Hidroelasticidade, MEF, MVF

To Luís and Henrique

Acknowledgements

I wish to express my sincere gratitude:

To my supervisors, Prof. Pedro Leal Ribeiro, for his shrewd comments and attention to detail that greatly benefited this text; and Gonzalo Redondo, for trusting me with this opportunity and facilitating the use of resources, without which this project would not have been possible;

To the team at D3 Applied Technologies, for sharing their experienced insights and receiving me in a very welcoming and fruitful environment. A special mention is due to Javier González, for his patience, comradery and humbling generosity;

To Ricardo Bencatel for his mentorship and interest throughout my academic path, which ultimately led to the writing of this dissertation;

To Cadflow and to Siemens Digital Industries Software, for granting a Simcenter STAR-CCM+ academic license. I would like to thank António Tomás and João Paulo Rodrigues, for the technical support throughout the process of applying for such license;

To my parents, my brother and sister, and my friends. To Sára.

Contents

Abstract	i
Resumo	iii
Acknowledgements	vii
Contents	ix
1 Introduction	1
1.1 Context and Motivation	1
1.2 Objectives	2
1.3 Layout	3
2 Literature Review	5
2.1 Introduction	5
2.2 Fluid-Structure Interaction Formulations	6
2.3 Coupling Techniques	7
2.4 Numerical Hydroelasticity and Applications	11
2.5 Fluid-Structure Interaction Instabilities	12
2.6 Vortex-Induced Vibrations	20
2.7 Chapter Recap	21
3 Mathematical Model	23
3.1 Introduction	23
3.2 Fluid domain	24
3.3 Solid domain	25
3.4 Conditions at the interface	26
3.5 Multiphase Flow	27
3.6 Chapter Recap	28
4 Numerical Methodology	29
4.1 Introduction	29
4.2 Finite Element Method	29
4.3 Finite Volume Method	34
4.3.1 Discretization of the Convection Term	34
4.3.2 Discretization of the Diffusion Term	35
4.3.3 Pressure-Velocity Coupling	37
4.3.4 Volume of Fluid Method	39
4.3.5 Wave Characteristics and Modelling	42
4.3.6 Time Discretization	44

CONTENTS

4.3.7	Iterative Methods	44
4.4	Fluid-Structure Coupling	45
4.5	Chapter Recap	46
5	Test Cases	49
5.1	Introduction	49
5.2	Validation Case	49
5.2.1	Model Setup	49
5.2.2	Meshing	51
5.2.3	Results and Discussion	54
5.3	Moth Rudder	62
5.3.1	Model Setup	62
5.3.2	Meshing	63
5.3.3	Results and Discussion	68
5.4	General Recommendations	74
6	Conclusion	79
6.1	Conclusions	79
6.2	Future Work	80
	References	81

List of Figures

2.1	Representation of Lagrangian, Eulerian and Arbitrary Lagrangian-Eulerian Formulation. Adapted from [24].	6
2.2	Diagram presenting and comparing conventional approaches and algorithms used to address coupled problems. Adapted from [51].	8
2.3	Flowchart representing the implicit algorithm for solving FSI systems with a partitioned approach.	10
2.4	Simplified model for the fluid-structure interaction of an immersed hydrofoil sailboat's rudder.	12
2.5	Influence of the velocity of the rigid body on the direction of the apparent flow velocity	17
2.6	General plot of Lift Coefficient vs. Angle of attack for a symmetrical profile based on the results from [79].	17
2.7	Lock-in effect.	21
3.1	Schematic representation of a general FSI problem.	23
3.2	Representation of the information exchange between solid and fluid domain through the interface conditions.	26
3.3	Examples of multiphase flows: (a) droplets of water suspended in air (dispersed flow), (b) wavy sea surface (stratified flow).	27
4.1	Linear tetrahedron [102].	31
4.2	Effects of numerical damping on the solution for the nonlinear equation of motion of a pendulum, using Backward-Euler with Newton-Raphson. Computed and plotted with Octave 6.3.0.	32
4.3	Two non-orthogonal adjacent cells.	37
4.4	Flowchart representing the SIMPLE algorithm for unsteady simulations. . .	39
4.5	Schematic representation of the normalized variable distribution. Adapted from [125].	41
4.6	(a) Map showing the different race courses for the AC36 [134]; and (b) detail of depth chart of the Hauraki Gulf [135].	43
5.1	Geometry of the validation case.	50
5.2	Variation of the solution for the first three natural frequencies with the mesh size, for the validation case.	52
5.3	Variation of the solution for the force coefficients with the mesh size, for the validation case.	52
5.4	Two-dimensional section of the fluid mesh for the validation case, close to the foil.	53
5.5	Solid mesh, with identification of the element type (12 - linear hexahedron). .	54

5.6	Comparison between numerical predictions of tip displacement and experimental data.	55
5.7	Comparison between force coefficients determined for rigid and flexible hydrofoil; (a) lift coefficient, (b) drag coefficient.	55
5.8	Velocity spectra obtained from first and second order time discretization.	56
5.9	Comparison between displacement response at the tip of the foil for flow velocities of 5, 7 and 10 m/s. The displacements are normalized with respect to the corresponding static displacements, for easier comparison.	58
5.10	Application of the Southwell method for estimating divergence velocity from numerical data.	59
5.11	Displacement response predicted by the three different models.	60
5.12	Velocity response predicted by the three different models.	60
5.13	Phase-space trajectory of the system as predicted by the three different models.	61
5.14	Lift coefficient versus Drag coefficient evolution as predicted by the three different models.	61
5.15	Change in the numerical solution with the time step size.	62
5.16	Geometry for the moth rudder simulation.	63
5.17	Variation of the solution for the first three natural frequencies with the mesh size, for the moth rudder case.	65
5.18	Variation of the solution for the fluid forces with the mesh size, for the moth rudder case.	65
5.19	Two-dimensional section of the fluid mesh for the moth rudder simulation; (a) whole domain, (b) close to the foil.	67
5.20	Details of the fluid mesh for the moth rudder simulation; (a) tip of the bulb, (b) trailing edge of the foil, (c) front-view detail of the foil tip.	68
5.21	Rudder mesh, with identification of the element type (10-linear tetrahedron; 13-linear wedge; 14-linear pyramid); general view with detailed view of the foil tip and trailing edge.	68
5.22	Location and identification of the probes on the moth rudder.	70
5.23	Trajectory of the different probes through three-dimensional space.	70
5.24	Trajectories of the probes projected on (a) Oxy plane, (b) Oxz plane (c) Oyz plane.	72
5.25	States of the probes regarding the motion in the (a) x direction, (b) y direction (c) z direction.	76
5.26	Von Mises stress displayed over deformed configuration (deformations multiplied by scale factor of 5), with undeformed configuration presented for comparison.	77
5.27	First three natural mode shapes of the steel rudder; (a) first mode shape, (b) second mode shape, (c) third mode shape.	77
5.28	Velocity spectra for the x , y and z direction, for the responses recorded (a) on the strut, (b) on the foil.	78
5.29	Velocity spectra for the x , y and z direction, for the one-way FSI responses recorded (a) on the strut, (b) on the foil.	78

List of Tables

4.1	Discretization schemes available in Siemens Star-CCM+ and their relevant characteristics [102].	36
5.1	Fluid boundary conditions for the validation case.	50
5.2	Solid boundary conditions for the validation case.	50
5.3	Fluid-structure coupling highlighted settings.	51
5.4	Important properties of remaining solvers.	51
5.5	Prism Layer Mesher settings.	53
5.6	Default Controls for the fluid mesh.	53
5.7	Custom Controls for the fluid mesh.	54
5.8	Default Controls for the solid mesh.	54
5.9	Experimentally determined in-water natural frequencies [75]	56
5.10	Comparison between "dry" and "wet" natural frequencies.	58
5.11	Fluid boundary conditions for the moth rudder simulation.	63
5.12	Solid boundary conditions for the moth rudder simulation.	64
5.13	Fluid-structure coupling highlighted settings for the moth rudder simulation.	64
5.14	Properties setup for the VOF solver and waves.	66
5.15	Important properties of remaining solvers for the moth rudder simulation.	66
5.16	Prism Layer mesher settings for the moth rudder simulation.	66
5.17	Default Controls for the fluid mesh around the moth rudder.	67
5.18	Custom Controls for the fluid mesh around the moth rudder.	69
5.19	Default Controls for the solid mesh of the rudder.	69
5.20	Custom Controls for the solid mesh of the rudder.	71
5.21	First three modes and corresponding "dry" natural frequencies of the steel rudder.	71
5.22	Mesh quality measures, their recommended values and corresponding values on the deformed mesh and undeformed mesh.	73
5.23	Comparison between results given by one- and two-way FSI analyses.	74

Symbols and Abbreviations

Roman Letters

A	Cross sectional area perpendicular to the flow;
$\mathbf{a}_f$	Surface area vector;
$\mathbf{A}_P$	Diagonal fluid system matrix from discretization;
$\mathbf{b}$	Body forces;
$\mathbf{B}$	Strain-displacement matrix;
B_o	Constant;
$\mathbf{c}$	Convective velocity;
C_D	Drag Coefficient;
C_L	Lift Coefficient;
$\mathbf{C}_f$	Fluid-induced damping matrix;
c_f	Fluid-induced damping;
C_o	Local Courant number;
C_y	Pseudo-Cauchy number;
D	Displacement number;
$\mathbf{D}$	Sensitivity of Second Piola-Kirchhoff tensor to Green tensor;
D_h	Sea depth;
E	Young's modulus;
$\mathbf{E}$	Green strain tensor;
$\hat{\mathbf{e}}$	Unitary vector;
$\circ_e$	Pertaining to element e ;
$\mathbf{f}$	Generalized force;
$\mathbf{f}$	Generalized forces vector;
$\mathbf{F}$	Deformation gradient tensor;
F	Blending function;
$\mathbf{f}^B$	Vector of volume forces;
$\mathbf{f}^{ext}$	Vector of external forces;
$\mathbf{f}^{int}$	Vector of internal forces;
$\mathbf{f}^S$	Surface traction;
f_v	Vortex-shedding frequency;
F_{vortex}	Vortex-induced load;
$\mathbf{g}$	Gravitational acceleration vector;
$\mathbf{H}$	Representative of fluid system matrix from discretization;
$\mathbf{I}$	Identity matrix;
J	Jacobian determinant of the deformation gradient tensor;

$\circ_k$	at iteration k ;
$\mathbf{K}_{eq}$	Equivalent stiffness matrix;
k_{eq}	Equivalent stiffness;
$\mathbf{K}_f$	Fluid-induced stiffness matrix;
k_f	Fluid-induced stiffness;
$\mathbf{K}^{\tan}$	Tangential stiffness matrix;
L	Characteristic length;
$\mathbf{M}$	Mass matrix;
M	Mass number;
m_s	Rigid body mass;
$\mathbf{n}$	Normal unitary vector;
$\mathbf{N}$	Vector of shape functions;
$\circ^n$	at time instant n ;
$\mathbf{o}$	General vectorial quantity;
$\mathbf{P}$	First Piola-Kirchhoff stress tensor;
p	Pressure;
$\mathbf{q}$	Generalized displacement vector;
q	Generalized displacement;
q_0	Characteristic generalized displacement;
Re	Reynolds number;
$\mathbf{R}$	Residual;
S	Surface area;
$\mathbf{S}$	Second Piola-Kirchhoff stress tensor;
St	Strouhal number;
T	Representative timescale; period;
t	Time;
$\mathbf{u}$	Displacement field;
$\bar{\mathbf{u}}$	Nodal displacements;
$\mathbf{U}_P$	Fluid velocity components at cell center;
U_r	Reduced velocity;
U_R	Ursell number;
$\mathbf{v}$	Velocity field;
$\hat{\mathbf{v}}$	Mesh velocity;
v_0	Characteristic velocity;
V_c	Donor cell volume;
W	Work;
$\mathbf{X}$	Material coordinates;
$\mathbf{x}$	Spatial coordinates;
$\mathbf{x}_o$	Vector storing general quantities.

Greek Letters

α	Angle of attack;
α_p	Pressure under-relaxation factor;
α_v	Velocity under-relaxation factor;
β	Volume fraction;
Γ	Interface;
γ	Angle between interface and cell surface;
Δ_o	Interval;
θ	Rotation;
λ	Wavelength;
λ_o	Transformed off-diagonal terms of fluid-induced stiffness matrix;
Λ	Matrix containing transformed off-diagonal terms of fluid-induced stiffness matrix;
λ_L, μ_L	Lamé constants;
μ_f	Dynamic viscosity;
ν_f	Kinematic viscosity;
ν_s	Poisson's ration;
ξ	Normalized volume fraction;
ξ	Natural coordinates;
Π	Pi group;
ρ_f	Fluid density;
ρ_s	Solid density;
σ	Cauchy stress tensor;
σ_f	Fluid stress tensor;
τ	Control volume;
ϕ_o	General representation of function;
ϕ_o^d	Representation of discretized function;
χ	Referential coordinates;
ω	Frequency;
Ω_o	Volume;
$\bar{\omega}$	Coalesced frequency;
ω_n	Natural frequency;
ω_v	Vortex-shedding frequency.

Other Symbols

$\dot{\circ}$	First derivative with respect to time;
$\ddot{\circ}$	Second derivative with respect to time;
$\tilde{\circ}$	Dimensionless (fluid);
$\bar{\circ}$	Dimensionless (solid);
$\nabla_{\circ}$	Nabla operator;
$\frac{\partial \circ}{\partial \circ}$	Partial derivative;
$\circ _{diag}$	Diagonal entries of a matrix;
$\circ _{off-diag}$	Off-diagonal entries of a matrix;
$\circ^T$	Transpose;
$\mathbb{I}m(\circ)$	Imaginary part;
$\mathbb{R}e(\circ)$	Real part;
e	Euler's number;
i	Imaginary unit.

Abbreviations

ALE	Arbitrary Lagrangean-Eulerian ;
AMG	Algebraic Multigrid;
AoA	Angle of Attack;
BCD	Bounded Central-Differencing;
CCM	Computational Continuum Mechanics;
CD	Central-Differencing;
CFD	Computational Fluid Dynamics;
CFL	Courant–Friedrichs–Lewy;
DNS	Direct Numerical Solution;
DoF	Degree-of-Freedom;
FEM	Finite Element Method;
FOU	First-Order Upwind;
FPK	First Piola-Kirchhoff;
FSI	Fluid-Structure Interaction;
FVM	Finite Volume Method;
GCL	Geometric Conservation Law;
HM3/CD	Hybrid MUSCL 3rd-Order/Central-Differencing;
HRIC	High-Resolution Interface Capturing;
LES	Large Eddy Simulation;
MUMPS	Multifrontal Massively Parallel sparse direct Solver;
NVD	Normalized Variable Diagram;
PDE	Partial Differential Equation;
PISO	Pressure Implicit Split Operation;
POM	Polyoxymethylene;
PVW	Principle of Virtual Work;
QUICK	Quadratic Upstream Interpolation for Convective Kinematics;
RANS	Reynolds-Averaged Navier-Stokes;
RBF	Radial Basis Function;
SOU	Second-Order Upwind;
SPK	Second Piola-Kirchhoff;
SIMPLE	Semi-Implicit Method for Pressure Linked Equations;
URANS	Unsteady Reynolds-Averaged Navier-Stokes;
URF	Under-Relaxation Factor;
VOF	Volume of Fluid.

Introduction

1.1 Context and Motivation

As a consequence of developments in manufacturing processes, materials, computer-aided engineering software and an increase in computational power, hydrofoiling has enjoyed a growth spurt in popularity throughout the beginning of the twenty-first century. This has been especially obvious in the context of the sailing sport; first, with the acceptance of hydrofoils within the International Moth Class, in 2000, and, more recently, within the America's Cup and the new SailGP. Nevertheless, the hydrofoil is not recent technology and dates back to the nineteenth-century. The amusing story of its origin is told by Mishel and Bolton:

"(...) an Englishman, Thomas Moy, attached wings under a boat in order to observe the flow of water over the surfaces and the swirls created under water. In 1861 on a Surrey canal to the amazement of Thomas Moy, the boat rose out of the water. Without intending to, he had invented the hydrofoil (...)" [1].

Since then, hydrofoiling boats have been used in both military applications and for passenger transportation, in addition to the aforementioned sportive or racing context, while the engineering requisites of such projects have grown increasingly ambitious. Taking sailing racing as an example, the operating conditions to which the hydrofoil and other appendages are exposed can be very demanding, given that high-speeds are attained. Even more so, because the hydrodynamic and weight-saving considerations, that are prioritized during the design of such vessels, often lead to thinner and lighter lifting and control surfaces and make them, in principle, more prone to deform significantly or even to ensue in unstable exchanges of energy with the flow [2].

Such interactions have already been seen in real racing conditions, with a well-known example being the capsize of Bernard Stamm's IMOCA 60 during "The Transat", in 2004, due to a loss of its keel caused by flutter. More recently, the capsize of American Magic's *Patriot* (at nearly 50 knots) was also speculated to be linked to some unstable interaction of the rudder with the water. The rudder's constituent material had previously been changed from a composite material to steel and it was proposed that the altered mass of the appendage could have been related to the unexpected instability. These examples highlight the importance of understanding the hydroelastic behavior of these structures, enabling accurate predictions so that the risks of catastrophic failure can be minimized.

However, extreme situations are far from constituting the only interest in studying Fluid-Structure Interaction (FSI). In fact, there has been a surging interest in FSI models whose predictions can be integrated in design and optimization processes [3; 4], both

structural and hydrodynamic. This bears a particular importance in the context of racing, where even the smallest advantage is sought-after and the budgets are not major concerns.

Earlier analysis of interactions between fluid and structure involved either a more theoretical approach, with simplified analytical or semi-analytical solutions, or resorted to experimentation with scaled models, together with scaling laws [5; 6]. Currently, studies of the behaviour of structures interacting with a fluid increasingly rely on numerical tools which, combined with the theoretical and experimental approaches, have contributed to detailed descriptions of these multiphysics problems. Good examples of such descriptions can be found in the well-developed field of aeroelasticity, which studies the interaction between air flows and structures.

Nevertheless, even though a vast knowledge base exists for aeroelasticity, these insights are not directly transferable to hydroelasticity, the field that studies the interaction between water flows and structures, as the largely different properties of water, compared to air, significantly alter the nature of the fluid-structure coupling. Furthermore, hydroelasticity comes with its own set of particular challenges, which include the possible occurrence of dispersed multiphase flow, related to phenomena such as cavitation and ventilation; the interaction with a free surface; the additional numerical stability concerns; and the cost and complexity of proper experimental setups. All of these factors contribute to the underdevelopment of the field, especially in what regards to its theoretical foundations. As Korobkin, Părău and Vanden-Broeck noted, in 2011:

”We found that the theoretical part of research on hydroelasticity at present is less developed than the numerical and experimental studies. This inadequacy in the body of theory is the main obstacle to progress in this discipline, including progress in computations and experiments.”[7]

Notwithstanding this diagnosis, FSI in general (and hydroelasticity in particular) is a mature-enough field to be applied to real world problems and numerical tools yield, in most cases, suitable predictions. Even some commercial software, such as Siemens Simcenter STAR-CCM+ or Ansys, is capable of conducting this type of analyses. Furthermore, state-of-the-art procedures in naval design include FSI analysis, albeit an unidirectional one [8]. This means that the coupling of fluid and structure is only established from one domain to the other (usually, from fluid to solid) with no feedback and is known as one-way FSI. In this work, a more advanced approach is developed, consisting of two-way FSI analysis, that accounts for exchange of information in both directions, and its merits and disadvantages are assessed.

1.2 Objectives

This dissertation aims at designing advanced tools for the study of FSI established between the appendages of high-speed boats and the flow of seawater around them. At the conclusion of the project, two models shall be developed using the commercial multiphysics software Siemens Simcenter STAR-CCM+:

- **a simplified version of a boat appendage**, consisting of a fully-submerged extruded NACA profile for the purposes of providing a simple case that can be validated using available data from the literature and a quick way to test the developed methodology concerning sensitivity to relevant choices of parameters/methods;

- **a representative scenario of operating conditions and geometry of surface-piercing appendages of high-speed boats**, using the relatively accessible rudder of an International Moth Class foiling boat, with the purpose of demonstrating that the developed methodology is suitable for the analysis of the hydroelastic response of partially-submerged appendages in a demanding racing environment.

Additionally, the two models shall be as similar to each other as possible (while attending to the limitations imposed by available experimental data), so that the conclusions taken from the simple model can be extended to the complete version.

The end goal is to design a robust and efficient tool that can be incorporated in the project of high-speed boats and is capable of

- predicting the mechanical response of the boat appendages resulting from the interaction with the flow of water around them;
- predicting changes to the flow caused by the deformation of the structure and the effects on the hydrodynamic performance of the appendages;
- capturing possible unstable behaviour arising from the bi-directional interactions between structure and fluid;
- accounting for additional relevant phenomena in the context of sail racing, such as interaction with a wavy free surface and ventilation;
- making these predictions as efficiently as possible, so that use of these tools is viable in practice.

Further objectives of this study are related to the characterization of the proposed methodology, with focus on

- understanding its limitations and shortcomings;
- comparing the two-way FSI approach to the one-way FSI used in the industry.

1.3 Layout

To achieve the proposed objectives, a series of steps was followed and these are successively covered throughout the different chapters of this dissertation in the following order:

- Chapter 2 compiles the fundamental concepts and theoretical background needed to address an FSI problem; first, FSI is introduced in more detail and the different formulations suitable for describing such systems are addressed; then, the challenges of numerically solving the FSI system of equations are presented, together with an overview of available methods and a particular focus on the coupling between structure and fluid; additionally, a brief mention of previous studies in the literature using such methods is made; and, finally, a simple mathematical description of the FSI exchanges leading to unstable or dangerous vibrational behaviour is presented, with the purpose of understanding and identifying extreme hydroelastic phenomena possibly occurring in a sailing racing context;
- Chapter 3 aims at establishing the system of equations governing the FSI between rudder and seawater. Each domain, solid and fluid, is described separately in detail, after which the coupling conditions are established. The chapter ends with some considerations regarding the multiphase characteristics of the flow;

- Chapter 4 discusses Computational Continuum Mechanics, with special focus on the Finite Element and Finite Volume Methods, which are used for this project. The theoretical foundations of these methods are detailed to allow for an adequate description of the developed methodology and to justify the options taken when designing this tool;
- Chapter 5 describes the specific model setups pertaining to the simplified case and the moth rudder simulation. The results from the runs of the models are then presented and discussed, with focus on validating and characterizing the methodology;
- Chapter 6 completes the dissertation, exposing the main conclusions drawn throughout the project.

Literature Review

2.1 Introduction

Fluid-Structure Interaction belongs to the class of multiphysics coupled problems, which designates the cases where two or more fields of different physical nature interact with each other. Depending on how this interaction occurs, multiphysics coupled problems can be further classified as

- *Surface coupled or geometrically-coupled problems* - the fields involved are separated by an interface, through which the interaction is established, thus occupying different spatial domains;
- *Volume coupled or materially-coupled problems* - the relevant physical variables of the problem are defined in the same domain and interact at each material point of that domain.

The difference between both can be further illustrated by pointing out the case of thermo-mechanical problems, where the deformation and temperature fields interact in the whole body, as an example of a volume coupled problem; and the case of FSI, where the coupling occurs only at the interface, thus constituting a surface coupled problem. This is an important distinction to make, since it has implications on how the model is set up and treated numerically.

For example, as a surface coupled problem, FSI is more suitable for a partitioned solution than the thermo-mechanical problem. This is a direct consequence of the fact that both domains can be separated and solved individually, with coupling conditions at the interface.

The presence of the different domains in FSI, fluid and solid, also brings another challenge concerning the description of the phenomena. This is, since a Lagrangian description is preferable for the solid domain and an Eulerian description is more adequate for the fluid domain, there is a need to establish a compatible formulation that is capable of accommodating the particular characteristics of each.

This way, some of the points of major importance when addressing computational fluid-structure interaction are related to either the type of description that is used to represent both domains, or to how the fully established model can be discretized and solved numerically. These points are to be tackled in Sections 2.2 and 2.3 of this work, respectively. In Section 2.4, a brief overview of the applications of these concepts is made, with particular emphasis on high-speed sailboats and their appendages. Finally, in Sections 2.5 and 2.6, simple models are used to illustrate the mathematical intricacies of unstable

hydroelasticity, selecting and focusing on instabilities that are most relevant within the scope of this dissertation.

2.2 Fluid-Structure Interaction Formulations

To tackle the incompatibility between conventional descriptions of fluid and solid problems, several formulations have been proposed in the field of FSI. Among these are the Smooth Particle Hydrodynamics methods [9–12], which are Lagrangian mesh-free methods that have been successfully applied and further developed in the context of violent flow FSI; the Space-Time Finite Element method [13–15], also designated as Deforming-Spatial-Domain/Stabilized Space-Time formulation; and the Arbitrary Lagrangian-Eulerian (ALE) formulation. Although the first-mentioned methods have their merits and constitute active fields of research, ALE is still used for a great range of FSI applications and is frequently employed by commercial software, which is why the focus will be kept on ALE hereafter.

After the pioneering works by Noh [16] and by Franck and Lazarus [17] on coupled Eulerian-Lagrange frameworks, ALE was first proposed by Hirt, Amsden, and Cook in [18], in an attempt to combine the more interesting characteristics of the Lagrangian formulation (good representation of large compression or expansion deformations; ability to handle multimaterial interfaces) and the Eulerian formulation (advantageous when large shear deformations occur). Initially introduced with structured meshes and finite-difference schemes, ALE was later applied to arbitrarily polygonal meshes [19; 20], to the Finite Element Method [21; 22] and to the Finite Volume Method [23].

When using an ALE formulation, the two domains, structure and fluid, are discretized separately. On one hand, a Lagrangian description is employed on the solid nodes. This is more convenient in solid mechanics and also facilitates the tracking of the fluid-structure interface at all times, which is needed because its motion is not previously known and depends on the solution. On the other hand, the fluid nodes are treated in an arbitrarily Lagrangian-Eulerian way; i.e., as shown in Figure 2.1, they are neither fixed, as in the Eulerian formulation, nor accompany the deformation of the continuum, as in the Lagrangian formulation.

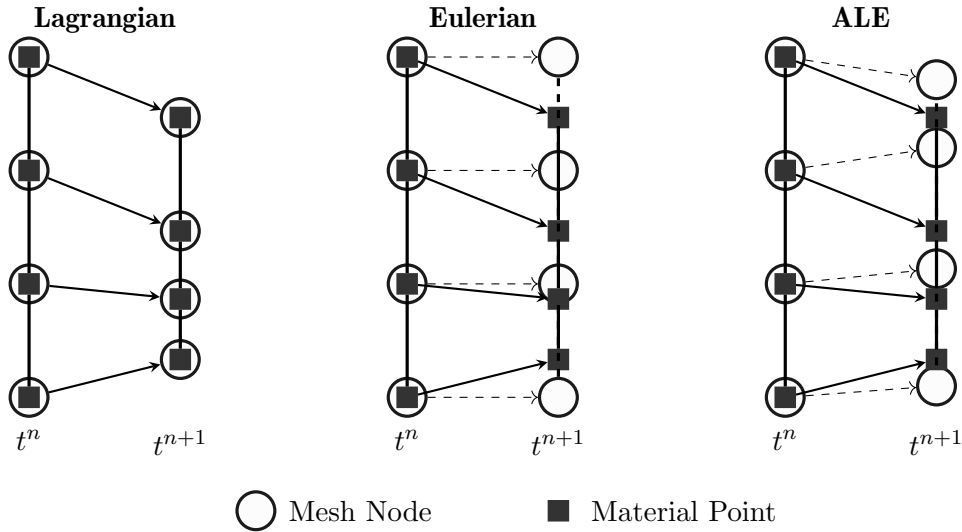


Figure 2.1: Representation of Lagrangian, Eulerian and Arbitrary Lagrangian-Eulerian Formulation. Adapted from [24].

This constitutes the main disadvantage of ALE: the mesh motion needs to be defined by a previously established mesh-update procedure, which increases the computational cost. This mesh rezoning is commonly dependent on the structural response and several techniques may be employed, with considerable influence on the results [24; 25]. These techniques may be classified as either algebraic approaches or elliptical approaches [26]. Elliptical methods consist on the solution of partial differential equations of the elliptical type so that the mesh forms lines of equal potential [24]. Examples include the use of the harmonic [27] (considered to be very reliable if the deformations are small) or the bi-harmonic [28] (although more costly, it is deemed to be a good alternative, especially when close to reentrant edges) extension, as well as the practice of treating the mesh-deformation problem as a pseudo-elastic one, thus solving the (non)linear elasticity equations to obtain the mesh displacements. The latter is a state-of-the-art method in the field [29]. Algebraic methods, on the other hand, include Transfinite Mapping Interpolation [30] (cheap procedure that does not introduce geometric errors but imposes mesh topology restrictions; only applicable to structured meshes [31]) and other mesh smoothing and interpolation techniques which are very general (applicable to unstructured meshes as well) and simple [24]. STAR-CCM+, used in this dissertation, employs multiquadric interpolation for computing the mesh deformation based on the information extracted at control points [32; 33].

The above-described methods are all related to mesh deformation (or nodal relocation), which requires interface tracking and frequent mesh-updates, but more easily preserves mesh quality near the boundaries [34]. Nevertheless, it is also appropriate to point out, for completeness sake, that, besides the approaches to FSI that involve mesh deformation, other methods exist using fixed grids such as the Immersed Boundary method [35], the Immersed Interface method [36], the Coupled Momentum method [37], Transpiration techniques [38; 39], or methods based on the eXtended Finite Elements with Lagrange multipliers [40]. For a recent review on immersed methods, see [41].

On a brief final note, the concept of the Geometric Conservation Law (GCL) is commonly brought up in relation to the deformation of the mesh. Initially proposed by Thomas and Lombard [42], GCL was, at the time, deemed to be an important condition for numerical schemes on moving meshes as means to assure that the motion of the mesh affects the solution in a minimal way. It imposes the conservation of volume of the domain throughout its evolution so that no part of the control volume is "lost" during the analysis [43]. Despite some further controversy [24; 44] related to its utility, GCL is still mentioned in some literature as having an important role on the setting up of more accurate (and possibly more stable) schemes [43–47], thus it is acknowledged in this review.

2.3 Coupling Techniques

When considering FSI problems, the two main methods used for solving the coupled system of partial differential equations (PDE's) are the monolithic approach and the partitioned (or staggered) approach. In the former, the entire problem is solved at once, with all the domains being advanced in time by one implicit time-stepping scheme. In the latter, each component of the problem is considered individually, with possibly different schemes for each domain, and the information from the solutions of the individual components is exchanged at the interface, in order to ensure coupling.

Monolithic solvers are considered to be more robust [48], and are associated to implicit schemes, thus minimizing stability concerns. These solvers are also advantageous when the system is highly nonlinear [49]. However, there is the significant drawback associated to

the fact that a monolithic solution requires specialized software and there is no available and validated general code for this effect. For this reason, the monolithic methods are mainly considered in the research field [50].

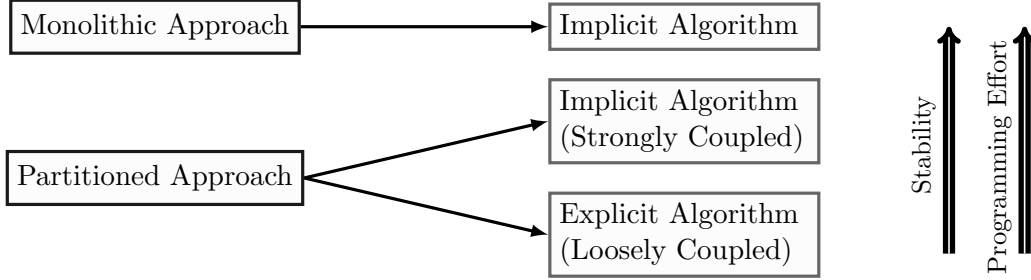


Figure 2.2: Diagram presenting and comparing conventional approaches and algorithms used to address coupled problems. Adapted from [51].

Partitioned approaches, on the other hand, due to their modular nature, allow for different, fully developed solvers to be used for each domain. This is a great advantage in terms of programming effort and the primary reason for the popularity of such methods. Nevertheless, although they are very efficient for weakly-coupled systems, partitioned approaches face severe stability issues, the more so when the systems are strongly coupled. Thus, due to possibly necessary sub-iterations or stabilization procedures, the solution may turn out to be computationally costly [49]. For this reason, it is important to analyse the algorithms that can be employed in solving FSI with partitioned approaches. Figure 2.2 presents a summarized, graphic representation of the distinctions made above.

For illustrating the differences between the algorithms and their characteristics, a general partitioned system is considered [52]

$$\begin{bmatrix} \dot{\mathbf{x}}_1 \\ \dot{\mathbf{x}}_2 \end{bmatrix} = \begin{bmatrix} \phi_1(\mathbf{x}_1, \mathbf{x}_2) \\ \phi_2(\mathbf{x}_2, \mathbf{x}_1) \end{bmatrix}. \quad (2.1)$$

To remain in the context of FSI, $\mathbf{x}_1$ can be seen as a vector containing the variables of interest in the fluid domain, while $\mathbf{x}_2$ stores the variables of interest in the solid domain.

A very simple (and simplified) partitioned approach would consist of

- 1) setting a value for the solid quantities, $\mathbf{x}_2^n$, and fluid quantities, $\mathbf{x}_1^n$, for the current time step, t^n , from initial conditions or from the previous time step;
- 2) advancing the solution of the fluid domain, $\mathbf{x}_1^{n+1}$, using the fluid and solid quantities at the current time step, $\mathbf{x}_1^n$ and $\mathbf{x}_2^n$;
- 3) advancing the solution of the solid domain, $\mathbf{x}_2^{n+1}$, using the solution of the fluid domain, $\mathbf{x}_1^{n+1}$, obtained in 2), and the solid quantities at the current time step, $\mathbf{x}_2^n$;
- 4) repeat steps 1) to 3) until a designated limit for the time step is attained.

This can alternatively be represented by

$$\begin{bmatrix} \mathbf{x}_1^{n+1} \\ \mathbf{x}_2^{n+1} \end{bmatrix} = \begin{bmatrix} \phi_1^d(\mathbf{x}_1^n, \mathbf{x}_2^n) \\ \phi_2^d(\mathbf{x}_2^n, \mathbf{x}_1^{n+1}) \end{bmatrix}. \quad (2.2)$$

From (2.2) it is clear that this algorithm solves the full system in an explicit way. As such, it may be susceptible to stability issues, with limitations on the maximum time step size. Another interesting takeaway is that, even if implicit solvers are used for each of the two domains, as shown in equation (2.3),

$$\begin{bmatrix} \mathbf{x}_1^{n+1} \\ \mathbf{x}_2^{n+1} \end{bmatrix} = \begin{bmatrix} \phi_1^d(\mathbf{x}_1^{n+1}, \mathbf{x}_1^n, \mathbf{x}_2^n) \\ \phi_2^d(\mathbf{x}_2^{n+1}, \mathbf{x}_2^n, \mathbf{x}_1^{n+1}) \end{bmatrix}, \quad (2.3)$$

the resulting solution of the system is still obtained through an explicit scheme, as the quantities of interest in the fluid domain for a given time step are computed considering only the solid quantities at the previous time step [53]. For this reason, in a partitioned approach using an explicit or weakly-coupled algorithm, one of the solvers always lags the other, which makes it difficult for energy conservation at the interface to be satisfied during the exchange of information, thus contributing to the already-mentioned stability concerns [49; 54].

Even though these explicit algorithms may be successfully applied to problems where the coupling is relatively weak (such as some aeroelastic phenomena), there is still the need for more robust algorithms capable of solving systems exhibiting strong coupling. Thus, the implicit or strongly-coupled algorithm is considered.

To solve the coupled system of equations in an implicit way, the term $\mathbf{x}_2^n$ in the computation of the fluid quantities, $\mathbf{x}_1^{n+1}$, in (2.3), needs to be substituted by $\mathbf{x}_2^{n+1}$, yielding

$$\begin{bmatrix} \mathbf{x}_1^{n+1} \\ \mathbf{x}_2^{n+1} \end{bmatrix} = \begin{bmatrix} \phi_1^d(\mathbf{x}_1^{n+1}, \mathbf{x}_1^n, \mathbf{x}_2^{n+1}) \\ \phi_2^d(\mathbf{x}_2^{n+1}, \mathbf{x}_2^n, \mathbf{x}_1^{n+1}) \end{bmatrix}. \quad (2.4)$$

At first, such a scheme seems to be possible only when considering a monolithic approach. However, this strong coupling may also be achieved by performing sub-iterations within each time step, thus allowing for multiple exchanges of information between domains (vide Figure 2.3). This contributes to the convergence of the coupling quantities and to good energy conserving properties at the interface, promoting better stability. It also improves accuracy but at the cost of more computational effort [52], which may even be greater than that of the monolithic approach if a considerable number of sub-iterations is needed, with the additional possibility of convergence not being attained in some cases [28]. To improve on the computational cost of the implicit algorithm, relaxation methods can be employed, such as Aitken Relaxation, which is commonly used in the context of FSI [28; 55–58].

It has been shown how implicit and explicit schemes differ from each other in a partitioned approach to FSI, and it was mentioned that the main motivation for developing an implicit scheme was the stability problems faced by the explicit algorithms when the fluid and structure domains are tightly coupled. A concept that is commonly associated to such instabilities is the so-called added-mass effect. A detailed mathematical analysis of this effect will not be given here (many in-depth works about this exist in the literature; vide [59; 60], for example), but it can be presented as a cause of severe numerical challenges for partitioned schemes with various interesting characteristics, as it depends on physical parameters and not discretization parameters.

In fact, this effect is found when solving FSI problems with incompressible flows and has been shown to be associated to the density ratio between fluid and solid, i.e., with a higher fluid density when compared to the solid density, the effect intensifies [58]. This is the reason why explicit coupling schemes may be successfully applied to aeroelasticity problems, but commonly suffer from instability in the context of naval architecture [61]. Furthermore, the added-mass effect has also been found to depend on the geometry of the

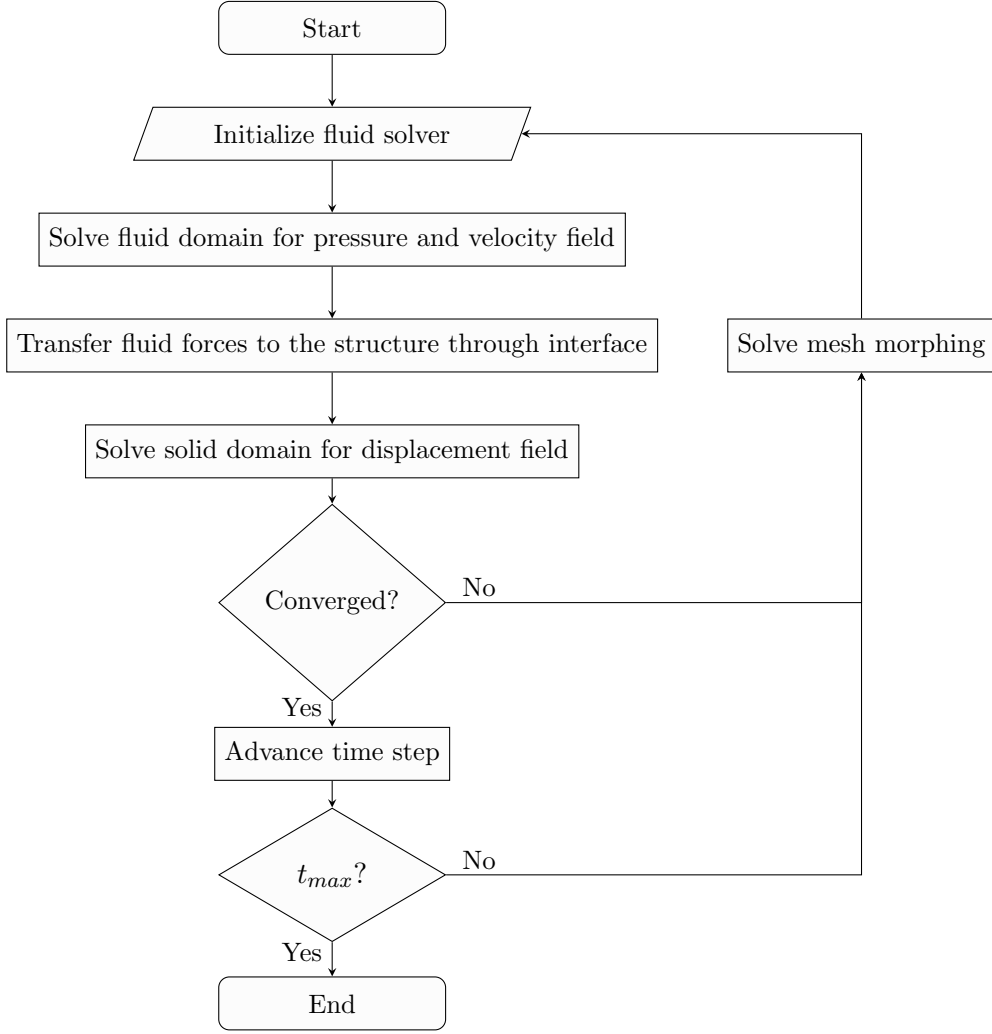


Figure 2.3: Flowchart representing the implicit algorithm for solving FSI systems with a partitioned approach.

problem [60]. Also interesting is the fact that decreasing the size of the time step does not necessarily prevent this instability and, in some cases, it may actually promote it [59].

Due to the added mass-effect, both implicit and explicit schemes may face difficulties, with the worst-case scenario being non-convergence, for the former, and unconditionally unstable behavior for the latter. Moreover, in the case of the implicit schemes, even if convergence is attained, the computational effort may be too costly [57]. For this reason, many new approaches have been proposed. While the improvements on implicit algorithms, using relaxation methods, have already been mentioned, other possibilities exist such as the semi-implicit methods [34; 58; 62]. These methods are based on separating the pressure stresses from the convective-viscous terms and geometrical nonlinearities and performing the coupling differently for each of these two blocks. On one hand, the pressure stresses are coupled implicitly to the structure, which helps maintain stability. On the other hand, the remaining terms are coupled explicitly, thus ensuring a more computationally efficient solution to the FSI problem [63] without notably affecting stability, since these schemes have been shown to be numerically stable in a large range of physical and discrete parameters [64].

2.4 Numerical Hydroelasticity and Applications

One of the first applications of FSI in the context of hydroelasticity is attributed to Miller [65], who calculated the critical velocities associated to static instabilities in reactor fuel-plate assemblies. In his work, simple models were used to obtain analytical solutions, considering beam theory to model the structural response and the Bernoulli's equation to compute the pressure difference caused by the cross flow of the coolant. Since then, other more complex models have been proposed for treating hydroelastic problems, including in the design of marine vessels and structures.

In [66], Parolini and Quarteroni described how a complete description of a racing sailing yacht behaviour should account for hydrodynamics, aerodynamics, interaction with the free surface, fluid-structure interaction and disturbances caused by nearby boats. In these models, the Reynolds Averaged Navier-Stokes (RANS) equations are used to avoid a very costly direct numerical solution (DNS) of the Navier-Stokes equation and, for solving the volume-coupled problem of pressure and velocity fields in incompressible flow, the Semi-Implicit Method for Pressure Linked Equations (SIMPLE) algorithm is applied. However, this description was still considered to be too complex for an efficient integration in the design process at the time. Thus, the different phenomena were isolated before simulation and studied separately with simplified models. The same authors, in [67], further developed the FSI analysis in racing sailboats by coupling the flow solver with a 6-DoF dynamical system representing the motion of the boat. This allowed designers to better understand the dynamics of the boat in both calm and wavy waters. Furthermore, the flow equations were established by implementing an ALE formulation.

With the advent of foiling, a growing number of sources in the literature were starting to address the use of FSI tools in the design process. Especially in competitive applications, the need to replace the traditional rigid-structure Computational Fluid Dynamics (CFD) simulations for more accurate models became increasingly apparent, as a way to obtain better performance predictions [68]. When the new foiling multihulls appeared at the 2013 America's Cup, attention started to turn towards the complications brought forward by the use of lightweight appendages under demanding circumstances. Lothodé *et al.* [69] studied the fluid-structure interaction problem concerning the behaviour of a C-foil of a Groupama 3 under operating conditions, employing a partitioned approach with an implicit algorithm for solving the coupling between fluid and solid domains, with the method devised by Durand [70], which also relies on the ALE formulation. The possibility of dynamic instabilities arising, namely flutter, was also successfully investigated. A very similar procedure was used by Sacher *et al.* [3] for solving the FSI problem involved in the optimization of a flexible hydrofoil compliant with the rules of the 35th America's Cup, thus demonstrating how FSI tools can be used to enhance the performance of such boats (benefits regarding drag and cavitation were attained), rather than just being useful for predicting and avoiding unstable behavior. A comparable, more detailed study was performed by Giovannetti [54]. Prior to that work, others had also developed methods for use in the project of racing yacht appendages considering their flexibility. Balze *et al.* [71], for example, proposed a simple and efficient way of evaluating the response of geometrically complex appendages, on an early stage of the design process, based on a Timoshenko beam model and a three dimensional nonlinear lifting line [72] for computing the hydrodynamic loads. Their approach showed significant advantages in terms of computational cost and time, although at the cost of some accuracy.

Knight *et al.* [73] used STAR-CCM+ with a simple explicit partitioned coupling scheme to investigate the behavior of the daggerboard of the AC45F class catamaran. The dag-

gerboard was modeled as partially submerged and constituted by an isotropic material. It allowed for a study on the influence of material properties such as Young's Modulus, Poisson's ratio and density on the hydrodynamics of the appendage. Even though their final model was not validated, the simulation procedure had been previously compared to the literature using a quasi-2D NACA0012 aerofoil with good results. It was also mentioned that the main interest of such tools would be in their application to composite materials, which was then explored by Temou [50]. Other works followed on the study of partially submerged appendages, with special focus on more complicated phenomena associated to multi-phase flow. Hosseinzadeh and Tabri [74] studied the effects of cavitation on the response of a three-dimensional hydrofoil using STAR-CCM+ and an implicit partitioned coupling algorithm. In their work, several turbulence models were also employed and compared, with the resulting solutions being validated by the experimental data published in [75]. The Volume of Fluid (VOF) method was also applied to solve the transport equation of the vapor volume fraction. Furthermore, the importance of three-dimensional effects on the fluid-structure interaction of hydrofoils was shown to be notable.

2.5 Fluid-Structure Interaction Instabilities

FSI spans a very vast and diversified field of applications, from the modeling of blood vessels to atmospheric re-entry of spacecraft. Throughout the large set of problems that FSI applies to, conditions vary greatly and, although the equations governing the models are roughly identical, the phenomena they describe can be very different from case to case, with distinct effects emerging as dominant.

For these reasons, it is useful to build an understanding of the specific case under study prior to moving on to complex models and predictions. Especially so when studying the instabilities that arise from the coupling of fluid and structure, since these can be of several sorts, as shall be seen. By acquiring these insights, one expects to later have a more focused approach when developing the models. This way, the objective in this Section is not to analytically or numerically solve the system of fluid and structure equations in order to obtain predictions, but rather to physically comprehend the instabilities and develop some kind of mathematical intuition towards the causes. With these purposes in mind, a simplified model is presented, resorting to dimensional analysis and modelling the rudder as a discrete system, closely following the approach proposed by de Langre in [76; 77].

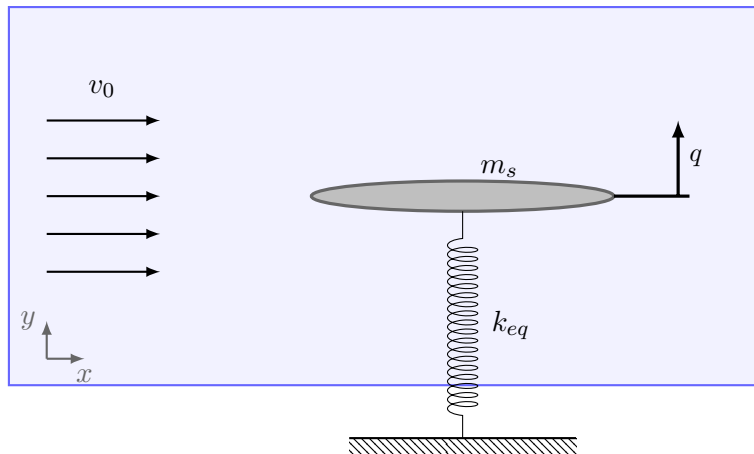


Figure 2.4: Simplified model for the fluid-structure interaction of an immersed hydrofoil sailboat's rudder.

Having mentioned this, the simplified model, represented in Figure 2.4, is described. The original problem pertains to the fluid-structure interaction between the rudder of a Moth sailboat and the water flowing around it. The initially three-dimensional flow is modeled to be two-dimensional around a rigid body with a shape close to an ellipse, as an approximation to the average cross-section of the rudder and ignoring the foil at the tip. The deformation of the rudder is approximated by the motion of this rigid section, which is modeled by a 1-DoF undamped harmonic oscillator with a total mass m_s and stiffness k_{eq} representing the bending stiffness of the rudder in the more compliant direction. The fluid domain is considered to have non-changing boundary conditions (besides those at the interface), i.e., assuming the sailboat does not tack/jibe in the time interval under study.

To start the analysis, the first step is to identify the important parameters playing a role in the descriptive laws. These parameters are, naturally, composed by a mixture of those related to the fluid and those related to the solid:

- t - time;
- $\mathbf{v}$ - velocity field;
- μ - dynamic viscosity;
- L - characteristic length;
- p - pressure;
- ρ_f - fluid density;
- v_0 - characteristic velocity.
- q - generalized displacement;
- k_{eq} - equivalent stiffness;
- m_s - rigid body mass;
- q_0 - characteristic generalized displacement.

Note that there are two variables of interest: the velocity field, $\mathbf{v}$, and the generalized displacement, q ; thus, there is the need to specify two expressions, each one relating one variable of interest to all the other identified parameters. By applying the Buckingham Pi Theorem [78], the number of necessary Pi groups is found to be equal to seven for both relations, which come as

$$\Phi_{fluid,solid}\left(\frac{\mathbf{v}}{v_0}, \frac{v_0 t}{L}, \frac{\rho_f v_0 L}{\mu}, \frac{p}{\rho_f v_0^2}, \frac{q_0}{L}, t\sqrt{\frac{k_{eq}}{m_s}}, \Pi_7\right) = 0, \quad (2.5)$$

$$\Phi_{solid,fluid}\left(\frac{q}{q_0}, \frac{v_0 t}{L}, \frac{\rho_f v_0 L}{\mu}, \frac{p}{\rho_f v_0^2}, \frac{q_0}{L}, t\sqrt{\frac{k_{eq}}{m_s}}, \Pi_7\right) = 0. \quad (2.6)$$

besides the dimensionless form of the velocity field, $\tilde{\mathbf{v}} = \frac{\mathbf{v}}{v_0}$ and time, $\tilde{t} = \frac{v_0 t}{L}$, other well-known dimensionless numbers appear. These are the Reynolds number, $Re = \frac{\rho_f v_0 L}{\mu}$, and the Euler number, $\tilde{p} = \frac{p}{\rho_f v_0^2}$. The first one is often interpreted as the ratio of inertial forces to viscous forces, while the latter is commonly viewed as the ratio of pressure forces to inertial forces. There are also the dimensionless form of the generalized displacement, $\bar{q} = \frac{q}{q_0}$, the displacement number, $D = \frac{q_0}{L}$, and the characteristic time of the solid region, $\bar{t} = t\sqrt{\frac{k_{eq}}{m_s}}$.

It is now possible to develop expressions (2.5) and (2.6) by recalling the basic governing equations for each of the domains. Bellow, the flow equations are written, which follow from momentum and mass conservation and take the form

$$\rho_f \frac{\partial \mathbf{v}}{\partial t} + \rho_f (\mathbf{v} \cdot \nabla) \mathbf{v} = -\nabla p + \mu \nabla^2 \mathbf{v} - \rho_f \mathbf{g} \quad (2.7)$$

$$\nabla \cdot \mathbf{v} = 0. \quad (2.8)$$

These are the incompressible Navier-Stokes equations, constituted by the momentum conservation equations (2.7) and the continuity equation (2.8). Note that, because an incompressible and isothermal flow is assumed, the need to solve the energy equation is eliminated; thus, it is not included here. It is also possible to manipulate these expressions into the dimensionless form, after which they come as

$$\frac{D\tilde{\mathbf{v}}}{D\tilde{t}} = -\nabla \tilde{p} + \frac{1}{Re} \nabla^2 \tilde{\mathbf{v}} \quad (2.9)$$

$$\nabla \cdot \tilde{\mathbf{v}} = 0. \quad (2.10)$$

Furthermore, for the purposes of this Section, the buoyancy term can be removed from the momentum conservation equations. Additionally, in (2.9), the total (or material) derivative notation was used, for the sake of compactness and simplicity.

Regarding the structural domain, the response is here modeled by the motion of a 1-DoF undamped oscillator, which is described by

$$m_s \frac{d^2 q}{dt^2} + k_{eq} q = f. \quad (2.11)$$

Expression (2.11) may be manipulated and written in a dimensionless form. For this, the dimensionless variable for the generalized displacement is introduced and its derivative with respect to the dimensionless time is taken by employing the chain rule, yielding

$$\frac{d^2 \bar{q}}{d\bar{t}^2} + \bar{q} = \frac{f}{q_0 k_{eq}} = \bar{f} \quad (2.12)$$

where $\bar{f}$ is the newly introduced dimensionless generalized force.

Having established the governing equations of each domain, the coupling needs to be addressed. In both expressions (2.5) and (2.6), the seventh Pi group, Π_7 , remains to be defined, and represents the coupling terms. It can be constructed by any dimensionless assembly of quantities that relates both problems (see that, before Π_7 , every other Pi group relates either to solid or to fluid, but never both). Some of the numbers that can be used to illustrate this coupling are the mass number, $M = \frac{\rho L^3}{m_s}$, the reduced velocity, $U_r = \frac{v_0}{L \sqrt{\frac{k_{eq}}{m}}}$, and some form of a pseudo-Cauchy number, $Cy = \frac{\rho v_0^2 L}{k_{eq}}$.

The conditions are now established to write the system of dimensionless equations that couples both fluid and solid problems. This will be comprised of the already seen Navier-Stokes equations and the equation of motion of a 1-DoF undamped harmonic oscillator. Additionally, conditions at the interface between solid and fluid must be defined. There is the kinematic condition, which establishes that the velocities in both domains shall be the same at the interface, coming from the *no-slip* and the *no-penetration* conditions. It can be expressed as

$$\mathbf{v} = \frac{d(\hat{\mathbf{e}}_j q)}{dt} = \hat{\mathbf{e}}_j \frac{dq}{dt}. \quad (2.13)$$

where $\hat{\mathbf{e}}_j$ is the unitary vector of the direction in which the rudder section oscillates.

There is also the dynamic condition, stating that along the interface, Γ , there must be continuity of stresses. Thus, the equilibrium is written as

$$\int_{\Gamma} (\boldsymbol{\sigma}_f \cdot \mathbf{n}) \cdot \hat{\mathbf{e}}_j dS = f \quad (2.14)$$

with the fluid stress tensor being given as $\boldsymbol{\sigma}_f = -p\mathbf{I} + \mu(\nabla \mathbf{v} + \nabla \mathbf{v}^T)$ for incompressible, viscous flow.

To further simplify the model, it is useful to evaluate the dynamics of both fluid and solid problems, i.e., understand how the timescales of the physical phenomena relate to each other. The recently introduced reduced velocity, U_r , is a good measure of this relation between timescales, since it can also be interpreted as the ratio of characteristic periods associated to the solid and the fluid. To obtain a rough estimation of this reduced velocity, one may consider the orders of magnitude of the following quantities to be, in the context of the application under study (see Chapter 5),

$$v_0 \approx 10^1, \quad L \approx 10^{-1}, \quad \omega_n = \sqrt{\frac{k_{eq}}{m}} \approx 10^1 - 10^2,$$

which means that the reduced velocity takes a value of around $U_r = \frac{T_{solid}}{T_{fluid}} \approx 10^0 - 10^1$. This indicates not only that both characteristic timescales are comparable in magnitude, but also that the dynamics associated to the solid domain seem to be a bit slower than those of the fluid domain. Thus, although it is not possible to completely neglect the dynamics of the slower domain, it is possible to approximate the evolution in time of a quantity of interest in the solid domain by a linear function. This is equivalent to saying, for the motion of the rigid body, that the velocity remains approximately constant or the acceleration is negligible, over a certain period. This constitutes the pseudo-static fluid-elasticity approximation.

Bearing this consideration in mind, the full system of dimensionless equations may be written, after substituting the adequate dimensionless quantities, in the form¹

$$\frac{D\tilde{\mathbf{v}}}{D\tilde{t}} = -\nabla \tilde{p} + \frac{1}{Re} \nabla^2 \tilde{\mathbf{v}} \quad (2.15)$$

$$\nabla \cdot \tilde{\mathbf{v}} = 0 \quad (2.16)$$

$$U_r^2 \frac{d^2 \bar{q}}{d\tilde{t}^2} + \bar{q} = \bar{f} \quad (2.17)$$

$$\tilde{\mathbf{v}} = \hat{\mathbf{e}}_j D \frac{d\bar{q}}{d\tilde{t}} \quad (2.18)$$

$$Cy \int_{\Gamma} ([-\tilde{p}\mathbf{I} + \frac{1}{Re}(\nabla \tilde{\mathbf{v}} + \nabla \tilde{\mathbf{v}}^T)] \cdot \mathbf{n}) \cdot \hat{\mathbf{e}}_j dS = D\bar{f}. \quad (2.19)$$

Notice that, because the dynamics of the fluid were identified to be the more relevant ones in the context at study, all the time derivatives in equations (2.15), (2.17) and (2.18) are expressed with respect to the dimensionless characteristic period of the fluid.

As mentioned earlier, there is no interest in solving this system of equations for now, as the objective of this study is merely to provide insights into the physical phenomena at play and the types of instabilities that may arise. Thus, starting with the pair action-reaction

¹Here, equations (2.15) and (2.16) are the same as (2.9) and (2.10), but are repeated for the sake of writing the full system governing the FSI.

at the interface, let $\bar{f}$ be given by a function of the Reynolds number, the displacement and the velocity. The acceleration is disregarded, as previously stated when proposing the pseudo-static fluid-elasticity approximation. This way, a new expression is used to describe the force across the interface

$$\bar{f} = Cy \cdot \phi(Re, \bar{q}, \dot{\bar{q}}). \quad (2.20)$$

where $\dot{\bar{q}}$ denotes the derivative of the dimensionless displacement, $\bar{q}$, with respect to the fluid characteristic time, $\tilde{t}$. Next, a Taylor expansion may be performed around the static equilibrium to yield

$$\bar{f} \approx Cy\bar{f}|_0 + Cy\left(\frac{\partial \bar{f}}{\partial \bar{q}}\right)|_0 \bar{q} + Cy\left(\frac{\partial \bar{f}}{\partial \dot{\bar{q}}}\right)|_0 \dot{\bar{q}} \quad (2.21)$$

with the Reynolds Number assumed to remain mostly constant. Two special terms can be immediately identified: 1) the term where the including the displacement, and 2) the term containing the velocity. Inserting this expansion into equation (2.17) results in

$$U_r^2 \ddot{\bar{q}} + \bar{q} = Cy\bar{f}|_0 + Cy\left(\frac{\partial \bar{f}}{\partial \bar{q}}\right)|_0 \bar{q} + Cy\left(\frac{\partial \bar{f}}{\partial \dot{\bar{q}}}\right)|_0 \dot{\bar{q}} \Leftrightarrow U_r^2 \ddot{\bar{q}} - c_f \dot{\bar{q}} + (1 - k_f) \bar{q} = Cy\bar{f}|_0. \quad (2.22)$$

By looking at equation (2.22) it is possible to conclude that the effect of the flow on the solid can be interpreted as additional stiffness, $k_f = Cy\left(\frac{\partial \bar{f}}{\partial \bar{q}}\right)|_0$, and damping, $c_f = Cy\left(\frac{\partial \bar{f}}{\partial \dot{\bar{q}}}\right)|_0$. It is by studying this flow-induced damping and flow-induced stiffness that the instabilities arising from the developed FSI model may begin to be understood.

First, the effect of the flow-induced stiffness is investigated. It is obvious from (2.22) that there is a critical value for k_f after which the flow-induced stiffness cancels out with the stiffness of the structure and the natural frequency takes a null value, causing the displacement of the structure to increase in an uncontrolled manner. This unstable behaviour is akin to buckling and is commonly known as divergence. This divergence can be static or dynamic, depending on the excitation to which the structure is subjected to and the fluid-solid mass ratio. It is associated with a critical flow velocity, which in this model is "hidden" in the pseudo-Cauchy number. By recalling that Cy is related to the square of the characteristic velocity of the fluid, it is verified that $k_f \propto v_0^2$ and thus the critical velocity will be attained at the instant in which the flow-induced stiffness is equal to the structural stiffness.

Another immediate conclusion from the analysis of equation (2.22) is the fact that if $c_f > 0$ the "damping" of the rigid body harmonic motion will be negative. This means that, instead of dissipating energy, this term is adding energy to the system, which makes it behave unstably, as the oscillations grow in amplitude. However, since the above condition is imposed, a more thorough study is needed to assess the sign of c_f . Since the pseudo-Cauchy number is always positive, the term $\frac{\partial \bar{f}}{\partial \dot{\bar{q}}}$ is evaluated. Considering $\bar{f}$ to be proportional to the lift coefficient, C_L , which varies with the angle of attack, α , the following relation is obtained using the chain rule

$$\frac{\partial \bar{f}}{\partial \dot{\bar{q}}} = \frac{\partial \bar{f}}{\partial C_L} \frac{\partial C_L}{\partial \alpha} \frac{\partial \alpha}{\partial \dot{\bar{q}}} \quad (2.23)$$

As the first term on the right-hand side of (2.23), $\frac{\partial \bar{f}}{\partial C_L}$, is always positive, the sign of $\frac{\partial \bar{f}}{\partial \dot{\bar{q}}}$ is given by the remaining two terms.

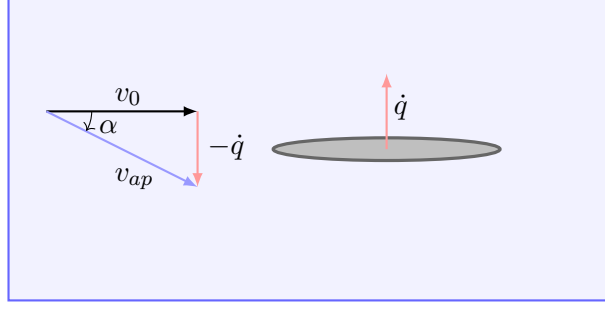


Figure 2.5: Influence of the velocity of the rigid body on the direction of the apparent flow velocity

First, the relation between the angle of attack and the modal velocity is determined. As established before, in this simple model the rigid body only moves in the vertical direction, without altering its angular position. However, because the flow velocity and the foil section velocity are comparable in magnitude, the motion of the rigid section will change the apparent velocity of the flow as "seen" from the perspective of the body. This is illustrated in Figure 2.5, from which the following expression is obtained

$$\tan(\alpha) = -\frac{\dot{q}}{v_0} \Leftrightarrow \alpha = -\arctan\left(\frac{\dot{q}}{v_0}\right). \quad (2.24)$$

And its derivative with respect to $\dot{q}$ comes

$$\frac{\partial \alpha}{\partial \dot{q}} = -\frac{1}{1 + \left(\frac{\dot{q}}{v_0}\right)^2} \quad (2.25)$$

and thus negative for any value of $\dot{q}$. Note that relations (2.24) and (2.25) were obtained using $\dot{q}$, for convenience. However, the results are equivalent, in terms of the sign, to using $\ddot{q}$.

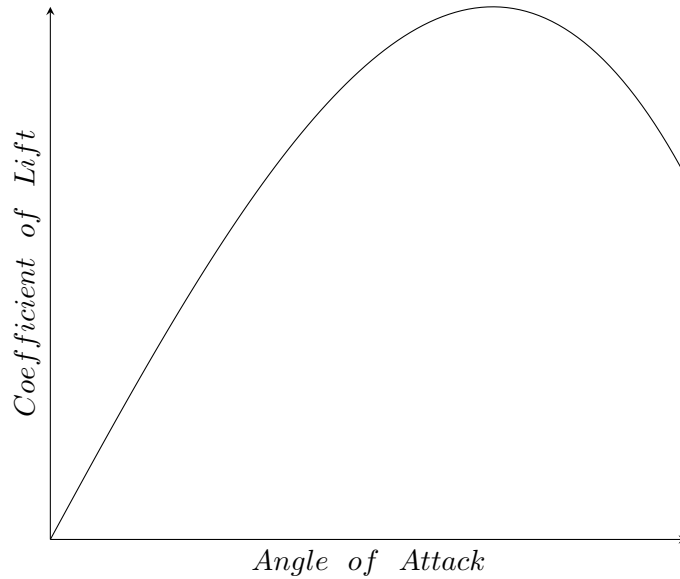


Figure 2.6: General plot of Lift Coefficient vs. Angle of attack for a symmetrical profile based on the results from [79].

At this point, the term $\frac{\partial C_L}{\partial \alpha}$ is the deciding factor. If it is positive, then $\frac{\partial \bar{f}}{\partial \bar{q}}$ is negative and so is c_f . In this case, there is no unstable behavior. However, if it is negative, this results in $\frac{\partial \bar{f}}{\partial \bar{q}} > 0$ and, consequently, $c_f > 0$, which would result in the instability of the system. To understand the sign of $\frac{\partial C_L}{\partial \alpha}$ Figure 2.6, representing the evolution of the lift coefficient with the angle of attack for a symmetrical profile foil, is analysed.

The plot shows that, initially, the coefficient of lift increases almost linearly with an increase of the angle of attack. However, there is a critical point where the coefficient of lift suddenly decreases. This phenomenon, which is known as stall, is due to flow separating from the surface, drastically reducing the lift. At this critical value of the angle of attack, the derivative $\frac{\partial C_L}{\partial \alpha}$ turns negative and thus, as was seen, instability will ensue. Since its occurrence is closely related to the stall phenomenon, this behavior is known as stall flutter.

These are already some interesting insights, but more conclusions can be taken from this fluid-solid coupling if some changes are made to the model. Namely, by considering the the motion of the rigid section to be described by two degrees of freedom, instead of just one. In this case, besides the dimensionless vertical displacement, $\bar{q}$, the dimensionless angular displacement, $\bar{\theta} = \frac{\theta}{\theta_0}$, would also be considered, with θ_0 being some characteristic angular displacement. The force across the interface can then be approximated by

$$\bar{\mathbf{f}} = \begin{bmatrix} Cy \cdot \phi_1(Re, \bar{q}_1, \bar{\theta}, \dot{\bar{q}}_1, \dot{\bar{\theta}}) \\ Cy \cdot \phi_2(Re, \bar{q}_1, \bar{\theta}, \dot{\bar{q}}_1, \dot{\bar{\theta}}) \end{bmatrix}. \quad (2.26)$$

A new Taylor series expansion around static equilibrium yields

$$\bar{\mathbf{f}} \approx \begin{bmatrix} Cy \bar{f}_1|_0 + Cy(\frac{\partial \bar{f}_1}{\partial \bar{q}_1}|_0) \bar{q}_1 + Cy(\frac{\partial \bar{f}_1}{\partial \bar{\theta}}|_0) \bar{\theta} + Cy(\frac{\partial \bar{f}_1}{\partial \dot{\bar{q}}_1}|_0) \dot{\bar{q}}_1 + Cy(\frac{\partial \bar{f}_1}{\partial \dot{\bar{\theta}}}|_0) \dot{\bar{\theta}} \\ Cy \bar{f}_2|_0 + Cy(\frac{\partial \bar{f}_2}{\partial \bar{q}_1}|_0) \bar{q}_1 + Cy(\frac{\partial \bar{f}_2}{\partial \bar{\theta}}|_0) \bar{\theta} + Cy(\frac{\partial \bar{f}_2}{\partial \dot{\bar{q}}_1}|_0) \dot{\bar{q}}_1 + Cy(\frac{\partial \bar{f}_2}{\partial \dot{\bar{\theta}}}|_0) \dot{\bar{\theta}} \end{bmatrix} \quad (2.27)$$

and, by rearranging the expression above, it is possible to write

$$\bar{\mathbf{f}} \approx \begin{bmatrix} Cy \bar{f}_1|_0 \\ Cy \bar{f}_2|_0 \end{bmatrix} + \begin{bmatrix} Cy(\frac{\partial \bar{f}_1}{\partial \bar{q}_1}|_0) & Cy(\frac{\partial \bar{f}_1}{\partial \bar{\theta}}|_0) \\ Cy(\frac{\partial \bar{f}_2}{\partial \bar{q}_1}|_0) & Cy(\frac{\partial \bar{f}_2}{\partial \bar{\theta}}|_0) \end{bmatrix} \begin{bmatrix} \bar{q}_1 \\ \bar{\theta} \end{bmatrix} + \begin{bmatrix} Cy(\frac{\partial \bar{f}_1}{\partial \dot{\bar{q}}_1}|_0) & Cy(\frac{\partial \bar{f}_1}{\partial \dot{\bar{\theta}}}|_0) \\ Cy(\frac{\partial \bar{f}_2}{\partial \dot{\bar{q}}_1}|_0) & Cy(\frac{\partial \bar{f}_2}{\partial \dot{\bar{\theta}}}|_0) \end{bmatrix} \begin{bmatrix} \dot{\bar{q}}_1 \\ \dot{\bar{\theta}} \end{bmatrix} = \bar{\mathbf{f}}_s + \mathbf{K}_f \bar{\mathbf{q}} + \mathbf{C}_f \dot{\bar{\mathbf{q}}}. \quad (2.28)$$

The expression written for the modal force in (2.28) can now be plugged into the equation of motion of a 2-DoF undamped harmonic oscillator, resulting in

$$U_r^2 \ddot{\bar{\mathbf{q}}} + \bar{\mathbf{q}} = \bar{\mathbf{f}}_s + \mathbf{K}_f \bar{\mathbf{q}} + \mathbf{C}_f \dot{\bar{\mathbf{q}}} \Leftrightarrow U_r^2 \ddot{\bar{\mathbf{q}}} - \mathbf{C}_f \dot{\bar{\mathbf{q}}} + (\mathbf{I} - \mathbf{K}_f) \bar{\mathbf{q}} = \bar{\mathbf{f}}_s. \quad (2.29)$$

Once again, there is a possibly negative "damping" term. It was already seen, for the simpler 1-DoF case, that this shall give rise to unstable behaviour. Right now, the focus is placed on the effects of the coupling between the two DoF induced by the flow, so the "damping" term may be discarded for simplicity. Besides, to better grasp the physical meaning behind the next operations, equations (2.29) are reverted back to their original form, instead of their dimensionless form. This changes the equations to

$$\mathbf{M} \ddot{\mathbf{q}} + (\mathbf{K}_{eq} - \mathbf{K}'_f) \mathbf{q} = \mathbf{f}_s \quad (2.30)$$

where $\mathbf{M}$ is a diagonal matrix containing the masses associated to each DoF, $\mathbf{K}_{eq}$ is a diagonal matrix composed of their respective stiffnesses, and $\mathbf{K}'_f$ is the new form of the flow-induced stiffness. Furthermore, all the time derivatives are now expressed with

respect to "normal" time, t . Separating the diagonal and the off-diagonal terms in $\mathbf{K}'_{\mathbf{f}}$ and neglecting the force exerted by the flow at the static equilibrium, $\mathbf{f}_s$, for simplicity, yields

$$\mathbf{M}\ddot{\mathbf{q}} + (\mathbf{K}_{\mathbf{eq}} - \mathbf{K}'_{\mathbf{f}|diag})\mathbf{q} = \mathbf{K}'_{\mathbf{f}|off-diag}\mathbf{q}. \quad (2.31)$$

By pre-multiplying each term in (2.31) by the inverse of the mass matrix, the following expression is obtained

$$\ddot{\mathbf{q}} + \mathbf{M}^{-1}(\mathbf{K}_{\mathbf{eq}} - \mathbf{K}'_{\mathbf{f}|diag})\mathbf{q} = \mathbf{M}^{-1}\mathbf{K}'_{\mathbf{f}|off-diag}\mathbf{q} \Leftrightarrow \ddot{\mathbf{q}} + \begin{bmatrix} \omega_1'^2 & 0 \\ 0 & \omega_2'^2 \end{bmatrix} \mathbf{q} = \mathbf{\Lambda}\mathbf{q} \quad (2.32)$$

where $\mathbf{\Lambda}$ is the transformed term corresponding to $\mathbf{K}'_{\mathbf{f}|off-diag}$ after the rearrangement of (2.31). It is now clear that, similarly to what was seen for the 1-DoF case, the flow-induced stiffness alters the stiffness properties of the system, thus changing the natural frequency associated to each DoF. With the values of the two natural frequencies changing, they either diverge from each other or converge into a point where both "wet" natural frequencies are identical. For now and to analyse the possibility of unstable behavior, it is assumed that this is the effect of the $\mathbf{K}'_{\mathbf{f}|diag}$ term on the original natural frequencies of the system, i.e., they evolve to a common value, $\omega_1'^2 = \omega_2'^2 = \bar{\omega}$. With this assumption, (2.32) is re-written as

$$\ddot{\mathbf{q}} + \bar{\omega}^2\mathbf{q} = \mathbf{\Lambda}\mathbf{q} = \begin{bmatrix} 0 & \lambda_1 \\ \lambda_2 & 0 \end{bmatrix} \mathbf{q}. \quad (2.33)$$

But, in order to really determine the new characteristics of the system after the coupling between the two DoF, one has to consider the effect of off-diagonal terms in the natural frequencies as well. With this purpose, the following eigenproblem is considered

$$\begin{bmatrix} \bar{\omega}^2 - \omega^2 & -\lambda_1 \\ -\lambda_2 & \bar{\omega}^2 - \omega^2 \end{bmatrix} \mathbf{q} = 0 \quad (2.34)$$

giving the solution

$$\omega^2 = \bar{\omega}^2 \pm \sqrt{\lambda_1\lambda_2}. \quad (2.35)$$

From (2.35), two possibilities arise: 1) $\lambda_1\lambda_2 > 0$, thus both solutions are purely real, they do not coincide and both modes are stable; 2) $\lambda_1\lambda_2 < 0$, which means that the solutions will have an imaginary part (for one, it is negative, while for the other, it is positive) and the real part is common to both. For the latter case, if these are substituted in the harmonic term of the response of the oscillator, it yields

$$e^{i(\text{Re}(\omega) + i\text{Im}(\omega))t} = e^{i\text{Re}(\omega)t} e^{-\text{Im}(\omega)t}. \quad (2.36)$$

Through equation (2.36) it is obvious that, due to the imaginary part of one of the solutions being negative, the oscillatory motion of the mode it is associated to will grow exponentially and unstable behavior will occur. On the other hand, the stable mode (whose exponent has a positive imaginary part) will have its amplitude damped. The unstable phenomenon here described is called coupled-mode flutter and shows, in a simplified way, how the coupling between structural modes enable the extraction of energy from the fluid flow, bringing the system to a critical point.

Coupled-mode flutter and divergence are regarded to be the most important instabilities affecting hydrofoils and similar appendages. Depending on the characteristics of the

specific FSI, such as mass ratio between fluid and solid, structure geometry and mass distribution, one phenomenon may be more important to predict than the other. For many applications of hydroelasticity, static divergence is generally seen to have a lower critical velocity [80; 81]. However, to ensure a wider range of applicability, the objective is to include, in the proposed methodology, the ability to predict both instabilities. Furthermore, it is known that the flutter critical velocity can be inadvertently made lower than that of divergence with the addition of heavy masses at the tip of boat appendages [82]. This effect can be seen in canting keels, for example [83].

2.6 Vortex-Induced Vibrations

In the previous Section, the coupling between fluid and solid was outlined and several types of instability were shown to arise from this coupling under the conditions associated to the case study. Vortex-induced vibrations constitute a slightly different phenomenon, in the sense that they need not be "extracted" from the equations governing the fluid-solid interaction, but can rather be seen as a consequence of periodic irregularities on the flow caused by the presence of the structure. This periodic irregularities may be adequately modeled as an external harmonic excitation acting on the structure [84]

$$F_{vortex} = \frac{1}{2} \rho v_0^2 C_L A \sin(2\pi f_v t) \quad (2.37)$$

where C_L is the lift coefficient, A is the cross-sectional area perpendicular to the flow velocity and f_v is the vortex shedding frequency. This excitation force is plugged into the equation of motion of a 1-DoF undamped harmonic oscillator to yield

$$m_s \ddot{q} + k_{eq} q = \frac{1}{2} \rho v_0^2 C_L A \sin(2\pi f_v t). \quad (2.38)$$

It is clear from (2.38) that if the vortex shedding frequency, $\omega_v = 2\pi f_v$, happens to coincide with the natural frequency of the system, $\omega_n = \sqrt{\frac{k_{eq}}{m_s}}$, the system experiences resonance and the magnitude of the oscillation amplitude will reach a maximum value. This vortex shedding frequency can be computed from the Strouhal number, $St = \frac{f_v L}{v_0}$, which in turn is a function of the geometry of the immersed structure and the Reynolds number of the flow. Because the amplitude of the oscillations around the natural frequency is large, it is expected that the motion of the body will further disturb the fluid flow around it, thus influencing the vortex shedding and, consequently, the excitation of the structure itself. The vortex-induced is then anticipated to evolve in a periodic fashion in time according to some law of the type

$$B_1 \ddot{F}_{vortex} + B_2 F_{vortex} = B_3 \Upsilon \quad (2.39)$$

where B_1 , B_2 and B_3 are constants and Υ is the response of the structure: either $\ddot{q}$, $\dot{q}$ or q , as authors diverge in opinion [85]. Here, $\ddot{q}$ will be considered. Establishing an analogy between equation (2.38) and equation (2.39), it is obvious that B_2/B_1 will be given by the square of some kind of natural frequency. This is, of course, the vortex shedding frequency, ω_v . Thus, equation (2.39) is rewritten as

$$\ddot{F}_{vortex} + \omega_v^2 F_{vortex} = B'_3 \ddot{q} \quad (2.40)$$

Assuming a constant Strouhal number [86], one can conclude that as the flow velocity increases, the shedding of the vortices will approach the resonant frequency of the struc-

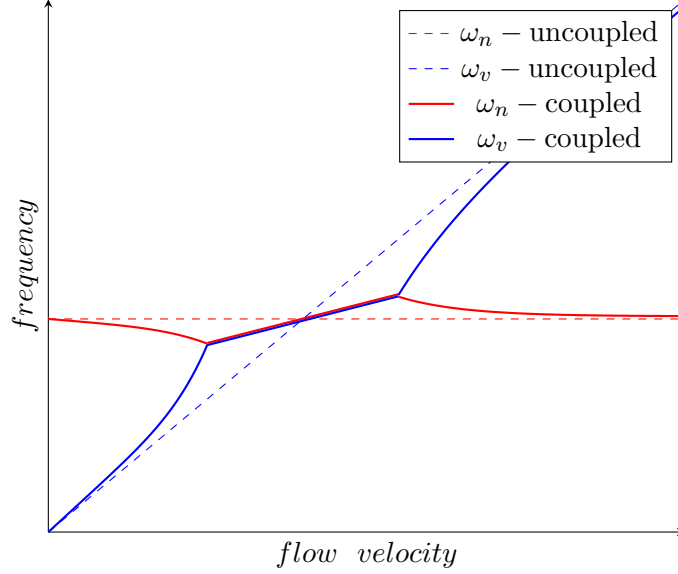


Figure 2.7: Lock-in effect.

ture. However, because the fluid-solid system of equations is now coupled and takes the form

$$\begin{bmatrix} m_s & 0 \\ -B'_3 & 1 \end{bmatrix} \begin{bmatrix} \ddot{q} \\ \ddot{F}_{vortex} \end{bmatrix} + \begin{bmatrix} k_{eq} & 1 \\ 0 & \omega_v^2 \end{bmatrix} \begin{bmatrix} q \\ F_{vortex} \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \end{bmatrix} \quad (2.41)$$

the phenomenon is more complex than initially stated; i.e., the fluid mode and the solid mode interact with each other and, through the coupling terms, modify the modal frequencies of the system, akin to what was seen in the coupled-mode flutter analysis. As a consequence, in the frequency range neighboring the natural frequency of the structure, the two frequencies will merge into the same value [77; 85]. This effect, illustrated in Figure 2.7, promotes resonant behaviour before and after the actual resonant frequency of the structure and is commonly known as lock-in.

2.7 Chapter Recap

In this Chapter, a general overview of the most important concepts associated to computational FSI was given. Regarding the possible formulations, SPH, DSD/SST and ALE were mentioned, besides fixed grid methods. The ALE is deemed to be the more relevant, given its popularity and its development as a tool for commercial software. Even though ALE is here already discussed in more detail than the other methods, further comments on the ALE formulation are to be made in Chapter 3, where the full mathematical model for the FSI is established.

Having addressed this, the ways of solving the resulting system of equations are examined. A distinction is made between partitioned and monolithic approaches, and the emphasis is put on the partitioned schemes. The advantages and disadvantages of implicit and explicit algorithms are highlighted, after which the more sophisticated semi-implicit algorithms are introduced. Special attention is also given to the added-mass effect as one of the main numerical challenges in computational FSI.

Applications in the literature utilizing these concepts are enunciated and briefly described, with particular focus on the study and simulation of the hydroelastic behaviour

of appendages in high-speed sailboats.

Finally, although several oversimplifying assumptions, such as pseudo-static fluid-elasticity, were employed, the use of a simple FSI model allowed for a clear identification of the possible unstable behavior occurring in the case studied, as well as a relatively thorough description of the mechanisms behind such instabilities. The possible unstable behaviour for a high-speed sailing boat appendage was found to be related to divergence, stall flutter, coupled-mode flutter, and vortex-induced resonance with lock-in, which is in agreement with existing literature [80; 87; 88]. Nevertheless, the focus is placed hereafter on divergence and coupled-mode flutter, as these were seen to be the most relevant instabilities in the context of this dissertation.

Mathematical Model

3.1 Introduction

As a multiphysics phenomenon, the governing equations of a general FSI problem are established considering the different nature of both domains, solid and fluid, schematically shown in Figure 3.1. Furthermore, knowing the importance of a correct coupling between these domains in the context of FSI, the interface needs to be defined with the prescription of the proper conditions, allowing the interchange of information.

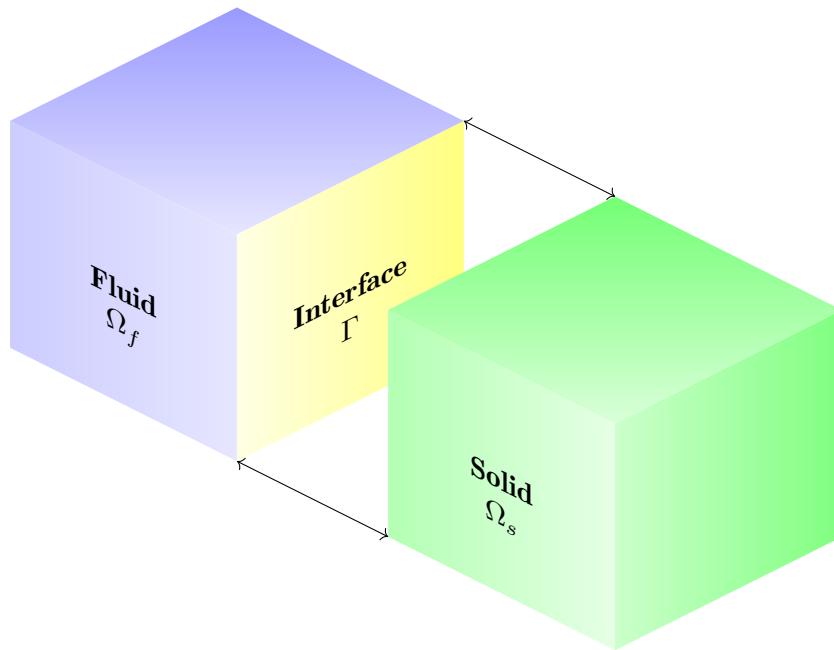


Figure 3.1: Schematic representation of a general FSI problem.

With this in mind, the present work aims at establishing the equations governing the FSI problem. Firstly, in Section 3.2, the Navier-Stokes equations are written and their ALE form is obtained, to address the compatibility issue between the Lagrangian and Eulerian descriptions. Then, in Section 3.3, the linear momentum balance is described, accounting for large deformations that can occur if unstable behaviour is found to occur in the case studies. In Section 3.4, the conditions at the interface are defined to ensure that both sub-problems are appropriately coupled. Finally, in Section 3.5, the matter of the multiphase flow is addressed and the modelling of the free surface is discussed.

3.2 Fluid domain

The fundamental governing equations for fluid dynamics are the Navier-Stokes equations, which are generally constituted by the continuity equation, the momentum equations and the energy equation. However, since the flow is assumed to be isothermal and incompressible, there is no need to consider the energy equation. Furthermore, the fluid can be assumed to be Newtonian. Thus, the Navier-Stokes equations come written as

$$\nabla \cdot \mathbf{v} = 0 \quad (3.1)$$

$$\rho_f \frac{\partial \mathbf{v}}{\partial t} + \rho_f (\mathbf{v} \cdot \nabla) \mathbf{v} = -\nabla p + \mu \nabla^2 \mathbf{v} - \rho_f \mathbf{g}. \quad (3.2)$$

These are established in an Eulerian framework, which was previously shown to be inadequate for the purposes of coupling the fluid equations to the solid equations, expressed in a Lagrangian framework. To solve this incompatibility, the ALE form of the Navier-Stokes equations is developed [89].

In continuum mechanics, it is usual to consider two basic configurations for the description of the kinematics: the material, R_X , and the spatial configuration, R_x . The material coordinates, $\mathbf{X}$, can be associated with the Lagrangian description, while the spatial coordinates, $\mathbf{x}$, are linked to the Eulerian description. To develop an Arbitrary Lagrangian-Eulerian description there is a need to define a third ("arbitrary") domain, R_χ , which is to be established as the reference. For this reason, it is called the referential configuration hereafter.

Since the referential coordinates, χ , are not attached to material particles (as would happen in the Lagrangian description), some convective terms will appear in the time derivatives. Recall that this equally occurs for the Eulerian description, since the spatial coordinates are independent from the material particles. Thus, for a given vectorial quantity, $\mathbf{o}$, the following relation is established

$$\frac{D\mathbf{o}}{Dt} = \frac{\partial \mathbf{o}}{\partial t} + (\mathbf{c} \cdot \nabla) \mathbf{o}. \quad (3.3)$$

In equation (3.3), on the left-hand side, is the material or substantial derivative, which describes the change of $\mathbf{o}$ with time for a given material particle. On the right-hand side, there is the local derivative, $\frac{\partial \mathbf{o}}{\partial t}$, which translates the change of $\mathbf{o}$ with time in a given spatial location, and there is the convective derivative, $(\mathbf{c} \cdot \nabla) \mathbf{o}$, which accounts for the change in $\mathbf{o}$ with time in some location due to the motion of the material particles from one position to another. In this last term, $\mathbf{c}$ is the convective velocity, representing the rate at which these particles are transported.

The concept of the substantial derivative is straightforward and was used to derive the momentum equation. Note that, in equation (3.2), on the left-hand side, both the local and the convective derivative for the flow velocity, $\mathbf{v}$, are present. Note also that, since equation (3.2) is written using an Eulerian description, the convective velocity is set equal to the flow velocity, $\mathbf{c} = \mathbf{v}$. This is because the Eulerian frame is completely static, and all convective effects are merely a result of the fluid motion.

For the ALE, however, this is not the case, as there is a need to account for the motion of the frame itself (which is neither static, as the Eulerian, nor accompanies the motion of the material particles, as the Lagrangian). To correct this, it is enough to simply consider the frame velocity, $\hat{\mathbf{v}}$, in the convective velocity, which comes

$$\mathbf{c} = \mathbf{v} - \hat{\mathbf{v}}. \quad (3.4)$$

Thus, by correcting the convective term in the momentum equation (3.2), the ALE form of the Navier-Stokes equations for isothermal and incompressible flow are obtained in non-conservative form as

$$\nabla \cdot \mathbf{v} = 0 \quad (3.5)$$

$$\rho_f \frac{\partial \mathbf{v}}{\partial t} + \rho_f ((\mathbf{v} - \hat{\mathbf{v}}) \cdot \nabla) \mathbf{v} = -\nabla p + \mu \nabla^2 \mathbf{v} - \rho_f \mathbf{g}. \quad (3.6)$$

After discretization, $\hat{\mathbf{v}}$ can be seen as the mesh velocity, which will be dependent on the solution of the FSI system at each time step. Thus, it will need to be calculated as the solution progresses. Some methods available for determining the motion of the mesh were already discussed in previous sections.

With the Navier-Stokes equations defined in the ALE form, it is possible to couple the fluid and the solid governing equations and solve the FSI system. Nevertheless, such a direct numerical simulation would be extremely costly, given the vastly different scales involved in the turbulent phenomena [90]. For this reason, it is useful to resort to the RANS equations or the Unsteady Reynolds Averaged Navier-Stokes (URANS) equations. These are derived from the Navier-Stokes equations by separating the flow velocity field into mean and fluctuating components. The same decomposition is also applied to the pressure field. The drawback of such procedure is that it introduces new nonlinear terms in the governing equations known as Reynolds stresses. This leads to a closure problem that can be solved by employing turbulence models. The most commonly used models, which are those based on the Eddy Viscosity Hypothesis [91], allow for the approximate calculation of the Reynolds stresses as functions of the mean quantities of the fluid flow problem. These include $k - \epsilon$ [92], $k - \omega$ [93], $k - \omega$ SST [94], and Spalart-Allmaras model [95]. In this work, the $k - \omega$ SST model is selected since it performs well in the presence of adverse pressure gradient and flow separation, which are expected for the cases at study. Furthermore, it is commonly used in similar applications [50; 54; 74; 87; 96–98].

3.3 Solid domain

To characterize the response of the structure to a given perturbation, the linear momentum balance is written as [28]

$$\nabla \cdot \mathbf{P} + \rho_s \mathbf{b} = \rho_s \ddot{\mathbf{u}} \quad (3.7)$$

where $\mathbf{P}$ is the first Piola-Kirchhoff (FPK) stress tensor, ρ_s is the density of the solid, $\mathbf{b}$ are the volumetric forces acting on the body and $\mathbf{u}$ is the displacement field. The FPK stress tensor is related to the more common Cauchy stress tensor, $\boldsymbol{\sigma}$, through the expression

$$\mathbf{P} = J \boldsymbol{\sigma} \mathbf{F}^{-T} \quad (3.8)$$

where $\mathbf{F}$ is the deformation gradient tensor, given by

$$\mathbf{F} = \mathbf{I} + \nabla \mathbf{u} \quad (3.9)$$

and J is its Jacobian determinant. In equation (3.9) the identity matrix is denoted by $\mathbf{I}$.

Introducing the second Piola-Kirchhoff (SPK) stress tensor, $\mathbf{S}$, the constitutive relations can be written in terms of the Lamé constants, λ_L and μ_L , and the Green strain tensor, $\mathbf{E}$. In the context of this work, a linear elastic material law with the assumption

of isotropic behaviour is appropriate, given the materials used in the case studies. Thus, it comes as

$$\mathbf{S} = \lambda_L \text{tr}(\mathbf{E})\mathbf{I} + 2\mu_L \mathbf{E}. \quad (3.10)$$

The relation between the FPK and SPK stress tensors is then established as

$$\mathbf{P} = \mathbf{F}\mathbf{S}. \quad (3.11)$$

Concluding the description of the structure dynamics, the kinematic relations are given by

$$\mathbf{E} = \frac{1}{2}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T + \nabla \mathbf{u} \cdot (\nabla \mathbf{u})^T) \quad (3.12)$$

relating the Green strain tensor to the displacement field. Here, the strain tensor is not linearized so that geometrical nonlinearities arising from large deformations are considered. This same reason motivated the use of the FPK and SPK stress tensors. As unstable behaviour caused by the interaction with the fluid may lead to a drastic response by the structure, it is important to capture these large deformations and nonlinear effects. Limit Cycle oscillations [99] can be pointed out as illustrative of this importance.

3.4 Conditions at the interface

Being a surface-coupled problem, FSI involves an interface that clearly separates the different physical domains. When modelling these problems, it is essential to ensure that the relevant information can flow between such domains, as to impose the desired coupling. This can be done through the setup of specific conditions at the interface, compliant with the physical requirements of the system. In this case, these are the *Kinematic Condition*, stating that both fluid and solid velocities are identical at the interface; and the *Dynamic Condition*, which requires the stress equilibrium to be satisfied at the interface. Moreover, describing FSI with a partitioned-approach thinking, the interface conditions can be attributed to each of the separated problems. This is, to solve the structure problem both the fluid force (which is transmitted through the dynamic condition) and the boundary conditions are needed. Its solution provides the displacement field. This is then used, through the kinematic condition, together with the fluid boundary conditions, to solve the fluid equations and obtain the flow pressure and velocity fields. Again this solution is fed to the structure solver. The repeating cycle is represented in Figure 3.2.

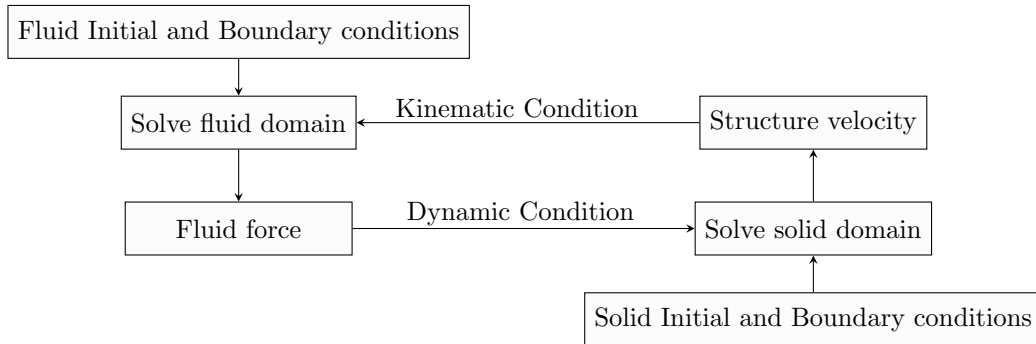


Figure 3.2: Representation of the information exchange between solid and fluid domain through the interface conditions.

Having loosely characterized the coupling conditions and highlighted their importance, a more precise mathematical description is in order. As the fluid equations were previously written in ALE form, it is possible to directly establish the conditions at the interface. Thus, considering a no-slip interface, the kinematic condition can be represented by

$$\mathbf{v} = \dot{\mathbf{u}} \quad (3.13)$$

while the dynamic condition is reduced to

$$(-p\mathbf{I} + \mu(\nabla\mathbf{v} + (\nabla\mathbf{v})^T)) \cdot \mathbf{n}_\Gamma = (J^{-1}\mathbf{P}\mathbf{F}^T) \cdot \mathbf{n}_\Gamma \quad (3.14)$$

where $\mathbf{n}_\Gamma$ is the normal vector on the interface.

3.5 Multiphase Flow

From sedimentation and smoke formation to free surfaces and bubbly flows, there are several physical phenomena that fall into the category of multiphase flows. This class designates the flows where two or more phases are present. It can also denote flows where different materials are interacting, both in the same phase but with different chemical and physical properties.

Attending to the wide scope of multiphase flow problems, it is useful to further categorize them according to the scale of the interface between phases (see Figure 3.3). On one hand, there are *dispersed flows*, which are characterized by the existence of particles, bubbles or droplets of finite dimensions dispersed (i.e., discontinuously distributed) amidst a continuous phase. On the other hand, the flows consisting of multiple immiscible streams of continuous phases are labeled as *stratified* or *separated flows* [100].

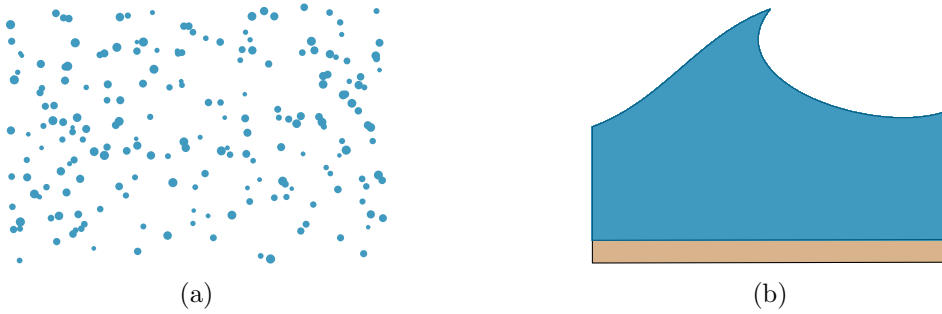


Figure 3.3: Examples of multiphase flows: (a) droplets of water suspended in air (dispersed flow), (b) wavy sea surface (stratified flow).

Although it is very common that both regimes occur simultaneously, this distinction remains important because it constrains the selection of the model used to describe the multiphase flow.

For example, dispersed flows are often solved by resorting to *trajectory models*, this is, modelling the dispersed phase as particles whose motion is determined through their momentum balance, corresponding to a Lagrangian approach. Another possibility is to use *two-fluid models*, which assume the dispersed phase to behave as a second continuous phase whose properties are defined through some averaging process, and constitute an Eulerian approach. These models are complex, given the necessity to establish a good equivalence between the dispersed phase and the modelled continuous phase, in the context of the problem, as well as the concerns that arise when setting up the boundary conditions [100].

On contrary, stratified flows can be solved in simpler ways. In principle, for the case of free surfaces, similar interface conditions to those seen for the FSI problem (kinematic and dynamic conditions) could be employed [100], but other slightly more intricate methods exist, such as the VOF method which has been successfully applied to these cases and is widely used to capture free surface interfaces. This method constitutes an Eulerian or *interface-capturing* approach, which is ideal for modeling the free surface in the present work. Note that Lagrangian methods (also known as *interface-tracking methods*, in this context) are not able to withstand the changes in topology of the domain caused by the breaking of the waves after contact with the rudder, and are mainly used for modelling the motion of small amplitude waves [101].

Attending to the physical problem at hand (the hydroelastic behaviour of a surface-piercing boat appendage), both the stratified and dispersed flow regimes occur, if a complete description of the phenomena is to be considered. While cavitation falls into the dispersed flow class, the free surface that interacts with the rudder constitutes a clear case of a stratified flow. In this work, however, only the latter is considered. The assumption is that no cavitation is expected to occur, as targeted by naval architects [3]. This allows for the already complex model to remain as simple and efficient as possible, without greatly impacting accuracy. With this considerations in mind, the VOF method is chosen to solve for the free surface, with the additional advantage of also being able to solve for ventilation, expected in some degree considering that the strut of the rudder is piercing the free surface.

3.6 Chapter Recap

In this Chapter, the mathematical model of the FSI problem under study was described. Regarding the fluid domain, the incompressibility and constant temperature assumptions were considered, besides the Newtonian nature of the fluid. Furthermore, the RANS equation were selected as the most efficient approach to solve the Navier-Stokes equation with the $k - \omega$ SST model as an adequate turbulence closure model. In the solid domain, the linear momentum balance was established, with the solid material assumed to exhibit isotropic, linear elastic behaviour. Moreover, the requirement of capturing large deformations with the structural model was satisfied. Furthermore, the coupling conditions were set, ensuring kinematic and dynamic equilibrium at the interface. Regarding the presence of a free surface, a more conceptual description was made. The free surface effects were isolated from cavitation and the VOF method was selected as the more useful approach for solving this type of stratified multiphase flow is described. Further information about this method is presented on Chapter 4.

Numerical Methodology

4.1 Introduction

Having established the mathematical model, it is important to formulate a numerical methodology that enables the computation of a solution, hence, Computational Continuum Mechanics (CCM) is discussed.

Given that CCM is a vast area with multiple intricacies, there is no pretension to elaborate an exhaustive description of the field in this subsection. However, an overview of the Finite Volume (FVM) and Finite Element (FEM) methods is essential and should be detailed enough to provide a good foundation on which to support the several decisions taken throughout the setup of this methodology. Furthermore, the coupling between structure and fluid, in the context of STAR-CCM+, is also addressed, considering the fundamental concepts that were discussed in previous sections and the particularities of the software.

4.2 Finite Element Method

To arrive at a Finite Element Method discretization, recall the Principle of Virtual Work (PVW)

$$\delta W_e + \delta W_i + \delta W_j = 0 \quad (4.1)$$

which states that the sum of the virtual work done by external forces, δW_e , by the internal forces, δW_i , and by the inertial forces, δW_j , at a given instant is equal to zero for any virtual displacement, $\delta \mathbf{u}$, of the system [84].

The external virtual work can be defined by considering the virtual work done by volume forces, $\mathbf{f}^B$, and surface tractions, $\mathbf{f}^S$, which are the only loads expected for the current application, as

$$\delta W_e = \int_{\Omega} \delta \mathbf{u}^T \mathbf{f}^B d\Omega + \int_{\Gamma} \delta \mathbf{u}^T \mathbf{f}^S d\Gamma \quad (4.2)$$

and the internal virtual work comes as

$$\delta W_i = - \int_{\Omega} \delta \mathbf{E}^T \mathbf{S} d\Omega \quad (4.3)$$

Finally, the virtual work done by inertial forces is expressed as

$$\delta W_j = - \int_{\Omega} \delta \mathbf{u}^T \rho_s \ddot{\mathbf{u}} d\Omega. \quad (4.4)$$

Having established the PVW, the displacement field can be approximated by the displacement, $\bar{\mathbf{u}}_e$, at the nodes of the element e on the discretized domain and the vector of shape functions, $\mathbf{N}_e$, used to interpolate the displacements inside the element

$$\mathbf{u} = \mathbf{N}_e \bar{\mathbf{u}}_e \quad (4.5)$$

and the discretized form of the variation of the Green-Lagrange tensor can also be obtained

$$\delta \mathbf{E} = \mathbf{B}_e \delta \bar{\mathbf{u}}_e \quad (4.6)$$

by resorting to the strain-displacement matrix, $\mathbf{B}$, which is derived from the shape functions in $\mathbf{N}$ attending to the kinematic relations.

Using this approximations, the PVW can now be rewritten for element e in the discretized form

$$\delta \bar{\mathbf{u}}_e^T \left[- \int_{\Omega_e} \mathbf{B}_e^T \mathbf{S} d\Omega - \int_{\Omega_e} \mathbf{N}_e^T \rho_s \mathbf{N}_e \ddot{\mathbf{u}} d\Omega + \int_{\Omega_e} \mathbf{N}_e^T \mathbf{f}^B d\Omega + \int_{\Gamma_e} \mathbf{N}_e^T \mathbf{f}^S d\Gamma \right] = 0. \quad (4.7)$$

With $\delta \bar{\mathbf{u}}_e$ taking an arbitrary value (except in the boundaries where Dirichlet conditions are set), a new equilibrium equation is obtained

$$\mathbf{M}_e \ddot{\bar{\mathbf{u}}}_e + \mathbf{f}_e^{int} = \mathbf{f}_e^{ext} \quad (4.8)$$

where $\mathbf{f}_e^{ext}$ combines the effects of the body and surface forces, and $\mathbf{M}_e$ is the element's mass matrix defined as

$$\mathbf{M}_e = \int_{\Omega_e} \mathbf{N}_e^T \rho_s \mathbf{N}_e d\Omega. \quad (4.9)$$

The remaining term, $\mathbf{f}_e^{int}$, is a nonlinear term given by

$$\mathbf{f}_e^{int} = \int_{\Omega_e} \mathbf{B}_e^T \mathbf{S} d\Omega \quad (4.10)$$

which does not allow the definition of the usual linear stiffness matrix. However, the tangential stiffness matrix, $\mathbf{K}_e^{tan}$, can still be constructed by resorting to the first-order term of a Taylor expansion, the derivative of the internal forces with respect to the nodal displacements

$$\mathbf{K}_e^{tan} = \frac{\partial \mathbf{f}_e^{int}}{\partial \bar{\mathbf{u}}_e} \quad (4.11)$$

and further decomposed into the contribution from the material stiffness and the nonlinear effects, the first and second term, respectively, on the right-hand side of the expression bellow

$$\mathbf{K}_e^{tan} = \int_{\Omega_e} \mathbf{B}_e^T \mathbf{D} \mathbf{B}_e d\Omega + \int_{\Omega_e} \mathbf{N}'_e^T \mathbf{S} \mathbf{N}'_e d\Omega. \quad (4.12)$$

On the geometric term, $\mathbf{N}'_e$ is given by the spatial gradient of the vector containing the shape functions. On the material term, $\mathbf{D}$ is the sensitivity of the SPK stress tensor

to the Green-Lagrange strain tensor and comes from linearizing the internal forces and discretizing the Green-Lagrange strain tensor as

$$\int_{\Omega_e} \mathbf{B}_e^T \frac{\partial \mathbf{S}}{\partial \mathbf{E}} \mathbf{E} d\Omega = \int_{\Omega_e} \mathbf{B}_e^T \mathbf{D} \mathbf{B}_e d\Omega \bar{\mathbf{u}}_e. \quad (4.13)$$

Having set the tangential stiffness matrix, it is now possible to rewrite equation (4.8), for each time instant, as

$$\mathbf{M}_e \ddot{\mathbf{u}}_e + \mathbf{K}_e^{tan} \bar{\mathbf{u}}_e = \mathbf{f}_e^{ext} \quad (4.14)$$

which is used to solve for the nodal displacements in element e [102].

The process can also be extended to other elements in the discretized domain, with the respective stiffness and mass matrices being computed from equations (4.9) and (4.12). Then, the matrices of all the elements are combined into the global matrices, $\mathbf{M}$ and $\mathbf{K}^{tan}$, in a process called assembly, which accounts for the information pertaining to element connectivity. This allows for the setup of the global system of equations, for each time instant,

$$\mathbf{M} \ddot{\mathbf{u}} + \mathbf{K}^{tan} \bar{\mathbf{u}} = \mathbf{f}^{ext} \quad (4.15)$$

used, together with the boundary conditions, to compute the nodal displacements over the entire discretized domain [103].

Note that no structural damping is considered throughout the derivation of the discretized system of equations, the reason being that it is assumed to be negligible when compared with the fluid-induced damping. Furthermore, its exclusion from the model will only result in more conservative predictions [8].

Regarding the solid mesh, even though STAR-CCM+ is not a software specifically focused on finite element analysis, it still provides a range of element types for the discretization of the domain. These include, in three dimensions, tetrahedra, hexahedra, pyramid and wedge elements, each of which can be linear or quadratic if mid-side nodes are added.

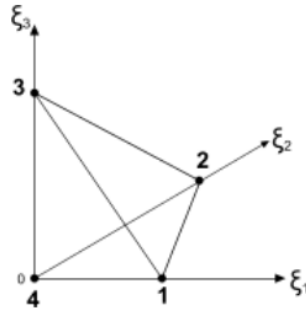


Figure 4.1: Linear tetrahedron [102].

Considering the geometry of the rudder, the Automated Mesh procedure has to be selected. This limits the choice of the element type to the tetrahedron only. The risk of shear locking and the difference in accuracy are evaluated latter to assess the need for quadratic elements. For now, the linear tetrahedra elements (Figure 4.1) are selected. These are characterized by their shape functions

$$\mathbf{N} = \begin{bmatrix} \xi_1 \\ \xi_2 \\ \xi_3 \\ 1 - \xi_1 - \xi_2 - \xi_3 \end{bmatrix} \quad (4.16)$$

where $\boldsymbol{\xi} = [\xi_1 \ \xi_2 \ \xi_3]$ is the natural coordinate system. With the shape functions defined in natural coordinates, it is possible to compute the mass and tangential stiffness matrices through numerical integration, using Gaussian quadrature. While the process is straightforward regarding the mass matrix, Newton iterative methods are used additionally to obtain the tangential stiffness matrix, given that it depends on the unknown displacement field.

Furthermore, since the problem is dynamic, there is also the need for a time integration scheme. For this, the Backward-Euler is used. This scheme is, admittedly, not ideal due to numerical damping, but its selection was dictated by the compatibility with the time discretization chosen for the fluid domain. To illustrate the effects of this numerical dissipation consider the nonlinear equation of a pendulum whose solution (Figure 4.2) was obtained using the Backward-Euler scheme with the Newton-Raphson method (procedure described below). The main takeaway from this simple example is that the dissipation tends to increase with the time step size. Given that small time steps are used for the FSI simulation, the effect is expected to be minimized to some degree.

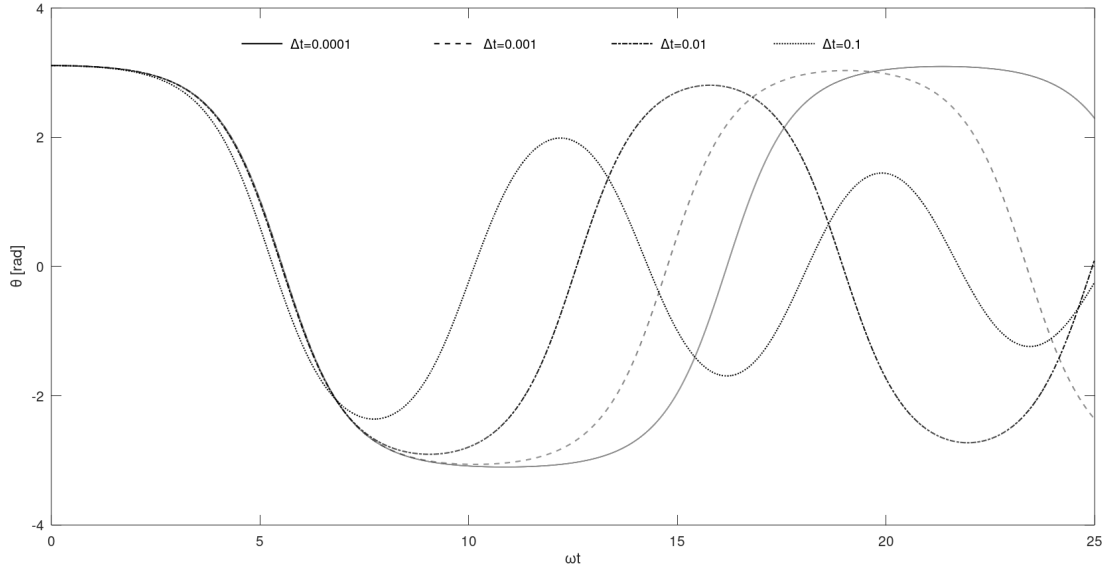


Figure 4.2: Effects of numerical damping on the solution for the nonlinear equation of motion of a pendulum, using Backward-Euler with Newton-Raphson. Computed and plotted with Octave 6.3.0.

Regarding the solution procedure, the approximations for velocity and acceleration at time $t = t^n$ in the Backward-Euler scheme are given by [102]

$$\dot{\mathbf{u}}^n = \frac{\mathbf{u}^n - \mathbf{u}^{n-1}}{\Delta t} \quad (4.17)$$

$$\ddot{\mathbf{u}}^n = \frac{\dot{\mathbf{u}}^n - \dot{\mathbf{u}}^{n-1}}{\Delta t} = \frac{1}{(\Delta t)^2} (\mathbf{u}^n - 2\mathbf{u}^{n-1} + \mathbf{u}^{n-2}). \quad (4.18)$$

Substituting the approximations in the equation of motion for the instant $t = t^n$ and emphasizing the dependency of the internal forces with respect to the displacement field yields

$$\mathbf{M} \frac{1}{(\Delta t)^2} (\bar{\mathbf{u}}^n - 2\bar{\mathbf{u}}^{n-1} + \bar{\mathbf{u}}^{n-2}) + \mathbf{f}^{int}(\bar{\mathbf{u}}^n) = \mathbf{f}^{ext,n}. \quad (4.19)$$

As mentioned before, the system is nonlinear and the Newton-Raphson method is employed to solve it iteratively [104; 105].

Firstly, the residual is taken at iteration k as

$$\mathbf{R}_k^n = \mathbf{M} \frac{1}{(\Delta t)^2} (\bar{\mathbf{u}}_k^n - 2\bar{\mathbf{u}}^{n-1} + \bar{\mathbf{u}}^{n-2}) + \mathbf{f}^{int}(\bar{\mathbf{u}}_k^n) - \mathbf{f}^{ext,n} = \mathbf{0} \quad (4.20)$$

and, for convenience, the terms that are not changing with the successive iterations are joined in the load $\bar{\mathbf{f}}$, resulting in

$$\mathbf{R}_k^n = \mathbf{M} \frac{1}{(\Delta t)^2} \bar{\mathbf{u}}_k^n + \mathbf{f}^{int}(\bar{\mathbf{u}}_k^n) - \bar{\mathbf{f}}^n = \mathbf{0}. \quad (4.21)$$

Then, a Taylor expansion for the residual at iteration k is performed with respect to the previous iteration, $k - 1$, yielding

$$\mathbf{R}_{k-1}^n + \frac{\partial \mathbf{R}_{k-1}^n}{\partial \bar{\mathbf{u}}} \big|_{k-1}^n (\bar{\mathbf{u}}_k^n - \bar{\mathbf{u}}_{k-1}^n) = \mathbf{0} \quad (4.22)$$

and, attending to equation (4.21)

$$\mathbf{M} \frac{1}{(\Delta t)^2} \bar{\mathbf{u}}_{k-1}^n + \mathbf{f}^{int}(\bar{\mathbf{u}}_{k-1}^n) - \bar{\mathbf{f}}^n + \left(\mathbf{M} \frac{1}{(\Delta t)^2} + \frac{\partial \mathbf{f}^{int}(\bar{\mathbf{u}}_{k-1}^n)}{\partial \bar{\mathbf{u}}} \big|_{k-1}^n \right) (\bar{\mathbf{u}}_k^n - \bar{\mathbf{u}}_{k-1}^n) = \mathbf{0}. \quad (4.23)$$

Recalling the definition of the tangential stiffness matrix in (4.11) and rearranging the equation above, an expression to determine the displacement at the current iteration is obtained

$$\bar{\mathbf{u}}_k^n = -(\mathbf{M} \frac{1}{(\Delta t)^2} + \mathbf{K}_{k-1}^{tan,n})^{-1} (\mathbf{M} \frac{1}{(\Delta t)^2} \bar{\mathbf{u}}_{k-1}^n + \mathbf{f}^{int}(\bar{\mathbf{u}}_{k-1}^n) - \bar{\mathbf{f}}^n + \bar{\mathbf{u}}_{k-1}^n). \quad (4.24)$$

Note that, every term on the right-hand side is known; this includes the tangential stiffness matrix at the previous iteration, $\mathbf{K}_{k-1}^{tan,n}$, which depends implicitly on the known displacement at the previous iteration, $\bar{\mathbf{u}}_{k-1}^n$, and is computed for each element according to equation (4.12), followed by the assembly of the global matrix. After the value of $\bar{\mathbf{u}}^n$ converges, a step is taken forward in time and the cycle repeats.

One of the most costly steps of this procedure is the update of the stiffness matrix and the calculation of its inverse for every iteration k . However, this effect can be minimized because, even though it is important to consider the nonlinear behavior due to the large displacements, the nonlinearity is not expected to be very severe for the present case. Thus, the Modified Newton method is considered suitable. This cuts down the computational cost because the tangential stiffness matrix is only updated for the first few iterations.

Additionally, STAR-CCM+ does not solve the linearized system of equations in (4.22) or (4.23) in the simple way that is shown in (4.24) (merely for illustrative purposes). In fact, it takes advantage of $\frac{\partial \mathbf{R}_{k-1}^n}{\partial \bar{\mathbf{u}}} \big|_{k-1}^n$ being a sparse matrix to employ the Multifrontal Massively Parallel sparse direct Solver (MUMPS) [102; 106].

4.3 Finite Volume Method

Fluid flows are generally described by the laws of conservation of mass, momentum and energy, with additional state equations and established relations for transport coefficients. To derive the relevant system of equations for employing a finite volume discretization, one may start by applying this set of conservation laws to a control volume, τ , enclosed by a surface area, S . As already mentioned when establishing the mathematical model, there is no need to include the energy equation, in this context; the application of the remaining conservation principles yields the integral form of the system of equations for an isothermal, incompressible flow

$$\int_S \mathbf{v} \cdot \mathbf{n} dS = 0 \quad (4.25)$$

$$\rho_f \int_\tau \frac{\partial \mathbf{v}}{\partial t} d\tau + \int_S [\rho_f \mathbf{v} \mathbf{v} + \boldsymbol{\sigma}] \cdot \mathbf{n} dS = \rho_f \int_\tau \mathbf{g} d\tau. \quad (4.26)$$

For water in these conditions, the stress tensor may be defined as

$$\boldsymbol{\sigma}_f = p\mathbf{I} - \mu_f \nabla \mathbf{v} \quad (4.27)$$

which can be plugged into equation (4.26) to yield

$$\rho_f \int_\tau \frac{\partial \mathbf{v}}{\partial t} d\tau + \int_S [\rho_f \mathbf{v} \mathbf{v} + p\mathbf{I} - \mu_f \nabla \mathbf{v}] \cdot \mathbf{n} dS = \rho_f \int_\tau \mathbf{g} d\tau. \quad (4.28)$$

Rearranging the terms in equations (4.25)-(4.26) and applying the divergence theorem to the pressure term, the following system is obtained

$$\int_\tau \nabla \cdot \mathbf{v} d\tau = 0 \quad (4.29)$$

$$\int_\tau \frac{\partial \mathbf{v}}{\partial t} d\tau + \int_S [\mathbf{v}(\mathbf{v} \cdot \mathbf{n})] dS = \int_S \nu_f (\nabla \mathbf{v} \cdot \mathbf{n}) dS + \int_\tau \mathbf{g} d\tau - \int_\tau \frac{1}{\rho_f} \nabla p d\tau. \quad (4.30)$$

The system above¹ can be solved by dividing the fluid domain into finite volumes, or cells, each assigned with a value for the relevant quantities of the flow problem. Besides the constant source term (second term on the right-hand side of equation (4.30)) whose integration is trivial, specific approaches are taken for discretizing each term, and are described bellow, considering a mesh comprised of generic three-dimensional polyhedral cells.

4.3.1 Discretization of the Convection Term

For the second term on the left-hand side, the convection term, it is possible to split the surface integral into a sum of integrals over the area of each of the N_c cell faces

$$\int_S [\mathbf{v}(\mathbf{v} \cdot \mathbf{n})] dS = \sum_{i=1}^{N_c} \int_{S_i} [\mathbf{v}_i(\mathbf{v}_i \cdot \mathbf{n}_i)] dS_i. \quad (4.31)$$

¹To simplify the exposition that follows, the ALE formulation and the Reynolds-Average decomposition are not considered here. However, this system of equations suffices for the points made in this section and the relevance of the attained conclusions still holds for the complete description. For similar reasons, the volume fraction transport equation is also treated separately.

Using numerical integration with one Gauss point located at each cell face center [107], the following approximation is obtained

$$\sum_{i=1}^{N_c} \int_S [\mathbf{v}_i(\mathbf{v}_i \cdot \mathbf{n}_i)] dS_i \approx \sum_{i=1}^{N_c} \mathbf{v}_{f_i}(\mathbf{v}_{f_i} \cdot \mathbf{n}_{f_i}) S_i \quad (4.32)$$

meaning that, by evaluating the fluid quantities at the faces of the cell (hence the subscript f), the integral can be replaced by a summation. However, given that, in FVM, the flow variables are stored in the cell centers, some interpolation scheme is required to calculate the values at the faces. STAR-CCM+ offers a large range of options for this interpolations, exposed in Table 4.1 bellow with the corresponding relevant characteristics.

From these discretization schemes, only First-Order Upwind (FOU), Second-Order Upwind (SOU) and Hybrid MUSCL 3rd-Order/Central-Differencing (HM3/CD) are available when the flow is solved with a segregated approach (see Section 4.3.3). The upwind schemes are widely used and are able to cover most demands in CFD applications because of the good compromise between accuracy and robustness [108], and, in this case, as far as the fluid domain is concerned, the problem is not complex enough to justify the cost of employing more advanced schemes such as HM3/CD. Thus, it is rejected, with the additional advantage of not having to tune the blending factor for this specific application. Regarding the upwind schemes, while FOU is more stable due to its boundedness property [109; 110], it is also less accurate than SOU. Furthermore, FOU is characterized by excessive numerical diffusion when the mesh lines are not aligned with the flow [110], which is often the case. Thus, to ensure the accuracy of the solution SOU is chosen, with additional remarks to be made about its stability.

The Second-Order Upwind interpolation is based on the approximation that the flow quantities vary linearly over the cell. This is, the values at the cell faces, $\mathbf{v}_f$ are computed from the value and gradient at the center of the cell located upwind, C , in the following manner

$$\mathbf{v}_f = \mathbf{v}_C + (\nabla \mathbf{v})_C \cdot \mathbf{r} \quad (4.33)$$

where $\mathbf{r}$ is the vector connecting the center of the cell and the center of the face. Note that, if the gradient is not limited in some way, this interpolation can lead to local maxima or minima at the cell faces, constituting nonphysical oscillations in the solution. This is the reason for the lack of convergence robustness of SOU.

To address this, gradient limiters such as the one developed by Venkatakrishnan [111] or the MinMod limiter [112] are made available in STAR-CCM+. Since the Venkatakrishnan limiter was proposed as an alternative to MinMod for improved convergence, it is the limiter selected in this work, even though it can be more dissipative in some cases [113]. Moreover, it is the generally recommended option in the user-guide [102].

4.3.2 Discretization of the Diffusion Term

The diffusion term is initially approached in a similar way to the convection term and the surface integral is replaced by a summation through considering the integral at each face individually and applying numerical integration

$$\int_S \nu_f (\nabla \mathbf{v} \cdot \mathbf{n}) dS \approx \sum_{i=1}^{N_c} \nu_f (\nabla \mathbf{v}_{f_i} \cdot \mathbf{n}_{f_i}) S_i. \quad (4.34)$$

Table 4.1: Discretization schemes available in Siemens Star-CCM+ and their relevant characteristics [102].

Discretization Scheme	Characteristics
First-Order Upwind (FOU)	• Only 1st order accurate • Introduces numerical diffusion • Unconditionally bounded thus stabilizing
Second-Order Upwind (SOU)	• 2nd order accurate • Stability concerns require gradient limiters
Central-Differencing (CD)	• 2nd order accurate • Dispersive errors • Raises stability concerns for most steady-state cases • Applicable for Large Eddy Simulations (LES)
Bounded Central-Differencing (BCD)	• 2nd order accurate • More dissipative than CD (the more so with coarser meshes) • Compromise between accuracy and robustness • Recommended for LES of complex turbulent flows
Hybrid MUSCL 3rd-Order/Central-Differencing (HM3/CD)	• Needs user-defined blending factor based on the physical problem • Mostly 3rd order accurate • Reduced numerical dissipation • Robust due to boundedness • Versatile: applicable to steady and unsteady; from incompressible to high-speed compressible • Hybrid of SOU with CD and BCD is also available

There is now a need to express the gradient at the cell face center, $\nabla \mathbf{v}_{f_i}$, in terms of the velocities and velocity gradients at the centers of the cells sharing that face (see Figure 4.3) and compute the dot product. For this purpose, the following expression is used by STAR-CCM+ [102]

$$\nabla \mathbf{v}_{f_i} \cdot \mathbf{n}_{f_i} = (\mathbf{v}_1 - \mathbf{v}_0) \frac{|\mathbf{n}_{f_i}|^2}{\mathbf{n}_{f_i} \cdot \mathbf{d}} + \left(\frac{\nabla \mathbf{v}_0 + \nabla \mathbf{v}_1}{2} \right) \cdot \mathbf{n}_{f_i} + \left(\left(\frac{\nabla \mathbf{v}_0 + \nabla \mathbf{v}_1}{2} \right) \cdot \mathbf{d} \right) \frac{|\mathbf{n}_{f_i}|^2}{\mathbf{n}_{f_i} \cdot \mathbf{d}}. \quad (4.35)$$

Note that, the first term alone would constitute a good approximation for the dot product if the cells were perfectly orthogonal. However, given that this is not the general case, some non-orthogonal correctors need to be applied. These are the second and third term, and are required to ensure the accuracy of the diffusion term discretization for meshes displaying some non-orthogonal cells. Nevertheless, because these non-orthogonal corrector terms can contribute to unboundedness, they are a cause for instability and their weight on the discretized calculations should be limited [107]. Thus, when constructing the mesh, it is advisable to keep the cells as orthogonal as possible to guarantee better convergence.

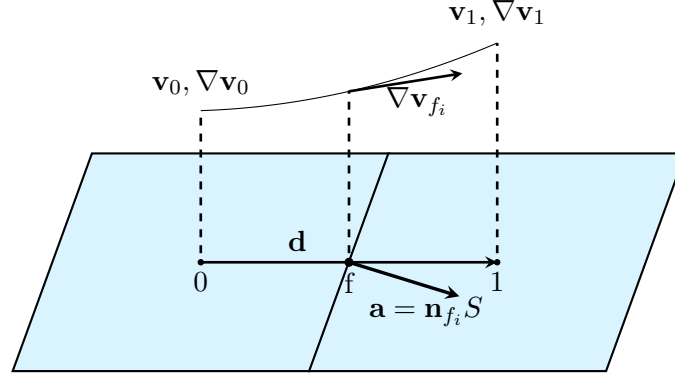


Figure 4.3: Two non-orthogonal adjacent cells.

4.3.3 Pressure-Velocity Coupling

As seen before, to get a solution to the Navier-Stokes equations for isothermal, incompressible flow, one has to solve for four unknowns: three components of the velocity, $\mathbf{v}$, and the pressure, p . Four equations are also available. These are the three momentum equations and the mass conservation equation. However, note that the latter is more of a restriction to the velocity field and that there is no explicit equation for pressure. This arises from the incompressibility of the flow, which decouples velocity from density in the mass conservation equation [110].

Thus, additional steps must be taken to solve the system of fluid equations, there being two ways of tackling this: the coupled and the segregated approach. The first solves all of the discretized equations under one large system, while the latter solves each equation separately, in a sequential manner. The coupled solver is more accurate, allows for larger time step sizes without diverging [114] and is more capable of solving high Mach number flows. On the other hand, the segregated solver is commonly used for incompressible flows [115], implies lower memory requirements since only one equation has to be stored at a time [116], and is faster per time step [114]. Attending to this information and to the application at hand, the segregated approach is deemed to be adequate.

Having chosen the segregated flow solver, two pressure-velocity coupling algorithms are available in STAR-CCM+. These are the SIMPLE [117] and the Pressure Implicit Split Operation (PISO) [118] algorithm. At short time steps, both methods have the same accuracy, although PISO is faster. However, for larger time step sizes (CFL condition not satisfied and convective Courant number rising above 10), PISO becomes unstable [102]. Since there is no prior requirement for the current application regarding the CFL condition², PISO would be overly-restrictive. Thus, the SIMPLE algorithm is selected.

SIMPLE Algorithm

The SIMPLE algorithm begins with the extraction of a pressure equation from the momentum and mass conservation equation. For this purpose, consider the semi-discretized algebraic system that is to be solved to obtain the solution for the flow

$$\mathbf{A}_P \mathbf{U}_P = \mathbf{H}(\mathbf{U}_P) - \nabla p. \quad (4.36)$$

Note that equation (4.36) is taken from the discretized momentum conservation equations, making $\mathbf{A}_P$ the diagonal matrix containing the coefficients that result from the discretization processes defined above. Furthermore, $\mathbf{U}_P$ is the vector storing the velocity components at the centers of the cells, and $\mathbf{H}$ represents the remaining terms coming from the matrix decomposition that originated matrix $\mathbf{A}_P$. Thus, $\mathbf{H}$ includes the coefficients for neighbouring cells (and their multiplication by the velocity values at the cell center) and the source part of the transient term and others (in this case, gravity). This equation can be rearrange to express $\mathbf{U}_P$ as

$$\mathbf{U}_P = \mathbf{A}_P^{-1}(\mathbf{H}(\mathbf{U}_P) - \nabla p) \quad (4.37)$$

without much cost given that $\mathbf{A}_P$ is a diagonal matrix. Substituting the expression for $\mathbf{U}_P$ in the mass conservation equation, one arrives at an equation for pressure

$$\nabla \cdot (\mathbf{A}_P^{-1}(\mathbf{H}(\mathbf{U}_P))) = \nabla \cdot (\mathbf{A}_P^{-1} \nabla p) \quad (4.38)$$

which, paired with equation (4.36) constitutes the new algebraic system to be solved. For a solution to be computed, first an approximation of the velocity field is calculated from equation (4.36) with a predictor for the pressure gradient. Then, a new pressure distribution is obtained from solving equation (4.38) using the approximated velocity field. The newly computed pressure field is used to correct the velocity so that it satisfies the continuity equation. However, the momentum equations are not satisfied at this point and the cycle is repeated [107] (vide Figure 4.4 for a flowchart representation of the algorithm).

A brief final mention to the under-relaxation factors (URFs) is made. These are applied to the pressure and velocity corrections as

$$p_P = p_P^* + \alpha_p p' \quad (4.39)$$

$$\mathbf{v}_P = \mathbf{v}_P^* + \alpha_v \mathbf{v}' \quad (4.40)$$

to achieve stable convergence. Earlier experimentation suggested that the default values of $\alpha_p = 0.2$ and $\alpha_v = 0.8$ are adequate for this application.

²Note that implicit time integration is used and, for the Volume of Fluid solver, the multi-stepping option is enabled, effectively eliminating the need to satisfy the CFL condition.

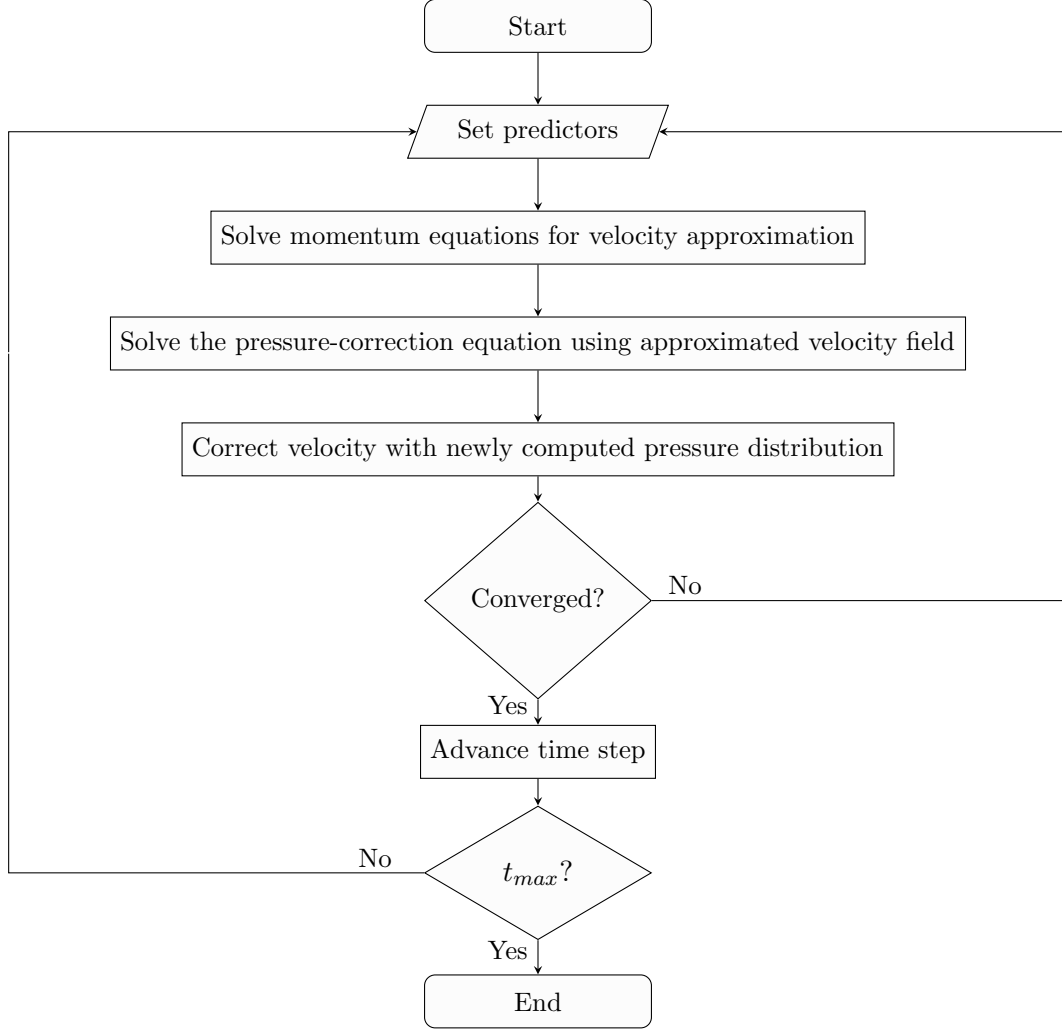


Figure 4.4: Flowchart representing the SIMPLE algorithm for unsteady simulations.

4.3.4 Volume of Fluid Method

As already mentioned, VOF method [119; 120] is an *interface-capturing* or *volume-tracking* technique, as opposed to *interface-tracking* methods. This means that, instead of following the motion of the interface, the volume of each phase is tracked and their interface is reconstructed from that information. For this purpose, the volume fraction of each phase i , α_i , is introduced as an additional variable. It is defined by the ratio

$$\beta_i = \frac{V_i}{V} \quad (4.41)$$

where V_i denotes the volume of each phase i occupying a given volume V . The value of β_i is bounded from zero, when phase i is not present, to one, when the volume is fully occupied by that phase. Furthermore, the volume V will contain a portion of the interface if it exhibits a volume fraction value between zero and one.

Given that the sum of volumes of the phases must not exceed the volume that encloses them, the following relation applies

$$\sum_{i=1}^N \beta_i = 1 \quad (4.42)$$

and, as a consequence, the multiphase problem is fully described when using one volume fraction variable per pair combination of the N phases present in the flow. In the present case, having two fluids, water and air, interacting on the free surface, only one volume fraction is required.

Even though there are two phases, only one set of the Navier-Stokes equations needs to be solved, with the flow properties being defined by averaging processes such as

$$\rho = \beta_i \rho_i + (1 - \beta_i) \rho_j, \quad (4.43)$$

$$\mu = \beta_i \mu_i + (1 - \beta_i) \mu_j. \quad (4.44)$$

However, the introduction of the volume fraction calls for a new equation to be added to the fluid system for closure. Similarly to other scalar fields pertaining to the flowing fluids, the volume fraction is subject to advective effects [121]. Thus, the following transport equation is established

$$\frac{\partial \beta_i}{\partial t} + \nabla \cdot (\mathbf{v} \beta_i) = 0 \quad (4.45)$$

together with the condition for initializing the interface, $\beta_i(\mathbf{x}, t_0)$. At this point, a suitable discretization scheme is needed for obtaining a numerical solution of equation (4.45), with a special emphasis placed on the convective term. In fact, the schemes that are usually applied for solving convective terms are not adequate, such as the upward differencing scheme which introduces numerical diffusion or the central differencing scheme that adds numerical dispersion. These schemes influence the solution of equation (4.45), smearing the interface or affecting stability [122]. As this is not desirable, more advanced techniques are used, with the High-Resolution Interface Capturing (HRIC) scheme being available in STAR-CCM+.

High-Resolution Interface Capturing (HRIC)

Similarly to other discretization schemes, HRIC [123] aims at interpolating the value taken by a given variable at the cell face from the values stored at the centre of surrounding cells. These face values are then used to numerically integrate different terms in the fluid equations, as shown before.

HRIC is based on the Normalized Variable Diagram (NVD) [124], thus the volume fractions are normalized according to

$$\xi = \frac{\beta_i - \beta_i^U}{\beta_i^D - \beta_i^U} \quad (4.46)$$

where ξ designates the normalized volume fraction at a given point and superscripts U , C and D denote the centres of the upwind, centre (donor) and downwind cells, respectively (see Figure 4.5, below).

To interpolate the value at the cell face, ξ_f is determined by the relations

$$\xi_f = \begin{cases} \xi_C & \text{if } \xi_C < 0 \vee \xi_C > 1 \\ 2\xi_C & \text{if } 0 \leq \xi_C < 0.5 \\ 1 & \text{if } 0.5 \leq \xi_C < 1 \end{cases} \quad (4.47)$$

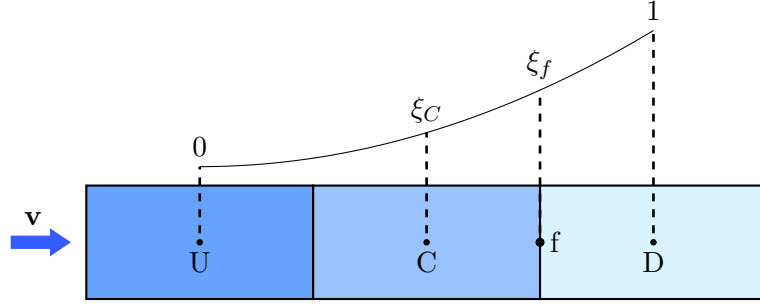


Figure 4.5: Schematic representation of the normalized variable distribution. Adapted from [125].

as an initial value. The calculated ξ_f is then blended with the face value calculated by the Quadratic Upstream Interpolation for Convective Kinematics (QUICK) [126] scheme, ξ_f^{blend} , in the following manner

$$\xi'_f = F(\gamma)\xi_f + (1 - F(\gamma))\xi_f^{blend}. \quad (4.48)$$

Here, $F(\gamma)$ is a blending function whose value depends on the angle, γ , between the normal vector of the interface and the surface area vector, $\mathbf{a}_f$, of face f shared by the cell that contains the interface and the downwind cell, and is defined as

$$F(\gamma) = \sqrt{|\cos \gamma|}. \quad (4.49)$$

QUICK is used here for its capability of dealing with flow parallel to the interface, avoiding artificial wrinkles in the free surface [102; 127]. Additionally, the blending expressed in equation (4.48) is necessary because the transition between schemes has to be smooth.

Next, a last correction is required to ensure the Courant–Friedrichs–Lewy (CFL) condition is satisfied [128], i.e., that the amount of fluid passing through a cell face does not exceed the amount held by the donor cell. For this purpose, the local Courant number is defined as

$$Co = \frac{\mathbf{v} \cdot \mathbf{a}_f}{V_C} \Delta t \quad (4.50)$$

where V_C is the volume of the donor cell and Δt is the time step used for the temporal discretization. The corrected normalized volume fraction is obtained from

$$\xi''_f = \begin{cases} \xi'_f & \text{if } Co < Co_l \\ \xi_C + (\xi'_f - \xi_C) \frac{Co_u - Co}{Co_u - Co_l} & \text{if } Co_l \leq Co < Co_u \\ \xi_C & \text{if } Co_u \leq Co \end{cases} \quad (4.51)$$

with Co_u and Co_l being adjustable parameters that control another transition, this time between HRIC and First-Order Upwind (FOU) schemes. The latter scheme is advantageous in terms of stability when large local Courant numbers are attained and the interface cannot be captured with fine resolution [102; 125].

Finally, the value of the volume fraction in face f is determined from the corrected normalized value as

$$\beta_i^f = \xi''_f(\beta_i^D - \beta_i^U) + \beta_i^U \quad (4.52)$$

after which the computation of the solution of the second term in equation (4.45) proceeds as with any other convective term solved with the Finite Volume Method.

Multi-Stepping

To be able to capture a sharp interface with the HRIC scheme, it is recommended that the CFL condition is fulfilled so that an accurate solution for the transport of the volume fraction can be obtained. However, this is often significantly more restrictive for the time step size selection than any other requirement of the simulation.

Attending to this matter, the Multi-Stepping option is selected. Multi-Stepping allows for several sub-iterations to be performed within a time step when solving for the volume fraction. As a consequence, the restrictions on the time step size are relaxed. This generally reduces the computation time, even when accounting for the additional sub-steps [102; 129].

4.3.5 Wave Characteristics and Modelling

Having established the free surface, it is also important to model the waves given that the wave-induced loads are expected to produce non-negligible responses. Furthermore, it is assumed that the frequency at which they excite the rudder is sufficiently close to the fundamental frequency of the structure for some dynamic amplification to occur. To incorporate them in the FSI model, it is useful to understand the mechanics behind these waves and clearly point out which relevant characteristics shall be required for a complete description.

Sea waves can be classified into several types, depending on the generating mechanism and the restoring forces acting to produce the oscillations [130]. When wind or breeze begin to blow above the water surface, small short-period waves or ripples first appear. At this scale, it is the surface tension that acts as the restoring force, and for that reason these are called *capillary waves*. However, as the wind continues to add energy to the water, the waves grow in amplitude, further increasing the surface area on which the wind acts. The waves then reach a point where the main restoring force is gravity, and surface tension effects become negligible. These are known as *gravity waves*. To the transitional regime where surface tension and gravity have comparable influence on the oscillations some authors call *ultragravity waves* [131].

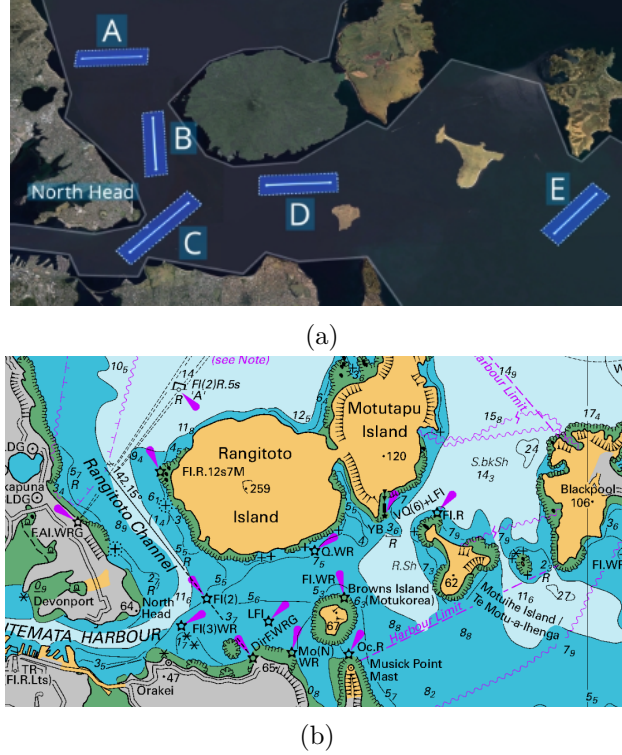
These three types of wave share the same generating force, the aerodynamic pressure caused by the wind, but they differ in scale. This difference is not only noticeable in the wave height but also shows in the period or wavelength of the oscillations. In fact, these parameters are tightly correlated with the wave type, and bands of periods can be clearly associated to each one. For example, capillary waves often exhibit periods smaller than 0.1 seconds, while gravity waves oscillate with periods between 1 and 20 seconds. Ultragravity waves, on the other hand, show periods between 0.1 and 1 second [131]. The importance of establishing these ranges of periods lies on the fact that the distinction between wave types can then be done criteriously. Thus, the influence of gravity and surface tension can be objectively determined, which bares relevance when setting up the model.

Besides the three aforementioned categories, others exist associated with different phenomena such as storms, earthquakes, atmospheric pressure gradients, earth's rotation, etc. These will not be considered in this work; in some cases, because the conditions exceed those in which the boat is designed to perform; in other cases, because the large timescales (tidal waves being a good example) are significantly different from those being studied.

Furthermore, other distinctions may be used to characterize the waves. For example, the influence of the seabed on the waves defines the classification of *deep* and *shallow* waves [132]. This is important because, for deep waves, i.e. when the depth is larger than one half of the wavelength, the interaction with the seabed can be neglected and thus the

model can be simplified.

Having this in mind, the description of the waves used in the model is briefly made. The characteristics used in this work for the waves are based on the sea conditions at the venue of the 36th America's Cup (AC36) - Auckland, and more specifically, the Hauraki Gulf. In [133], although Brookes and Green mention the complexity of the wave climate in Auckland, they were able to successfully describe the conditions in the Hauraki Gulf through a combination of experimental data and mathematical models. Even though most of their data pertains to open-waters, there is also register of inshore-waters with wave heights between 0.15 and 0.35 meters and periods ranging from 2 to 2.5 seconds. Having been recorded for just one day, this values are not completely representative of the true inshore conditions, but can serve as reference for the inputs of the model.



where g is the gravitational acceleration and T is the period [132]. For the aforementioned band of wave periods the corresponding wavelength interval would be of around 6 to 10 meters. Given that the minimum sea depth is greater than half of the largest expected wavelength, the waves can be mostly considered to be deep waves, thus also validating the use of equation (4.53).

Having briefly described wave mechanics and the sea conditions to be considered, the model of the waves needs to be established. For this purpose, a fifth-order approximation to the Stoke's wave theory is used, based on the work developed in [136], shown to be more accurate than other previous fifth-order approximations such as [137; 138].

The fifth-order approximation allows for a more precise representation of the wave loads on the structure than the first-order one without carrying a significant additional cost for the model, computationally [102]. At the same time, given that no specific wave data is being studied, it is not too precise for this application, as other tools such as the JONSWAP [139] or Pierson-Moskowitz [140] spectra would be. For this reason, the fifth-order model is selected. To set it up, the essential parameters are the wave height, the wave period or wavelength and the water depth, all of each have been addressed above.

Furthermore, a supplementary concern to have when using the fifth-order approximation is ensuring the validity of its application by keeping the Ursell number [141], U_R , under 30 [102]. The Ursell number is given by

$$U_R = \frac{h\lambda^2}{D_h^3} \quad (4.54)$$

where h is wave height and D_h is the sea depth. For the sea conditions here described, the Ursell number remains well below the unity, indicating that the fifth-order model is applicable.

4.3.6 Time Discretization

In STAR-CCM+, the discretization of the transient term is subject to two decisions: selecting implicit or explicit and first or second-order schemes. For this work, the implicit first-order scheme is selected.

The implicit option is conditioned by the choice to employ a segregated approach with the SIMPLE algorithm, being the only available selection.

The first-order scheme is used mainly because of its compatibility with the Multi-stepping approach of the Volume of Fluid solver. Note that Multi-stepping is not available for second-order time discretization and, for such a scheme, the Courant number would need to be kept below 0.5 for accuracy [102]. This would impose a significant restriction on the time step size and slow down an already heavy simulation. Thus, in this matter, noticeable savings in computational time are prioritized over temporal accuracy, which is affected by numerical diffusion [109]. Furthermore, the first-order scheme is more stable than its higher-order counterpart, which may be useful to stabilize the fluid-structure interaction simulation.

4.3.7 Iterative Methods

Although simple, Jacobi and Gauss-Seidel are among the most commonly used iterative methods in CFD to solve the linear system of equations

$$\mathbf{A}\mathbf{w} = \mathbf{b} \quad (4.55)$$

originating from the discretization of the fluid equations, where $\mathbf{A}$ is a matrix of coefficients (related to both $\mathbf{A}_P$ and $\mathbf{H}$ in equation (4.36)), $\mathbf{w}$ is a vector storing the unknowns at all the n cell centers in the grid and $\mathbf{b}$ is representative of the residuals at those cell centers. This system can also be represented as

$$\begin{bmatrix} a_{1,1} & a_{1,2} & a_{1,3} & \cdots & a_{1,n} \\ a_{2,1} & a_{2,2} & a_{2,3} & \cdots & a_{2,n} \\ a_{3,1} & a_{3,2} & a_{3,3} & \cdots & a_{3,n} \\ \vdots & \vdots & \vdots & \ddots & \vdots \\ a_{n,1} & a_{n,2} & a_{n,3} & \cdots & a_{n,n} \end{bmatrix} \begin{bmatrix} w_1 \\ w_2 \\ w_3 \\ \vdots \\ w_n \end{bmatrix} = \begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ \vdots \\ b_n \end{bmatrix}. \quad (4.56)$$

Both Jacobi and Gauss-Seidel evaluate each row of the system individually to solve for the unknown associated to that row. However, a considerable difference arises from the fact that Jacobi considers only the values from the previous iteration, $k - 1$, as shown below

$$w_i^k = \frac{1}{a_{i,i}} \left(b_i - \sum_{\substack{j=1 \\ j \neq i}}^n a_{i,j} w_j^{k-1} \right) \quad (4.57)$$

while Gauss-Seidel uses the values of w_j at the new iteration k in case they have already been computed. Because of this, Gauss-Seidel shows improved converge, but it is non-parallelizable [142].

Even though these iterative methods are much more useful in CFD than direct methods (whose time and memory cost increases rapidly with n , besides being very susceptible to accumulated round off errors), their convergence rates are still affected by the size of the system and can be impracticably slow.

To improve this, STAR-CCM+ resorts to a multigrid approach, in combination with iterative solvers. Multigrid techniques are characterized by the transfer of the residuals from the original finer grid to a set of successively coarser grids (repeating a process called restriction), where they are corrected and transferred back, level by level in a step known as prolongation, to the finer grid. The main reason to do so is the fact that iterative solvers are very good at reducing the high-frequency components of the numerical error, but not so capable of smoothening low-frequency components. Thus, by transferring the residual to a coarser mesh, the ratio between the wavelength of the error and the size of the cells decreases, the error frequency "appears" to be higher and the iterative algorithms perform better, resulting in accelerated convergence [109]. In STAR-CCM+, the Algebraic Multigrid (AMG) [143] method is used ("Algebraic" referring to the way the coarser grids are constructed, as opposed to "Geometric"). Considering all the information above, an approach combining the Gauss-Seidel with the AMG solver was used in this work. An exception is made for the pressure solver, for which a conjugate-gradient method [144] is used (with the AMG solver as a preconditioner) to accelerate convergence, being the recommended procedure for solving incompressible flows with the segregated approach [102].

4.4 Fluid-Structure Coupling

Since the general numerical aspects of fluid-structure coupling have already been covered in previous sections, this subsection is focused merely on the materialization of such coupling inside STAR-CCM+.

Firstly, an interface between fluid and solid region is defined so that information can be exchanged. The transferred data depends on the type of coupling desired, which is controlled through the *morpher specification* and *traction specification*. In this work, the former is set to *solid displacement*, ensuring that the displacement field of the solid region is used as an input to the fluid mesh morpher, and the latter is set to *fluid load* so that the shear and pressure computed at the fluid boundary can be used as external loads on the structural solver. This yields a two-way fluid-structure interaction.

An additional difficulty concerning this transfer of data arises from the fact that the fluid and solid meshes are non-conformal, i.e., the vertices on one mesh do not exactly match with the vertices of the other at the interface (and, even if that was the case, FEM calculates the values at the element nodes, while FVM uses the values at the face/cell center). Thus, some mapping procedure is required and the Mapped Contact Interface type, which enables the interpolation of the data transferred between the non-conformal meshes, must be selected. In the present case, of node-to-face mapping, the interpolation is done with the use of the shape functions associated to the element type(s) involved [102].

To deform the fluid mesh so that the velocity field can be solved accounting for the displacements of the structure the *Morpher motion* is used. In STAR-CCM+, the morpher selects a set of control points at the mesh boundaries, where the displacements are known (either through applied constraints or through data transferred from the solid region). This information is used to generate an interpolation field that is applied to all vertices in the mesh, ensuring a correct deformation. In a final step, some corrections may be made on the displacement of vertices at or near the boundaries. Two different methods can be used for the interpolation: Radial Basis Function (RBF) [4] and BSpline [145]. For this work, BSpline is selected for its superior ability to scale on various processors and better preserve the prism layers [102].

As mentioned before, FSI simulations may be difficult to stabilize, especially when the densities of fluid and structure are similar and the system is tightly coupled. For this reason, STAR-CCM+ provides two options for the solution stabilization: the *Constant Displacement Under-relaxation* method, which consists simply on applying an under-relaxation factor to the deformation of the fluid mesh; and the *Boundary Interface Added Mass* method, which makes an early prediction of the traction at the interface by estimating the mass of fluid that is displaced. The latter method, besides stabilizing the solution, helps to accelerate the convergence and is the recommended option for unsteady two-way FSI [102]. Due to the aforementioned benefits, it is selected for use in this methodology. For a more in-depth discussion on the early estimation of the added mass vide [146].

To further stabilize the solution, the fluid traction is also ramped. This helps to avoid nonphysical behaviour at the beginning of the simulation when the flow is badly resolved, and smoothens the response of the structure at an early stage.

4.5 Chapter Recap

In this Chapter, the numerical methodology is designed and described. First, a brief account of the Finite Element Method is given, starting from the Principle of Virtual Work and including geometrically nonlinear effects. The main element type used in the finite element discretization is described and some remarks are made on the time integration scheme, the Backward-Euler scheme, and the iterative scheme, the Modified Newton method. Then, a more detailed presentation of the Finite Volume Method is delivered, with an overview on the discretization of the most important terms such as the temporal,

convective and diffusive terms. Comments on the face-interpolation schemes and on the non-orthogonality of the cells are highlighted. The SIMPLE algorithm, for dealing with the decoupling of pressure and velocity fields is selected and described. The importance of the Volume of Fluid method for capturing the free surface is shown, with focus on the need for higher-resolution discretization schemes and the benefits of multi-stepping; and the modelling of the waves is further referenced. A brief overview is given on the iterative methods used. Finally, the coupling of fluid and structure is explained, in the practical context of STAR-CCM+, highlighting the mesh morphing and the stabilization techniques available.

Test Cases

5.1 Introduction

Given the difficulties in obtaining experimental data for the hydroelastic behaviour of the moth rudder both for time and cost-related reasons, the FSI analysis developed in this dissertation is tested with existing data in the literature regarding a similar problem. Since the idea is to validate the procedure designed to analyse the hydroelastic effects on the moth rudder, it is desirable that the validation case is as comparable to the rudder simulation as possible, so that the main features of the simulation tools can be tested and validated. This procedure is similar to what was done in [73], where the numerical models were validated using the well-studied and well-documented NACA0012 hydrofoil before analyzing the daggerboard on the AC45F.

In this case, the experiments in [75], studying the hydroelastic behaviour of a hydrofoil with a NACA66312 section, are selected to evaluate the numerical predictions. After validating the FSI tools, other simulations with the NACA66312 section are run to evaluate the sensitivity of the model to both physical and numerical parameters. These tests are only run for the validation case and not for the rudder simulation, because the former model is lighter and faster to run than the latter, while the conclusions are still general and applicable to the rudder case. Finally, the hydroelastic response of the moth rudder is simulated and the results are discussed.

5.2 Validation Case

5.2.1 Model Setup

The geometry of the domain used for running the validation case is presented below, in Figure 5.1.

The relevant dimensions are the chord of the hydrofoil, $c = 0.148$ m, the wingspan of the hydrofoil, $b = 0.191$ m and the side of the squared cross-section of the water tunnel, $h = 0.192$ m. The pitch of the hydrofoil is also varied from 7° to 12° , between simulations.

The fluid is water, with a constant density, $\rho_f = 997.56$ kg/m³, and dynamic viscosity, $\mu_f = 8.888\text{E-}4$ Pa·s. The hydrofoil is constituted by polyoxymethylene (POM) with density $\rho_s = 1480$ kg/m³, Young's Modulus $E = 3$ GPa and Poisson's Ratio $\nu_s = 0.35$.

The Reynolds-averaged Navier-Stokes equations are solved using the $k - \omega$ SST turbulence model. An All $y+$ Wall Treatment was selected, meaning that the solution of the boundary layers is computed using wall functions.

To simulate the experimental setup, the boundary conditions shown in Table 5.1 were

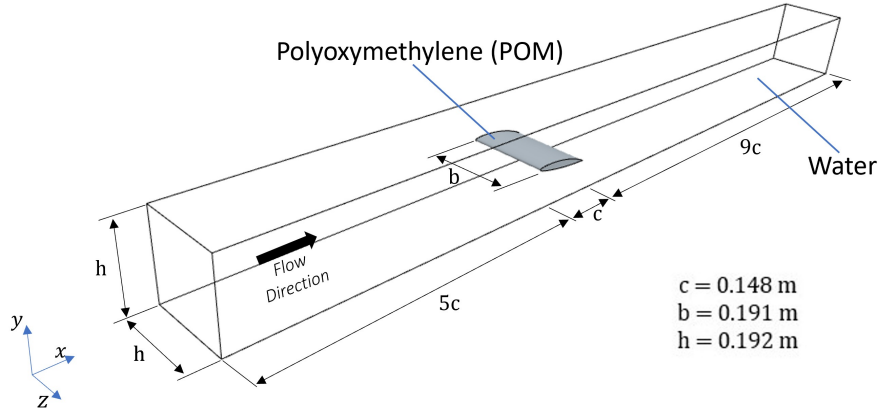


Figure 5.1: Geometry of the validation case.

applied to the fluid domain. Regarding the structure, the imposed boundary conditions are shown in Table 5.2.

Table 5.1: Fluid boundary conditions for the validation case.

Boundary	Conditions
Inlet	Velocity Inlet Velocity= 5 m/s; Turbulence Intensity= 2.95 %
Outlet	Pressure Outlet Pressure= 0 (Gauge)
Top, Side, Bottom Walls	Wall No-slip; smooth
Foil Surface	Wall No-slip; smooth

Table 5.2: Solid boundary conditions for the validation case.

Boundary	Conditions
Fixed Tip	Fixed Constraint
Foil Surface	Fluid Traction
Free Tip	Fluid Traction

Regarding the fluid-structure coupling, the relevant selected properties are shown in Table 5.3, below.

Additional information pertaining to different solvers that was considered relevant is presented in Table 5.4.

Table 5.3: Fluid-structure coupling highlighted settings.

Property	Value
FSI Load Ramping - Start	0.00 <i>s</i>
FSI Load Ramping - End	0.01 <i>s</i>
Stabilization Method	Boundary Interface Added Mass
Mesh Morpher	BSpline
Recompute Interfaces	Yes
Morph at Inner Iterations	Yes

Table 5.4: Important properties of remaining solvers.

Property	Value
Time Discretization	Implicit First-Order
Time Step Size	0.0005 <i>s</i>
Maximum Inner Iterations	15
Convection Discretization	Second-Order Upwind
Stiffness Matrix Update (Update Frequency)	Modified Newton (First two iterations)

5.2.2 Meshing

Mesh Independence Study

Concerning the FSI models, two different meshes are involved in the numerical solution, the solid and the fluid. Thus, an analysis on the effect of the sizes of both of these meshes is needed to assess the mesh independence. This could be done by monitoring the evolution of one relevant output of the FSI simulation while varying the size of the two meshes. Nevertheless, it is quicker to evaluate each mesh individually, with decoupled models, and do one independence study for each. This also allows for a more careful analysis of each mesh. Thus, to test the fluid grid, the external flow over the foil is studied and the force coefficients are monitored. On the other hand, to study the independence of the solid mesh, the first three natural modes are extracted through modal analysis and their variation with successive refinements of the mesh is evaluated.

The results are presented in Figures 5.2 and 5.3. With this information the solid mesh constituted by 0.6M elements and the fluid mesh composed of 1.2M cells were selected. Note that, while the lift coefficient was seen to be independent from mesh size, the drag coefficient still shows a difference of a little over 3% when compared to the finer mesh. However, given that the computational resources are limited, this results were deemed to be acceptable.

Fluid

For the fluid region, an automated mesh (Figure 5.4) was generated using the Trimmed Cell mesher and the Prism Layer mesher [102]. The latter mesher's settings, in Table 5.5,

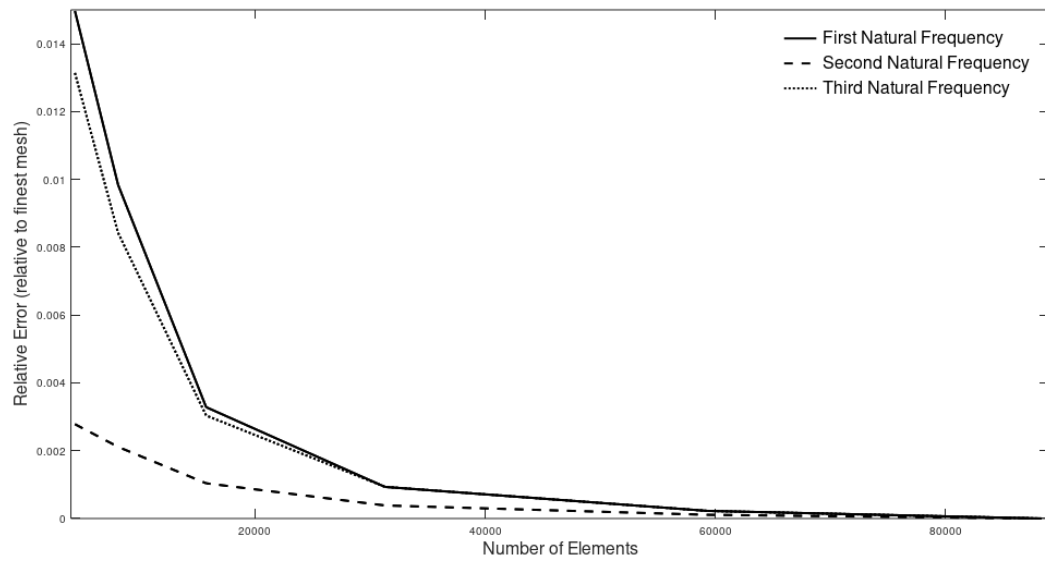


Figure 5.2: Variation of the solution for the first three natural frequencies with the mesh size, for the validation case.

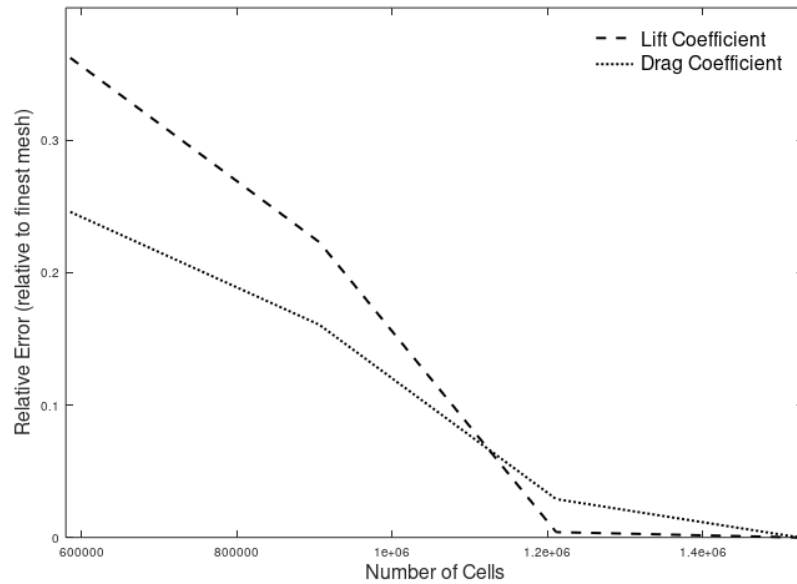


Figure 5.3: Variation of the solution for the force coefficients with the mesh size, for the validation case.

were tuned to keep the prism layers from collapsing at the thin trailing edge.

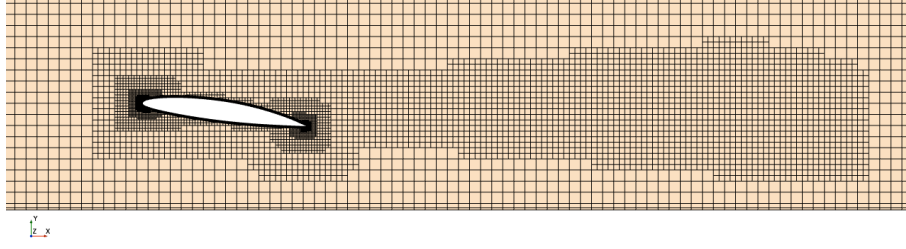


Figure 5.4: Two-dimensional section of the fluid mesh for the validation case, close to the foil.

Table 5.5: Prism Layer Mesher settings.

Property	Value
Minimum Thickness Percentage	0.01
Layer Reduction Percentage	30
Boundary March Angle	10
Concave Angle Limit	0.0
Convex Angle Limit	250.0

The mesh is defined by two different types of controls: the *Default Controls*, which set the general default values for the whole mesh, and the *Custom Controls*, that control the mesh properties in specific regions. The Default and Custom Controls for this mesh are shown in Tables 5.6 and 5.7, respectively.

Table 5.6: Default Controls for the fluid mesh.

Property	Value
Base Size	0.01 m
Target Surface Size	0.01 m
Minimum Surface Size	$1.0 \cdot 10^{-5}$ m
Number of Prism Layer	2
Prism Layer Total Thickness	0.002 m
Prism Layer Near-Wall Thickness	0.0003 m
Surface Growth Rate	1.15
Volume Growth Rate	Slow

Solid

The mesh in the solid region was generated using a Directed Mesh. A total of 30 layers of elements were distributed from the fixed tip to the free tip, using the Quadrilateral

Table 5.7: Custom Controls for the fluid mesh.

Region	Property	Value
Foil Edges	Target Surface Size	0.0005 m
Foil Surface	Number of Prism Layer	7
Wake	Wake Refinement	0.005 m
	Distance	0.5 m
	Spread Angle	5°

mesher. The mesh parameters are defined in Table 5.8. The element type used is the linear hexahedron, as shown in Figure 5.5, which is identified in STAR-CCM+ by the code number 12.

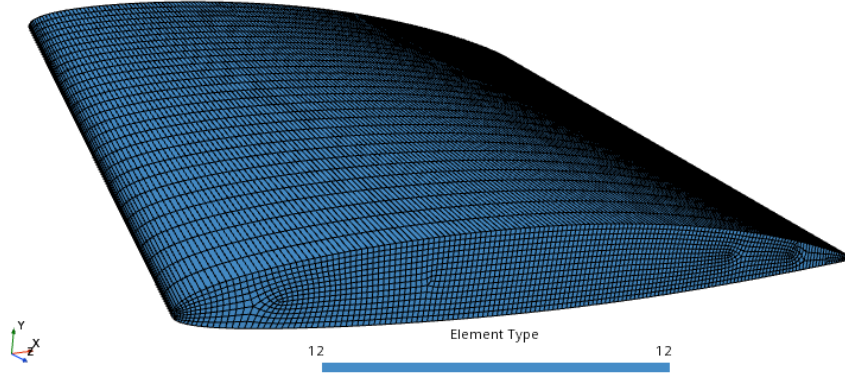


Figure 5.5: Solid mesh, with identification of the element type (12 - linear hexahedron).

Table 5.8: Default Controls for the solid mesh.

Property	Value
Base Size	0.001 m
Target Surface Size	0.001 m
Minimum Surface Size	0.0001 m
Surface Growth Rate	1.3

5.2.3 Results and Discussion

With the setup and meshes defined, the simulations were initially run for 0.15 s of simulated time, and for different angles of attack (AoA): 7°, 8°, 10° and 12°. The vertical displacements on the leading edge, at the tip section, were measured for the last time instant of the simulation and compared to those in the literature [75], to validate the predictions. The results are compiled and presented in Figure 5.6

The displacements at the tip are shown to increase with the angle of attack, both for experiments and numerical solutions. The latter compare well with the experimental data,

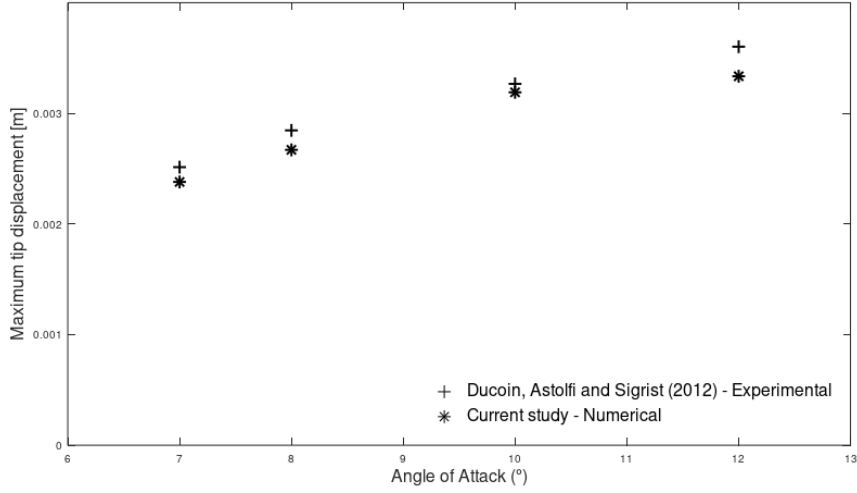


Figure 5.6: Comparison between numerical predictions of tip displacement and experimental data.

with errors kept below 7.5% for all angles of attack. However, the results indicate a slight tendency for under-estimating the structural displacements, yielding non-conservative predictions.

The force coefficients of the FSI simulation were also compared to those computed by a normal hydrodynamic analysis, and the different solutions are shown in Figure 5.7

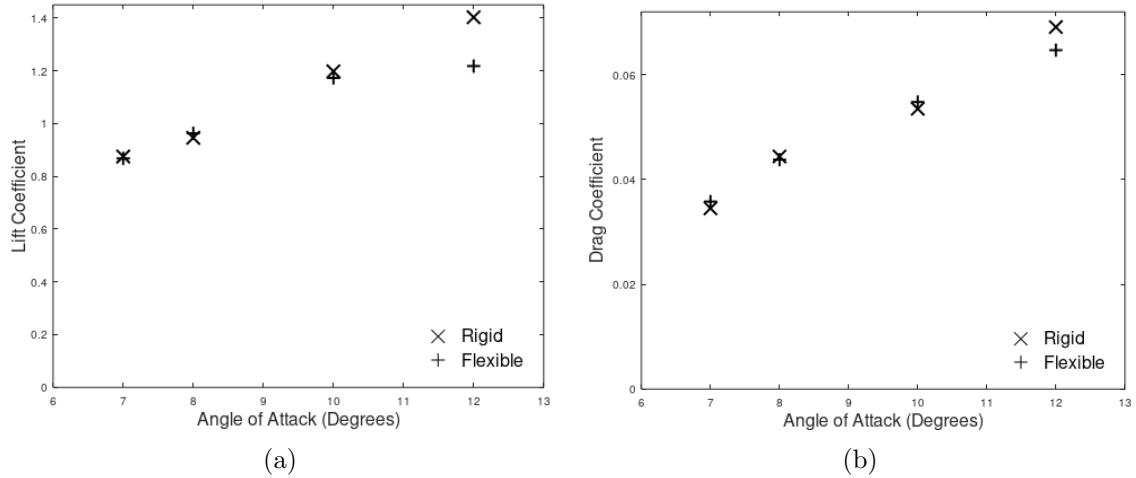


Figure 5.7: Comparison between force coefficients determined for rigid and flexible hydrofoil; (a) lift coefficient, (b) drag coefficient.

From the data collected, it is verifiable that the analysis that includes the hydroelastic effects on the flow does not yield substantially different results, with the exception of the 12° angle of attack case. Besides this, both force coefficients show increases in value with the angle of attack. Larger differences between predictions for rigid and flexible case at 12° of AoA are attributed to the proximity to stall conditions (13° AoA, for this profile, under the presented flow conditions [75]), which are exceeded by the hydrofoil after deformation. The reason for the difference in the drag coefficient is harder to pinpoint since, in theory, it should increase with AoA (thus, with the deformation of the structure). It could be related to complex flow phenomena arising from large deformations at the tip or, simply,

possible struggles by the model to make good predictions under stall conditions.

In addition to analysing the steady-state hydroelastic effects, it is necessary to evaluate the predictions for transient hydroelastic behaviour. Even though no time history of the displacements is shown in [75], a spectral analysis is presented in said work, with the purpose of determining the in-water natural frequencies of the structure, in this particular flow conditions. Table 5.9 shows these experimental values for the in-water natural frequencies.

Table 5.9: Experimentally determined in-water natural frequencies [75]

Mode	Frequency
First Bending Mode	43 Hz
First Torsional Mode	171 Hz
Second Bending Mode	291 Hz
First Mixed Bending-Torsional Mode	560 Hz

To assess how the model predictions compare to these measurements, the vertical velocity responses were measured at 81 probe points, arranged in a 9x9 matrix, replicating the experimental setup. This data is then used for a spectral analysis of the velocity response to understand how the flow of water changed the natural frequencies of the system, in the numerical solution.

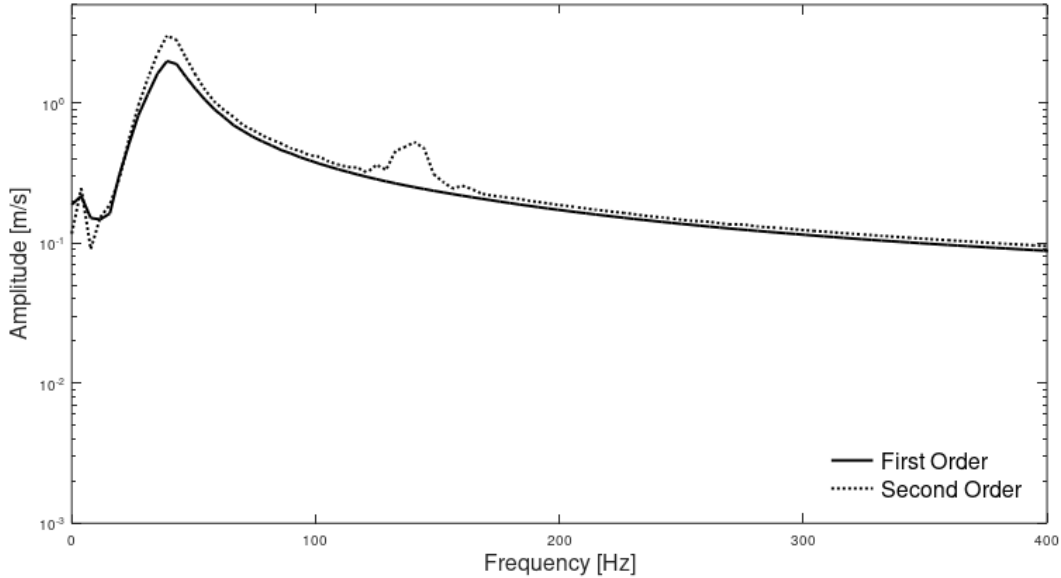


Figure 5.8: Velocity spectra obtained from first and second order time discretization.

The spectral analysis, using the Fast Fourier transform [147], was performed for a series of 512 time instants, each separated by a time step size of 0.0005 s. With these values, the frequency resolution of the spectral analysis is

$$\Delta f = \frac{1}{512 \cdot 0.0005} \approx 3.9 \text{ Hz} \quad (5.1)$$

meaning that the uncertainty associated to the measure of the dominant frequencies is

± 1.95 Hz. The coefficients obtained for the different 81 probe responses were averaged and plotted in Figure 5.8.

The application of the "standard" method (with the first-order time discretization) yielded a good prediction of the first in-water natural frequency, giving a value of $f_1^{num} = 39.1 \pm 1.95$ Hz and a relative error of 9 ± 4.5 %. However, through the analysis of the spectrum, it is clear that the time scheme used, together with the time step size, is too dissipative to capture any other transient phenomena at higher frequencies.

For comparison, the same analysis was performed for the results given by the second-order time discretization. The results are also presented in Figure 5.8. This comparison evidences the clear superiority of the higher-order scheme to capture higher frequency phenomena. While the value estimated for f_1 remains the same (within the resolution of the spectral analysis), the higher-order scheme also allows for a prediction of the first torsional natural frequency. It is calculated to be $f_2^{num} = 140.6 \pm 1.95$ Hz, meaning a relative error of 17.8 ± 1.1 %. Besides the first bending and first torsional frequency, no other natural frequencies were detected for the in-water case, with the second-order time discretization.

However, the reason for this should be attributed mainly to the size of the time step, and not numeric dissipation. Note that, the time step of $\Delta t = 0.0005$ s is only a little over ten times smaller than the period of the first torsional mode, $t_2 = \frac{1}{171} \approx 0.006$ s. This difference is reduced even more if one considers the second bending mode, whose period is $t_3 = \frac{1}{291} \approx 0.003$ s, or the first mixed bending-torsional mode, which has a period of $t_4 = \frac{1}{560} \approx 0.002$ s. These are timescales too close to the time step to be accurately captured. Note that general recommendations point to the use of a time step at least ten times smaller than the lowest structural period of interest, while other works suggest that this difference should be more than a hundredfold if full temporal accuracy is targeted [8]. Thus, it is not surprising that this numerical analysis could only capture the first two natural frequencies. This matter may also be a possible source of error for f_2^{num} .

Nevertheless, the capability of calculating the change of the two first natural frequencies is already enough, as far as instability prediction is concerned. It was shown before, in Section 2.5, that two modes are sufficient to describe any hydroelastic unstable or resonant behaviour expected in this context (while divergence, stall flutter and lock-in can be described with only one DoF, coupled-mode flutter requires two modes). And although those conclusions were taken from a simplified model, numerous other works describe the coupling in these terms [5; 6; 81; 148].

No data is available concerning a critical velocity for flutter to occur for this setup. However, it is assumed, considering the aforementioned reasons, that flutter can be predicted if the first natural frequencies of the submerged structure are correctly determined. As shown, this is possible using a second-order time discretization, although with some error, but not with the first-order scheme. Finally, to further emphasize this change in frequencies caused by the fluid-structure interaction, Table 5.10 compares the "dry" natural frequencies, obtained with the Solid Stress Normal Modes solver of STAR-CCM+, with the "wet" or in-water natural frequencies. The data shows a pronounced change of the natural frequencies of vibration due to submergence in water. Besides accounting for effects of submergence, it is also possible to verify that the effects of flow velocity on the dynamic behaviour of the structure are predicted. Figure 5.9 is a clear indication of this, showing that, as the flow velocity increases, so do the periods of the response.

Even though it is important to ensure that the methodology can capture the hydroelastic effects that lead to flutter, for submerged flexible hydrofoils the critical velocity for divergence is typically lower than that of flutter [81]. Thus, it is of important to evaluate

Table 5.10: Comparison between "dry" and "wet" natural frequencies.

Dry Frequency (Num.)	Wet Frequency (Num.)	Wet Frequency (Exp.) [75]
98.5 Hz	39.1 ± 1.95 Hz	43 Hz
308.9 Hz	140.6 ± 1.95 Hz	171 Hz
574.6 Hz	-	291 Hz
621.6	-	560 Hz

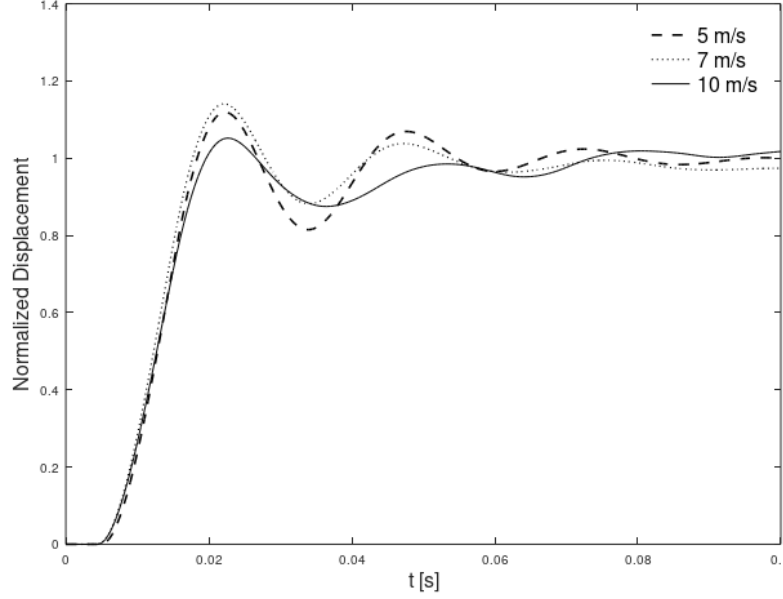


Figure 5.9: Comparison between displacement response at the tip of the foil for flow velocities of 5, 7 and 10 m/s. The displacements are normalized with respect to the corresponding static displacements, for easier comparison.

how the methodology fares in predicting this type of instability.

This can be done with the current setup since the NACA66312 hydrofoil considered here is one example of a structure whose hydroelastic unstable failure mode is divergence. Its critical velocity was estimated, by Ducoin and Young through a simplified analysis, to be of 29.6 m/s, for an initial angle of attack of 8° [149].

To understand if the model can predict divergence, tests were run to get a estimation of the critical point and compare it with the aforementioned result. To do this, the Southwell method [149–151] is used together with the numerical solutions. Here, some assumptions are considered such as linear hydrodynamics (meaning that $\frac{\partial C_L}{\partial \theta} \approx \text{constant}$), negligible coefficient of hydrodynamic moment and location of the center of mass on the elastic axis. With these simplifications the Southwell method gives the following relation between dynamic pressure, $Q = \frac{1}{2}\rho_f v^2$, divergence dynamic pressure, $Q_D = \frac{1}{2}\rho_f v_D^2$, angle of deformation, θ , and initial angle of attack, α

$$\theta = \frac{Q\alpha}{Q_D - Q}. \quad (5.2)$$

Considering that the fluid density remains constant, and with further manipulation of the expression above, it yields

$$\frac{1}{\theta} = \frac{v_D^2}{\alpha} \frac{1}{v^2} - \frac{1}{\alpha} \quad (5.3)$$

meaning that the inverse of the deformation angle is proportional to the inverse of the square of the flow velocity. Thus, by collecting a set of points from the numerical solution, such a line can be fitted to the data; the interception of this fitted curve to the abscissa gives the inverse of the square of the divergence speed. To execute this procedure, simulations were run for 4, 4.5, 5, 6, 7 and 10 m/s, at an initial angle of attack of $\alpha = 8^\circ$, and using the first-order discretization scheme. The angular deformations were extracted and plotted in Figure 5.10, with the fitted line.

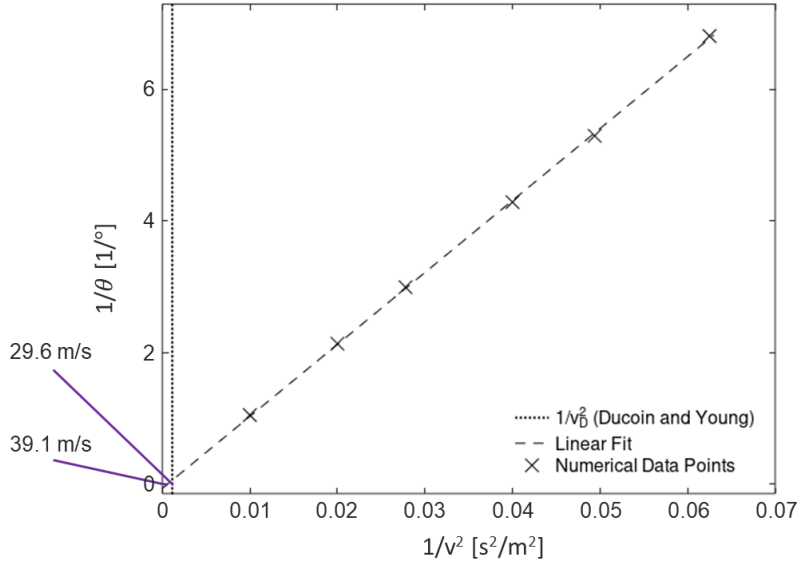


Figure 5.10: Application of the Southwell method for estimating divergence velocity from numerical data.

The result of this estimation yields a divergence velocity of 39.1 m/s, which compares reasonably with the prediction in the literature. However, the fitted line is seen to intercept the ordinate at $\frac{1}{\theta} \approx \frac{1}{14^\circ}$. Comparing this with the value of $\frac{1}{\theta} \approx \frac{1}{8^\circ}$ indicated by (5.3), it is verified that some error is associated to this estimation. This deviation may be explained by possible errors introduced by the assumptions inherent to the Southwell method. Especially linear hydrodynamics, considering that the deformations at speeds close to v_D will likely change $\frac{\partial C_L}{\partial \theta}$.

Having seen how numerical predictions compare to experimental data and estimations in the literature, further investigation is made regarding two of the main features of the model: the temporal discretization order and the "directionality" of the coupling. Hence, the selected method (two-way FSI with first-order temporal discretization) is compared to a two-way FSI model with second-order discretization and a one-way FSI model with first-order discretization. The tests were run for the 8° angle-of-attack case and, since the interest in this comparison is mainly qualitative and the computational resources are limited, slightly coarser meshes were used than those employed in the validation of the hydroelastic displacements.

The displacement and velocity responses were measured at the leading edge, on the free tip, and compared. Figures 5.11, 5.12 and 5.13 show this comparison of the three solutions in terms of structural behavior, i.e., the displacement response, the velocity

response and the phase space of the system, respectively. Unsurprisingly, the one-way FSI model under-performed in terms of accurately predicting the transient behaviour of the structure and deviates significantly from the solutions obtained with the two-way FSI models. Nevertheless, it appears to be suitable for predicting steady-state displacements, for this simple case. Furthermore, the concern towards numerical dissipation introduced by the first-order temporal scheme is verified by the results. It is clear, again, that the higher frequency components of the velocity and displacement response were more affected by this artificial damping than their lower frequency counterparts.

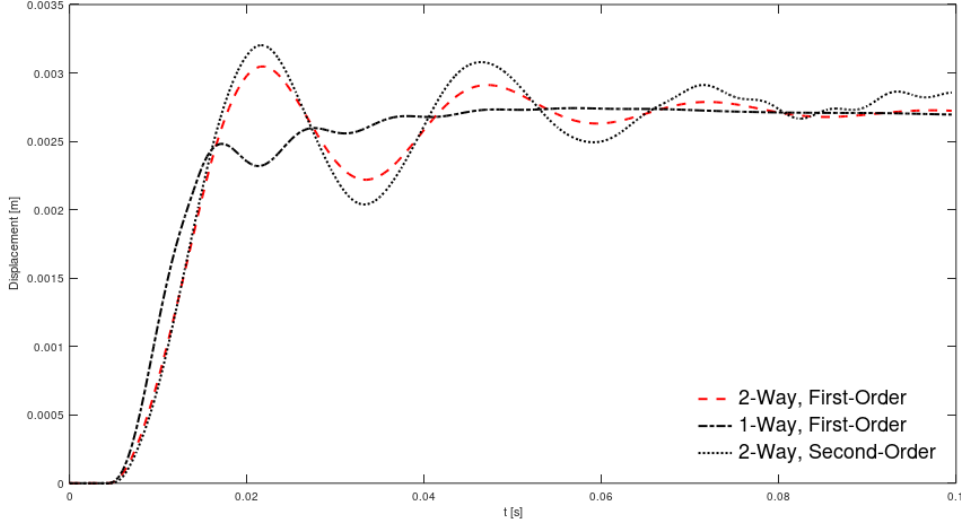


Figure 5.11: Displacement response predicted by the three different models.

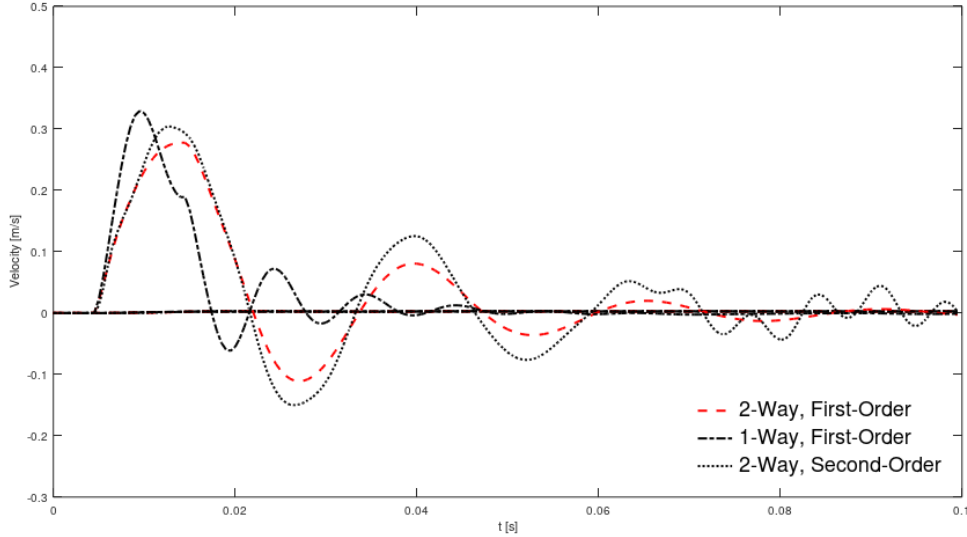


Figure 5.12: Velocity response predicted by the three different models.

Additionally, Figure 5.14 plots the lift coefficient against the drag coefficient to monitor how the fluid traction evolves throughout the solution for the different models. Given that, for the one-way FSI, the coupling is established as fluid-to-solid, it is to be expected that the motion of the structure does not affect the evolution of the flow solution, here represented by the force coefficients. On the contrary, for both of the two-way FSI models,

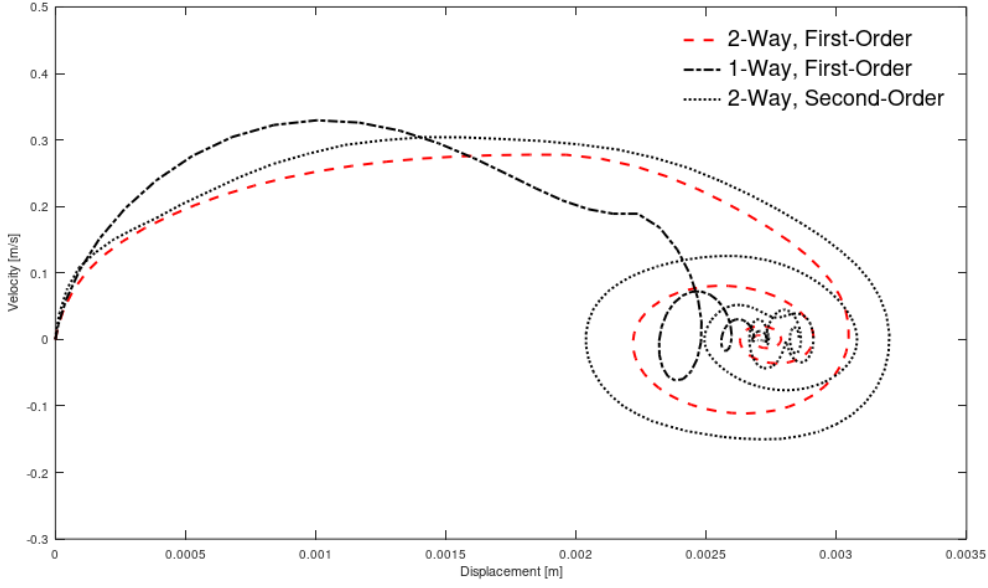


Figure 5.13: Phase-space trajectory of the system as predicted by the three different models.

the influence of the solid transient behaviour on the force coefficients is obvious. Once again, the existence of numerical dissipation is evidenced by the difference shown between first and second-order time discretization. Moreover, even though the transient effects are very different when comparing the force coefficients, the solution still tends to relatively similar values in steady-state, for all the analyses. This means that, for this specific case, the flow is not considerably altered by the deformations of the structure, and also explains why the steady-state displacements predicted by one- and two-way FSI are very similar.

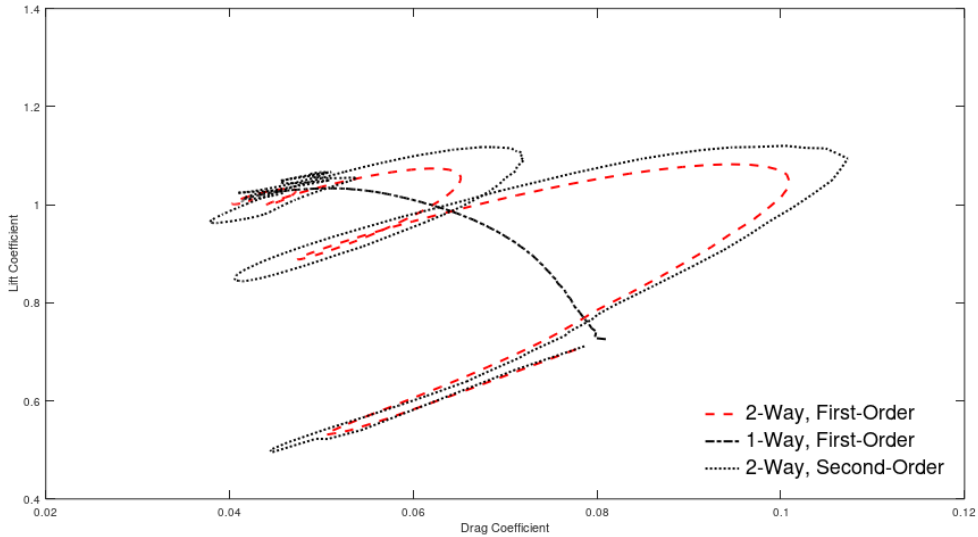


Figure 5.14: Lift coefficient versus Drag coefficient evolution as predicted by the three different models.

To conclude these additional tests, further studies were conducted on the influence of the time step size. Three simulations were run for 0.1 s, with time step sizes of $\Delta t = 10^{-3}$ s,

$\Delta t = 5 \cdot 10^{-4}$ s and $\Delta t = 2.5 \cdot 10^{-4}$ s. The resulting displacement responses are plotted in Figure 5.15. As expected, the numerical damping is more important for the larger time step and decreases as the time step size shrinks. Furthermore, the time step size does not seem to bear relevance on the steady-state solution, given that all solutions predict oscillations around the same equilibrium point.

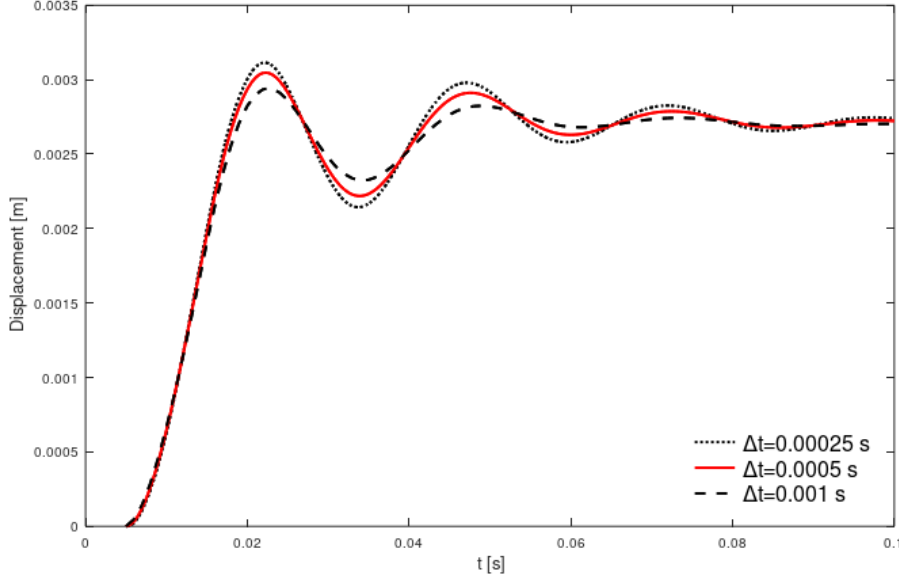


Figure 5.15: Change in the numerical solution with the time step size.

5.3 Moth Rudder

5.3.1 Model Setup

To model the hydroelastic behavior of the moth rudder, the geometry shown in Figure 5.16 is used.

The characteristic dimensions are the span of the foil, $b_m \approx 0.7$ m, the average chord of the foil, $c_m \approx 0.06$ m and the length of the rudder strut, $h_m \approx 1$ m. Regarding the orientation of the rudder, the roll and pitch were set to -29.5° and 2.1° , respectively.

The fluids constituting the multiphase flow are seawater, with a constant density, $\rho_w = 1025.0$ kg/m³, and dynamic viscosity, $\mu_w = 1.218\text{E-}3$ Pa·s; and air, with constant density, $\rho_a = 1.15$ kg/m³, and dynamic viscosity, $\mu_a = 1.8\text{E-}5$ Pa·s. The rudder is constituted by steel with density $\rho_s = 7830.0$ kg/m³, Young's Modulus $E = 200$ GPa and Poisson's Ratio $\nu_s = 0.285$.

Similarly to the validation case, the Reynolds-averaged Navier-Stokes equations are solved using the $k - \omega$ SST turbulence model and an All $y+$ Wall Treatment was selected to enable the solution of the boundary layers with wall functions.

The boundary conditions shown in Table 5.11 were applied to the fluid domain. Here, the Inlet includes also the bottom, top and side walls. Regarding the rudder structure, the imposed boundary conditions are shown in Table 5.12.

Regarding the fluid-structure coupling, the relevant selected properties are shown in Table 5.13, below.

The important information pertaining to the setup of the VOF solver, as well as the VOF waves, is presented in Table 5.14.

Table 5.12: Solid boundary conditions for the moth rudder simulation.

Boundary	Conditions
Fixed Top	Fixed Constraint
Foil Surface	Fluid Traction
Strut Surface	Fluid Traction

Table 5.13: Fluid-structure coupling highlighted settings for the moth rudder simulation.

Property	Value
FSI Load Ramping - Start	0.1 s
FSI Load Ramping - End	0.2 s
Stabilization Method	Boundary Interface Added Mass
Mesh Morpher	BSpline
Recompute Interfaces	Yes
Morph at Inner Iterations	Yes

results are presented in Figures 5.18 and 5.17. Attending to this information, the fluid mesh with 6.1M cells was selected. Regarding the solid mesh, the 1.4M elements were initially deemed to be suitable in terms of the independence of the solution from the mesh size. However, further consideration led to a refinement to 1.9M elements, for reasons related to compatibility between fluid and solid meshes at the interface.

Fluid

For the fluid region, an automated mesh (Figure 5.19; details on Figure 5.20) was generated using the Trimmed Cell mesher and the Prism Layer mesher. Similarly to what was done for the validation case, the settings of the Prism Layer mesher, in Table 5.16, were tuned to keep the prism layers from collapsing at the thin trailing edge of the foil and strut.

The mesh is defined by two different types of controls: the *Default Controls*, which set the general default values for the whole mesh, and the *Custom Controls*, that control the mesh properties in specific regions. The Default and Custom Controls for this mesh are shown in Tables 5.17 and 5.18, respectively.

Solid

The solid mesh of the moth was generated using an Automated Mesh with the Tetrahedral and the Thin meshers. As a result, the most common element type is the linear tetrahedron, as shown in Figure 5.21, which is identified in STAR-CCM+ by the code number 10. However, linear wedge elements (code number 13) are also found near the trailing edge, as well as a few linear pyramid elements (code number 14). The general mesh parameters are defined in Table 5.19, and the Custom Controls are defined in Table 5.20.

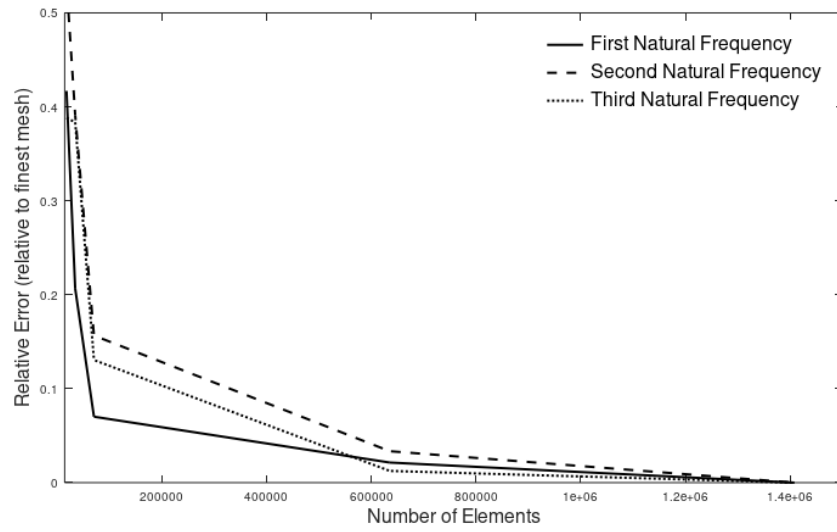


Figure 5.17: Variation of the solution for the first three natural frequencies with the mesh size, for the moth rudder case.

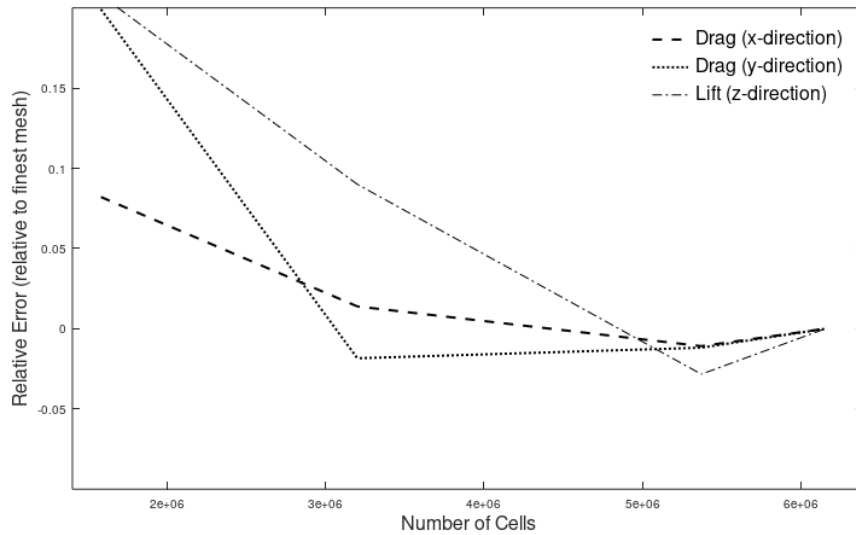


Figure 5.18: Variation of the solution for the fluid forces with the mesh size, for the moth rudder case.

Table 5.14: Properties setup for the VOF solver and waves.

Property	Value
Discretization Scheme	HRIC
Solution Strategy	Implicit Multi-Step
Number of steps	3
VOF Waves	Fifth-Order Waves
Point on Free Surface	[0.0,0.0,0.0]
Vertical Direction	[0.0,0.0,1.0]
Advancing Direction	$[\cos(50^\circ), \sin(50^\circ), 0.0]$
Current	[0.0,0.0,0.0]
Wind	$[12.86\cos(50^\circ), 12.86\sin(50^\circ), 0.0]$ m/s
Additional Velocity	[12.86,0.0,0.0] m/s
Wave Height	0.25 m
Water Depth	10 m
Wave Period	2.0 s
Light Fluid	Air
Heavy Fluid	Seawater

Table 5.15: Important properties of remaining solvers for the moth rudder simulation.

Property	Value
Time Discretization	Implicit First-Order
Time Step Size	0.003888s
Maximum Inner Iterations	10
Convection Discretization	Second-Order Upwind
Stiffness Matrix Update (Update Frequency)	Modified Newton (First two iterations)

Table 5.16: Prism Layer mesher settings for the moth rudder simulation.

Property	Value
Minimum Thickness Percentage	0.01
Layer Reduction Percentage	30
Boundary March Angle	10
Concave Angle Limit	0.0
Convex Angle Limit	360.0

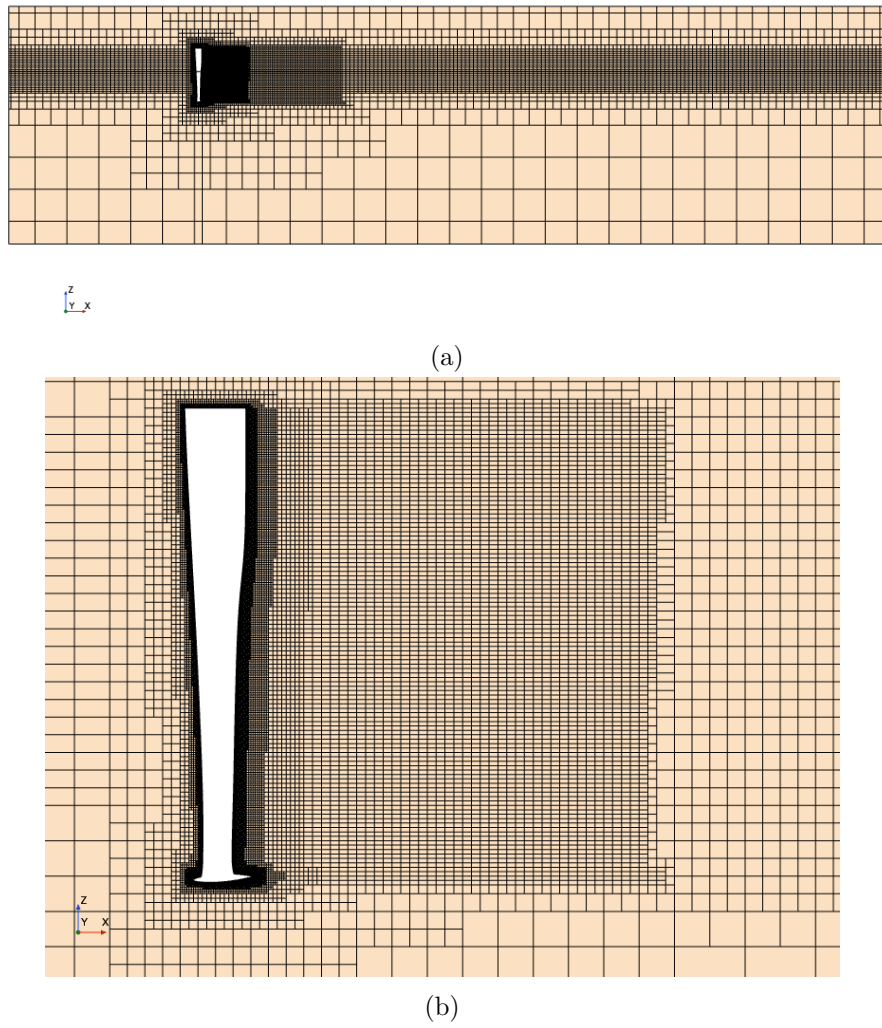


Figure 5.19: Two-dimensional section of the fluid mesh for the moth rudder simulation; (a) whole domain, (b) close to the foil.

Table 5.17: Default Controls for the fluid mesh around the moth rudder.

Property	Value
Base Size	22.0 m
Target Surface Size	22.0 m
Minimum Surface Size	$6.6 \cdot 10^{-4}$ m
Number of Prism Layers	5
Prism Layer Total Thickness	0.002 m
Prism Layer Stretching	1.3
Surface Growth Rate	3.0
Volume Growth Rate	Medium

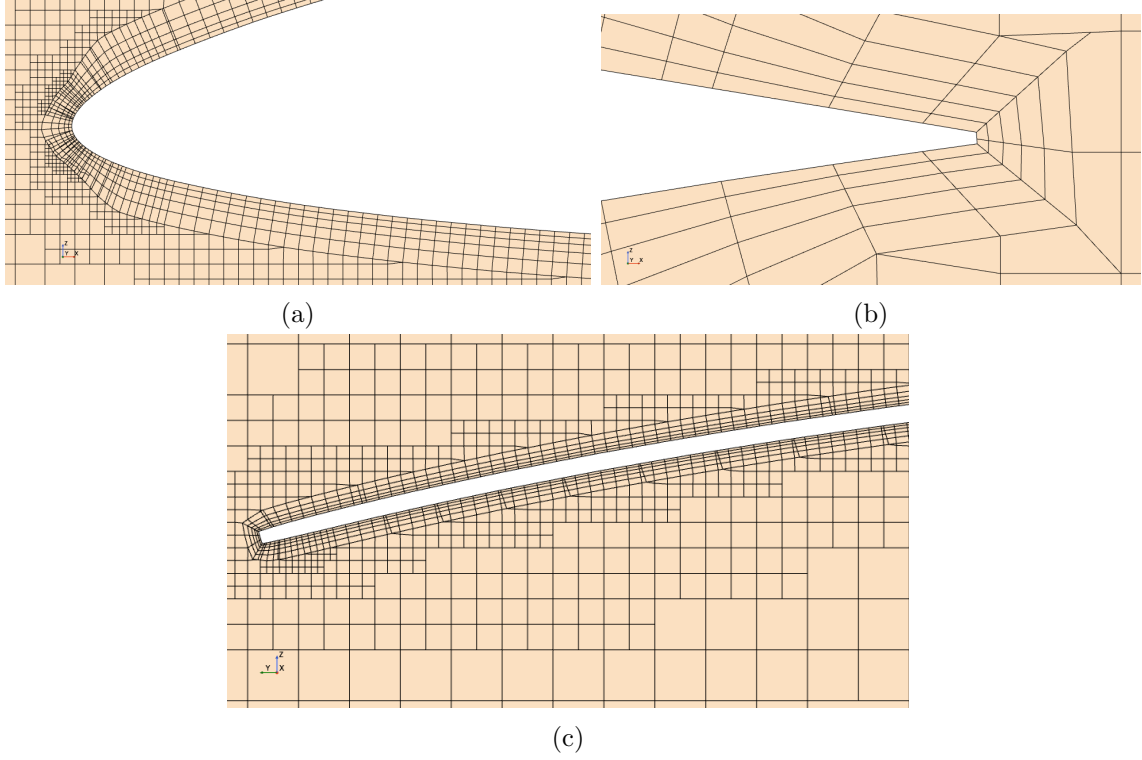


Figure 5.20: Details of the fluid mesh for the moth rudder simulation; (a) tip of the bulb, (b) trailing edge of the foil, (c) front-view detail of the foil tip.

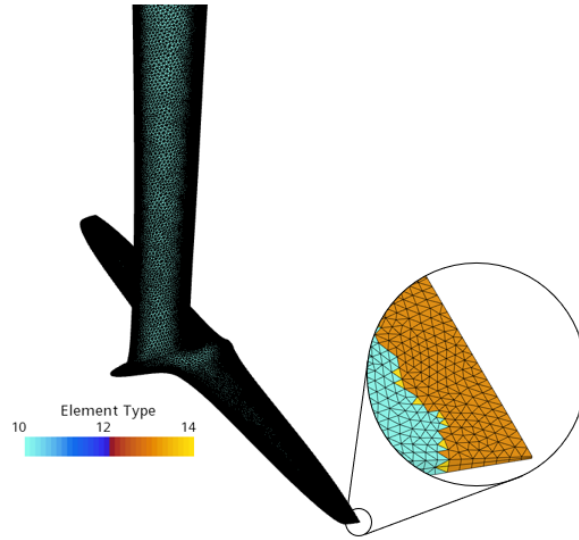


Figure 5.21: Rudder mesh, with identification of the element type (10-linear tetrahedron; 13-linear wedge; 14-linear pyramid); general view with detailed view of the foil tip and trailing edge.

5.3.3 Results and Discussion

Different time-series of the response of the moth rudder were recorded by placing eight probes in distinct points on the structure, collecting information on the components of the

Table 5.18: Custom Controls for the fluid mesh around the moth rudder.

Region	Property	Value
Foil and Strut Edges	Target Surface Size	0.0005 m
	Minimum Surface Size	0.00132 m
Foil and Strut Surfaces	Target Surface Size	0.0088 m
	Minimum Surface Size	$1.1 \cdot 10^{-4}$ m
Fixed Top Surface	Surface Growth Rate	1.05
	Prism Layer Stretching	1.5
Wake	Isotropic Size	0.066 m
	Distance	3.0 m
Close Wake	Anisotropic X Size	0.033 m
	Anisotropic Y Size	0.0165 m
	Anisotropic Z Size	0.0165 m
	Distance	1.0 m
Strut Region - Above Free Surface	Custom Size	2.2 m
	Number of Prism Layers	1
	Prism Layer Total Thickness	0.0001 m
Strut Region - Bellow Free Surface	Custom Size	0.2 m
Close Strut Region - Bellow Free Surface	Custom Size	0.11 m
Hydrofoil Bulb Region	Custom Size	0.0012 m
Free Surface Region	Anisotropic X Size	0.077 m
	Anisotropic Y Size	0.077 m
	Anisotropic Z Size	0.035 m

Table 5.19: Default Controls for the solid mesh of the rudder.

Property	Value
Base Size	0.001 m
Target Surface Size	0.001 m
Minimum Surface Size	0.0001 m
Surface Growth Rate	1.3

displacement and velocity in the x , y and z direction. The location of these eight probes and their respective identification is shown in Figure 5.22.

To visualize the response of the moth rudder, the trajectories taken by each probe through three-dimensional space were plotted in Figure 5.23, and the respective projections on different planes are shown in Figure 5.24. To include information on the velocity response, the successive states of the probes were plotted in phase space, with each direction being considered individually; this is presented in Figure 5.25.

From any of these data representations it is clear that the moth rudder undergoes cyclical, stable and sustained oscillations. However, this does not constitute a case of limit cycle oscillation, but rather a forced response, excited by the motion of the waves, as will

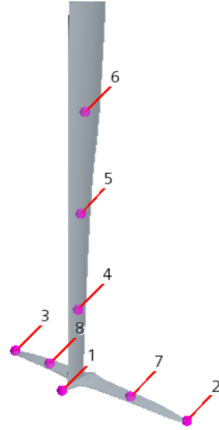


Figure 5.22: Location and identification of the probes on the moth rudder.

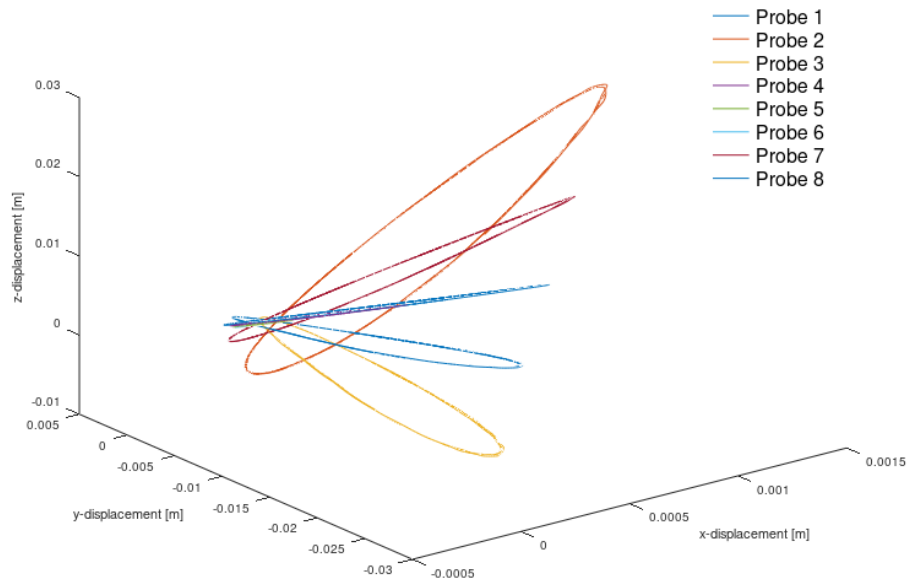


Figure 5.23: Trajectory of the different probes through three-dimensional space.

Table 5.20: Custom Controls for the solid mesh of the rudder.

Region	Property	Value
Foil and Strut Edges	Target Surface Size	0.1 m
	Minimum Surface Size	0.001 m

be discussed ahead. The points where maximum amplitudes are recorded vary according to the direction of the motion that is considered. Probe 2, located at the left tip of the rudder foil, has a trajectory with the largest maximum displacement in the x direction, around 1.24 mm, and in the z direction, 27.6 mm. On the other hand, the right tip of the rudder foil, represented by probe 3, displays the largest maximum displacement in the y direction, with a magnitude of 27.7 mm. Regarding the velocities, similar results are observed, with the maximum values in the x and z direction associated to probe 2, and the maximum values in the y direction being recorded by probe 3. Moreover, higher frequency phenomena is more evident in the phase space of the motion in the x direction.

Regarding the remaining data, probes 7 and 8 exhibit smaller amplitudes of motion than the ones placed at the tips, as expected, given that (i) the rigid body motion of the foil as the strut bends and (ii) the bending of the foil itself promote higher displacements away from the strut. Additionally, progressively larger trajectories are drawn by the probes located on the strut as their distance to the fixed top increases. The displacements found on the location of probe 6 are almost negligible, due its proximity to the fixed surface and because the upper region of the strut is not submerged.

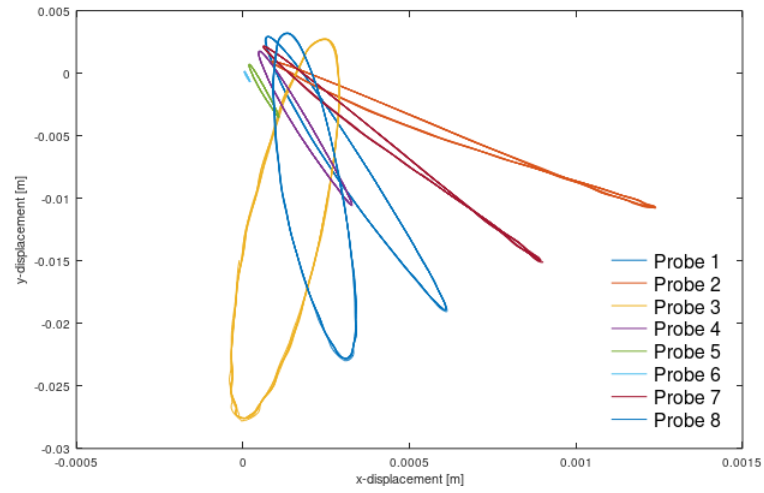
The stresses in the moth rudder caused by hydroelastic effects can also be studied. The Von Mises stress for a time instant of maximum deformation is displayed over the deformed structure in Figure 5.26. The middle of the strut is identified to be the maximum stress region, resulting from the bending of the strut (evidenced in the aforementioned figure) and wave impact. The Von Mises stress nears 90 MPa, at maximum in said region. This is typically far bellow the fatigue strength of steels, meaning that no fatigue life analysis is needed. Nevertheless, the methodology presented here allows for such studies to be conducted, since the loading history can be easily obtained.

To evaluate the effects that the water-rudder coupling exerts on the transient behaviour of the structure, a modal analysis is first performed, for a "dry" configuration. The first three obtained mode shapes and natural frequencies are represented in Figure 5.27 and Table 5.21.

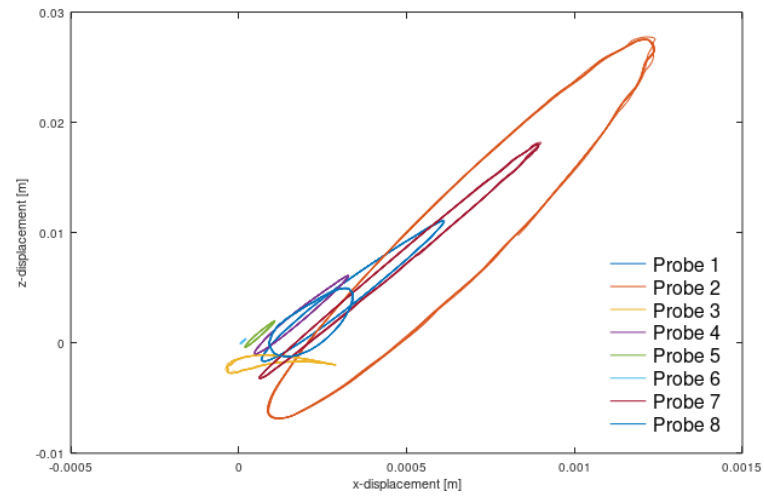
Table 5.21: First three modes and corresponding "dry" natural frequencies of the steel rudder.

Mode	Mode Shape Description	Frequency
First Mode	Mainly bending of strut	12.0 Hz
Second Mode	Bending of strut + anti-symmetric bending of foil	42.6 Hz
Third Mode	Mainly torsion of strut	49.5 Hz

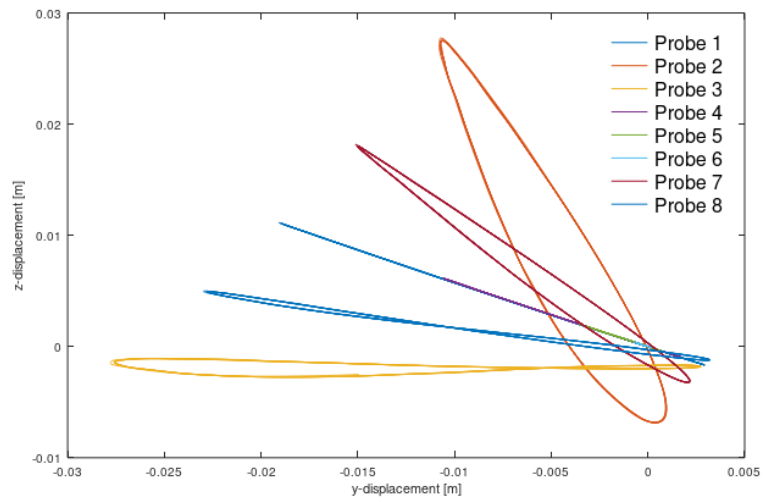
A Fast Fourier transform is then applied to the components of the velocity responses to search for peaks in the spectra. This analysis is made separately for the strut (probes 1, 4, 5 and 6) and for the foil (probes 2, 3, 7, 8). The resulting spectra are shown in Figure 5.28. The velocity responses were composed of 512 data points, separated by a time step



(a)



(b)



(c)

Figure 5.24: Trajectories of the probes projected on (a) Oxy plane, (b) Oxz plane (c) Oyz plane.

size of $\Delta t = 0.003888$ s, meaning a resolution of

$$\Delta f = \frac{1}{512 \cdot 0.003888} \approx 0.5 \text{ Hz.} \quad (5.4)$$

and indicating an uncertainty of 0.25 Hz in the measurements of the frequency of the peaks. Two relevant local maxima are identified at 2 ± 0.25 Hz and at 3.5 ± 0.25 Hz. However, no conclusions can be directly drawn from such identification, since these peaks also appear in the the spectral analysis of the fluid loads. This could be for one of two reasons: (i) the vibrations of the structure affect the flow and induce variations in the loads at these frequencies, or (ii) these peaks represent cyclical phenomena intrinsic to the flow (such as the waves). This distinction is important since it indicates if the peaks correspond to the "wet" natural frequencies or merely represent the response to a forced excitation at a given frequency. To check which of the two hypothesis is correct, the spectral analysis of the fluid loads can be studied for a rigid structure or, alternatively, a one-way FSI analysis can be performed and the results compared. The latter is shown later.

Since structural deformations in FSI can substantially affect the fluid mesh, it is important to assess the quality of such mesh after morphing, to conclude on the effects that this deformation may have on the flow solution. Table 5.22 compiles a set of recommendations on the minimum or maximum values of the main mesh quality measures, and compares these with the the values characterizing the mesh of this model at maximum deformation.

Table 5.22: Mesh quality measures, their recommended values and corresponding values on the deformed mesh and undeformed mesh.

Mesh Quality Measure	Recommended [102]	Deformed Mesh	Undeformed Mesh
Face Validity	=1	1	1
Cell Quality	$> 10^{-5}$	$7 \cdot 10^{-5}$ to 1	$7 \cdot 10^{-5}$ to 1
Volume Change	$> 10^{-2}$	$1.1 \cdot 10^{-2}$ to 1	$1.1 \cdot 10^{-2}$ to 1
Skewness Angle	$< 85^\circ$	0° to 84.89°	0° to 84.89°

This information shows that, even though the mesh is being deformed as the solution progresses, this morphing does not negatively impact the quality of the mesh. In fact, it did not alter the maximum and minimum values of the mesh quality measures, when compared to the undeformed grid.

Here, the face validity is a measure of the direction of the normal at the cell faces (pointing away from the cell centre results in a good face validity, of value 1); cell quality is a function of the orientation of the cell faces and distribution of centroids of neighbouring cells (the perfect cell has a cell quality of 1); volume change indicates a ratio between the volume of a cell and that of the largest adjacent cell; skewness angle is a measure of the non-orthogonality of the cell, which was detailed in Sub-section 4.3.2.

It is verified that, for the deformed mesh, all the quality measures remain within the recommended thresholds. To complete the description presented in Table 5.22, it may be mentioned that over 96% of cells in the deformed mesh showed a volume change between 1 and 0.1; 99.5% of the cells displayed a skewness angle inferior to 60° , and 92% showed skewness angles bellow 30° ; 93% of cells had a cell quality over 0.5, while 66% were over 0.9 regarding this measure.

The results from the present methodology are now compared to the state-of-the-art analysis in naval design, that accounts for one-way FSI. The same data that was collected

in the preceding part of this section for the two-way FSI was also computed for the one-way FSI analysis. The comparison between the representative values of both structural solutions is shown in Table 5.23. Additionally, the spectral analysis for the velocity response is presented, in Figure 5.29.

Table 5.23: Comparison between results given by one- and two-way FSI analyses.

Quantity	One-way FSI	Two-way FSI
Maximum Displacement		
<i>x</i> direction	1.24 mm (Probe 2)	1.24 mm (Probe 2)
<i>y</i> direction	28.9 mm (Probe 3)	27.7 mm (Probe 3)
<i>z</i> direction	28.5 mm (Probe 2)	27.6 mm (Probe 2)
Maximum Velocity		
<i>x</i> direction	9.61 mm/s (Probe 2)	7.52 mm/s (Probe 2)
<i>y</i> direction	256 mm/s (Probe 3)	196 mm/s (Probe 3)
<i>z</i> direction	267 mm/s (Probe 2)	216 mm/s (Probe 2)
Maximum Von Mises Stress		
Magnitude	89.1 MPa	89.6 MPa
Region	Strut middle	Strut middle

From this condensed information, it is verifiable that the structure deforms in similar ways for the one-way and for the two-way FSI analyses. This is shown by the similarity in maximum displacements and the fact that the probes predicted to be displaced the most are consistent among the analyses. Moreover, the maximum Von Mises stresses obtained by the two solutions are identical and located on the same region. The main differences arise in the maximum velocity predictions, which are significantly higher for the one-way FSI analysis. This is consistent with the spectral analyses that show that the one-way FSI has larger velocity amplitudes and is much noisier than the two-way FSI. These results are a consequence of the inability of the one-way FSI to accurately model fluid-induced damping and inertia.

Further investigation of the one-way FSI velocity spectrum also verifies that the two main peaks are, in fact, due to cyclical phenomena intrinsic to the flow (waves; possibly ventilation) and not necessarily the "wet" natural frequencies of the structure. This can be affirmed since both peaks are presented in the two different analyses of one- and two-way FSI. The results are, here, inconclusive in regards to the determination of the in-water natural frequencies. These frequencies are certainly "hidden" in the spectra, but do not stand out due to the characteristics of the excitation and, thus, are not clearly identifiable. However, if the determination of the "wet" natural frequencies was to be done unequivocally, another run of the simulation without modelling the waves would suffice.

5.4 General Recommendations

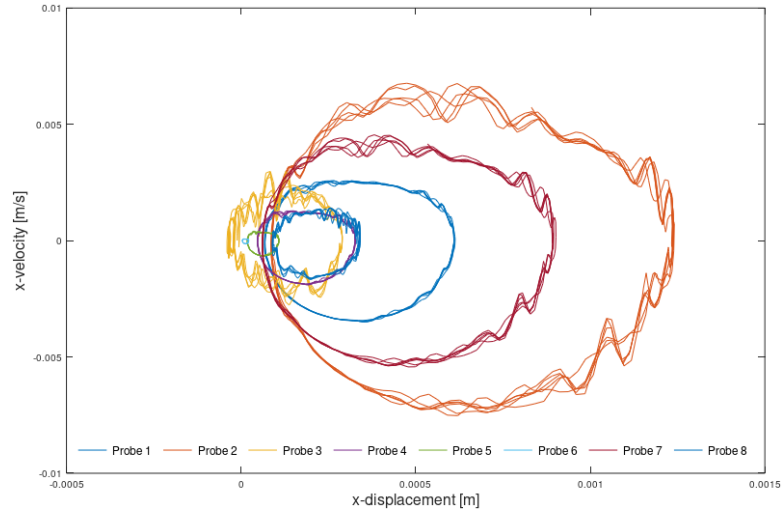
Considerable troubleshooting was needed in the successive stages of the project in order to get robust, fully-working models. Notwithstanding the fact that the errors or bad practices are usually obvious when spotted, it may be useful to register some brief practical suggestions, should this methodology be used by a third-party in future analyses.

Some challenges may be found in achieving a stable numerical solution, depending on the density ratio between fluid and structure, on flow velocity, and on geometry. Based on

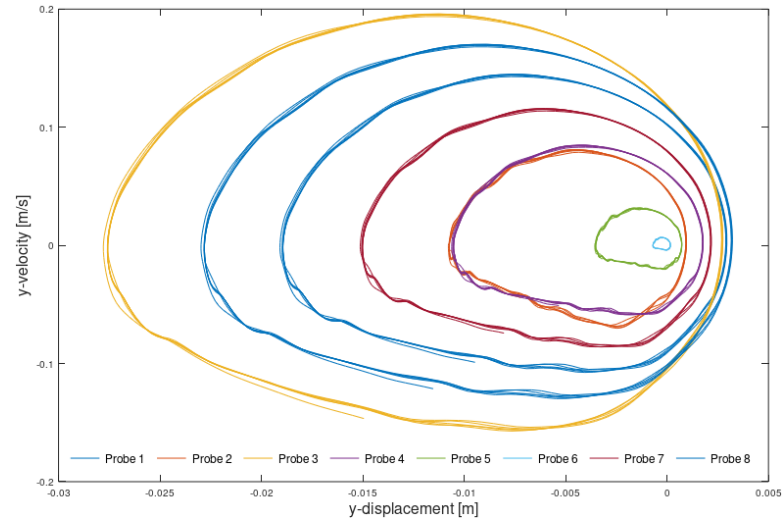
experimentation and research throughout this project, to fix such numerical instabilities, it is suggested to *(i)* decrease (or increase) the time step size, *(ii)* introduce/change the ramping of the fluid traction, *(iii)* consider ramping the flow velocity, *(iv)* increase the number of sub-iterations within each time step to tighten the coupling, *(v)* obtain a solution with a steady solver and use it to initialize the unsteady simulation.

Although the morphing of the fluid mesh was found to be very robust, it can, in some cases, lead to bad-quality or even negative-volume cells. In face of such difficulties, the useful approaches included *(i)* reconsidering the size or geometry of the fluid domain, *(ii)* approximately matching the mesh sizes at the interface, *(iii)* checking if the interface is correctly defined, *(iv)* assessing the adequacy of the mesh morpher boundary conditions, *(v)* ensure that prism layers are nicely generated and not collapsed.

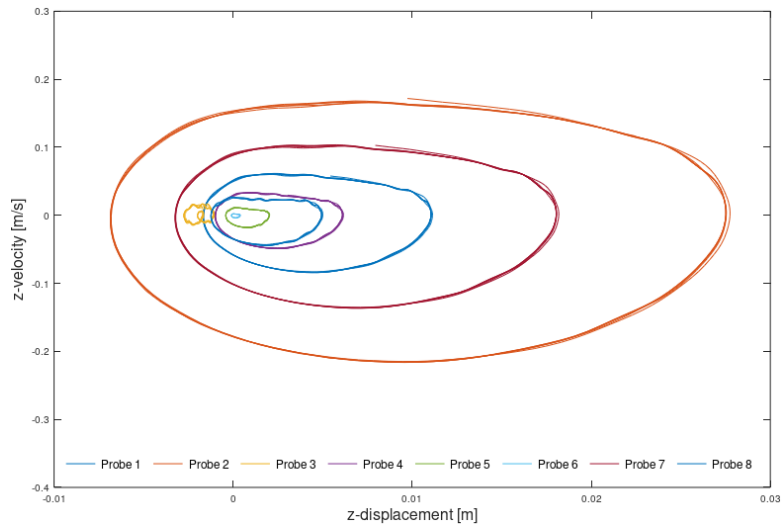
Additionally, two-way FSI simulations can easily turn out to be too computationally expensive, especially if care is not taken to limit the costs. One is advised to consider adequate mesh sizes, since these strongly influence the computational time needed to obtain the solution. Even though this is also true for standard fluid-only and structure-only simulations, note that, for the particular case of FSI, additional steps exist whose computation is influenced by mesh size. For example, the mapping of information from one mesh to the other, and the deformation of the fluid grid. What is more, both of this steps are done at every sub-iteration, if a tight coupling is required.



(a)



(b)



(c)

Figure 5.25: States of the probes regarding the motion in the (a) x direction , (b) y direction (c) z direction.

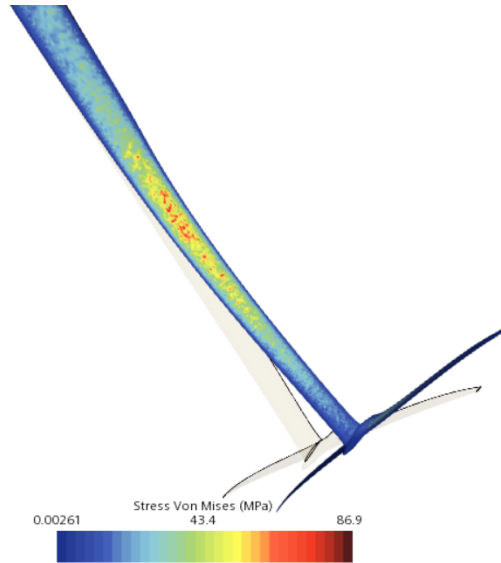


Figure 5.26: Von Mises stress displayed over deformed configuration (deformations multiplied by scale factor of 5), with undeformed configuration presented for comparison.

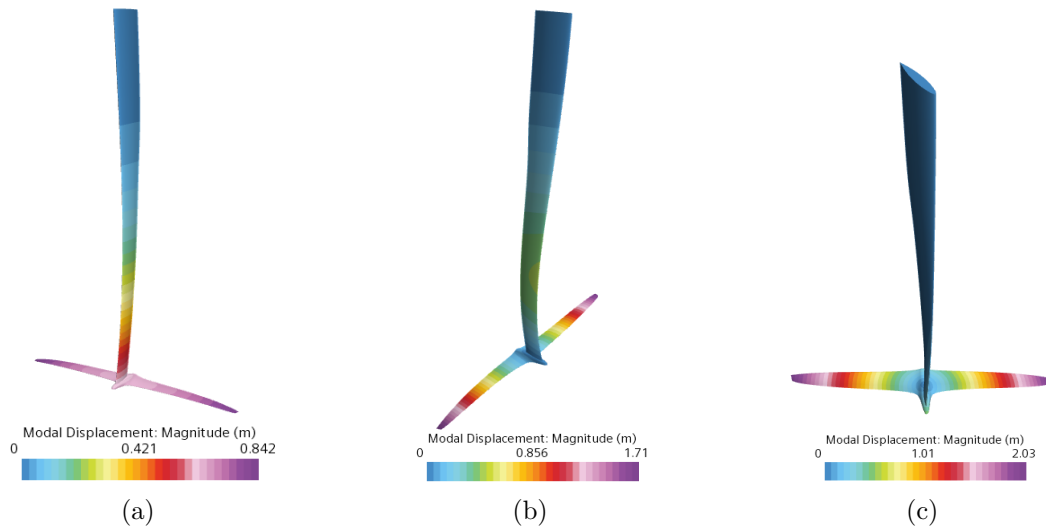


Figure 5.27: First three natural mode shapes of the steel rudder; (a) first mode shape, (b) second mode shape, (c) third mode shape.

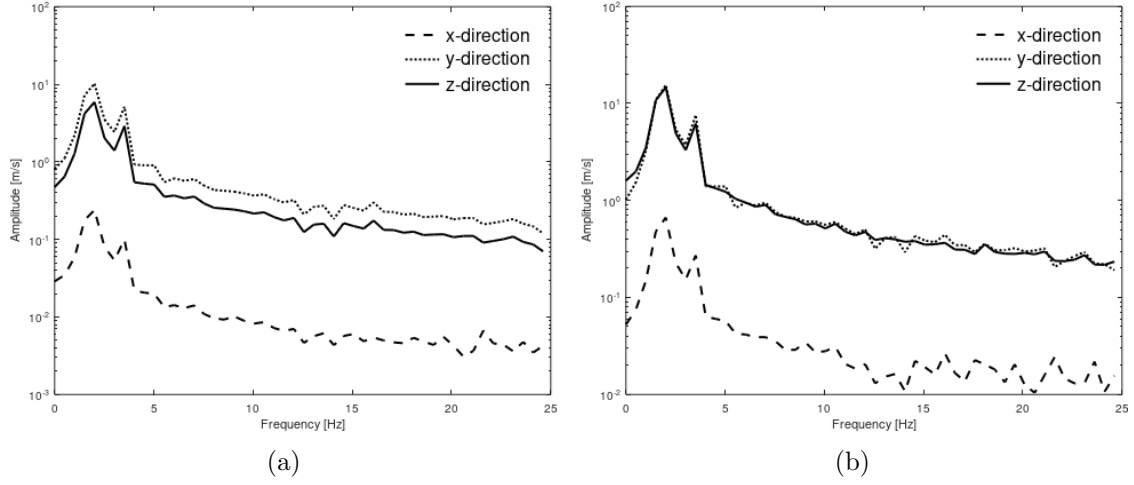


Figure 5.28: Velocity spectra for the x , y and z direction, for the responses recorded (a) on the strut, (b) on the foil.

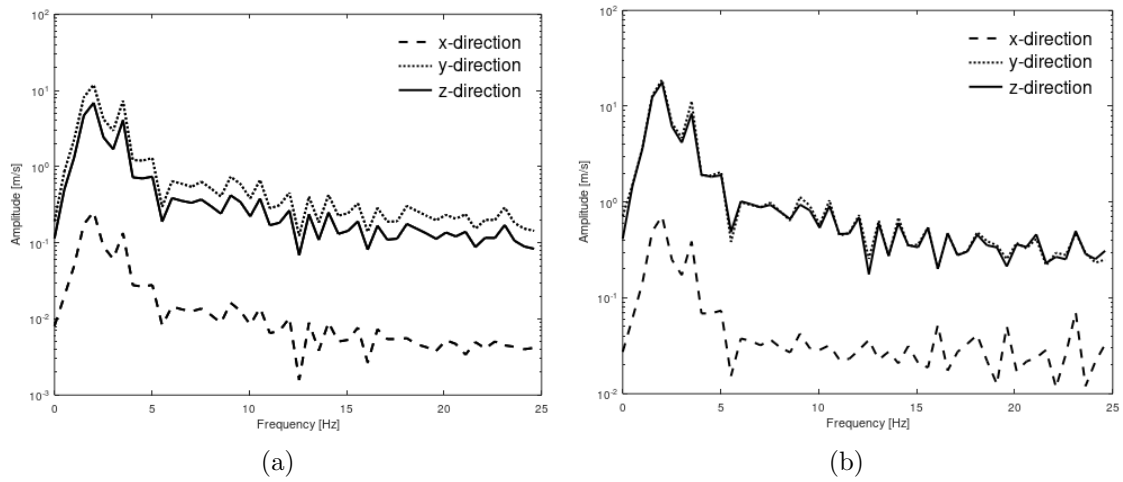


Figure 5.29: Velocity spectra for the x , y and z direction, for the one-way FSI responses recorded (a) on the strut, (b) on the foil.

Conclusion

6.1 Conclusions

In this dissertation, a numerical methodology was developed to analyse the hydroelastic effects on appendages of high-speed boats under racing conditions. The methodology was based on the use of the Finite Volume and Finite Element methods. To describe the fluid domain, an Arbitrary Lagrangian-Eulerian formulation of the unsteady Reynolds-Averaged Navier-Stokes equations was used; the Volume of Fluid method was chosen to solve the volume fraction equation and model the free surface; and a geometrically nonlinear finite element formulation was employed to model the structural domain. A partitioned approach was used to numerically solve the fluid-structure interaction system, and tight coupling was enforced through sub-iterations.

The proposed methodology was applied to a case study in the literature and hydroelastic displacements predicted by the numerical model were shown to generally agree with experimental data. However, the accurate prediction of other effects was found to be dependent on specific configurations:

- The use of the first-order temporal discretization was seen to be useful for two-way FSI models when sharp capturing of the free surface is needed without overly-restrictive requirements for the time step size. It was demonstrated that it is possible to get estimations for the static divergence speed, using this temporal scheme. The stabilizing effect of using this lower-order scheme was also mentioned. Nevertheless, the excessive numerical dissipation evidenced in the solutions obtained with this configuration constitutes a significant downside. Higher-frequency phenomena were quickly damped out, making the first-order temporal discretization unfit for most transient analysis;
- The second-order temporal discretization was shown to yield better predictions for transient responses, with the possibility of capturing higher-frequency phenomena being limited only by the time step size. This upside makes this higher-order scheme more suitable for prediction of transient unstable phenomena, such as flutter. In fact, the solutions obtained with this scheme predicted, with reasonable accuracy, the "wet" natural frequencies of the structure; this was previously shown to be key for flutter modelling and is a good indicator that the methodology is suitable for design against this instability. However, the use of the second-order scheme implies smaller time step sizes, if the wavy free surface is to be captured sharply, and may lead to substantial increases in computation time;

- One-way FSI was shown to perform well both in steady state and steady cyclic conditions, while also being a less time-consuming analysis. The comparison between solutions obtained with one- and two-way FSI indicated that it is a suitable approach for hydroelastic analysis if structural deformation does not significantly affect the flow. However, when this is not the case, the predictions may critically differ from those predicted by two-way FSI and real behaviour of the system. Examples of these situations range from the onset of stall to the more dangerous instability phenomena.

Overall, two-way FSI, albeit a very powerful and, in some situations, fundamental tool, still leads to expensive models. Consequently, the decision to employ such an analysis must be based on the specific problem at study and a sound engineering judgement.

6.2 Future Work

Notwithstanding the work here present, regarding the design of the methodology and the demonstration of its applicability to the context of interest, other topics can be considered in future works. Among these, it may be relevant to highlight:

- Collect experimental data on the hydroelastic behaviour of the appendages of high-speed boats, for more adequate validation of the numerical methodology. Data regarding flutter and divergence should also be included to better understand how the methodology fares in face of unstable phenomena;
- Assess the capabilities of the methodology for capturing stall-flutter and vortex-induced vibrations. Even though these were identified to be less relevant for the current application than flutter or divergence, it would be beneficial to understand the merits or limitations of the methodology under these conditions;
- Include the prediction of cavitation and its effects. Although the boats appendages are designed to operate for sub-cavitation regimes, the phenomenon can still occur and it is interesting to comprehend how it affects the dynamic behaviour of these structures;
- Resort to the co-simulation capabilities of STAR-CCM+ to couple this software with more specialised FEM tools, such as Abaqus, allowing for more control over the structural domain discretization;
- Integrate the developed methodology in hydrodynamic and/or structural optimisation processes;
- Apply the proposed methodology to composite appendages.

References

- [1] Boris Mishel and Group Captain David Bolton. Hydrofoils at war. The RUSI Journal, 127(3):16–21, 1982.
- [2] Xueyan Zhao, Chao Cao, and Pengfei Guo. Simulation analysis study on the flutter characteristics of control rudder in high speed flight vehicle. Journal of Physics: Conference Series, 1053:012056, 2018.
- [3] Matthieu Sacher, Mathieu Durand, Élisabeth Berrini, Frédéric Hauville, Régis Duviigneau, Olivier Le Maître, and Jacques-André Astolfi. Flexible hydrofoil optimization for the 35th America’s Cup with constrained EGO method. Ocean Engineering, 157:62–72, 2018.
- [4] Stefan Jakobsson and Olivier Amoignon. Mesh deformation using radial basis functions for gradient-based aerodynamic shape optimization. Computers & Fluids, 36(6):1119–1136, 2007.
- [5] Peter K Besch and Yuan-Ning Liu. Hydroelastic design of subcavitating and cavitating hydrofoil strut systems. Technical report, David W. Taylor Naval Ship Research and Development Center Bethesda MD, 1974.
- [6] Peter K Besch and Yuan-Ning Liu. Bending flutter and torsional flutter of flexible hydrofoil struts. Technical report, David W. Taylor Naval Ship Research and Development Center Bethesda MD, 1973.
- [7] Alexander Korobkin, Emilian I Părau, and Jean-Marc Vanden-Broeck. The mathematical challenges and modelling of hydroelasticity. Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences, 369(1947):2803–2812, 2011.
- [8] Eetu Kivelä. The use of FSI models for the prediction of wind-induced vibration of modern cruise ship deck outfitting structures. Master’s thesis, Aalto University. School of Engineering, 2019.
- [9] M. Souli R. Messahel. SPH and ALE Formulations for Fluid Structure Coupling. Computer Modeling in Engineering & Sciences, 96(6):435–455, 2013.
- [10] Abbas Khayyer, Hitoshi Gotoh, Hosein Falahaty, and Yuma Shimizu. An enhanced ISPH–SPH coupled method for simulation of incompressible fluid–elastic structure interactions. Computer Physics Communications, 232:139–164, 2018.
- [11] Joseph O’Connor and Benedict D. Rogers. A fluid–structure interaction model for free-surface flows and flexible structures using smoothed particle hydrodynamics on a GPU. Journal of Fluids and Structures, 104:103312, 2021.

- [12] Steven J Lind, Benedict D Rogers, and Peter K Stansby. Review of smoothed particle hydrodynamics: towards converged lagrangian flow modelling. Proceedings of the Royal Society A, 476(2241):20190801, 2020.
- [13] Tayfun E. Tezduyar, Sunil Sathe, Ryan Keedy, and Keith Stein. Space-time finite element techniques for computation of fluid-structure interactions. Computer Methods in Applied Mechanics and Engineering, 195(17):2002–2027, 2006. Fluid-Structure Interaction.
- [14] Tayfun E. Tezduyar and Sunil Sathe. Modelling of fluid-structure interactions with the space-time finite elements: Solution techniques. International Journal for Numerical Methods in Fluids, 54(6-8):855–900, 2007.
- [15] Kenji Takizawa and Tayfun E Tezduyar. Space-time fluid-structure interaction methods. Mathematical Models and Methods in Applied Sciences, 22(supp02):1230001, 2012.
- [16] W.F. Noh. CEL: A Time-Dependent, Two-Space-Dimensional, Coupled Eulerian-Lagrange Code. Technical report, Office of Scientific and Technical Information (OSTI), 1963.
- [17] Robert M Franck and Roger B Lazarus. Mixed eulerian-lagrangian method. Methods in computational physics, 3:47–67, 1964.
- [18] C.W Hirt, A.A Amsden, and J.L Cook. An arbitrary Lagrangian-Eulerian computing method for all flow speeds. Journal of Computational Physics, 14(3):227–253, 1974.
- [19] J.C. Campbell and M.J. Shashkov. A Tensor Artificial Viscosity Using a Mimetic Finite Difference Algorithm. Journal of Computational Physics, 172(2):739–765, 2001.
- [20] Raphaël Loubère and Mikhail J. Shashkov. A subcell remapping method on staggered polygonal grids for arbitrary-Lagrangian-Eulerian methods. Journal of Computational Physics, 209(1):105–138, 2005.
- [21] J Donea, P Fasoli-Stella, and S Giuliani. Lagrangian and Eulerian finite element techniques for transient fluid-structure interaction problems. North-Holland, Netherlands, 1977.
- [22] J. Donea, S. Giuliani, and J.P. Halleux. An arbitrary Lagrangian-Eulerian finite element method for transient dynamic fluid-structure interactions. Computer Methods in Applied Mechanics and Engineering, 33(1):689–723, 1982.
- [23] J.Y. Trépanier, M. Reggio, H. Zhang, and R. Camarero. A finite-volume method for the Euler equations on arbitrary lagrangian-eulerian grids. Computers & Fluids, 20(4):399–409, 1991.
- [24] Jean Donea, Antonio Huerta, J.-Ph. Ponthot, and A. Rodríguez-Ferran. Arbitrary Lagrangian-Eulerian Methods, chapter 14. John Wiley & Sons, Ltd, 2004.
- [25] P Kjellgren and J Hyvärinen. An arbitrary Lagrangian-Eulerian finite element method. Computational Mechanics, 21(1):81–90, February 1998.
- [26] Michael Schäfer. Implicit Partitioned Coupling in Fluid-Structure Interaction, chapter 3, pages 109–164. John Wiley & Sons, Ltd, 2013.

-
- [27] Yuqi Wu and Xiao-Chuan Cai. A fully implicit domain decomposition based ALE framework for three-dimensional fluid–structure interaction with application in blood flow computation. Journal of Computational Physics, 258:524–537, 2014.
- [28] Thomas Richter. Numerical Methods for Fluid-Structure Interaction Problems. PhD thesis, University of Heidelberg, Germany, Institute for Applied Mathematics, 2010.
- [29] Alexander Shamanskiy and Bernd Simeon. Mesh moving techniques in fluid-structure interaction: robustness, accumulated distortion and computational efficiency. Computational Mechanics, 67(2):583–600, Feb. 2021.
- [30] Robert Haber, Mark S. Shephard, John F. Abel, Richard H. Gallagher, and Donald P. Greenberg. A general two-dimensional, graphical finite element preprocessor utilizing discrete transfinite mappings. International Journal for Numerical Methods in Engineering, 17(7):1015–1044, 1981.
- [31] R. Boman and J.-P. Ponthot. Efficient ALE mesh management for 3D quasi-Eulerian problems. International Journal for Numerical Methods in Engineering, 92(10):857–890, 2012.
- [32] Robert E. Cole and Wayne L. Neu. Validation of a commercial fluid-structure interaction solver with applications to air cushion vehicle flexible seals. Ocean Engineering, 189:106287, 2019.
- [33] Andrew Bloxom. Numerical Simulation of the Fluid-Structure Interaction of a Surface Effect Ship Bow Seal. PhD thesis, Virginia Polytechnic Institute and State University, 2014.
- [34] Haeseong Cho. Efficient semi-implicit coupling fluid-structure interaction analysis via model-order reduction of dynamic grids. Aerospace Science and Technology, 121:107356, 2022.
- [35] Charles S Peskin. Numerical analysis of blood flow in the heart. Journal of Computational Physics, 25(3):220–252, 1977.
- [36] Ming-Chih Lai and Charles S. Peskin. An Immersed Boundary Method with Formal Second-Order Accuracy and Reduced Numerical Viscosity. Journal of Computational Physics, 160(2):705–719, 2000.
- [37] C. Alberto Figueroa, Irene E. Vignon-Clementel, Kenneth E. Jansen, Thomas J.R. Hughes, and Charles A. Taylor. A coupled momentum method for modeling blood flow in three-dimensional deformable arteries. Computer Methods in Applied Mechanics and Engineering, 195(41):5685–5706, 2006. John H. Argyris Memorial Issue. Part II.
- [38] Thierry Fanion, Miguel Angel Fernández, and Patrick Le Tallec. Deriving Adequate Formulations for Fluid Structure Interaction Problems : from ALE to Transpiration. Research Report RR-3879, INRIA, 2000.
- [39] Miguel Angel Fernández and Patrick Le Tallec. Linear stability analysis in fluid-structure interaction with transpiration. Part I: formulation and mathematical analysis. Research Report RR-4570, INRIA, 2002.

- [40] Axel Gerstenberger and Wolfgang A. Wall. An eXtended Finite Element Method/Lagrange multiplier based approach for fluid–structure interaction. Computer Methods in Applied Mechanics and Engineering, 197(19):1699–1714, 2008. Computational Methods in Fluid–Structure Interaction.
- [41] Boyce E. Griffith and Neelesh A. Patankar. Immersed Methods for Fluid–Structure Interaction. Annual Review of Fluid Mechanics, 52(1):421–448, 2020.
- [42] P. D. Thomas and C. K. Lombard. Geometric Conservation Law and Its Application to Flow Computations on Moving Grids. AIAA Journal, 17(10):1030–1037, 1979.
- [43] Alireza Naseri. Developing numerical methods for fully-coupled nonlinear fluid-structure interaction problems. PhD thesis, Universitat Politècnica de Catalunya, 2019.
- [44] S. Étienne, A. Garon, and D. Pelletier. Perspective on the geometric conservation law and finite element methods for ALE simulations of incompressible flow. Journal of Computational Physics, 228(7):2313–2333, 2009.
- [45] Luciano Garelli. Fluid structure interaction using an arbitrary lagrangian eulerian formulation. PhD thesis, Universidad Nacional Del Litoral, 2011.
- [46] Rong Ma, Xinghua Chang, Laiping Zhang, Xin He, and Ming Li. On the Geometric Conservation Law for Unsteady Flow Simulations on Moving Mesh. Procedia Engineering, 126:639–644, 2015. Frontiers in Fluid Mechanics Research.
- [47] Dongmi Luo, Shiyi Li, Weizhang Huang, Jianxian Qiu, and Yibing Chen. A Quasi-Conservative discontinuous galerkin method for multi-component flows using the non-oscillatory kinetic flux II: ALE framework. Journal of Scientific Computing, 90(1):46, December 2021.
- [48] Y. Bazilevs, K. Takizawa, and T.E. Tezduyar. Advanced FSI and Space–Time Techniques, chapter 6, pages 139–169. John Wiley & Sons, Ltd, 2013.
- [49] Bernd Markert. A survey of selected coupled multifield problems in computational mechanics. Journal of Coupled Systems and Multiscale Dynamics, 1(1):22–48, 2013.
- [50] Vanilla Temtching Temou. Experimental and numerical study of fluid-structure interactions on flexible hydrofoils in composites. PhD thesis, Université de Bretagne occidentale-Brest, 2020.
- [51] Simone Bna’. Multilevel Domain Decomposition Algorithms for Monolithic Fluid-Structure Interaction Problems with Application to Haemodynamics. PhD thesis, Alma Mater Studiorum Università di Bologna, 2014.
- [52] Jean-Marc Vassen, Pascal DeVincenzo, Charles Hirsch, and Benoit Leonard. Strong coupling algorithm to solve fluid-structure-interaction problems with a staggered approach. In 7th European symposium on aerothermodynamics, volume 692, page 128, 2011.
- [53] Hermann G. Matthies and Jan Steindorf. Strong Coupling Methods. In Wolfgang Wendland and Messoud Efendiev, editors, Analysis and Simulation of Multifield Problems, pages 13–36, Berlin, Heidelberg, 2003. Springer Berlin Heidelberg.

-
- [54] Laura Marimon Giovannetti. Fluid structure interaction testing, modelling and development of passive adaptive composite foils. PhD thesis, University of Southampton, 2017.
 - [55] Miguel A Fernández. Coupling schemes for incompressible fluid-structure interaction: implicit, semi-implicit and explicit. SeMA Journal, 55(1):59–108, 2011.
 - [56] Y Yu. Numerical Methods for Fluid-Structure Interaction: Analysis and Simulations. PhD thesis, Brown University, 2014.
 - [57] Sergio Idelsohn, Facundo Del Pin, and Riccardo Rossi. Avoiding Instabilities Caused by Added Mass Effects in Fluid-Structure Interaction Problems, chapter 4, pages 165–220. John Wiley & Sons, Ltd, 2013.
 - [58] A Naseri, O Lehmkuhl, I Gonzalez, and A Oliva. Partitioned semi-implicit methods for simulation of biomechanical fluid-structure interaction problems. J. Phys. Conf. Ser., 745:032020, September 2016.
 - [59] Christiane Förster, Wolfgang A. Wall, and Ekkehard Ramm. Artificial added mass instabilities in sequential staggered coupling of nonlinear structures and incompressible viscous flows. Computer Methods in Applied Mechanics and Engineering, 196(7):1278–1293, 2007.
 - [60] P. Causin, J.F. Gerbeau, and F. Nobile. Added-mass effect in the design of partitioned algorithms for fluid-structure problems. Computer Methods in Applied Mechanics and Engineering, 194(42):4506–4527, 2005.
 - [61] C. Yvin, A. Leroyer, M. Visonneau, and P. Queutey. Added mass evaluation with a finite-volume solver for applications in fluid-structure interaction problems solved with co-simulation. Journal of Fluids and Structures, 81:528–546, 2018.
 - [62] Miguel Angel Fernández, J-F Gerbeau, and Céline Grandmont. A projection semi-implicit scheme for the coupling of an elastic structure with an incompressible fluid. International Journal for Numerical Methods in Engineering, 69(4):794–821, 2007.
 - [63] Mikel Landajuela, Marina Vidrascu, Dominique Chapelle, and Miguel Angel Fernández. Coupling schemes for the FSI forward prediction challenge: comparative study and validation. Research Report RR-8824, Inria, December 2015.
 - [64] Miguel A. Fernández and Jean-Frédéric Gerbeau. Algorithms for fluid-structure interaction problems, pages 307–346. Springer Milan, Milano, 2009.
 - [65] Daniel R Miller. Critical flow velocities for collapse of reactor parallel-plate fuel assemblies. Technical report, Knolls Atomic Power Lab., Schenectady, NY, 1958.
 - [66] Nicola Parolini and Alfio Quarteroni. Mathematical models and numerical simulations for the America’s Cup. Computer Methods in Applied Mechanics and Engineering, 194(9):1001–1026, 2005.
 - [67] Nicola Parolini and Alfio Quarteroni. Modelling and numerical simulation for yacht engineering. In Proceedings of the 26th Int. Symp. on Naval Hydrodynamics, 2007.
 - [68] Laura Marimon Giovannetti, Ali Farousi, Fabian Ebbesson, Alois Thollot, Alex Shiri, and Arash Eslamdoost. Fluid-structure interaction of a foiling craft. Journal of Marine Science and Engineering, 10(3):372, 2022.

REFERENCES

- [69] C Lothodé, M Durand, Y Roux, Alban Leroyer, Michel Visonneau, L Dorez, and Groupama Sailing Team. Dynamic fluid structure interaction of a foil. In Innov'Sail, pages 1–6, 2013.
- [70] Mathieu Durand. Interaction fluide-structure souple et legere, application aux voiliers. PhD thesis, Ecole Centrale de Nantes (ECN), 2012.
- [71] R Balze, Nedeleg Bigi, Kostia Roncin, Jean-Baptiste Leroux, Alain Nême, Vincent Keryvin, A Connan, H Devaux, and D Gléhen. Racing Yacht Appendages Optimisation Using Fluid-Structure Interactions. In 4th Innov'Sail conference, 2017, 2017.
- [72] Chloé Duport, Jean-Baptiste Leroux, Kostia Roncin, Christian Jochum, and Yves Parlier. Benchmarking of a 3D non-linear lifting line method against 3D RANSE simulations. La Houille Blanche, (5-6):70–73, 2019.
- [73] Jason Knight, Matthew Richard McConnell, Anthony Ledger, and Rodrigo Azcueta. Fluid structure interaction of hydrofoils. In 1st International Conference on Marine Engineering and Technology. Zenodo, 2019.
- [74] Saeed Hosseinzadeh and Kristjan Tabri. Numerical investigation of hydroelastic response of a three-dimensional deformable hydrofoil. In HSMV 2020, pages 77–86. IOS Press, 2020.
- [75] Antoine Ducoin, Jacques André Astolfi, and Jean-François Sigrist. An experimental analysis of fluid structure interaction on a flexible hydrofoil in various flow regimes including cavitating flow. European Journal of Mechanics-B/Fluids, 36:63–74, 2012.
- [76] Emmanuel De Langre. Analyse dimensionnelle en interaction fluide-structure. La Houille Blanche - Revue internationale de l'eau, (3-4):14–18, 2000.
- [77] Emmanuel De Langre, Xavier Amandolese, and Olivier Doaré. Fundamentals of Fluid-Solid Interactions [MOOC]. <https://www.coursera.org/learn/fluid-solid-interaction/home/>, 2016.
- [78] Alberto N. Conejo. Buckingham's π Theorem, pages 69–76. Springer Singapore, Singapore, 2021.
- [79] Miguel R. Visbal and J. S. Shang. Investigation of the flow structure around a rapidly pitching airfoil. AIAA Journal, 27(8):1044–1051, 1989.
- [80] Eun Jung Chae, Deniz Tolga Akcabay, and Yin Lu Young. Dynamic response and stability of a flapping foil in a dense and viscous fluid. Physics of Fluids, 25(10):104106, 2013.
- [81] Eun Jung Chae and Yin Lu Young. Influence of spanwise flexibility on steady and dynamic responses of airfoils vs hydrofoils. Physics of Fluids, 33(6):067124, 2021.
- [82] Peter K Besch and Yuan-Ning Liu. Flutter and divergence characteristics of four low mass ratio hydrofoils. Technical report, David W. Taylor Naval Ship Research and Development Center Bethesda MD, 1971.
- [83] Luc Mouton, Alban Leroyer, Ganbo Deng, Patrick Queutey, Thibaut Soler, and Britton Ward. Towards unsteady approach for future flutter calculations. Journal of Sailing Technology, 3(1):1–19, 2018.

-
- [84] José Dias Rodrigues. Apontamentos de Vibrações de Sistemas Mecânicos. U.PORTO-FEUP-DEMec, 2020.
- [85] M.L. Facchinetti, Emmanuel De Langre, and F. Biolley. Coupling of structure and wake oscillators in vortex-induced vibrations. Journal of Fluids and Structures, 19(2):123–140, 2004.
- [86] Jung-Chi Liao. Vortex-induced vibration of slender structures in unsteady flow. PhD thesis, Massachusetts Institute of Technology, 2002.
- [87] Eun Jung Chae. Dynamic response and stability of flexible hydrofoils in incompressible and viscous flow. PhD thesis, University of Michigan, 2015.
- [88] Y.L. Young, T. Wright, H. Yoon, and C.M. Harwood. Dynamic hydroelastic response of a surface-piercing strut in waves and ventilated flows. Journal of Fluids and Structures, 94:102899, 2020.
- [89] Jean Donea and Antonio Huerta. Finite Element Methods for Flow Problems, chapter 1, pages 1–32. John Wiley & Sons, Ltd, 2003.
- [90] Chris Baker, Terry Johnson, Dominic Flynn, Hassan Hemida, Andrew Quinn, David Soper, and Mark Sterling. Computational techniques. In Chris Baker, Terry Johnson, Dominic Flynn, Hassan Hemida, Andrew Quinn, David Soper, and Mark Sterling, editors, Train Aerodynamics, pages 53–71. Butterworth-Heinemann, 2019.
- [91] François G. Schmitt. About Boussinesq’s turbulent viscosity hypothesis: historical remarks and a direct evaluation of its validity. Comptes Rendus Mécanique, 335(9):617–627, 2007. Joseph Boussinesq, a Scientist of bygone days and present times.
- [92] Colleen D Scott-Pomerantz. The k-epsilon model in the theory of turbulence. PhD thesis, University of Pittsburgh, 2004.
- [93] David Wilcox. A half century historical review of the k-omega model. In 29th Aerospace Sciences Meeting, page 615, 1991.
- [94] Laszlo Konozy. The k- ω Shear-Stress Transport (SST) Turbulence Model, pages 57–66. Springer International Publishing, 2019.
- [95] Philippe Spalart and Steven Allmaras. A one-equation turbulence model for aerodynamic flows. In 30th Aerospace Sciences meeting and exhibit, page 439, 1992.
- [96] Suzanne Ruth Hutchison. Numerical modelling of hydrofoil fluid-structure interaction. PhD thesis, University of Tasmania, 2012.
- [97] Shuling Chen. Hydrodynamic behaviour of gliding hydrofoil crafts. PhD thesis, City University London, 2013.
- [98] Fabien Gaugain. Analyse expérimentale et simulation numérique de l’interaction fluide-structure d’un hydrofoil élastique en écoulement subcavitant et cavitant. PhD thesis, Ecole nationale supérieure d’arts et métiers-ENSAM, 2013.
- [99] Debabrata Dutta and J. K. Bhattacharjee. Limit Cycle Oscillations, pages 125–135. Springer Netherlands, Dordrecht, 2008.

REFERENCES

- [100] Christopher E. Brennen. Fundamentals of Multiphase Flow. Cambridge University Press, 2005.
- [101] Ruben Scardovelli and Stéphane Zaleski. Direct Numerical Simulation of Free-Surface and Interfacial Flow. Annual Review of Fluid Mechanics, 31(1):567–603, 1999.
- [102] Siemens Digital Industries Software. Simcenter STAR-CCM+ User Guide, version 2022.1. Siemens, 2022.
- [103] Olek C Zienkiewicz and Robert Leroy Taylor. The finite element method for solid and structural mechanics. Elsevier, 2005.
- [104] Tianyun Liu, Chongbin Zhao, Qingbin Li, and Lihong Zhang. An efficient backward Euler time-integration method for nonlinear dynamic analysis of structures. Computers & structures, 106:20–28, 2012.
- [105] Kim Jie Koh and Airil Yasreen Mohd Yasreen. On Newton-Raphson formulation and algorithm for displacement based structural dynamics problem with quadratic damping nonlinearity. In MATEC Web of Conferences, volume 111, page 01004. EDP Sciences, 2017.
- [106] Patrick R Amestoy, Iain S Duff, and J-Y L’excellent. Multifrontal parallel distributed symmetric and unsymmetric solvers. Computer methods in applied mechanics and engineering, 184(2-4):501–520, 2000.
- [107] Hrvoje Jasak. Error analysis and estimation for the finite volume method with applications to fluid flows. PhD thesis, Imperial College London (University of London), 1996.
- [108] Guan Heng Yeoh and Jiyuan Tu. Three-Phase Flows. In Guan Heng Yeoh and Jiyuan Tu, editors, Computational Techniques for Multiphase Flows, pages 541–566. Butterworth-Heinemann, Oxford, 2010.
- [109] Marwan Darwish and Fadl Moukalled. The Finite Volume Method in Computational Fluid Dynamics: An Advanced Introduction with OpenFOAM® and Matlab®. Springer, 2021.
- [110] Henk Kaarle Versteeg and Weeratunge Malalasekera. An introduction to computational fluid dynamics: the finite volume method. Pearson education, 2007.
- [111] Venkat Venkatakrishnan. On the accuracy of limiters and convergence to steady state solutions. In 31st Aerospace Sciences Meeting, page 880, 1993.
- [112] Timothy Barth and Dennis Jespersen. The design and application of upwind schemes on unstructured meshes. In 27th Aerospace Sciences meeting, page 366, 1989.
- [113] Michael Mansour, Prafull Khot, Peter Kovats, Dominique Thevenin, Katharina Zähringer, and Gabor Janiga. Impact of computational domain discretization and gradient limiters on CFD results concerning liquid mixing in a helical pipe. Chemical Engineering Journal, 383:123121, 2020.
- [114] Francisco Pimenta and Manuel A Alves. A coupled finite-volume solver for numerical simulation of electrically-driven flows. Computers & Fluids, 193:104279, 2019.

-
- [115] Alejandro Alonzo-García, Claudia del Carmen Gutiérrez-Torres, and José Alfredo Jiménez-Bernal. Computational Fluid Dynamics in Turbulent Flow Applications. In Numerical Simulation-From Brain Imaging to Turbulent Flows. IntechOpen, 2016.
 - [116] Ahmed Kovacevic, Sham Rane, Nikola Stosic, Yu Jiang, Sam Lowry, and Michal Furmanczyk. Influence of approaches in CFD solvers on performance prediction in screw compressors. In 22nd International Compressor Engineering Conference at Purdue, 2014.
 - [117] Seok Ki Choi, Ho Yun Nam, and Mann Cho. Use of the momentum interpolation method for numerical solution of incompressible flows in complex geometries: choosing cell face velocities. Numerical Heat Transfer, 23(1):21–41, 1993.
 - [118] R.I Issa. Solution of the implicitly discretised fluid flow equations by operator-splitting. Journal of Computational Physics, 62(1):40–65, 1986.
 - [119] BD Nichols, CW Hirt, and RS Hotchkiss. SOLA-VOF: A solution algorithm for transient fluid flow with multiple free boundaries. Technical report, Los Alamos National Lab.(LANL), Los Alamos, NM (United States), 1980.
 - [120] Cyril W Hirt and Billy D Nichols. Volume of fluid (VOF) method for the dynamics of free boundaries. Journal of computational physics, 39(1):201–225, 1981.
 - [121] Nikolaos D. Katopodes. Volume of Fluid Method. In Nikolaos D. Katopodes, editor, Free-Surface Flow, pages 766–802. Butterworth-Heinemann, 2019.
 - [122] Tomasz Waławczyk and Tadeusz Koronowicz. Modeling of the flow in systems of immiscible fluids using Volume of Fluid method with CICSAM scheme. Turbulence, 8:267–276, 2005.
 - [123] Samir Muzaferija. A two-fluid Navier-Stokes solver to simulate water entry. In Proceedings of 22nd symposium on naval architecture, 1999, pages 638–651. National Academy Press, 1999.
 - [124] Brian Phillip Leonard. Simple high-accuracy resolution program for convective modelling of discontinuities. International journal for numerical methods in fluids, 8(10):1291–1318, 1988.
 - [125] Bruno Silva de Lima, Lucas de Souza Meira, and Francisco José de Souza. Numerical simulation of a water droplet splash: Comparison between PLIC and HRIC schemes for the VoF transport equation. European Journal of Mechanics-B/Fluids, 84:63–70, 2020.
 - [126] Brian P Leonard. A stable and accurate convective modelling procedure based on quadratic upstream interpolation. Computer methods in applied mechanics and engineering, 19(1):59–98, 1979.
 - [127] Yashar Mehmani. Wrinkle-Free Interface Compression for Two-Fluid Flows. arXiv: Fluid Dynamics, 2018.
 - [128] Tomasz Waławczyk and Tadeusz Koronowicz. Comparison of CICSAM and HRIC high-resolution schemes for interface capturing. Journal of theoretical and applied mechanics, 46:325–345, 2008.

REFERENCES

- [129] David Mann. Simcenter STAR-CCM+ V13.04: Get into your stride with VOF Multi-Step, 2018.
- [130] Chiang C Mei, Michael Aharon Stiassnie, and Dick K-P Yue. Theory and Applications of Ocean Surface Waves: Part 1: Linear Aspects. World Scientific, 2005.
- [131] Alessandro Toffoli and Elzbieta M. Bitner-Gregersen. Types of Ocean Surface Waves, Wave Classification, pages 1–8. John Wiley & Sons, Ltd, 2017.
- [132] Dominic Reeve, Andrew Chadwick, and Christopher Fleming. Coastal engineering: processes, theory and design practice. CRC Press, 2018.
- [133] Harvey D. Brookes and Malcolm O. Green. Determination of Wave Climate for the Auckland Region, New Zealand. Journal of Coastal Research, pages 23–29, 2001.
- [134] America’s Cup. 36th America’s Cup match dates and Race Course announced at America’s Cup Overture in Cowes. https://www.americascup.com/en/news/46_36th-AMERICAS-CUP-MATCH-DATES-AND-RACE-COURSE-ANNOUNCED-AT-AMERICAS-CUP-OVERTURE-IN-COWES, 2018 [Accessed May 2022].
- [135] New Zealand Govt. Bream head to slipper island including hauraki gulf / tīkapa moana, 1992 [Accessed May 2022].
- [136] John D Fenton. A fifth-order Stokes theory for steady waves. Journal of waterway, port, coastal, and ocean engineering, 111(2):216–234, 1985.
- [137] Lars Skjelbreia and James Hendrickson. Fifth order gravity wave theory. Coastal Engineering Proceedings, 1(7):10–10, 1960.
- [138] SC De. Contributions to the theory of Stokes waves. In Mathematical Proceedings of the Cambridge Philosophical Society, volume 51, pages 713–736. Cambridge University Press, 1955.
- [139] Klaus Hasselmann, Tim P Barnett, E Bouws, H Carlson, David E Cartwright, K Enke, JA Ewing, A Gienapp, DE Hasselmann, P Kruseman, et al. Measurements of wind-wave growth and swell decay during the Joint North Sea Wave Project (JONSWAP). Ergaenzungsheft zur Deutschen Hydrographischen Zeitschrift, Reihe A, 8:1–95, 1973.
- [140] Willard J Pierson Jr and Lionel Moskowitz. A proposed spectral form for fully developed wind seas based on the similarity theory of SA Kitaigorodskii. Journal of geophysical research, 69(24):5181–5190, 1964.
- [141] Norske Veritas. Environmental conditions and environmental loads. Det Norske Veritas Oslo, Norway, 2000.
- [142] James N Hawkes, Stephen R Turnock, Guilherme Vaz, Simon J Cox, and Alex B Phillips. Chaotic linear equation-system solvers for unsteady CFD. In MARINE VI: proceedings of the VI International Conference on Computational Methods in Marine Engineering, pages 931–942. CIMNE, 2015.
- [143] John W Ruge and Klaus Stüben. Algebraic multigrid. In Multigrid methods, pages 73–130. SIAM, 1987.

-
- [144] Magnus R Hestenes and Eduard Stiefel. Methods of Conjugate Gradients for Solving. Journal of research of the National Bureau of Standards, 49(6):409, 1952.
 - [145] Seungyong Lee, George Wolberg, and Sung Yong Shin. Scattered data interpolation with multilevel B-splines. IEEE transactions on visualization and computer graphics, 3(3):228–244, 1997.
 - [146] Emmanuel Lefrançois. How an added mass matrix estimation may dramatically improve FSI calculations for moving foils. Applied Mathematical Modelling, 51:655–668, 2017.
 - [147] James W Cooley and John W Tukey. An algorithm for the machine calculation of complex fourier series. Mathematics of computation, 19(90):297–301, 1965.
 - [148] Yuan-Ning Liu and Peter K Besch. Hydrofoil Flutter Analysis, Using a Modified Strip Theory. Technical report, David W. Taylor Naval Ship Research and Development Center Bethesda MD, 1971.
 - [149] Antoine Ducoin and Yin L Young. Hydroelastic response and stability of a hydrofoil in viscous flow. Journal of fluids and structures, 38:40–57, 2013.
 - [150] Rodney H Ricketts and Robert V Doggett Jr. Wind-tunnel experiments on divergence of forward-swept wings. Technical report, NASA Langley Research Center Hampton, VA, United States, 1980.
 - [151] Jennifer Heeg. Dynamic investigation of static divergence: Analysis and testing. Technical report, NASA Langley Research Center Hampton, VA, United States, 2000.