

ABSTRACT

Title of dissertation: A UNIFIED
FLUID-STRUCTURE-INTERACTION
FORMULATION FOR DIRECT NUMERICAL
SIMULATION OF DEFORMABLE
HYPERELASTIC SOLIDS IMMERSED IN
VISCOUS INCOMPRESSIBLE FLOW

Jiazhen Qiao, Doctor of Philosophy, 2026

Dissertation directed by: Professor Amir Riaz
Department of Mechanical Engineering

It is of great interest for the metal casting industry to understand the physics associated with the pouring of highly oxidative liquid metals, such as steel or aluminum alloys, to reduce dross formation for improved production quality. The most common approach for the prediction and control of dross formation currently practiced in the industry is trial and error, which usually results in excessive waste of material and energy, as well as degradation in product quality. Therefore, it is important to develop more reliable and systematic design tools to predict and control the formation and transport of dross. The ultimate objective is to develop high-fidelity direct numerical simulation tools, which can model the complete behavior of the pouring process, including the formation of dross and its interactions with the surrounding fluid.

As a first step towards that end, a multidimensional direct numerical simulation methodology is developed to model the dynamics of deformable elastic solids immersed in an incompressible viscous fluid. In this approach, the solid domain is represented by a signed

distance function that is defined on a fixed and uniform Cartesian grid. The strain field associated with the solid domain is tracked using a set of three reference level set functions that form a dynamic grid which allows the solid deformation to be determined with reference to the fixed underlying Cartesian grid. The initial state of the dynamic grid coincides with the Cartesian grid, representing the unstressed equilibrium state of the solid. The deformation gradient tensor is formed by comparing the reference level sets with the underlying Cartesian grid. A finite-strain constitutive model for hyper-elastic, Neo-Hookean materials is used to relate the deformation gradient tensor to the solid stress tensor. Due to the modular nature of the formulation, other material models can be incorporated in a straightforward manner. The divergence of the stress tensor is added to the momentum equation to represent the solid stress as a source term only in the solid region. The jump in normal stress across the solid-fluid boundary is captured implicitly by creating a finite width solid-fluid boundary across which the respective material stresses are smoothly blended together using a continuous Heaviside function. In order to ensure a smooth continuation of the stress fields in the blended boundary region, an improved extrapolation methodology is developed to propagate the dynamic grid from the solid to the fluid domain using Hamilton-Jacobi and topology projection strategies for reinitializing the solid level set indicator field to a signed distance function. In contrast to Lagrangian-Eulerian methods for immersed elastic solids, our methodology results in a unified formulation, where a single Poisson equation needs to be solved to obtain pressures and velocities within both the solid and the fluid domains. The implementation has been embedded within the FLASH 4 solver framework to enable scalable parallel computations using HYPRE with an algebraic multi-grid solver. We demonstrate the accuracy of the numerical implementation in comparison with Lagrangian-Eulerian methods using canonical test cases. We also demonstrate that the methodology is robust and easy to extend to the axisymmetric and 3-D domains.

A UNIFIED FLUID-STRUCTURE-INTERACTION FORMULATION
FOR DIRECT NUMERICAL SIMULATION OF DEFORMABLE
HYPERELASTIC SOLIDS IMMERSED IN VISCOUS
INCOMPRESSIBLE FLOW

by

Jiazhen Qiao

Dissertation submitted to the Faculty of the Graduate School of the
University of Maryland, College Park in partial fulfillment
of the requirements for the degree of
Doctor of Philosophy
2026

Advisory Committee:
Professor Amir Riaz, Chair/Advisor
Professor Kenneth Kiger
Professor James Duncan
Professor Johan Larsson
Professor James Baeder, Dean's Representative

© Copyright by
Jiazhen Qiao
2026

Dedication

To my parents, whose endless love, encouragement, and sacrifices have been my greatest source of strength.

Acknowledgments

It would not have been possible to complete this dissertation without the invaluable help and support of many people over the years. I would like to express my deepest gratitude to all of you.

First and foremost, I would like to express my deepest gratitude to my advisor, Professor Amir Riaz. His continuous guidance and unwavering support are what made this dissertation a reality. I am deeply thankful for the countless hours he invested in my development, his incredible patience during the challenging phases of my research, and his constant encouragement when I needed it most. He went above and beyond what is expected of an advisor, and I will always be grateful for everything he has done to help me grow both as a researcher and as a person.

I would also like to thank my friends and colleagues in Maryland for their friendship, support, and for making my graduate experience truly rewarding. I would like to express my gratitude to everyone in our lab, Aditya Sangli, Shantanu Vachhani, and Seyed Mohammadamin Taleghani for the valuable discussions and their help. I am also grateful to Professor Ken Kiger, Professor Elias Balaras at George Washington University and Professor Scott Giese at the University of Northern Iowa for their valuable insights and guidance.

Most importantly, I want to thank my family for the unconditional love and support they have given me. I owe a special, heartfelt thank you to my father and mother. I am deeply grateful for the values and insights you instilled in me as I was growing up. Your lifelong guidance has supported me every step of this journey and shaped me into the person I am today. I love you.

Finally, I gratefully acknowledge the financial sponsorship of this study by DLA-Troop Support in Philadelphia, PA, and the Defense Logistics Agency Information Operations, J68, Research & Development, Fort Belvoir, VA. I also wish to acknowledge the use

of the high-performance computing resources provided by the University of Maryland (<http://hpcc.umd.edu>), which were essential for obtaining the computational data reported in this study.

Table of Contents

Dedication	ii
Acknowledgements	iii
1 Introduction	1
1.1 Overview	1
1.2 Review of existing numerical methods	7
1.2.1 Numerical methods for Multiphase Flow	7
1.2.2 Numerical methods for Fluid-Structure Interaction	12
1.2.3 The Added-Mass Effect and Coupling Instabilities	16
1.3 Research Objectives	18
1.4 Thesis Outline	20
2 Mathematical model	22
2.1 Unified framework for momentum transport	22
2.2 Interfacial jump conditions	23
2.3 Distributed material properties	25
2.4 Neo-Hookean material model	27
3 Discretization and numerical implementation	29
3.1 Advection and re-initialization of the interface and the dynamic grid	31
3.2 Calculation and discretization of the dynamic grid scalar functions	34
3.2.1 Discretization in 2-D	34
3.2.2 Discretization in 2-D-Axisymmetric	35
3.2.3 Discretization in 3-D	35
3.3 Numerical algorithm of implementing the solid stress term	36
3.4 Extension of the solid region to a thin layer geometry	38
4 Fluid-Elastic Solid Interaction in 2-D	40
4.1 Parametric study for relaxation of a sine wave shaped layer	40
4.2 Dynamic grid extrapolation	43

4.3	Error Analysis	45
4.4	Immersed oscillating disk	49
4.5	Immersed oscillating ring	51
4.6	Elastic wall in a lid driven cavity	52
4.7	Deformable disk in a lid-driven cavity	59
4.8	Deformable solid ring in a lid-driven cavity	60
4.9	Rising solid disk in a vertical channel	63
4.10	Deformation of a Finite-Thickness Solid Layer Subjected to Inlet Shearing .	64
4.11	Deformation of a pre-deformed solid film	69
4.12	Deformation of a solid film under a point force	77
4.13	Deformation solid filaments with periodic inflow	80
5	Fluid–Elastic Solid Interaction in 2–D –Axisymmetric	85
5.1	Periodic flow through an axisymmetric vessel	85
5.2	Periodic flow through an axisymmetric nozzle	89
6	Fluid–Elastic Solid Interaction in 3–D	97
6.1	Immersed oscillating sphere	97
6.2	Immersed oscillating cylinder	100
6.3	3D solid film	103
7	Industrial Application and Solver Capability Evaluation: Internship Work at Das-	
	sault Systèmes	105
7.1	Multiphase Solver Validation for Industrial Lubrication Systems	106
7.1.1	FZG Gear Lubrication Simulation Setup	106
7.1.2	Torque Prediction and Oil Distribution Results	107
7.2	Validation of Sloshing Dynamics: Tuned Liquid Damper	109
7.2.1	Computational Setup and Methodological Variations	109
7.2.2	Results and Discussion	110
7.3	Development and Validation of an Immersed Boundary Lattice Boltzmann	
	Method (IBM-LBM) Scheme	114
7.3.1	Development of a Custom IBM-LBM Solver	114
7.3.2	Validation: 2D Uniform Flow Over a Solid Cylinder	116
7.4	Summary and Relationship to Main Dissertation Work	119
8	Conclusions	121
8.1	Contributions	121
8.2	Future Work	123
	Bibliography	125

Chapter 1: Introduction

1.1 Overview

Metals and alloys are usually found in the liquid state during their production processes. These liquid metals are usually highly oxidative and will react with oxygen in the air almost instantly to form a thin oxide film at the interface. During the filling process, the liquid metal is generally poured into the mold using the conventional top filling method, which is widely used in the industry due to its high flow rate owing to the gravity driven nature of the process. However, the impingement of the liquid jet on the free surface at high velocities may lead to the formation of cavities. When such cavities collapse, pieces of the oxide film and dross are entrained into the liquid metal along with air and become lodged as impurities in the solidified metal in the form of bubble trails and cracklike defects. Such inclusions can severely degrade the mechanical properties of the castings, creating a high risk of catastrophic failure. [1]. Fig. 1.1 shows secondary electron images of the two opposing surfaces of a tensile fracture, showing a young spinel doubled over bifilm opened along its non-bonded dry central interface [2].

Although different filling techniques such as bottom filling and low pressure casting have been developed to make the filling process as smooth as possible, it is still difficult to completely avoid perturbing the free surface of the liquid metal and prevent air and oxide film entrainment into the liquid metal [3][4]. Furthermore, there are a number of physical mechanisms related to the entrainment and impurity distribution that require a closer look.

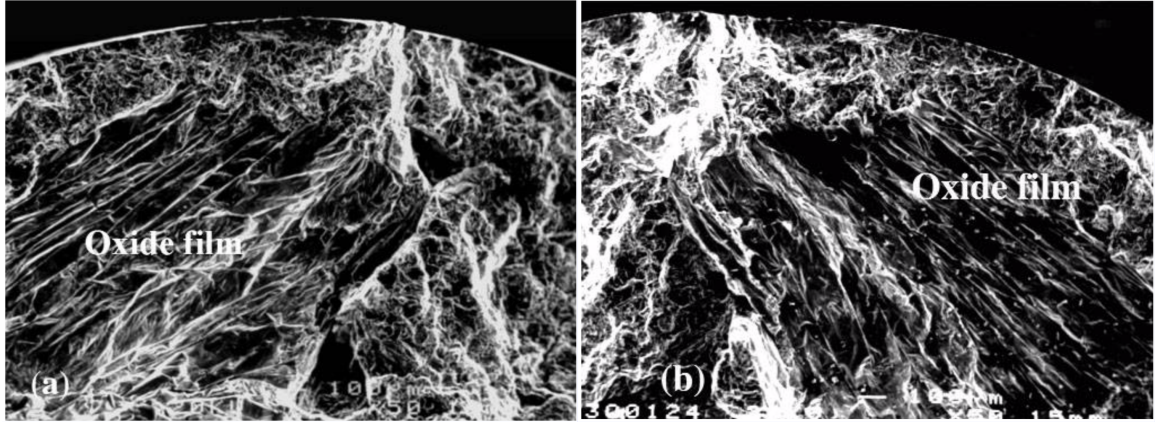


Figure 1.1: Secondary electron images of the two opposing surfaces of a tensile fracture. [2]

For example, the oxide film formed on the liquid metal jet can break up and tear due to fluid shear and jet instabilities and plunge into the liquid pool. Accurate prediction of such event can provide the necessary information regarding the prevention and mitigation of this source of entrainment. Moreover, vortices generated by jet impingement can also interact with the free surface and the entrained oxide film, potentially causing them to deform, fracture and redistribute. In the metal casting industry, a significant effort is spent on mitigating the entrainment using heuristic methods due to the lack of availability of accurate and efficient tool. Motivated by this need, we seek to develop a suite of numerical methods that can contribute toward the improvement of system level modeling of physical mechanisms governing the casting processes described above.

In order to accurately model the various aspects of the casting process involving the simultaneous flow of air, liquid metal, oxide films and dross, it is crucial to first carefully model the underlying fundamental mechanisms associated with the interactions between immersed deformable solids and the surrounding viscous incompressible flow. Accurate numerical simulation of such interactions depends crucially upon capturing the correct balance of viscous, elastic, inertial and gravitational forces created by the motion of the

immersed elastic solid within the surrounding fluid. While a number of numerical methods exist for modeling fluid-structure interactions for rigid solids, there is much less work available for modeling the interactions between fluid and deformable solids. For example, within commercial solvers, dross is represented by a passive scalar field that is advected with the flow. The force exerted by the deformation of the solid on the fluid is not accounted for. Currently, this technique is the most widely used entrainment defect prediction tool in the metal casting industry. Yang et al. [5] used this method to represent oxide films and to investigate the distribution of oxide film defects in the filling process of aluminium alloy castings. Based on the passive scalar method, Cao et al. [6] proposed a formation rate model taking into account the volume fraction of liquid metal, as well as the temperature and oxide inclusion density. Reilly et al. [7] used a particle tracking method to investigate oxide inclusion near the free surface with the Boolean logic criteria. Reilly et al. [7] and other researchers have pointed out a number of drawbacks of the cumulative scalar technique, which lacks models of film strength, buoyancy of the oxide film and adhesion to the mould walls. Our approach is to develop direct numerical simulation methods to model the two-way coupling in order to account for the influence of the motion of the deformable oxide film on the surrounding fluid and vice versa. Towards that end, our ultimate goal is to use direct numerical simulations to model in detail all the relevant physical mechanisms for an accurate prediction of related to the dynamics of the oxide film including its motion, rupture and entrainment.

Among the different conventional FSI methods, the non-body-fitted methods have gained popularity over the past a few decades compared to body-fitted methods. By using a fixed Cartesian mesh for Eulerian variables, a computationally expensive regeneration of the mesh that conforms to solid boundary is avoided. The effect of solid body is introduced into the fluid region in the vicinity of the solid boundary. These methods have high computational efficiency and are compatible with high fidelity Direct Numerical Simulation

(DNS) solvers. Non-body-fitted methods can be further categorized as fully Lagrangian method, Eulerian-Lagrangian method and fully Eulerian method. Traditionally, FSI problems are studied by Eulerian-Lagrangian methods, where the fluid is represented on a fixed Eulerian mesh while the solid is tracked separately as a Lagrangian element. The coupling between the two is achieved by periodically interpolating the fluid-solid interactions from the Lagrangian markers to the fixed Eulerian grid and vice versa. The immersed boundary methods on the other hand can be implemented in a fully Eulerian fashion and are also widely used to model solid-fluid interactions. The classical Immersed Boundary (IB) method was first proposed by C.S. Peskin [8] to study flows around the flexible leaflet of human heart valve. In this method, solid boundary is represented by a set of points connected with springs which are moved by the fluid. As these solid points change their locations, a force can be calculated by the constitutive law of the springs. The resultant force field is used to represent the effect of the solid boundary. Other IB methods can be classified as continuous forcing approach and discrete forcing approach [9]. In continuous forcing approach, the forcing function is included into the RHS of the continuous governing equation for the fluid which applies to the entire domain. This method is not well suited for flows with rigid boundaries because the constitutive laws used for elastic boundaries are not well posed in the rigid limit. A large elastic constant can result in stability problems. Peskin's Immersed Boundary Method [10] and the method of Bhalla et al. [11] fall in this category. In the discrete forcing approach, the governing equations are first discretized on the Cartesian grid without regard of the IB. Then, the discretization in the cells near IB is adjusted to account for the presence of the solid. Discrete forcing approach enables a sharp representation of the immersed space that does not introduce any extra stability constraints and does not need any arbitrary parameters in the forcing term. The methods proposed by Iaccarino and Verzicco [12] and Yang and Balaras [13] fall in this category. Among all the different methods, we are interested in an approach that can avoid the difficulty of solving

different set of governing equations on two separate meshes, leads to a robust and efficient implementation of solid-fluid interactions and accounts for solid deformation. Thus, a method to solve a unified form of equations on Eulerian mesh is favorable. Previous work has been done specifically on the interactions between fluid and deformable solid on an Eulerian mesh. Zhao et al. [14] has formulated a single set of equations of motion on a fixed Eulerian mesh. Although this method only needs to solve one set of equations which simplifies the problem, it requires an overlapping Lagrangian mesh to track solid deformation and compute elastic stresses. Exchange of information between Lagrangian mesh and Eulerian mesh is needed to both advect the Lagrangian mesh and apply the calculated solid stresses on the Eulerian mesh. He and Qiao [15] proposed a novel fully Eulerian method that uses level sets to track solid deformation. The method avoids using the Lagrangian mesh to further simplify the implementation even more. However, the advection of the level set functions used to calculate the solid stress leads to stability issues caused by the unphysical advection of the level sets outside the solid body. Valkov et al. [16] and Jain et al. [17] use a reference-map technique (RMT) to track solid deformation.

In addition to appropriate numerical models, suitable material models are also important for accurately describing the physics of the oxide film. Material properties of oxide-coated gallium and eutectic gallium Indium were measured using rheometers equipped with parallel plate geometries of different sizes [18]. The authors noted the measured values of the elastic modulus to be dependent on the radius of the plate geometry, which led them to conclude that the observed elastic response originates within the oxide skin. When the surface stress is less than the yield stress, surface stress is stored elastically in the oxide skin. When the surface stress is greater than the yield stress, the oxide skin will yield, or flow. These findings provide justification for the use of elastic materials to model the dynamical behavior of oxide films. Thus stress-strain (strain rate) relationship of the oxide film can be established by using elastic models for the solid oxide film. The Maxwell model (spring

and dash-pot in series) and the Kelvin (Voigt) Model (spring and dash-pot in parallel) are the simplest models for viscoelastic materials. Other models are usually combinations of Maxwell and Kelvin(Voigt) Model. For example, the Zener type of model [19] was used to study the interaction between a deformable elastic solid and a collapsing bubble.

In the present work, the focus is on modeling materials that are highly deformable but not too soft. Therefore, a Neo-Hookean model for finite-strain elasticity is used for the computation of elastic stresses. The Neo-Hookean model represents a complex hyperelastic material with a non-linear stress strain relationship. It can be used to describe the elastic response of materials when subjected to very large elastic strains. The structure of the hyperelastic material is based on the statistical thermodynamic representation of cross-linked polymer chains. It is deemed appropriate for modeling elastic materials with rubber-like behavior and is also capable of capturing plastic deformations. Other constitutive laws, such as those more specifically tuned for the rheology of dross and oxide films, can be easily incorporated into our numerical formulation to interpret experiments in which such materials are used.

In the context of casting processes, vortices generated by the plunging jet can produce large deformations and high stresses in dross and oxide films, which may even lead to their rupture and breakup. In order to accurately model the full chain of events, from the entrainment of dross and oxide films to their eventual breakup, fragmentation, and dispersal, the numerical method needs to be robust and modular, in addition to possessing the attributes of accuracy and computational efficiency. In this context, the level set formulation emerges as a suitable foundation on which to build the appropriate algorithms for tracking the solid-fluid domains and their respective boundaries, for applying the relevant interfacial conditions and for quantifying and tracking strain histories in elastic materials, all within an Eulerian framework.

The basic approach of using level sets to track strain fields involves the use of Eu-

lerian kinematic variables that retain the memory of a reference configuration, enabling the application of finite-strain elasticity models without Lagrangian meshes. Specifically, we employ Eulerian reference coordinates to represent the displacement field, which is embedded within a set of multiple level set functions called a dynamic grid [15]. In our formulation, the dynamic grid is advected using the local velocity field and the deformation with respect to an equilibrium state is used to calculate the strain field in the solid. A constant bulk shear modulus is used in our formulation to connect the stress field, using the Neo-Hookean model for finite-strain elasticity, with the strain field represented by the deformation gradient tensor, which is formed by calculating the derivative of the dynamic grid with respect to the fixed Cartesian grid. A unified Eulerian formulation results when the elastic and viscous stresses, expressed using domain specific constitutive laws, are added to the momentum equation to solve for a single set of pressure and velocity variables for the solid and fluid domains. Based on this simulation methodology the physics of the interaction between deformable immersed solids and the surrounding fluid can be reasonably accounted for, as demonstrated later in this dissertation using a wide variety of test cases.

1.2 Review of existing numerical methods

1.2.1 Numerical methods for Multiphase Flow

In multiphase flow, the Navier–Stokes equation needs to be solved in the presence of a deforming immersed solid body to resolve the complex physical dynamics associated with its interaction with the surrounding fluid. Liquid–air, liquid–liquid, and liquid–solid interfaces can occur, where addressing the physical behaviors such as large interfacial deformation, break–up, and coalescence is numerically challenging. Over the years, numerical methods have been developed to address the advection of the interface and apply appropriate boundary conditions across the interface. As shown in Fig. 1.2, the numerical methods

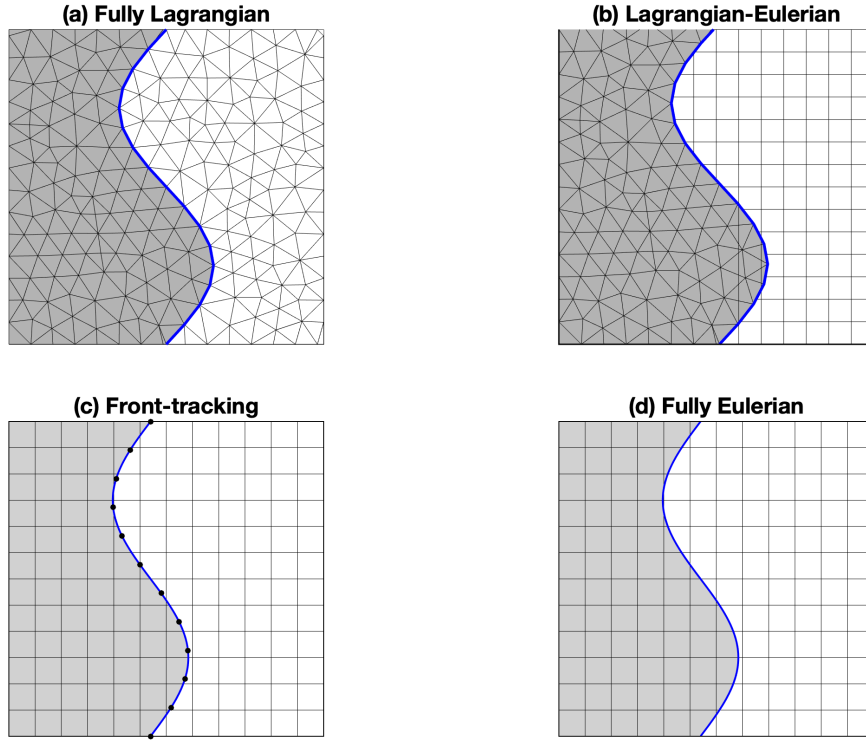


Figure 1.2: Categorization of numerical methods in multiphase flow

for flow with different phases can be categorized into fully Lagrangian methods, fully Eulerian methods, Front-tracking methods and Eulerian-Lagrangian methods.

Fig. 1.2(a) is the schematic of a Fully Lagrangian method, which uses a Lagrangian mesh for both phases and boundary fitted grids are used to track the individual phases. The advantage is that interface remains sharp and boundary dynamics can be applied directly to the interface which results in high accuracy. Ryskin and Leal [20, 21] used an orthogonal curvilinear grid to fit the shape of the boundary. Rising bubble and a bubble in extensional flow were simulated using the boundary-fitted grid. Feng et al. [22, 23] used a similar method to simulate the sedimentation of particles in a vertical channel and the motion of a circular particle in a Couette and a Poiseuille flow. Although Lagrangian methods tend to be an accurate approach, they are less used in recent developments due to their complexity to implement in three dimensions, difficulty in resolving rupture and coalescence, and limited

capability for handling complex geometries.

Fig. 1.2(b) shows the schematic of Lagrangian–Eulerian method, which connects between fully moving and fully fixed meshes by capitalizing on the strengths of both frameworks. This is achieved by discretizing one phase using a moving, body-fitted Lagrangian mesh that conforms exactly to the interface. The surrounding phase, meanwhile, is governed by a fixed Eulerian Cartesian grid. The primary advantage of this approach is its ability to sharply resolve the interfacial boundary and accurately capture near-wall hydrodynamics or boundary layers without requiring the entire computational domain to deform. However, the coupling between the two distinct grid topologies introduces significant computational complexity. Information must be accurately interpolated and transferred across the non-conforming mesh boundaries. Furthermore, while it mitigates the global remeshing required by fully Lagrangian methods, severe interfacial deformations can still distort the Lagrangian portion of the mesh, eventually necessitating localized remeshing to maintain numerical stability.

Fig. 1.2(c) shows the schematic of the front-tracking method which has the simplicity of solving the bulk fluid conservation equations (Navier-Stokes) across the entire domain. However, unlike fully Eulerian methods that capture the interface implicitly, front-tracking explicitly tracks the phase boundary using a separate, lower-dimensional Lagrangian grid composed of interconnected massless marker points. These marker points are advected by the local fluid velocity interpolated from the underlying fixed Eulerian grid. This hybrid approach provides a distinct advantage: it explicitly maintains the exact sub-grid location of the interface, completely eliminating the numerical diffusion and interfacial smearing that often corrupt fully Eulerian volume-capturing methods (such as Volume-of-Fluid or Level Set methods). Because the interface is explicitly defined, calculating interfacial forces, such as surface tension, is highly accurate. However, the major drawback of the front-tracking method lies in handling complex topological changes. When phases coalesce or break

apart occurs, the Lagrangian marker mesh must be algorithmically cut and reconnected—a process that is computationally expensive and exceptionally difficult to implement robustly in three dimensions.

Fig. 1.2(d) is the schematic of a Fully Eulerian method. It uses the stationary Cartesian grid for both phases while using a scalar function, such as the level set or the volume of fluid function to represent the interface. In the present study, Fully Eulerian method is used, and we present a more detailed discussion of the two most widely used methods, in this context, namely, the Volume of Fluid method (VOF) and the Level Set method (LSM).

VOF originated from the Marker-and-Cell method (MAC) and was first developed by Nichols, Hirt and Hotchkiss [24] to handle transient incompressible flow problems with multiple free surfaces. It introduces a volume fraction function C . When any given cell is fully occupied by the tracked phase, $C = 1$, and when the cell does not contain the tracked phase, $C = 0$. Interface cells are thus determined by the condition, $0 < C < 1$. The volume fraction function C is advected using a conservative advection equation and the interface is reconstructed using Simple Line Interface Calculation (SLIC). To address the orientation errors and improve geometric accuracy, Youngs [25] proposed the Piecewise Linear Interface Calculation (PLIC). This method has been widely used as the default method of interface reconstruction in the most of the classical VOF implementation. Gueyffier et al. [26] applied this method and extended it to 3-D. This basic approach has been extended in a various ways over the years to improve accuracy and robustness. Weymouth et al. [27, 28] developed a second-order construction method that conserves volume during advection. Pilliod and Puckett [29] proposed two second order volume-of-fluid interface reconstruction methods and a design criteria for reconstruction to be second-order accurate. S. Popinet [30] introduced an adaptive quad/octree spatial discretisation, geometrical VOF interface representation, continuum surface force formulation and height-function curvature estimation to improve accuracy in surface tension and pressure gradient for multiple

classical test cases. Overall, VOF is popular due to its strengths in mass conservation, robustness and the advantages related to implicit interface tracking. However, it also suffers from limitations associated with interface smearing, complex interface reconstruction and lower accuracy in the calculation of the quantities defined at the interface.

The level set method was first introduced by Osher and Sethian [31] to capture motion of interfaces and evolving shapes. The interface is tracked implicitly by a scalar function called the level set function, ϕ . The level set is initialized as a signed distance function from the interface and the interface is defined as, $\phi = 0$. The geometric properties such as surface normals and curvatures are calculated from the level set scalar function. The advection of the level set function is governed by a partial differential equation which can be solved numerically on a Cartesian grid. Details can be found in [32]. High order accuracy methods such as the weighted essentially non-oscillatory(WENO) scheme can be used to solve the advection equation [33, 34]. To maintain the signed distance function after advection, reinitialization of the level set function is needed. Sussman et al. [35] introduced the classical reinitializing method. Due to its basic characteristic, the level set does not enforce volume conservation explicitly when it is advected by an incompressible flow field. As a result, numerical errors associated with both the advection and the re-initialization process can cause numerical errors in the location of the interface to build up over time, which may potentially lead to large errors in mass conservation. In order to provide a remedy, Russo et al. [36] developed a second order accurate reinitialization method with better mass conservation. Du Chene et al. [37] independently developed a method to achieve second order accuracy in curvature calculations. Qin et al. [38] improved volume conservation and topology representation by decoupling the advection and reinitialization in the geometric projection method. Along with VOF, LSM is one of the most widely used methods for interface capturing due to its accurate representation of the interface geometry, accuracy of surface normals and curvatures calculation on Eulerian grids and robustness with respect

to topology changes and interface motions. The main drawback however is that the reinitialization process is inherently non-conservative and leads to errors, particularly on coarse grids, despite the remedies offered by the improved methods of reinitialization. One approach for overcoming this limitation is to combine the beneficial characteristics of VOF and LSM. Sussman et al. [39] developed a coupled level set/volume-of-fluid method that improves the accuracy with respect to, mass conservation, surface tension and topology changes compared with the use of either method on its own.

1.2.2 Numerical methods for Fluid-Structure Interaction

Among the various methods that address the interactions between fluid and solid, the ones that have been found to be most effective for large deformations of soft solids are the Eulerian-Lagrangian methods and fully Eulerian methods. In the present work, we focus the discussion on these methods. A simple categorization of these methods can be found in Fig. 1.3.

Interactions between a viscous fluid and a deformable solid occur in a wide variety of applications, including cardiovascular dynamics in heart valves [8, 10, 40–43], flow in blood vessels [44–46], flow past deformable structures [47–51], and locomotion of fish and swimming bodies [52–56]. Numerical simulation of such applications involve tracking the transport and deformation of elastic solids, computation of solid and viscous stresses within respective domains and accounting for the momentum exchange at solid-fluid interfaces. A wide range of computational approaches, methods and techniques have been developed to deal these these challenges. Among those, Cartesian grid methods have gained prominence as they offer some simplicity by avoiding the generation and re-meshing of complex body-fitted or generalized grids that need to follow the contours of moving and deformable solid bodies. Optimized, iterative linear solvers and multigrid methods can also be more easily

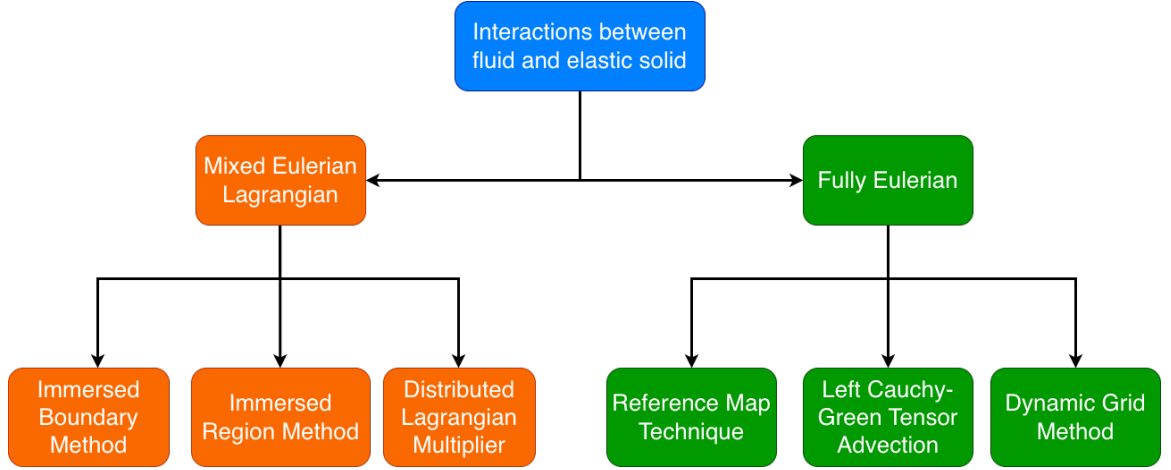


Figure 1.3: Classification of the numerical methods used to solve interactions between fluid and elastic solid in the literature. The orange boxes represent most frequently used methods that can be categorized as the mixed Eulerian Lagrangian methods. The green boxes represent fully Eulerian methods.

adapted for Cartesian grids.

Eulerian-Lagrangian methods have been widely used for modeling elastic FSI problems. The classical Immersed Boundary (IB) method of Peskin [8] was first proposed for studying flows around the flexible leaflet of a human heart valve. In this approach the fluid is represented on a Cartesian mesh while the elastic solid boundary is tracked using a set of spring-connected Lagrangian markers that are advected by the local velocity. The spring force resulting from the relative motion of the markers is added as a volumetric source term in the momentum equation. The coupling between the elastic boundary and the fluid is achieved by interpolating the fluid-solid kinematics and dynamics from the Lagrangian markers to the fixed Eulerian grid and vice versa [9, 57]. This approach was originally developed to more easily handle relatively large deformations compared with the arbitrary Lagrangian-Eulerian (ALE) method. The classical IB method has been widely implemented for problems involving biological systems and elastic fibres [10, 11, 58–60].

A separate class of Lagrangian-Eulerian IB methods was developed in response to the

difficulty of applying the classical IB method to rigid bodies [9, 12, 13, 61]. This class of methods utilizes the discretized velocity field on the Eulerian points near the solid boundary to compute a local forcing function that enforces appropriate boundary conditions.

The immersed region (IR) method represents another class of Lagrangian-Eulerian methods. In this context, Zhang et al. [62] proposed the immersed finite element method (IFEM), where both the fluid and solid domains are modeled with the finite element method but the solid domain is represented by a Lagrangian mesh that overlaps the Eulerian fluid domain. The continuity between the fluid and solid domains is enforced via the interpolation of the velocities and forces using the discrete delta function approach based on the reproducing kernel particle method (RKPM). The authors used their method to study particulate flows with flexible solids. Wang & Zhang [63] later developed the modified immersed finite element method (mIFEM). Compared with the IFEM, mIFEM solves the solid governing equation with appropriate boundary conditions generated from the surrounding fluid instead of imposing it based on the fluid velocity field. This method is useful when the solid behavior dominates the fluid flow. Liu et al. [64] proposed an immersed solid system and described the motion of the immersed solid by defining the Eulerian-Lagrangian mapping. They developed an implicit time integration algorithm with the finite element method and the streamline upwind/Petrov–Galerkin method that avoids inaccuracies from the smoothed approximation of the Dirac Delta function. Zhao et al. [14] have formulated a single set of equations of motion on a fixed Eulerian mesh. This method only needs to solve one set of equations which simplifies the problem. It requires an overlapping Lagrangian mesh to track solid deformation and compute elastic stresses. Exchange of information between the Lagrangian mesh and the Eulerian mesh is needed to advect the Lagrangian mesh and apply the calculated solid stresses on the Eulerian mesh.

The Distributed Lagrangian Multiplier (DLM) method is another instance of the Lagrangian-Eulerian method that was first used by Glowinski et al. [65] to simulate viscous incompress-

ible flow with suspended solid particles. The conservation of mass and momentum for the fluid is enforced both inside and outside the particle boundaries. The flow within the solid domain is constrained to be a rigid-body motion using the distributed Lagrange multipliers. The fluid-particle motion is treated implicitly by using a combined weak formulation in which the mutual forces cancel. Patankar et al. [66] later proposed a new formulation where the deformation rate tensor within the particle domain is constrained to be zero. The formulation leads to Lagrange multipliers for rigid motion analogous to the way that pressure arises as a Lagrange multiplier for incompressibility. Yu [67] extended the original DLM method for rigid solids to handle incompressible non-linear elastic solids. The solid stress field was solved using a finite element method in which the incompressibility constraint was implemented using the penalty function.

In general, the Lagrangian-Eulerian approach requires explicit coupling between the Lagrangian solid domain and the Eulerian fluid domain. This involves the frequent transfer of information between the two domains using some form of interpolation. As a result, errors can potentially appear in the conservation of mass and momentum between the solid and the fluid. The fully Eulerian approach [68–70] has been used as an alternative. It was first implemented by Dunne [68] to study the interactions between an incompressible fluid and a hyperelastic solid. Karmin et al. [71] first proposed the reference-map technique (RMT) that employed a unified equation of motion for the solid and the fluid. A continuous stress field for solids and fluids, with a smooth distribution of stresses at the interface. Other authors such as Valkov et al. [16], Jain et al. [17], Rycroft et al. [72] and Lin et al. [73] further developed this method.

Sugiyama et al. [74] have developed a method in which the conservation laws for both the fluid the solid are discretized on a fixed Cartesian grid. The multi-component geometry is described using the volume of fluid method and the solid deformation is quantified by updating the left cauchy-Green deformation tensor. Rosti et al. [75] used this method to per-

form simulations of a turbulent channel flow over a hyper-elastic wall. Mao & Jaiman [76] and Rath et al. [77] used an interface and geometry preserving (IGP) method to model fully Eulerian fluid-structure interactions with phase-field formulations.

He & Qiao [15] proposed a novel fully Eulerian method that uses level sets to track solid deformation, which we designate as the dynamic grid (DG) method. This method uses level sets to track the solid structure and avoids using the Lagrangian mesh, which simplifies the implementation. However, this approach suffers from stability issues caused by the excessive distortion of the dynamic grid outside the solid domain.

1.2.3 The Added-Mass Effect and Coupling Instabilities

A fundamental challenge in simulating fluid-structure interaction (FSI) is the accurate and stable resolution of the added-mass effect. This refers to the additional inertia a structure experiences as it accelerates, inherently displacing the surrounding fluid. While negligible at higher density ratios ($\rho_s/\rho_f \gg 1$), Causin et al. [78] shows it can cause a numerical scheme to become unconditionally unstable when fluid and structural densities are comparable ($\rho_s/\rho_f \approx 1$).

To mathematically illustrate this instability, consider a conventional partitioned FSI approach where the structural dynamics are governed by:

$$M_s \ddot{d} + C_s \dot{d} + K_s d = F_f \quad (1.1)$$

where M_s , C_s , and K_s are the structural mass, damping, and stiffness matrices, and F_f is the total fluid traction. The fluid force can be analytically decomposed into an added-mass force (proportional to instantaneous acceleration) and the remaining hydrodynamic forces, $\tilde{F}_f$ (such as viscous drag and convective pressure) [79]:

$$F_f = -M_a \ddot{d} + \tilde{F}_f \quad (1.2)$$

where M_a is the added-mass matrix. Substituting this decomposition yields:

$$(M_s + M_a) \ddot{d} + C_s \dot{d} + K_s d = \tilde{F}_f \quad (1.3)$$

In explicit partitioned solvers, calculating fluid forces using previous structural kinematics (step n) causes a temporal mismatch [78]:

$$M_s \ddot{d}^{n+1} \approx -M_a \ddot{d}^n + \dots \quad (1.4)$$

If $\rho_f \gtrsim \rho_s$, the spectral radius of the iteration matrix $M_s^{-1}M_a$ exceeds unity [78, 80]. The calculated acceleration artificially amplifies and rapidly diverges, triggering a catastrophic numerical breakdown known as the added-mass instability. While techniques like rigorous sub-iterations, convergence accelerators, or fictitious mass additions can stabilize these partitioned schemes, they significantly inflate computational cost and algorithmic complexity.

Fully Eulerian FSI formulations fundamentally circumvent this instability by treating both phases within a unified continuum, governed by a single momentum equation. For example, Sugiyama et al. [74] used the equation below:

$$\rho(\phi) \left(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right) = -\nabla p + \nabla \cdot \left[(1 - \phi_s) \boldsymbol{\sigma}^f + \phi_s \boldsymbol{\sigma}^s \right] + f_b \quad (1.5)$$

Here, u is the monolithic velocity field for the entire domain, $\sigma = (1 - \phi_s)\sigma^f + \phi_s\sigma^s$ is the global deviatoric stress tensor, and f_b represents body forces. The effective density $\rho(\phi)$ is continuously interpolated across the interface using a solid volume fraction function, ϕ :

$$\rho(\phi) = \phi\rho_s + (1 - \phi)\rho_f$$

Because this unified equation is solved implicitly over the entire domain, the local acceleration term $\partial u / \partial t$ simultaneously accounts for the inertia of both phases. The added-mass effect is intrinsically captured within the bulk matrix inversion without isolating the destabilizing M_a operator. This structural paradigm ensures robust algorithmic stability regardless of the density ratio, entirely bypassing the added-mass instability observed for Lagrangian-Eulerian methods.

1.3 Research Objectives

We focus on developing a solver for the DNS of immersed deformable solids with the ability to track and advect the solid-fluid interface, calculate the solid stress based on the appropriate material model and account for the two-way interaction of fluid and solid stresses.

The deformable solid is considered to be elastic and a unified framework with respect to equations of motion is used to solve for both fluid and solid dynamics, with viscous and solid stresses specified in their respective domains. Across the interface between the fluid and solid domains, the shear modulus transitions smoothly from the bulk shear modulus in the solid region to zero in the fluid domain over a few computational cells using a smoothed Heaviside function. A reconstruction method for the dynamic grid is carried out using a linear projection method [81] to resolve the stability issues caused by mesh distortion reported by [15]. The dynamic grid within the solid region is preserved and the dynamic grid out-

side the solid region is reconstructed at every time step based on local interfacial topology. With this method, the strain field outside the solid region is regularized without modifying the local velocity field. Validation cases are presented to demonstrate the accuracy of this method in comparison with other numerical results reported in the literature.

In the present work, we first introduce the numerical formulations implemented inspired by previous studies and then discuss about the improvements we made in terms of accuracy, ease of implementation, stability and robustness. The improvements in our method are (1): Improved robustness by reconstruction of the dynamic grid. (2): Improved stability by implementing continuous solid stress resulting from smoothed transition of the solid shear modulus. (3): Application to 3-D problems and problems with different density ratios. The detailed introduction of our method is discussed below.

We employ a fully Eulerian, unified formulation for the elastic solid and the incompressible fluid, both of which are represented on a uniform Cartesian grid. The solid level set [82] is used to track the solid domain and the solid-fluid interface. The deformation of the solid is represented by a dynamic grid system composed of three individual level set functions in 3-D (two level set functions in 2-D). The dynamic grid is advected with the local fluid velocity. The scalar function formation of the dynamic grid allows the divergence of the solid stress tensor to be added directly to the momentum equation as a source term for solid stress. This leads to a unified formulation for incompressible flow, where a single set of momentum and pressure equations is needed to characterize the velocities and pressures in both the solid and the fluid domains. Following the approach of He & Qiao [15], the strain field is determined from the set of n scalar functions that constitutes the dynamic grid, where n is the number of dimensions. The dynamic grid is initially aligned with the fixed uniform Cartesian grid for fluid variables. The misalignment between the two grids is used to measure the strain experienced by the solid material over time in the form of the Cauchy-Green deformation gradient tensor. The solid stress is then computed using the stress-strain

relationship for a given material. A Neo-Hookean model is used in this study for solid elastic stress calculations where we have used a constant bulk shear modulus. Other material models, such as those incorporating finite-time strain history, non-linear bulk modulus and other hyper-elastic models can be incorporated into our numerical method in a modular fashion.

We employ a finite width, smooth transition across the interface between the fluid and the solid. The shear modulus transitions smoothly from the bulk shear modulus in the solid to zero in the fluid using a smoothed Heaviside function, similar to the approach of Lee et al. [83]. We have addressed the stability issues caused by the unphysical distortion of the dynamic grid in the fluid region, reported by He & Qiao [15] using the linear projection method proposed by Aslam [81]. This approach allows the preservation of the dynamic grid within the solid region. The dynamic grid outside the solid is reconstructed at every time step based on local interfacial topology of the solid-fluid interface. With this method, the strain field outside the solid region is regularized without modifying the local velocity field in the neighborhood of the solid-fluid interface. We have implemented the method in our in-house solver, which is a scalable parallel computational code base using HYPRE with algebraic multigrid and adaptive grid refinement on uniform Cartesian grids [84, 85]. We show that our numerical implementation is as accurate as Lagrangian methods but is also robust and easy to implement in 3-D and antisymmetric code.

1.4 Thesis Outline

The thesis is structured as follows:

Chapter 1: Motivation and background of the study, literature review on different FSI methods, and objective of this study.

Chapter 2: Development and mathematical models used for the method in this study

from the unified form of governing equation to interface boundary conditions and discussion of material models.

Chapter 3: Specific numerical formulations used and details of algorithm implementation are discussed in 2-D and extend to Axisymmetric and 3-D.

Chapter 4: Results for validation and verification simulations for 2-D.

Chapter 5: Results for validation and verification simulations for 2-D–Axisymmetric.

Chapter 6: Results for validation and verification simulations for 3-D.

Chapter 7: Applications in the industry. Validation and development during the internship at Dassault Systèmes.

Chapter 8: Contributions of this work and discussion of the proposed future work.

Chapter 2: Mathematical model

2.1 Unified framework for momentum transport

We use a unified formulation to simultaneously solve for the pressure and velocity fields in the fluid and solid regions in a coupled manner. Both the fluid and the solid are taken to be incompressible. The fluid is assumed to be Newtonian while the solid is based on the neo-Hookean elastic material model. In the fluid region, the transport of momentum is governed by the standard Navier-Stokes equations, while in the solid region the viscous stress is replaced by a solid stress. For incompressible flow, the unified continuity and momentum equations can be expressed as:

$$\nabla \cdot \mathbf{u} = 0 \quad (2.1)$$

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \frac{1}{\rho} \nabla \cdot \boldsymbol{\tau} + \frac{1}{\rho} \nabla \cdot \boldsymbol{\sigma} + \mathbf{g} \quad (2.2)$$

where $\boldsymbol{\tau}$ and $\boldsymbol{\sigma}$ are the viscous and solid stress tensors, respectively, that are defined as,

$$\boldsymbol{\tau} = \mu_f [(\nabla \mathbf{u}) + (\nabla \mathbf{u})^T] \quad (2.3)$$

$$\boldsymbol{\sigma} = \mu_s (\mathbf{F} \mathbf{F}^T - \mathbf{I}) \quad (2.4)$$

where μ_f and μ_s are the fluid and solid stress coefficients, respectively. The tensor $(\nabla \mathbf{u}) + (\nabla \mathbf{u})^T$ is the strain-rate tensor of the fluid, $\mathbf{F}$ is the deformation gradient tensor in

the solid, and $\mathbf{I}$ is the second-order identity tensor. We implement the non-dimensional form of Eq. (2.2), which can be written as,

$$\begin{aligned} \frac{\partial \mathbf{u}^*}{\partial t^*} + \mathbf{u}^* \cdot \nabla \mathbf{u}^* = & -\frac{1}{\tilde{\rho}} \nabla p^* + \frac{1}{\tilde{\rho} Re} \nabla \cdot [(\nabla \mathbf{u}^*) + (\nabla \mathbf{u}^*)^T] \\ & + \frac{S}{\tilde{\rho}} \nabla \cdot (\mathbf{F}\mathbf{F}^T - \mathbf{I}) + \frac{1}{Fr^2} \mathbf{j} \end{aligned} \quad (2.5)$$

where non-dimensional variables are denoted with $*$. The non-dimensional numbers Re , S , Fr and density ratio $\tilde{\rho}$ are defined as,

$$Re = \frac{\rho_f UL}{\mu_f} \quad (2.6)$$

$$S = \frac{\mu_s}{\rho_f U^2} \quad (2.7)$$

$$Fr = \frac{U}{\sqrt{gL}} \quad (2.8)$$

$$\tilde{\rho} = \frac{\rho}{\rho_f} \quad (2.9)$$

ρ is the continuous density field defined in Eq. (2.17). ρ_f is the fluid density. Fig. 2.1 shows the schematic of the solid, $\mathbf{x} \in \Omega_s$, and the fluid, $\mathbf{x} \in \Omega_f$, domains that are separated by the solid-fluid interface, $\mathbf{x} \in \Gamma$.

2.2 Interfacial jump conditions

In the absence of melting or solidification, mass conservation requires the solid and fluid velocities to be continuous at the interface,

$$[\mathbf{u}] = 0 \quad (2.10)$$

where $[\cdot]$ indicates the jump across the interface. The viscous stress, $\boldsymbol{\tau}$, and the solid stress, $\boldsymbol{\sigma}$, act within their respective domains and undergo a step change to zero across the solid-

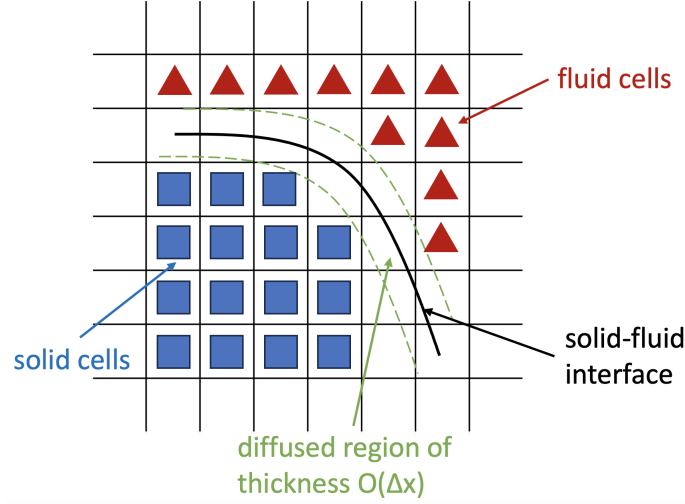


Figure 2.1: Domains of fluid and solid across which respective pressure and velocity fields are coupled in the unified formulation. The solid black line represents the solid-fluid interface. Fluid and solid material properties vary smoothing in the finite region between the two green dashed lines around the interface.

fluid interface, when the interface is sharp. The discontinuity in the stress field across the interface gives rise to a singular jump in pressure, which can be expressed as,

$$[p] = [\boldsymbol{\tau} \cdot \hat{\mathbf{n}} \cdot \hat{\mathbf{n}}] + [\boldsymbol{\sigma} \cdot \hat{\mathbf{n}} \cdot \hat{\mathbf{n}}] \quad (2.11)$$

where $\hat{\mathbf{n}}$ is the interfacial normal unit vector. The first term on the right hand side in Eq. (2.11) relates to the interfacial jump in viscosity, from $\mu_f = \mu_f^*$, $\mathbf{x} \in \Omega_f$ to $\mu_f = 0$, $\mathbf{x} \in \Omega_s$, where μ_f^* is the dimensionless fluid viscosity. The second term on the right hand side in Eq. (2.11) is due to the jump in the elastic modulus across the interface, from $\mu_s = \mu_s'$, $\mathbf{x} \in \Omega_s$ to $\mu_s = 0$, $\mathbf{x} \in \Omega_f$, where μ_s' is the elastic modulus. Eq. (2.11) indicates that the pressure jump occurs to compensate for the interfacial force density that arises due to the discontinuity of normal viscous and solid stresses across the interface. The pressure jump condition in Eq. (2.11) can be added to the discrete form of the momentum equation (Eq. (2.2)), using various Cartesian grid methods for sharp jumps, such as the ghost fluid

method or the immersed interface method.

The jump in the tangential viscous stress across the solid-fluid interface is continuous for elastic solid deformation:

$$[\boldsymbol{\tau} \cdot \hat{\mathbf{n}} \cdot \hat{\mathbf{t}}] + [\boldsymbol{\sigma} \cdot \hat{\mathbf{n}} \cdot \hat{\mathbf{t}}] = 0, \quad (2.12)$$

where $\hat{\mathbf{t}}$ is the unit tangent vector at the interface. We should also note that the solid-fluid domain coupling, as formulated in the current unified approach, is not applicable to a rigid body. In the limit, $\mu_s \rightarrow \infty$, the pressure jump, $[p] \rightarrow \infty$. Hence, the pressure in the fluid domain must be decoupled from the solid. The velocity in the two domains must then be computed separately, which is the approach used in immersed boundary methods when dealing with rigid bodies.

2.3 Distributed material properties

The sharp treatment of interfacial stresses can potentially lead to nonphysical pressure oscillations resulting in numerical instability. Distributing the stresses over a finite width around the interface, as shown in Fig. 2.1, can provide a remedy. This can be accomplished by imagining the fluid and solid stress coefficients to be continuous around the interface,

$$\mu_f(\mathbf{x}) = \mu'_f H(\phi(\mathbf{x})) + \mu_f^* [1 - H(\phi(\mathbf{x}))] \quad (2.13)$$

$$\mu_s(\mathbf{x}) = \mu'_s H(\phi(\mathbf{x})) \quad (2.14)$$

where μ_f^* is the fluid viscosity and μ'_f is a fictitious fluid viscosity in the solid region, which is generally set to 0. However, a small non zero value can be useful for numerical stability, particularly during the initial transient phase [14]. μ'_s is the solid elastic modulus. The heaviside function, H , which is a function of the signed distance function, ϕ , allows

the solid and fluid stress coefficients to take on their respective values within the solid and the fluid domains, and across the interface. The signed distance function is a scalar field that represents the normal distance to the solid-fluid interface. It is described in more detail in section 3.1. We take the heaviside function to be,

$$H(\phi(\mathbf{x})) = \frac{1}{2} \left[1 + \operatorname{erf} \left(\frac{\phi(\mathbf{x})}{\varepsilon} \right) \right] \quad (2.15)$$

where erf is the error function and ε is the spreading coefficient that controls the width of the smooth region around the interface. The signed distance function is positive in the solid domain, negative in the fluid domain and zero at the interface:

$$\begin{aligned} \phi(\mathbf{x}) &> 0 \quad , \quad \mathbf{x} \in \Omega_s \\ \phi(\mathbf{x}) &= 0 \quad , \quad \mathbf{x} \in \Gamma \\ \phi(\mathbf{x}) &< 0 \quad , \quad \mathbf{x} \in \Omega_f \end{aligned} \quad (2.16)$$

The heaviside function thus varies smoothly between 1 and zero, $H(\mathbf{x}) \in [1, 0]$, across the interface, Γ , from the solid to the fluid, respectively.

It is used to allow density to smoothly transition between its respective values in the solid and the fluid domains,

$$\rho(\mathbf{x}) = \rho_s H(\phi(\mathbf{x})) + \rho_f [1 - H(\phi(\mathbf{x}))], \quad (2.17)$$

where ρ_s and ρ_f are the solid and fluid densities, respectively.

The smoothing of stress coefficients given by Eq. (2.14) results in a unified stress tensor, $\tau_f + \sigma$, which is continuous around the interface. The singular jump in pressure (Eq. (2.11)) is relieved as a result, $[p] = 0$, and the pressure varies continuously over the finite width around the interface. Momentum is still conserved across the interface because the pressure

variation is recovered implicitly in the discretization of the continuous unified stress field in the momentum equation. This simplifies the discretization of the momentum equation, particularly in 3-D, and leads to robust numerical solutions. In section 4 – 6, we show that the numerical accuracy of the distributed stress approach is similar to that for the sharp interface case.

2.4 Neo-Hookean material model

The state of stress in the solid can be described using a Cauchy stress tensor, which is expressed as a function of the strain. In this work, we use an isotropic, Neo-Hookean hyperelastic material model as the constitutive law for the solid. For a hyperelastic material, the first Piola-Kirchhoff stress tensor $\mathbf{P}$ can be expressed as the derivative of the strain energy density function $\Psi(\mathbf{F})$ with respect to the deformation gradient tensor $\mathbf{F}$ [86] shown as:

$$\mathbf{P} = \frac{\partial \Psi(\mathbf{F})}{\partial \mathbf{F}} \quad (2.18)$$

Thus the Cauchy stress tensor for the solid can be expressed in terms of the deformation gradient tensor $\mathbf{F}$ by [86]:

$$\boldsymbol{\sigma}_s = J^{-1} \mathbf{P} \mathbf{F}^T = J^{-1} \frac{\partial \Psi(\mathbf{F})}{\partial \mathbf{F}} \mathbf{F}^T \quad (2.19)$$

where $J = \det(\mathbf{F})$. For a compressible Neo-Hookean solid, the strain energy density function can be written as [87]:

$$\Psi(\mathbf{F}) = C_1(I_1 - 3 - 2\ln J) + D_1(J - 1)^2, \quad (2.20)$$

where $D_1 = \lambda_L/2$, $C_1 = \mu_s/2$. λ_L is the first Lamé parameter. μ_s is the shear modulus of the solid or the second Lamé parameter. I_1 is the first invariant (trace) of the right Cauchy–Green deformation tensor $\mathbf{C}$. The right Cauchy–Green deformation tensor can be expressed as $\mathbf{C} = \mathbf{F}^T \mathbf{F}$. Substituting Eq. (2.20) into Eq. (2.19), we can get:

$$\begin{aligned}
\boldsymbol{\sigma} &= J^{-1} \left(2C_1 \mathbf{F} - 2C_1 \mathbf{F}^T + 2D_1 (J - 1) J \mathbf{F}^{-T} \right) \mathbf{F}^T \\
&= J^{-1} 2C_1 \left(\mathbf{F} \mathbf{F}^T - \mathbf{I} \right) + 2D_1 (J - 1) \mathbf{I} \\
&= J^{-1} \mu_s \left(\mathbf{F} \mathbf{F}^T - \mathbf{I} \right) + \lambda_L (J - 1) \mathbf{I}
\end{aligned} \tag{2.21}$$

Using the incompressibility constraint for the Neo-Hookean solid, $J = 1$, results in Eq. (2.4), which relates the solid stress to the deformation gradient tensor for an incompressible elastic solid.

Chapter 3: Discretization and numerical implementation

We use the Eulerian approach for both the fluid and the solid domains. The solid domain is tracked using a signed distance function, $\phi(\mathbf{x}, t)$, as defined in Eq. (2.16). The surface of the solid body is tracked by, $\phi(\mathbf{x}, t) = 0$, inside the solid, $\phi(\mathbf{x}, t) > 0$ and $\phi(\mathbf{x}, t) < 0$ in the fluid domain. The signed distance function represents the minimum distance between any point in space and the corresponding point on the interface, $\mathbf{x}_\Gamma$, which can be expressed as,

$$\phi(\mathbf{x}, t) = \min(|\mathbf{x} - \mathbf{x}_\Gamma(t)|) \quad (3.1)$$

where $\mathbf{x}_\Gamma(t)$ represents the set of all points that are located on the interface at any time t . The signed distance function implies that $|\nabla \phi| = 1$.

The deformation of the solid is tracked using a set of 3 scalar functions, $X(\mathbf{x}, t)$, $Y(\mathbf{x}, t)$ and $Z(\mathbf{x}, t)$, that we broadly label as the dynamic grid. Each of the elements of the dynamic grid is initially assigned to one Cartesian coordinate. The deformation gradient tensor can be defined as,

$$\mathbf{F} = \frac{\partial \mathbf{x}}{\partial \mathbf{X}} = \left(\frac{\partial \mathbf{X}}{\partial \mathbf{x}} \right)^{-1} \quad (3.2)$$

The gradients of X , Y and Z elements of the dynamic grid with respect to the spatial coordinates, x , y and z can be computed on the Cartesian grid to form $\mathbf{F}^{-1}$, which is then inverted to determine $\mathbf{F}$ for computing the solid stress in Eq. (2.4). We start the simulation with a stress free condition using $X(\mathbf{x}, 0) = x$, $Y(\mathbf{x}, 0) = y$ and $Z(\mathbf{x}, 0) = z$. In Fig. 3.1, (a) and (b) illustrate the initial state of the dynamic grid and the surface of a spherical solid

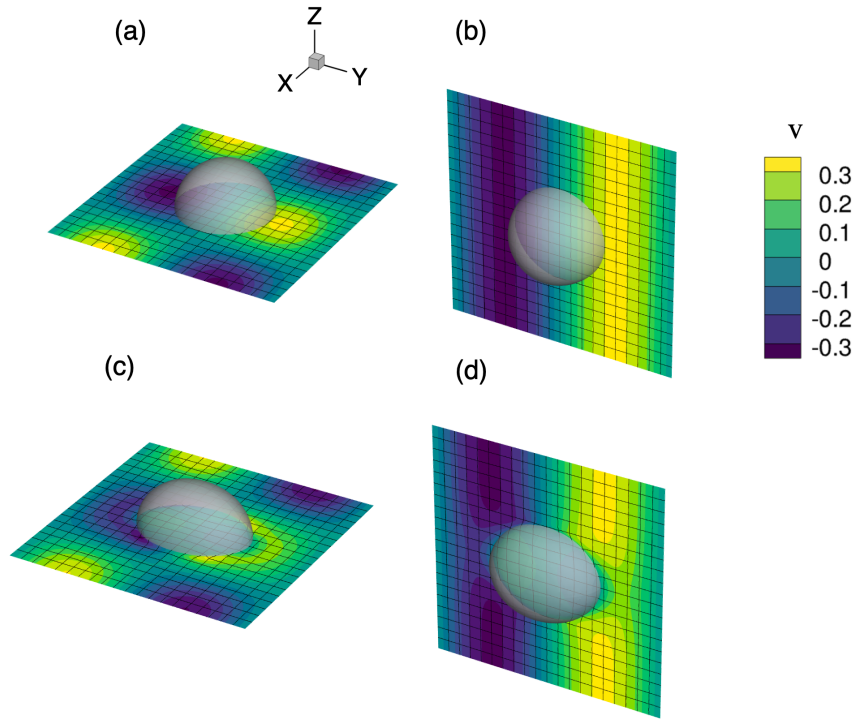


Figure 3.1: The solid-fluid interface (colored in grey) is initialized as a sphere at the center of the domain. (a) and (b) are slices in x-y and y-z plane at initial state. They overlap with the background Cartesian grid. (c) and (d) are x-y and y-z plane at a deformed state. The black solid lines illustrate dynamic grid and the contours are colored by velocity component in the y direction

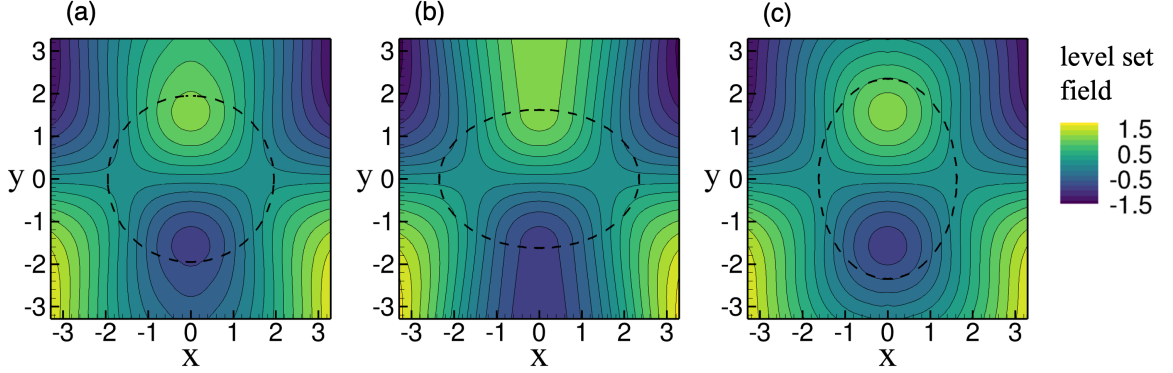


Figure 3.2: Extrapolation of the test level set field from inside the circular and elliptical interface to the outside of the interface. The interface is marked with dashed line. Contours are colored by level set values. We use the linear extrapolation scheme [81] in this study.

body, indicated by the iso-surface of $\phi = 0$; (c) and (d) illustrate the deformed dynamic grid and the surface of a spherical solid body.

3.1 Advection and re-initialization of the interface and the dynamic grid

The the signed distance function $\phi_s(\mathbf{x}, t)$ that tracks the solid–fluid interface is advected by the local velocity field $\mathbf{u}(\mathbf{x}, t)$ according to,

$$\frac{\partial \phi}{\partial t} + \mathbf{u} \cdot \nabla \phi = 0 \quad (3.3)$$

In Cartesian coordinate, the dynamic grid scalar functions $X(\mathbf{x}, t)$, $Y(\mathbf{x}, t)$ and $Z(\mathbf{x}, t)$ are advected by the same velocity field according to,

$$\frac{\partial X}{\partial t} + \mathbf{u} \cdot \nabla X = 0 \quad (3.4)$$

$$\frac{\partial Y}{\partial t} + \mathbf{u} \cdot \nabla Y = 0 \quad (3.5)$$

$$\frac{\partial Z}{\partial t} + \mathbf{u} \cdot \nabla Z = 0 \quad (3.6)$$

Since we have also implemented the algorithm in 2D–Axisymmetric, the advection in the Cylindrical–polar coordinate can be written as,

$$\frac{\partial Z}{\partial t} + \mathbf{u} \cdot \nabla Z = 0 \quad (3.7)$$

$$\frac{\partial R}{\partial t} + \mathbf{u} \cdot \nabla R = 0 \quad (3.8)$$

The advection terms in the above equations are discretized using 5th order weighted essentially non-oscillatory space(WENO5) scheme [34]. A reinitialization is needed for $\phi(\mathbf{x}, t)$ after advection to maintain the signed distance property, $|\nabla \phi| = 1$. The first order Hamilton-Jacobi equation is used for reinitialization,

$$\frac{\partial \phi}{\partial t'} + \text{Sgn}(\phi) (|\nabla \phi| - 1) = 0 \quad (3.9)$$

where ϕ at any given time, t , is marched in fictitious time, t' , to a steady state where $|\nabla \phi_s - 1|$ is equal to zero. $\text{Sgn}(\phi_s)$ is a sign function:

$$\text{Sgn}(\phi) = \frac{\phi}{|\phi| + \varepsilon} \quad (3.10)$$

where ε is a small value added to avoid singularity when $\phi = 0$.

The dynamic grid also needs re-initialization. Since the dynamic grid is used solely to track solid deformation, velocity in the fluid region can cause unphysical deformation of the grid and undermines the stability of the simulation. It is reported by He and Qiao [15] that using fluid velocity to advect the grid can lead to significant distortion of the grid outside solid. This error can propagate into the solid region over time during the simulation and leads to unphysical distortions in the solid body. To address this issue, we have implemented a linear extrapolation method [81] to re-initialize $X(x, y, z)$, $Y(x, y, z)$ and $Z(x, y, z)$

outside the solid boundary. Similar to the initialization method used for solid-fluid interface, this method also solves an advection equation to steady state. The equations are listed below, taking $X(x, y, z)$ as an example. The extrapolation procedures are the same for $Y(x, y, z)$ and $Z(x, y, z)$.

$$X_n = \hat{\mathbf{n}} \cdot \nabla X \quad (3.11)$$

$$\frac{\partial X_n}{\partial t} + H(\phi) \hat{\mathbf{n}} \cdot \nabla X_n = 0 \quad (3.12)$$

$$\frac{\partial X}{\partial t} + H(\phi) (\hat{\mathbf{n}} \cdot \nabla X - X_n) = 0 \quad (3.13)$$

$$\hat{\mathbf{n}} = \frac{\nabla \phi_s}{|\nabla \phi_s|} \quad (3.14)$$

$$H(\phi) = \begin{cases} 0 & \text{if } \phi > 0 \\ 1 & \text{if } \phi \leq 0 \end{cases} \quad (3.15)$$

Unit normal vector of the interface can be calculated using Eq. (3.14). The normal derivative of the scalar X is defined by Eq. (3.11). A step function defined by Eq. (3.15) restricts the advection only on the interface and outside the solid region where ϕ_s is less or equal to 0. Using the steady state solution of the normal derivative X_n from Eq. (3.12), one can then solve Eq. (3.13) to steady state and obtain the extrapolated field for X . Fig. 3.2 shows the extrapolation of a level set field from inside the circular and elliptical interface to the outside of the interface.

In this way, the solid strain history within the solid region is preserved and the dynamic grid outside the solid region is re-initialized based on the dynamic grid inside the solid and the local topology of the interface. Comparison of the grids before and after re-initialization

is provided in section 4.2.

3.2 Calculation and discretization of the dynamic grid scalar functions

3.2.1 Discretization in 2-D

The deformation gradient tensor is calculated using the derivatives of the dynamic grid given by Eq. (3.2). The calculation of the derivatives of $\mathbf{x}$ with respect to $\mathbf{X}$ is not straightforward. However, the derivatives of $\mathbf{X}$ with respect to $\mathbf{x}$ can be readily computed.

$$\mathbf{F}^{-1} = \frac{\partial \mathbf{X}}{\partial \mathbf{x}} = \begin{bmatrix} \frac{\partial X}{\partial x} & \frac{\partial X}{\partial y} \\ \frac{\partial Y}{\partial x} & \frac{\partial Y}{\partial y} \end{bmatrix} \quad (3.16)$$

The components of the inverse deformation gradient tensor are calculated using second order central differencing scheme. Take $\partial X / \partial x$ as an example, it is approximated using central differencing scheme as:

$$\frac{\partial X}{\partial x} = \frac{X_{i+1,j,k} - X_{i-1,j,k}}{2\Delta x} \quad (3.17)$$

We then inverse the $\mathbf{F}^{-1}$ tensor to get deformation gradient tensor $\mathbf{F}$ and transpose of deformation gradient tensor $\mathbf{F}^T$ which are needed in the solid strain tensor in Eq. (2.4).

In 2D, the left Cauchy–Green deformation tensor $\mathbf{FF}^T$ can be simply expressed in terms of the dynamic grid level set as a function of the background Eulerian grid as follows:

$$\mathbf{FF}^T = \frac{1}{[\det(\mathbf{F}^{-1})]^2} \begin{bmatrix} \left(\frac{\partial Y}{\partial y}\right)^2 + \left(\frac{\partial X}{\partial y}\right)^2 & -\frac{\partial Y}{\partial y} \frac{\partial Y}{\partial x} - \frac{\partial X}{\partial y} \frac{\partial X}{\partial x} \\ -\frac{\partial Y}{\partial x} \frac{\partial Y}{\partial y} - \frac{\partial X}{\partial x} \frac{\partial X}{\partial y} & \left(\frac{\partial Y}{\partial x}\right)^2 + \left(\frac{\partial X}{\partial x}\right)^2 \end{bmatrix} \quad (3.18)$$

3.2.2 Discretization in 2-D-Axisymmetric

In 2-D-Axisymmetric, deformation gradient tensor is calculated in cylindrical polar coordinates. We have mapped z in the cylindrical coordinates to x and r in the cylindrical coordinates to y : $\mathbf{X}(x,y) \rightarrow \mathbf{X}(z,r)$. As a result, the inverse of the deformation gradient tensor becomes:

$$\mathbf{F}^{-1} = \frac{\partial \mathbf{X}}{\partial \mathbf{x}} = \begin{bmatrix} \frac{\partial Z}{\partial z} & \frac{\partial Z}{\partial r} \\ \frac{\partial R}{\partial z} & \frac{\partial R}{\partial r} \end{bmatrix} \quad (3.19)$$

The left Cauchy-Green deformation tensor $\mathbf{FF}^T$ can then be expressed as follows:

$$\mathbf{FF}^T = \frac{1}{[\det(\mathbf{F}^{-1})]^2} \begin{bmatrix} \left(\frac{\partial Z}{\partial r}\right)^2 + \left(\frac{\partial R}{\partial r}\right)^2 & -\frac{\partial R}{\partial r} \frac{\partial R}{\partial z} - \frac{\partial Z}{\partial r} \frac{\partial Z}{\partial z} \\ -\frac{\partial Z}{\partial r} \frac{\partial Z}{\partial z} - \frac{\partial R}{\partial r} \frac{\partial R}{\partial z} & \left(\frac{\partial R}{\partial z}\right)^2 + \left(\frac{\partial Z}{\partial z}\right)^2 \end{bmatrix} \quad (3.20)$$

The solid stress tensor can then be calculated using Eq. (2.4).

3.2.3 Discretization in 3-D

In 3-D, the derivatives of $\mathbf{X}$ with respect to $\mathbf{x}$ can be readily computed.

$$\mathbf{F}^{-1} = \frac{\partial \mathbf{X}}{\partial \mathbf{x}} = \begin{bmatrix} \frac{\partial X}{\partial x} & \frac{\partial X}{\partial y} & \frac{\partial X}{\partial z} \\ \frac{\partial Y}{\partial x} & \frac{\partial Y}{\partial y} & \frac{\partial Y}{\partial z} \\ \frac{\partial Z}{\partial x} & \frac{\partial Z}{\partial y} & \frac{\partial Z}{\partial z} \end{bmatrix} \quad (3.21)$$

The left Cauchy-Green deformation tensor $\mathbf{FF}^T$ can be calculated using the dynamic grid scalar functions $X(\mathbf{x},t)$, $Y(\mathbf{x},t)$ and $Z(\mathbf{x},t)$. In 3D, the left Cauchy-Green deformation tensor is calculated as follow:

The matrix of minors $\mathbf{M}$ for $\mathbf{F}^{-1}$ can be defined as:

$$\mathbf{M} = \begin{bmatrix} \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} \\ \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} \\ \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} \\ \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} \\ \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} & \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Z} - \frac{\partial Y}{\partial X} \frac{\partial Z}{\partial Y} \end{bmatrix} \quad (3.22)$$

The left Cauchy–Green deformation tensor $\mathbf{B} = \mathbf{F}\mathbf{F}^T$ can be expressed in terms of the dynamic grid level set as a function of the background Eulerian grid as:

$$\mathbf{B} = \mathbf{F}\mathbf{F}^T = \frac{1}{[\det(\mathbf{F}^{-1})]^2} \begin{bmatrix} B_{11} & B_{12} & B_{13} \\ B_{21} & B_{22} & B_{23} \\ B_{31} & B_{32} & B_{33} \end{bmatrix} \quad (3.23)$$

where

$$\begin{aligned} \mathbf{B}_{11} &= (M_{11}^2 + M_{21}^2 + M_{31}^2) \\ \mathbf{B}_{12} &= (-M_{11}M_{12} - M_{21}M_{22} - M_{31}M_{32}) \\ \mathbf{B}_{13} &= (M_{11}M_{13} + M_{21}M_{23} + M_{31}M_{33}) \\ \mathbf{B}_{21} &= (-M_{12}M_{11} - M_{22}M_{21} - M_{32}M_{31}) \\ \mathbf{B}_{22} &= (M_{12}^2 + M_{22}^2 + M_{32}^2) \\ \mathbf{B}_{23} &= (-M_{12}M_{13} - M_{22}M_{23} - M_{32}M_{33}) \\ \mathbf{B}_{31} &= (M_{13}M_{11} + M_{23}M_{21} + M_{33}M_{31}) \\ \mathbf{B}_{32} &= (-M_{13}M_{12} - M_{23}M_{22} - M_{33}M_{32}) \\ \mathbf{B}_{33} &= (M_{13}^2 + M_{23}^2 + M_{33}^2) \end{aligned}$$

The solid stress tensor can then be calculated using Eq. (2.4).

3.3 Numerical algorithm of implementing the solid stress term

We use the fractional step method to solve the unified momentum equation, Eq. (2.2). An additional intermediate step is needed for the solid stress compared to the fractional

step method for the fluid only problems. The algorithm can be summarized in the following steps:

1. Advection of the interface level set function and dynamic grid scalar functions using the local velocity field using Eq. (3.3-3.6).
2. Reconstruction of the dynamic grid level set function outside the solid to ensure smoothness in the dynamic grid across the solid-fluid interface using Eq. (3.11-3.13).
3. Calculation of the Cauchy–Green deformation tensor with the information from the dynamic grid scalar functions using Eq. (3.20) and then calculate solid stress using using Eq. (2.4).
4. Explicit solution of the advection and diffusion terms for the fluid to obtain the intermediate velocity u^* using an Adams-Bashforth method. The intermediate velocity u^* is calculated by the equation below. The advection terms are discretized using a WENO3 scheme and the diffusion terms are discretized using the central differencing scheme:

$$\begin{aligned} \frac{\mathbf{u}^* - \mathbf{u}^n}{\Delta t} = & \frac{3}{2} \left[-\mathbf{u}^n \cdot \nabla \mathbf{u}^n + \frac{1}{\rho} \nabla \cdot \boldsymbol{\tau}^n \right] \\ & - \frac{1}{2} \left[-\mathbf{u}^{n-1} \cdot \nabla \mathbf{u}^{n-1} + \frac{1}{\rho} \nabla \cdot \boldsymbol{\tau}^{n-1} \right] + \mathbf{g} \end{aligned} \quad (3.24)$$

5. Solution of the solid stress term for the solid region to get the second intermediate velocity u^{**} .

$$\mathbf{u}^{**} = \mathbf{u}^* + \Delta t \nabla \cdot \boldsymbol{\sigma} = \mathbf{u}^* + \Delta t \nabla \cdot \mu_s (\mathbf{F}\mathbf{F}^T - \mathbf{I}) \quad (3.25)$$

6. Solving the Poisson equation to obtain the unified pressure field.

$$\nabla \cdot \left(\frac{\nabla p}{\rho} \right) = \frac{\nabla \cdot \mathbf{u}^{**}}{\Delta t} \quad (3.26)$$

7. Solution of the velocity field at time step n+1 using u^{**} and the pressure gradient.

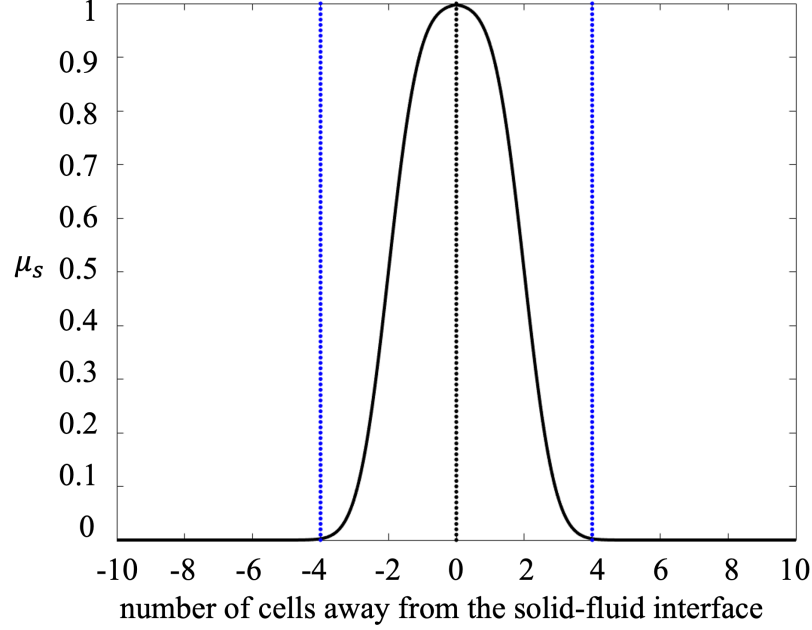


Figure 3.3: A thin layer distribution of the shear modulus μ_s . The black dashed line represents solid. The blue dashed lines represent solid interface, which are boundaries for distribution of shear modulus and solid stress. In this implementation, solid shear modulus goes from 1 to 0 within 4 cells away from the solid.

$$\frac{\mathbf{u}^{n+1} - \mathbf{u}^{**}}{\Delta t} = -\frac{\nabla p}{\rho} \quad (3.27)$$

8. Use Eq. (3.27) to solve for the updated velocity field $\mathbf{u}^{n+1}$.

3.4 Extension of the solid region to a thin layer geometry

We have extended our formulation to a thin layer geometry to better model metal oxide films. We have implemented a modified Heaviside function to make the shear modulus of the solid equal to 1 at the solid-fluid interface and going to 0 within a few grid cells on both sides of the interface. For example, the shear modulus of the solid can be distributed by the following functions:

$$\lambda_s(\mathbf{x}) = -|\phi_s(\mathbf{x})| + 4\Delta x \quad (3.28)$$

λ_s describes a supporting region of a length of $8\Delta x$. It is used to calculate solid stress.

$$H(\lambda_s(\mathbf{x})) = \frac{1}{2} \left[1 + \operatorname{erf} \left(\frac{\lambda_s(\mathbf{x}) - 2\Delta x}{2\Delta x} \right) \right] \quad (3.29)$$

$\mu_s = 1$ at the interface, $\mu_s(H) = \mu_s H(\lambda_s(\mathbf{x}))$ results in the distribution in Figure 2.4.

We have modified the oscillating disk problem to an oscillating ring problem to showcase our implementation. We have also used this formulation to set up simulations of an impinging liquid jet that enters a pool of liquid with solid layer at the free surface. To investigate the effects of oscillation in inlet flow on solid layer, we have set up simulations with solid film deforming under periodic velocity inlet. preliminary results are shown in the next section.

Chapter 4: Fluid–Elastic Solid Interaction in 2–D

In this section, a comprehensive suite of two-dimensional simulations is presented to validate and demonstrate the capabilities of the proposed fully Eulerian fluid–structure interaction (FSI) framework. The chapter begins by examining the fundamental numerical behavior of the methodology, featuring a parametric study of a relaxing sine-wave-shaped layer, an evaluation of the dynamic grid extrapolation technique utilized for stability, and a rigorous error analysis. Following this numerical validation, the solver is applied to a variety of canonical benchmark problems, including immersed oscillating bodies and deformable elastic structures interacting with lid-driven cavity and vertical channel flows. Finally, to highlight the method’s applicability to the thin, flexible structures characteristic of oxide films in liquid metal casting, the latter sections investigate highly dynamic scenarios. These include the transient response of pre-deformed solid films, films subjected to localized point forces, and the complex secondary flows generated by solid filaments under periodic oscillatory inflow. Through these diverse test cases, the robustness, accuracy, and versatility of the numerical implementation are established.

4.1 Parametric study for relaxation of a sine wave shaped layer

In this set of simulations, a 3 layer fluid system is set up in the shape of a sine wave as shown in Fig. 4.1. The thickness of the intermediate layer between two interfaces is set to be uniform in the entire domain. The interfaces change their shapes due to surface tension

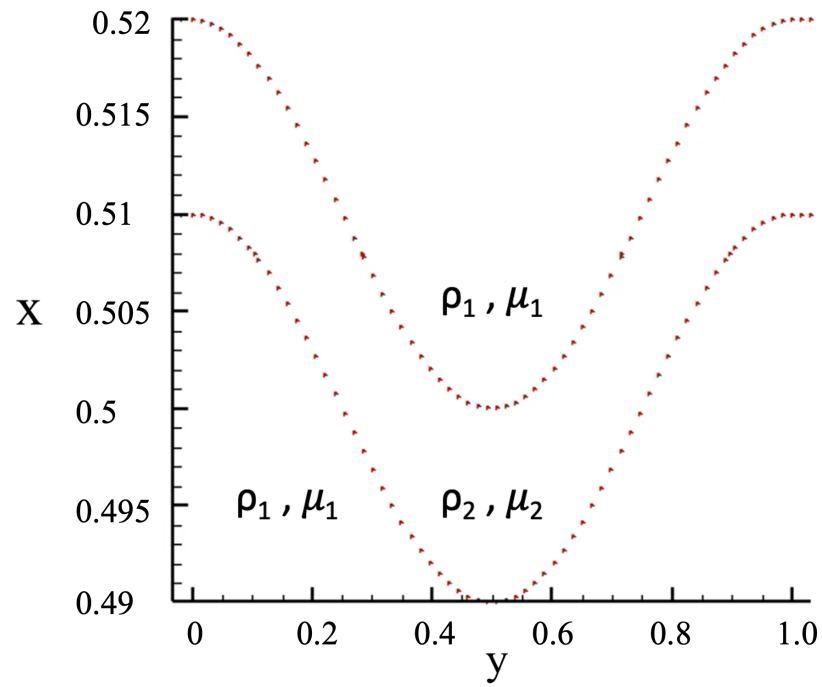


Figure 4.1: Initial shape of the 3 layer fluid system

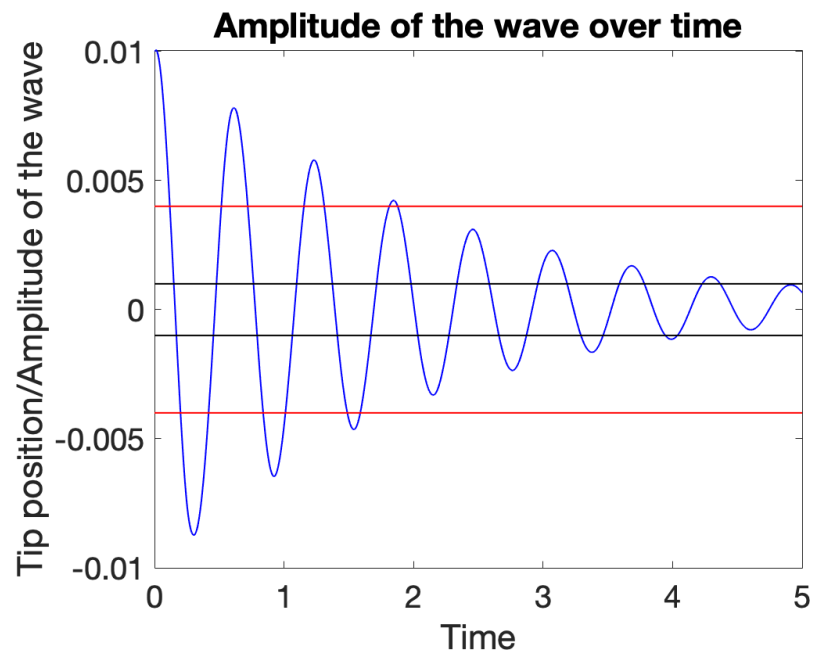


Figure 4.2: Amplitude of the wave over time

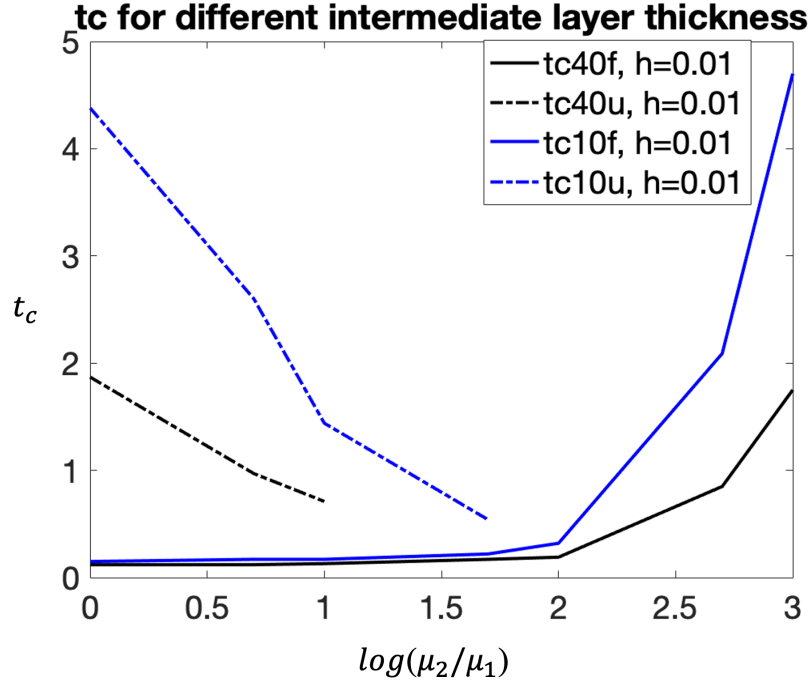


Figure 4.3: Critical time for viscosity ratio from 1 to 1000

and relax to a flat interface at steady state. The tip position on the bottom interface wave is tracked over time as the amplitude of the wave. It is then plotted as a function of time to find the critical time for relaxation t_c . For all the cases, we used the same fluid in the top and bottom layer and keep the density same for all 3 layers. A parametric study was performed for cases with different viscosity ratios μ_2/μ_1 and intermediate layer thicknesses to see their relaxation behaviors. As shown in Fig. 4.2, points on the blue curve are the tip positions on the capillary wave at different time. The red lines and black lines are marked at 40% and 10% of the initial amplitude. Critical time length t_{c40f} and t_{c10f} are defined as the time it takes for the wave to first reaches 40% and 10% of the initial amplitude. t_{c40u} and t_{c10u} are defined as the time it takes for amplitude of the wave to ultimately fall below 40% and 10% of the initial amplitude. These critical time lengths are used to describe relaxation behaviors for different cases.

Fig. 4.3 shows the results for cases with different viscosity ratios. Results have shown that there is no substantial difference between 40% and 10% criteria except that it takes longer time to reach 10% criteria. However, t_{c40u} and t_{c10u} decrease as μ_2/μ_1 increase while t_{c40f} and t_{c10f} increase as μ_2/μ_1 increase. This observation can be explained by the fact that the higher viscosity ratio leads to fewer oscillations, thus a shorter time to fall below t_{c40u} and t_{c10u} criteria. When it comes to t_{c40f} and t_{c10f} criteria, higher viscosity fluid is more resistance to the motion which brings the interfaces flat. A critical viscosity ratio μ_2/μ_1 between 100 and 200 is observed where t_{cu} is equivalent to t_{cf} because the amplitude of the wave falls below the percentage criteria in one oscillation. Simulations for different initial wave amplitude and intermediate layer thicknesses were also performed and the results agree with these observations.

This study has shown that the oscillating behaviours of the intermediate stress can be changed by adjusting viscosity ratios. However, this could only be used as relatively low accuracy approach to model the solid metal oxide layer because of the absence of an appropriate strain model for the solid. To address the problem of modeling the oxide layer accurately, a numerical approach of directly computing the solid stress and accounting for it in the solid region is required.

4.2 Dynamic grid extrapolation

The dynamic grid is well defined only within the solid domain. On the other hand, the calculation of the deformation gradient tensor in the finite-width interface region requires information about the strain field from outside the solid, where the solid strain field is not defined. The strain field in the fluid does not match smoothly with that in the solid at the solid-fluid interface, and hence cannot be used. Therefore, in order to avoid numerical stability issues near the solid-fluid interface, we use a multidimensional extrapolation

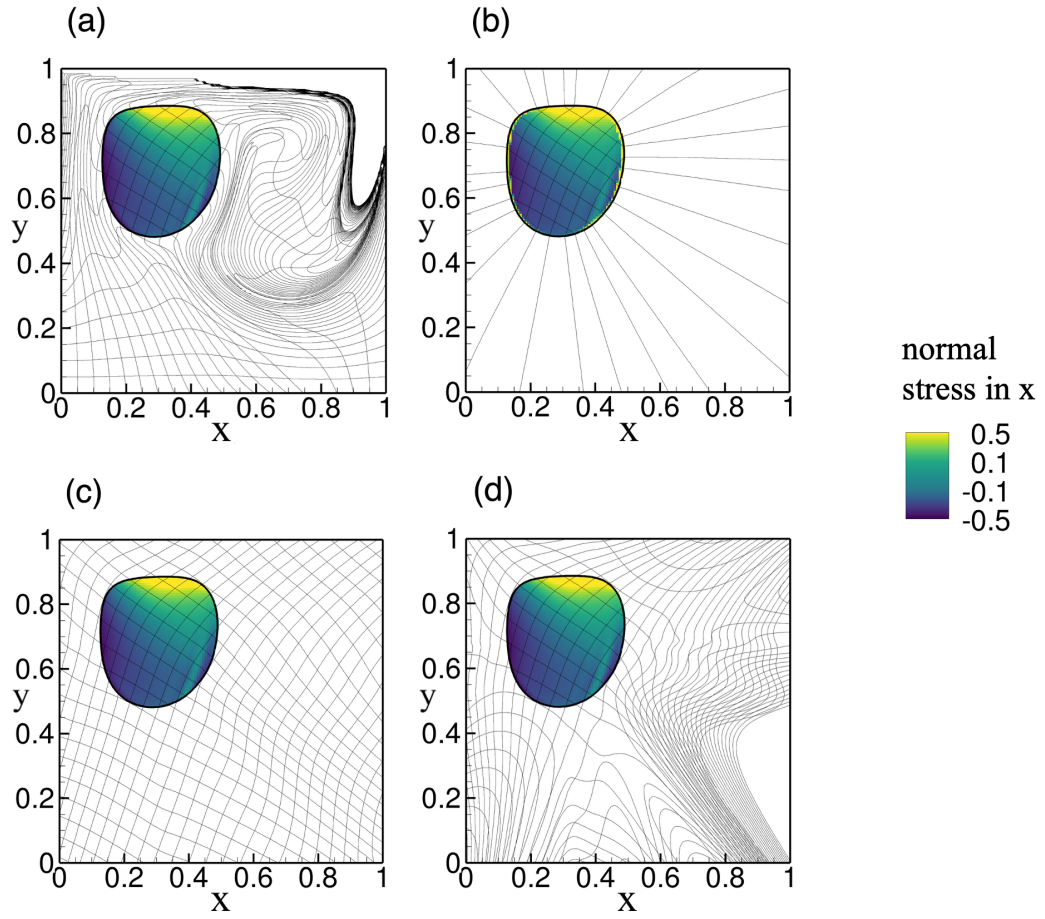


Figure 4.4: Dynamic grid for a solid disk in a lid-driven cavity simulation before(a), after constant extrapolation(b), after linear(c) and after quadratic extrapolation(d)[81]. The convoluted region outside solid is smoothed after extrapolation but constant extrapolation introduces unsmoothness of the derivative of the dynamic grid at the interface. Linear extrapolation is used as the default approach in this study.

method [81] to project the dynamic grid from inside the solid out into the fluid domain. We demonstrate this procedure for a solid disk in a lid-driven cavity problem. Fig. 4.4 shows the dynamic grid before and after the extrapolation. We test three different types of extrapolation schemes; constant, linear and quadratic [81], using the procedure described in Section 3.1. It can be observed that the dynamic grid inside the solid region is retained while the dynamic grid outside the solid is reconstructed based on the extrapolation of the dynamic grid inside the solid. It can also be observed that the method of constant extrapolation does not provide a smooth transition of the dynamic grid from the solid to the fluid domain while the quadratic extrapolation amplifies any local non-uniformity, leading to excessive distortions in the extrapolated field. We therefore use the linear extrapolation method as our default approach. Stability issues caused by the mesh distortion in the fluid domain reported by [15] are resolved with this approach.

4.3 Error Analysis

An error analysis is carried out by comparing the results obtained for different grid sizes with the reference results obtained on a finer mesh. In this study, the reference results for velocity, pressure, and stress were obtained on a mesh of size 1024×1024 . To gain a qualitative assessment of the rate of convergence, we define the following rms error estimate,

$$E_{\alpha}^2 = \frac{1}{N} \sum_{i=1}^N (\alpha_i - \tilde{\alpha}_i)^2 \quad (4.1)$$

where, α represents different flow variables such as velocity, pressure, and stress components obtained for mesh sizes smaller than the reference mesh size, and $\tilde{\alpha}$ denotes the

corresponding variable for the reference mesh. The subscript, i , denotes the index for each grid cell for a total of N cells. The error estimate, E_α^2 , is calculated for meshes for which the number of grid cells in the reference mesh is greater by a multiple of 4, in 2-D, than the number of grid cells in the coarser mesh. Consequently, the index, i , refers to identical spatial locations for both the coarse and the reference mesh.

In addition to the rms errors based on local values of velocity, pressure and stress, we further consider integrated errors associated with the overall kinetic energy of the fluid, E_{KE} , and the strain energy of the solid, E_{SE} , which are defined as,

$$E_{KE} = \left| KE/N - \widetilde{KE}/\widetilde{N} \right| \quad (4.2)$$

$$E_{SE} = \left| SE/N - \widetilde{SE}/\widetilde{N} \right| \quad (4.3)$$

where, $\sim$ indicates the quantities associated with the reference mesh. The kinetic energy of the fluid, KE , and the strain energy of the solid, SE , are defined according to

$$KE = \frac{1}{2} \int |\mathbf{u}|^2 dA \quad (4.4)$$

$$SE = \int \mu_s [tr(\mathbf{FF}^T) - 2] dA \quad (4.5)$$

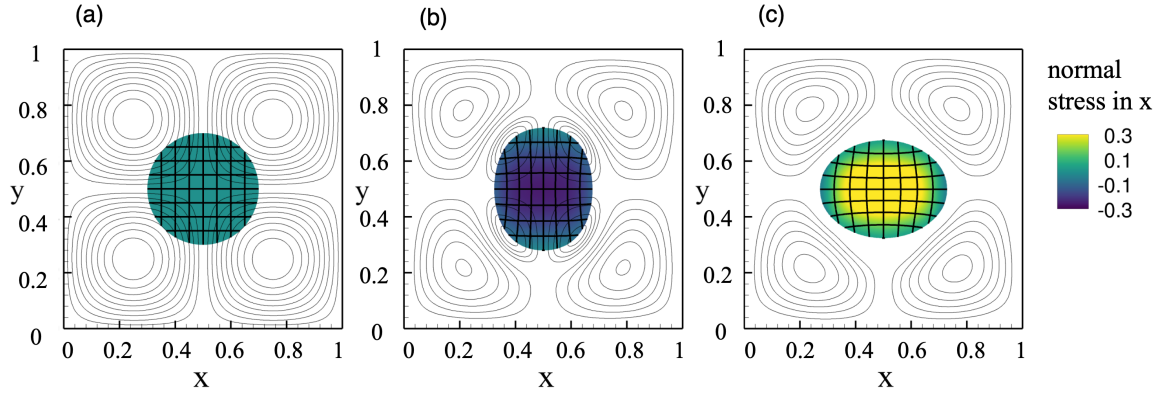


Figure 4.5: Deformation of an immersed oscillating disk and the flow streamlines at different times in a 1×1 domain with a 128×128 grid resolution. $Re = 1000$, $S = 1.0$. The fluid stretches the disk and the solid retracts due to elastic stresses, which sets up damped oscillations

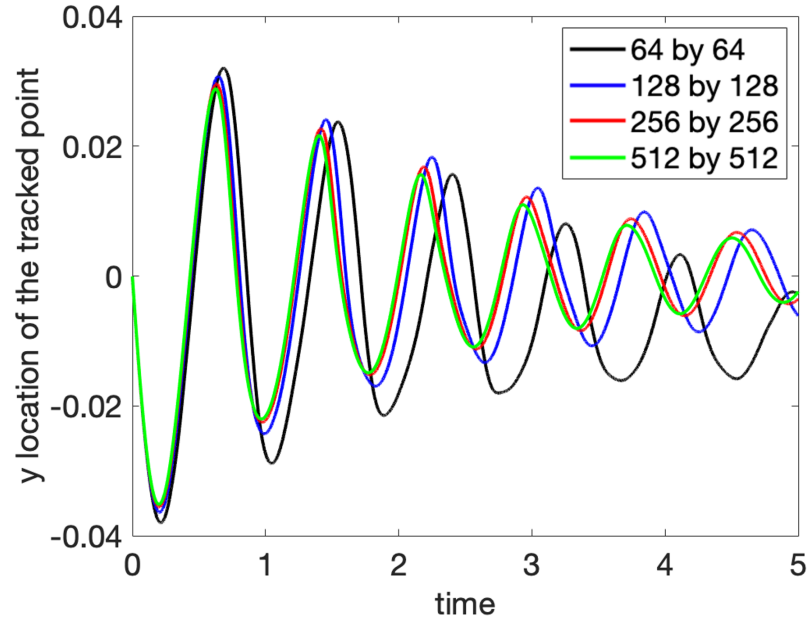


Figure 4.6: Change in y location of the tracked point on the boundary of the solid disk at $y = 0$ over time for different grid resolutions. The results show reasonable convergence with an increase in mesh size.

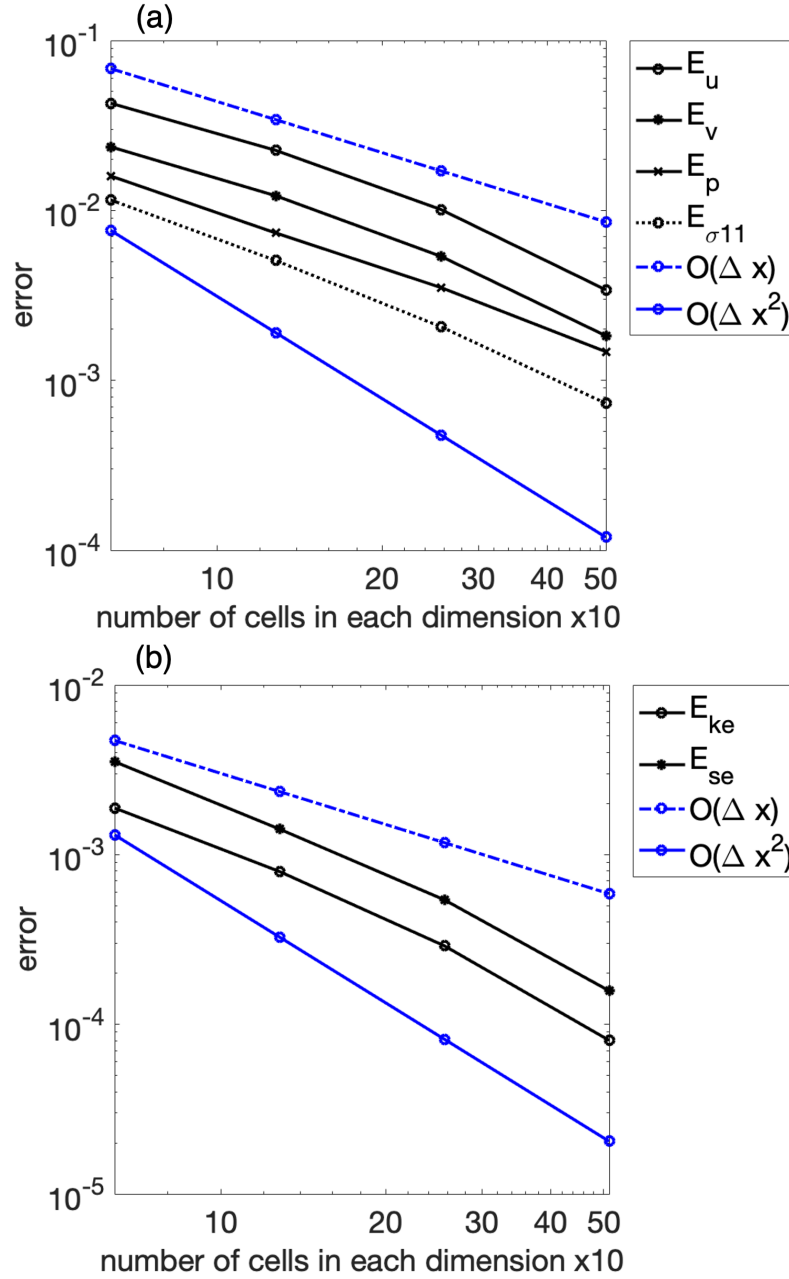


Figure 4.7: Convergence rate evaluated by the rms errors for u, v, p and σ_{11} (left), and integral errors for ke and se (right) using the equations described in section 4.3 at $t = 0.2$. The convergence rate approaches $O(\Delta x^2)$ with grid refinement from 64 to 512 in each direction.

4.4 Immersed oscillating disk

This test case is designed to assess the capability of the formulation to accurately capture the coupled dynamics between a deformable solid and the surrounding fluid. The imposed vortical velocity field generates significant deformation of the elastic disk, providing a stringent test of the method's ability to resolve solid stresses, interface motion, and fluid–structure coupling. The relatively high Reynolds number ($Re = 1000$) introduces stronger inertial effects in the fluid, which further challenges the stability and accuracy of the numerical scheme. In addition, performing a grid convergence study verifies the spatial accuracy of the formulation and demonstrates that the method achieves the expected close to second-order convergence rate. In this example, the disk is initially stress-free and has a circular shape with a radius of 0.2 placed in the center of a 1×1 domain with a 128×128 grid resolution. At $t = 0$, the system is subject to a velocity field that is defined by the stream function

$$\psi = \psi_0 \sin(k_x x) \sin(k_y y) \quad (4.6)$$

where $\psi_0 = 5 \times 10^{-2}$ and $k_x = k_y = 2\pi$. The Reynolds number of fluid is $Re = 1000$ and the stiffness parameter of the solid is $S = 1.0$. The system evolves like a damped oscillator. The shape-evolution and the streamlines at different times are presented in Fig. 4.5 corresponding to previously reported results [14]. The colored contours σ_{11} are solid normal stresses in the x direction.

A convergence study is performed for the case by determining the convergence rate using the error calculations from Eq.(4.1-4.5).

Fig. 4.6 shows the convergence of evolution of the tip position on the solid disk as the grid resolution increases. Fig. 4.7 shows that the convergence rate approaches $O(\Delta x^2)$ as

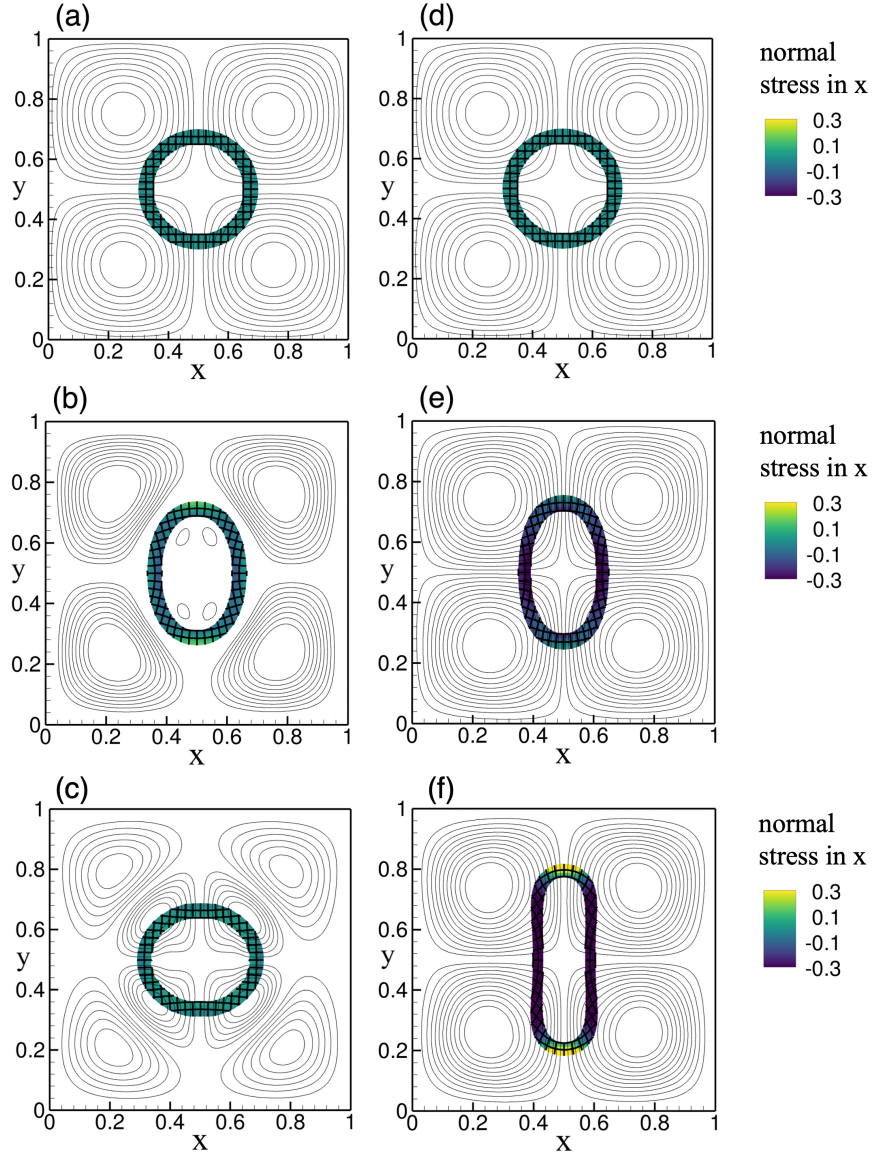


Figure 4.8: An oscillating ring with outer radius of 0.2 and a thickness of 0.05 surrounded by fluid with different solid shear modulus on a 128×128 grid. Similar to the disk in the previous simulations, in (a)-(c), the ring can restore its original shape and oscillates with solid stiffness parameter $S = 1.0$. As we increase the solid stiffness parameter to $S = 0.01$ in (d)-(f), the ring gets continuously stretched and cannot restore its original shape. Contours are colored by solid normal strain in x .

the grid resolution increases from 64 to 512 in each direction. This agrees with the results reported by Jain et al.[17] previously.

4.5 Immersed oscillating ring

This case extends the previous oscillating disk problem to a thin elastic ring in order to examine the robustness of the formulation when simulating slender deformable structures. Compared with a solid disk, the thin ring undergoes larger deformations and stronger local stretching, which challenges the numerical treatment of solid stresses and interface tracking. By varying the solid stiffness parameter S , this test evaluates the sensitivity of the method to changes in material elasticity and its ability to capture different deformation regimes ranging from elastic shape recovery to continuous stretching. In addition, this case demonstrates that the level set formulation can robustly handle severe deformation and even topological changes, such as rupture of the thin solid structure when the local thickness approaches the grid spacing. In this test problem, the disk in section 4.4 is replaced by an initially stress-free circular ring with outer radius of 0.2 and a thickness of 0.05. A grid resolution of 128×128 is used. The shape evolution of the solid ring with different solid stiffness parameter S at different time are shown in Fig. 4.8. The fluid has a Reynolds number, $Re = 1000$. For (a)-(c) in Fig. 4.8, the solid stiffness parameter, $S = 1.0$. For (d)-(f) in Fig. 4.8, the solid stiffness parameter, $S = 0.01$. The system is subject to a prescribed velocity field described by the same stream function in Eq. (4.6) with the velocity magnitude 20% of the velocity field in section 4.4. The solid ring can hold its shape when the shear modulus is large enough and can also continuously stretch and can even break when either the shear modulus is decreased or the velocity is increased. With the use of the level set method, the current algorithm is numerically able to capture the rupture of the solid in a stable manner when the thickness of the solid dips below the grid spacing. A physical model for sub-grid scale rupture can be added at this point to determine the actual point of rupture.

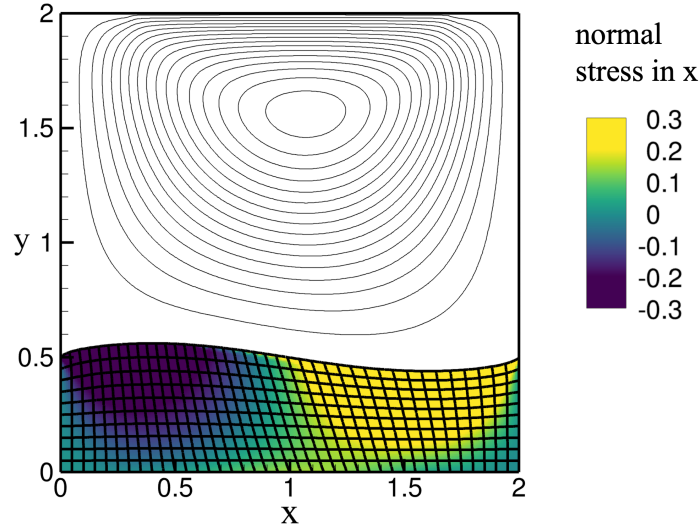


Figure 4.9: An elastic wall in a lid driven cavity at steady state with a 128×128 domain. $Re = 5$ and $S = 0.2$. Color contours represent normal solid stress in x direction. The yellow colored region shows the solid stress resulted from stretching of the solid wall in x direction while the purple colored region shows compression stress of the solid wall.

4.6 Elastic wall in a lid driven cavity

This case examines the ability of the formulation to accurately capture fluid–structure interaction driven by shear forces at a solid–fluid interface. The lid-driven cavity generates a well-known recirculating flow that produces sustained shear stress along the elastic substrate, leading to deformation of the solid wall. This configuration tests the capability of the numerical method to resolve the coupling between viscous fluid stresses and elastic solid response, particularly in a confined geometry with moving boundaries. In addition, the convergence study evaluates the spatial accuracy of the formulation for both fluid and solid variables, demonstrating that the method approaches the expected second-order accuracy as the grid is refined. In this example, an elastic Neo-Hookean solid substrate is placed in the bottom quarter of the domain. The top three quarters of the domain is filled with an incompressible fluid. The size of the domain is $\{0, 2\}$ in both the x and y directions. The

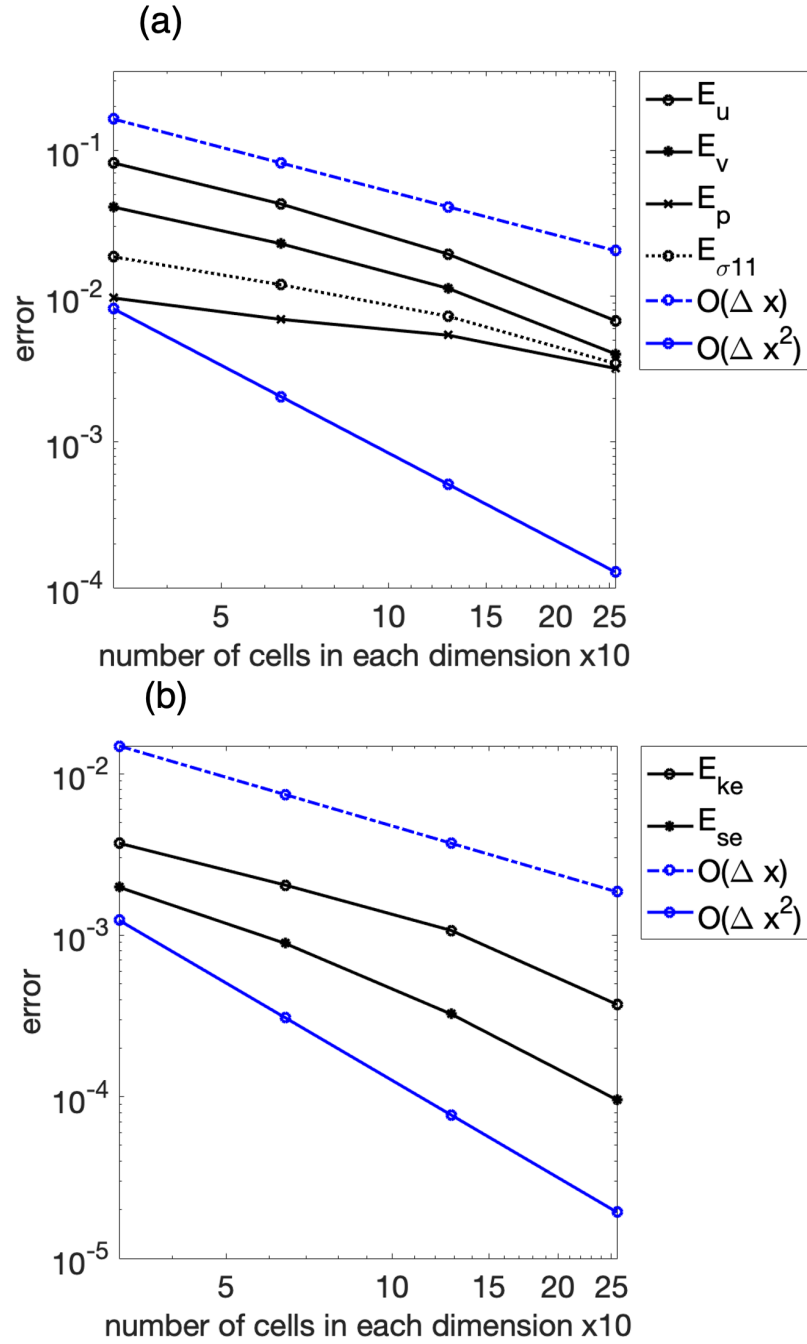


Figure 4.10: Convergence rate evaluated at steady state by the rms errors for u, v and σ_{11} (left), and the errors for ke and se (right) using the equations described in section 4.3 at steady state. The reduction in error approaches $O(\Delta x^2)$ on the finer mesh.

Reynolds number, $Re = 5$, and the solid stiffness parameter of the elastic wall, $S = 0.2$. Initially, the fluid is stationary and the solid is stress-free. At time $t > 0$, the top wall moves with horizontal velocity, u ,

$$u = 0.5 \begin{cases} \sin^2(\pi x/0.6), & x \in [0.0, 0.3] \\ 1, & x \in [0.3, 1.7] \\ \sin^2(\pi(x - 2.0)/0.6) & x \in [1.7, 2.0] \end{cases} \quad (4.7)$$

The solid substrate deforms as a result of the shear stress created by the flow along the substrate at the solid-fluid interface. Eventually, the system achieves a steady state of maximum deformation, indicating a balance between the elastic and shearing stresses at the solid-fluid interface. In this example, the domain is discretized into a 128×128 uniform Cartesian grid. In Fig. 4.9, the shape of the substrate and the streamlines of the flow field in steady state are plotted to provide a visual comparison with other simulation results [14]. The colored contours σ_{11} indicate the normal stress in the solid along the x -direction. Similar to results in the convergence study in section 4.4, Fig. 4.10 shows the convergence rate calculated from Eq.(4.1-4.5) start to be around $O(\Delta x)$ but approaches $O(\Delta x^2)$ as the grid resolution increases. Convergence study for the same simulation was previously reported [14]. Although in that study, a Lagrangian mesh density of 64×64 was used for convergence study of the Cartesian mesh while in the present study, a uniform resolution was used for both solid and fluid, the convergence rate for the present study agrees with the previous result reasonable especially when solid resolution becomes closer to the one in the previous study [14].

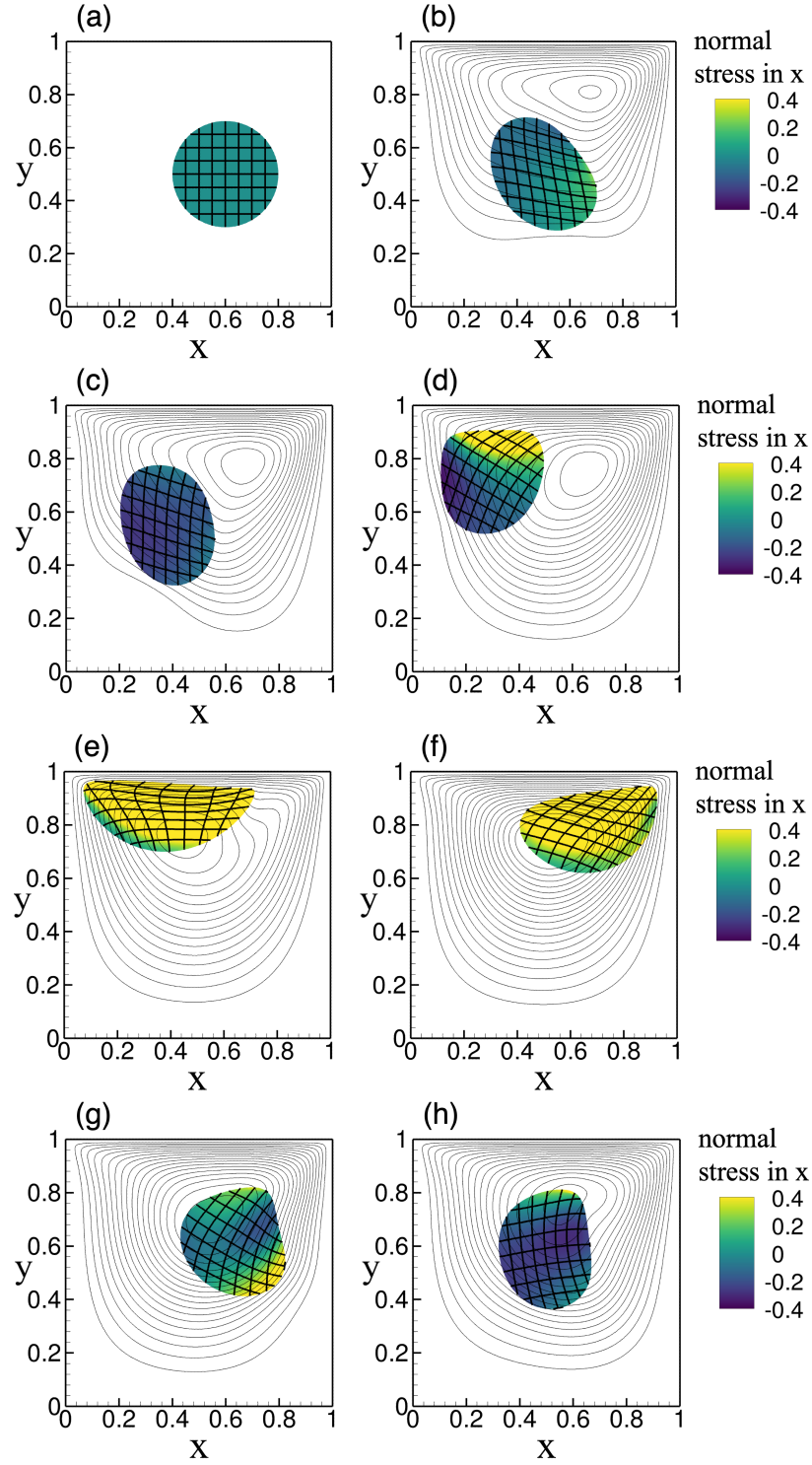


Figure 4.11: Evolution of the streamlines and the shape of a deformable disk in a lid driven cavity at different time with a 128×128 grid. $Re = 100$ and $S = 1.0$. Normal solid stress in x direction from Stretching and compression of the disk is visualized by the color contours.

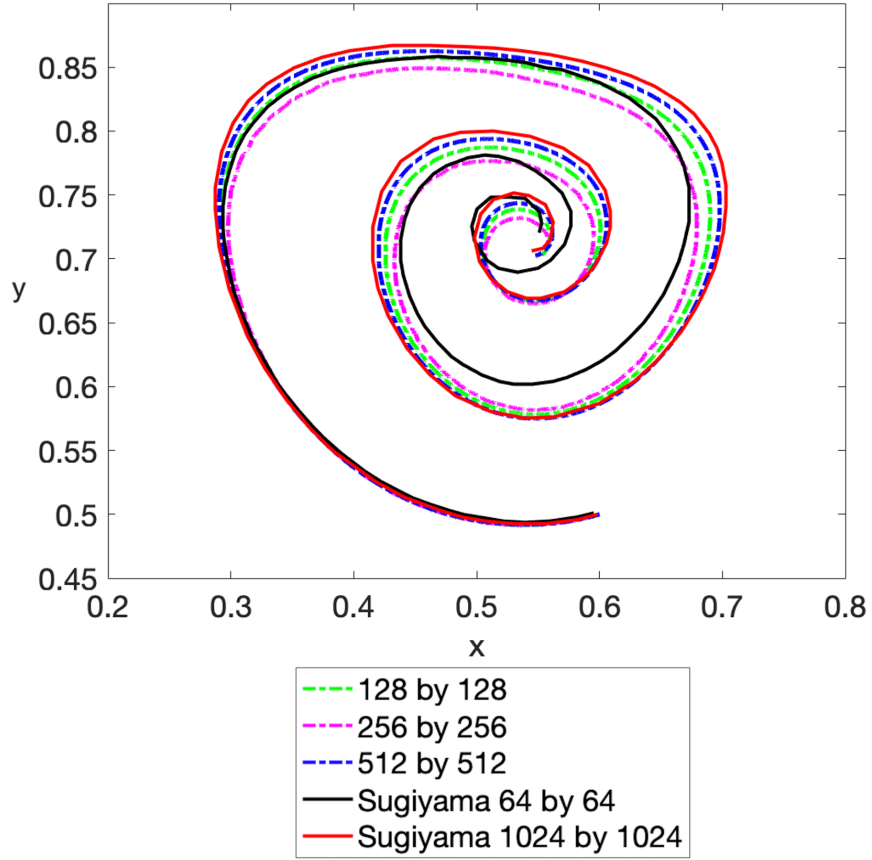


Figure 4.12: The dashed lines show the evolution for the centroid of the disk at 3 different grid resolutions (128×128 , 256×256 and 512×512) in the present study. The solid lines show the results for the centroid of the disk at 128×128 and 1024×1024 resolutions reported by Sugiyama et al.[74]. The present results approach the results of 1024×1024 resolution in Sugiyama et al. as the resolution increases.

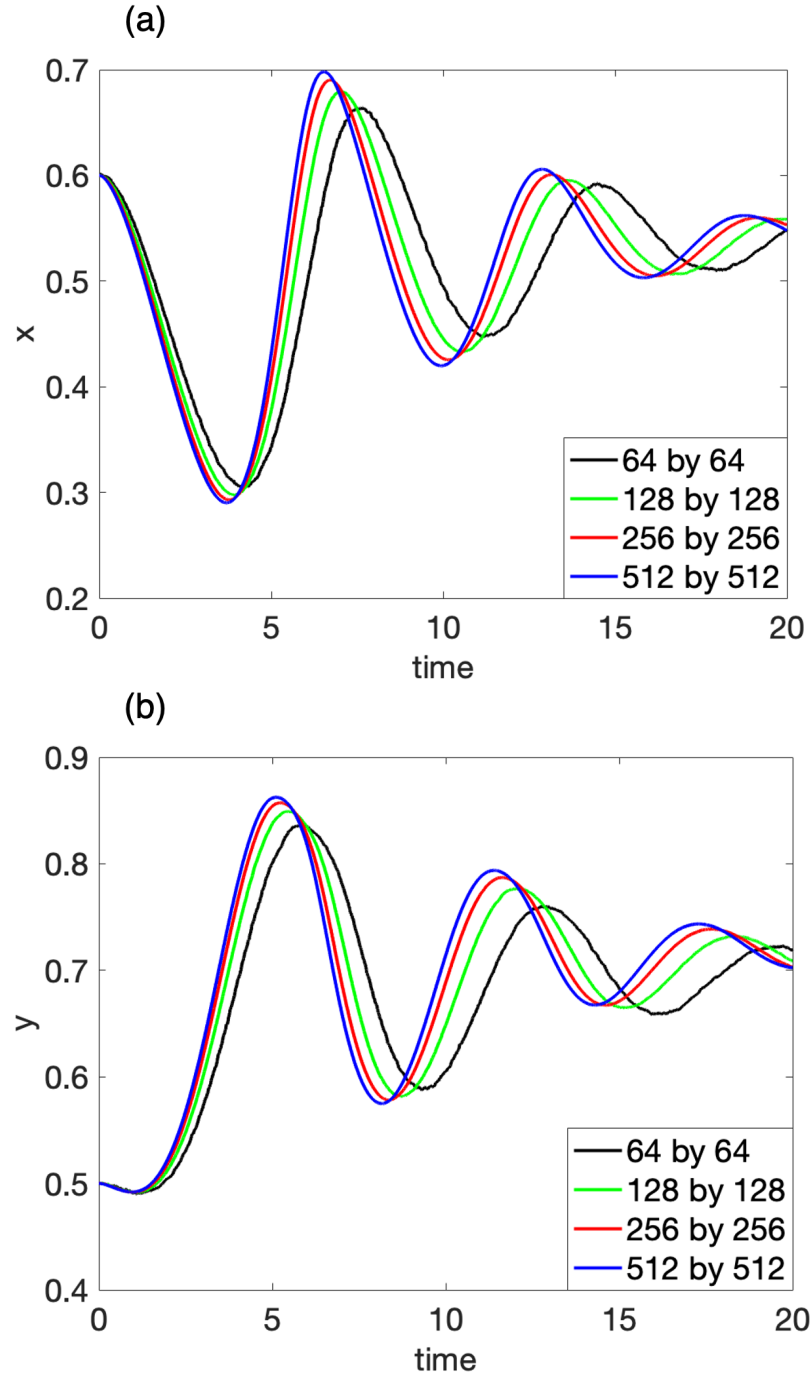


Figure 4.13: x locations (in Figure (a))and y locations (in Figure (b))for the centroid of the solid disk in a lid-driven cavity for different grid resolutions.

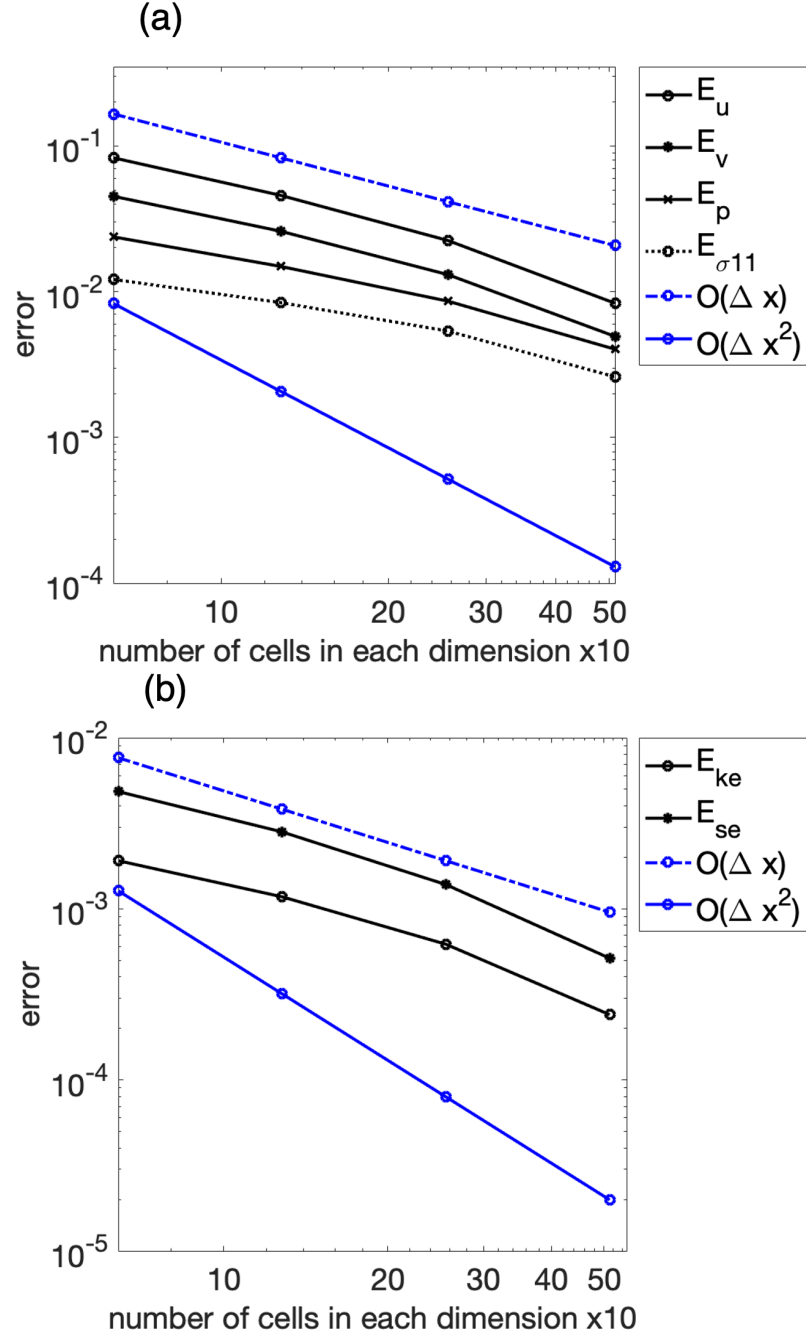


Figure 4.14: Convergence rate evaluated by the rms errors for u, v, p and σ_{11} (left), and the errors for ke and se (right) using the equations described in section 4.3 at $t = 2$. The reduction in error approaches $O(\Delta x^2)$ on the finer mesh.

4.7 Deformable disk in a lid-driven cavity

This case investigates the ability of the formulation to simulate fully coupled fluid–structure interaction involving both translation and deformation of an immersed elastic body. In contrast to the previous elastic-wall problem, the solid disk is free to move within the flow, making this a more challenging test of the method’s ability to capture momentum exchange between the fluid and the deformable structure. The lid-driven cavity generates complex recirculating flow that produces both normal and shear stresses on the disk, leading to coupled deformation and rigid-body motion. Comparison of the disk centroid trajectory and deformation with previously reported results validates the accuracy of the formulation in predicting the motion of deformable bodies in confined flows, while the grid refinement study confirms that the method approaches second-order spatial accuracy. In this example, an elastic disk is translated and deformed by fluid motion inside a lid-driven cavity, as shown in Fig. 4.11. The size of the computational domain is $\{0, 1\}$, both in the x and y directions. The domain is discretized using 128×128 grid cells. At time $t = 0$, the circular disk of radius $r = 0.2$ is centered at $(x, y) = (0.6, 0.5)$ with an initial radius of $r = 0.2$. At $t > 0$, the top wall moves with constant velocity, $u = 1$. The fluid motion exerts both shearing and normal stresses on the elastic disk, causing it to translate and deform. The Reynolds number for fluid, $Re = 100$, and the stiffness parameter for solid, $S = 10$. The shape evolution of the solid disk and the streamlines of the flow field are plotted in Fig. 4.11 at different times corresponding to the previous simulation results [14]. The colored contours σ_{11} are solid normal stresses along the x direction.

Fig. 4.12 shows the centroid of the disk at different resolution in comparison with the results in the previous study. As the resolution for the present method increases, the centroid approaches the results of Sugiyama et al[74]. Fig. 4.13 shows the evolution of the x and y coordinates for the centroid of the disk for the lid-driven cavity problem for different

grid sizes. The centroid coordinates can be observed to converge. Fig. 4.14 shows the convergence rate for the error estimates defined using the equations described in section 4.3. The convergence rate is around $O(\Delta x)$ for the coarse mesh but improves with an increase in the number of grid cells, approaching second-order accuracy. These results compare reasonably well with those previously reported in the literature.

4.8 Deformable solid ring in a lid-driven cavity

This case further examines the capability of the formulation to simulate deformable bodies with more complex geometries and varying material stiffness. Compared with the solid disk in the previous example, the hollow ring introduces additional structural flexibility and localized deformation, which provides a more stringent test of the numerical treatment of solid stresses and interface coupling. By varying the stiffness parameter S , this case evaluates the sensitivity of the method to different elastic responses and demonstrates that the solver can accurately capture deformation across a range of material stiffnesses. The results highlight the robustness of the formulation in handling deformable structures with internal cavities undergoing large deformation in confined flows. Similar to the previous example of the deformable disk in a lid-driven cavity, the domain in this case extends from 0 to 1 in both x and y directions and is discretized using 128×128 grid cells. The ring is initially centered at $(0.6, 0.5)$ in a stress-free state. It has a thickness of 0.1 and an outer radius of 0.2. The fluid Reynolds number, $Re = 100$, and we consider three different values for the solid stiffness parameter, $S = 0.1, 1.0$ and 10 . For $t > 0$, the top wall moves with a velocity, $u = 1$. The evolution of the ring is shown in Fig. 4.15. These results show that as the solid stiffness parameter S decreases, the solid ring becomes more resistant to deformation. These results demonstrate the ability of the solver to be applicable to stiffer solids.

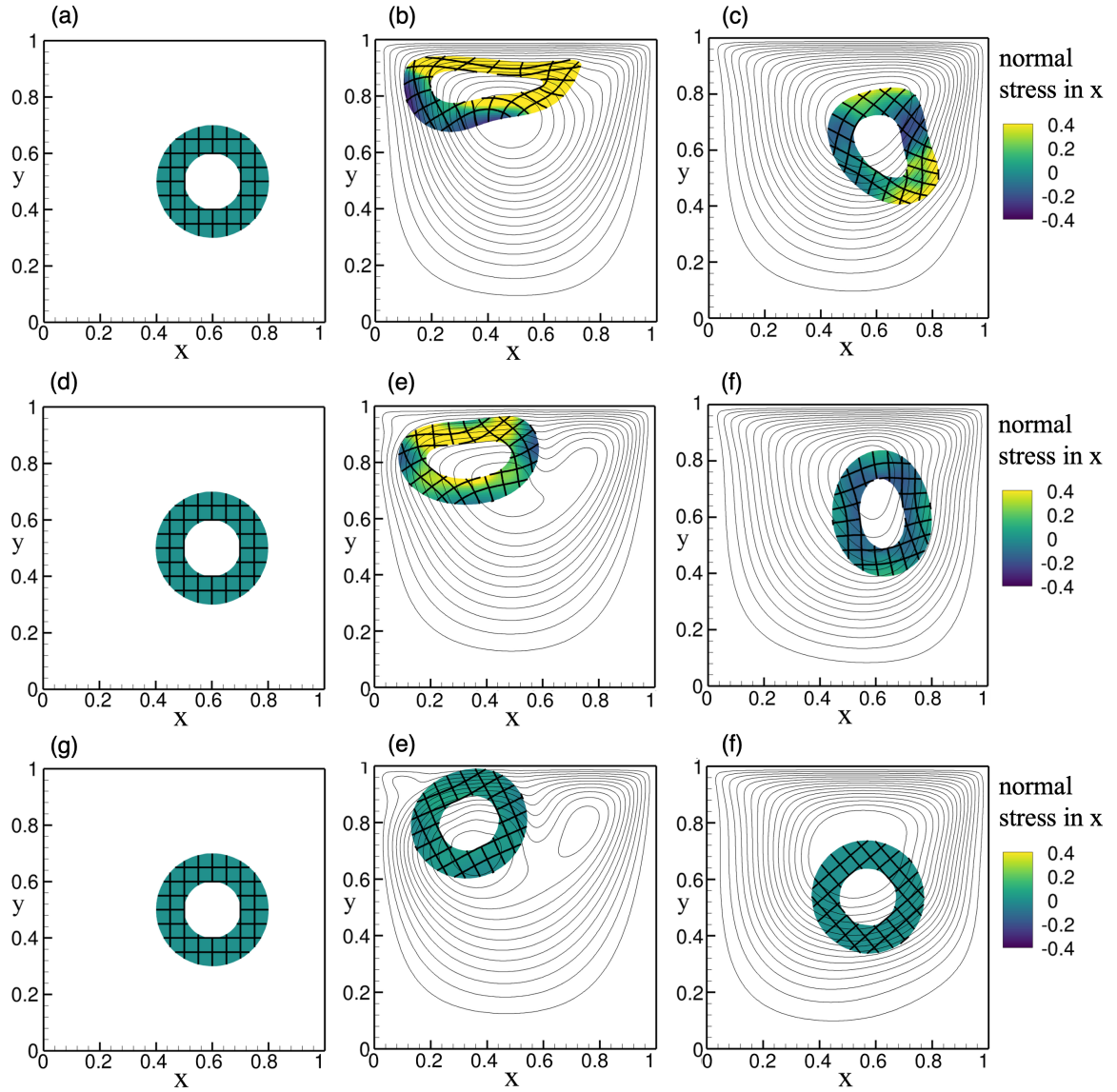


Figure 4.15: A deformable solid ring in a lid-driven cavity with different solid stiffness parameter S at different time with a 128×128 grid. Figure (a), (b) and (c) are with $S = 0.1$. Figure (d), (e) and (f) are with $S = 1.0$. Figure (g), (h) and (i) are with $S = 10$.

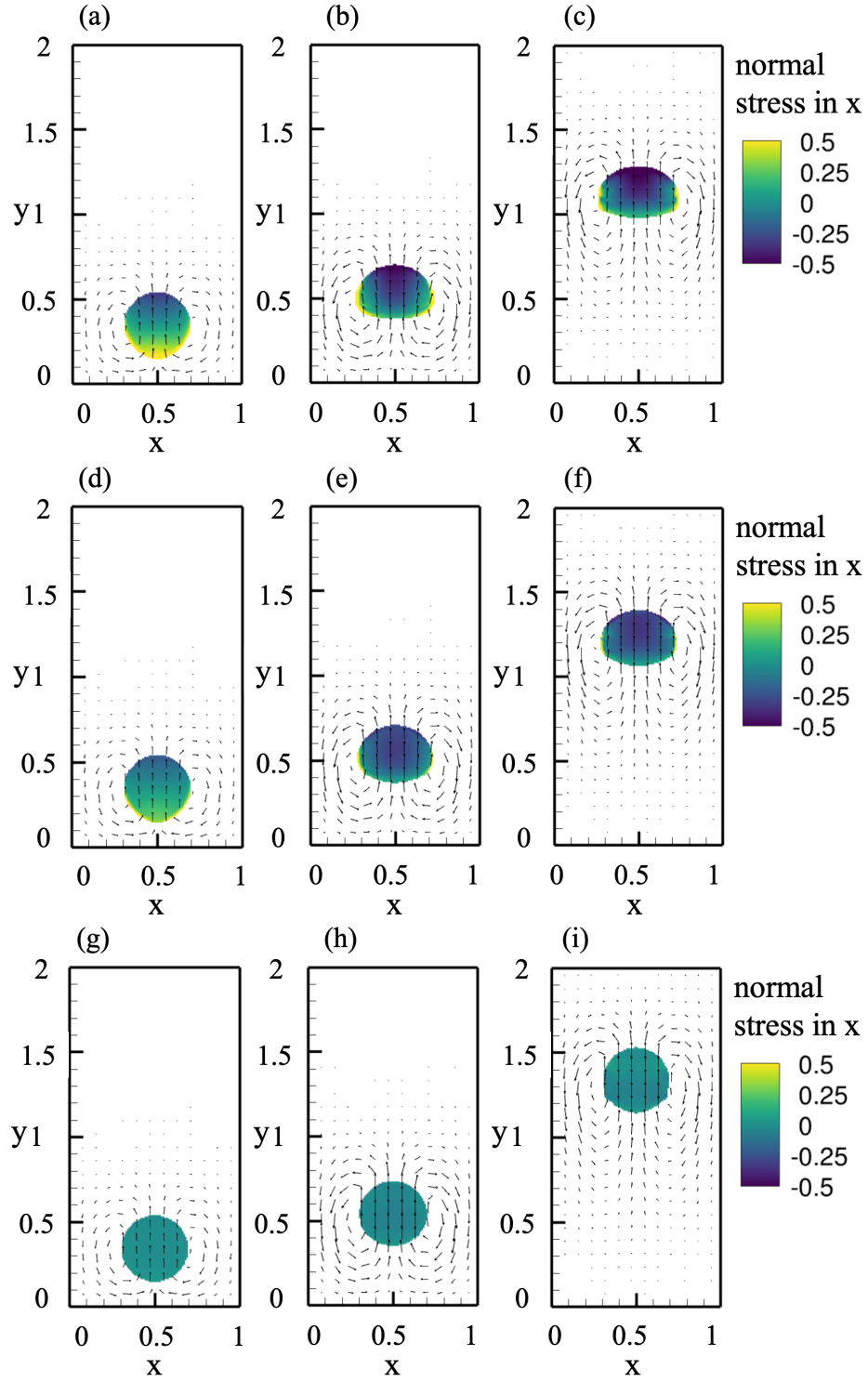


Figure 4.16: A light solid disk with density ratio $\frac{\rho_s}{\rho_f} = 0.1$ and different shear modulus rising in a channel at different time with a grid resolution of 128×256 grid. Figure (a)–(c) are with $S = 0.5$. Figure (d)–(f) are with $S = 1.0$. Figure (g)–(i) are with $S = 10$.

4.9 Rising solid disk in a vertical channel

This case evaluates the ability of the formulation to simulate buoyancy-driven motion of a deformable solid in a vertical channel. The density ratio $\rho_s/\rho_f = 0.1$ introduces strong coupling between fluid inertia and solid motion, making this configuration particularly challenging due to pronounced added-mass effects. Compared with previous studies, which typically considered more moderate density ratios, the present formulation remains stable and accurate for significantly lighter solids. By varying the stiffness parameter S , the test also demonstrates the capability of the method to capture the interaction between buoyancy forces, elastic deformation, and fluid-induced oscillations. These results highlight the robustness of the solver for problems involving large density contrasts and strongly coupled fluid–structure dynamics. The domain in this case extends from 0 to 1 in x and 0 to 2 in y direction and is discretized using 128×256 grid cells. The disk is initially centered at $(0.5, 0.3)$ in a stress-free state with a radius of 0.2. The Reynolds number, $Re = 100$, Froude number $Fr = 1$ and we consider three different values of the solid stiffness parameter, $S = 0.5, 1.0$ and 10 . The density ratio is kept as $\frac{\rho_s}{\rho_f} = 0.1$ with the solid disk lighter than the surrounding fluid. At $t = 0$, the disk will start to rise. Fig. 4.16 shows the results for solid stress and velocity vectors at five different time for the disks with different shear modulus. It can be observed that the rising speed increases with decrease of stiffness parameter S as a result of less kinetic energy converted to the potential and kinetic energy of the solid oscillations.

4.10 Deformation of a Finite-Thickness Solid Layer Subjected to Inlet Shearing

This case evaluates the capability of the formulation to simulate the deformation of a finite-thickness solid layer subjected to controlled hydrodynamic shearing. By prescribing a fully developed velocity profile on both sides of the solid film, the test isolates the structural response of the solid to fluid-induced shear stresses while minimizing inertial mismatch effects. Varying the stiffness parameter S allows the method to capture different deformation regimes ranging from stable elastic bending to extreme stretching and rupture for highly compliant materials. In addition, this configuration provides a useful framework to examine the influence of numerical grid resolution on the effective thickness of the Eulerian representation of the solid layer and motivates the proposed scaling of the shear modulus to maintain consistent macroscopic stiffness across different grid resolutions.

As illustrated in Fig. 4.17, a fully developed parabolic velocity profile is prescribed on both sides of a centrally located solid layer to evaluate controlled hydrodynamic shearing in the y -direction. The maximum fluid inlet velocity is set to 1. To isolate the structural response from inertial mismatch effects, the fluid-to-solid density ratio is held at 1, and the dynamic viscosities are matched. The computational domain spans 3×4 spatial units and is uniformly discretized into a 300×400 Eulerian grid. The fluid operates at a moderate Reynolds number of $Re = 50$, ensuring a steady, laminar shear regime. To determine the influence of the shear modulus on the structural behavior, solid stiffness parameters of $S = 20$ (rigid) and $S = 0.2$ (highly compliant) are evaluated.

The resulting elastodynamic deformations are depicted in Fig. 4.18. When the solid shear modulus is exceptionally low ($S = 0.2$), the layer lacks the requisite internal elastic restoring forces to counterbalance the external fluid traction. Consequently, it cannot sus-

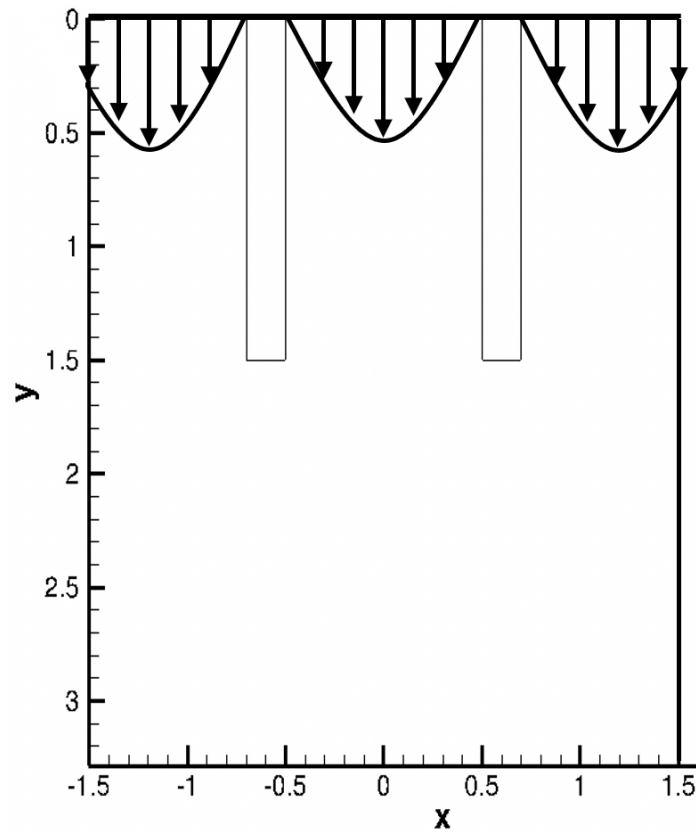


Figure 4.17: Boundary conditions for the simulation of a centrally located solid film deforming under a fully developed velocity profile applied simultaneously to both sides.

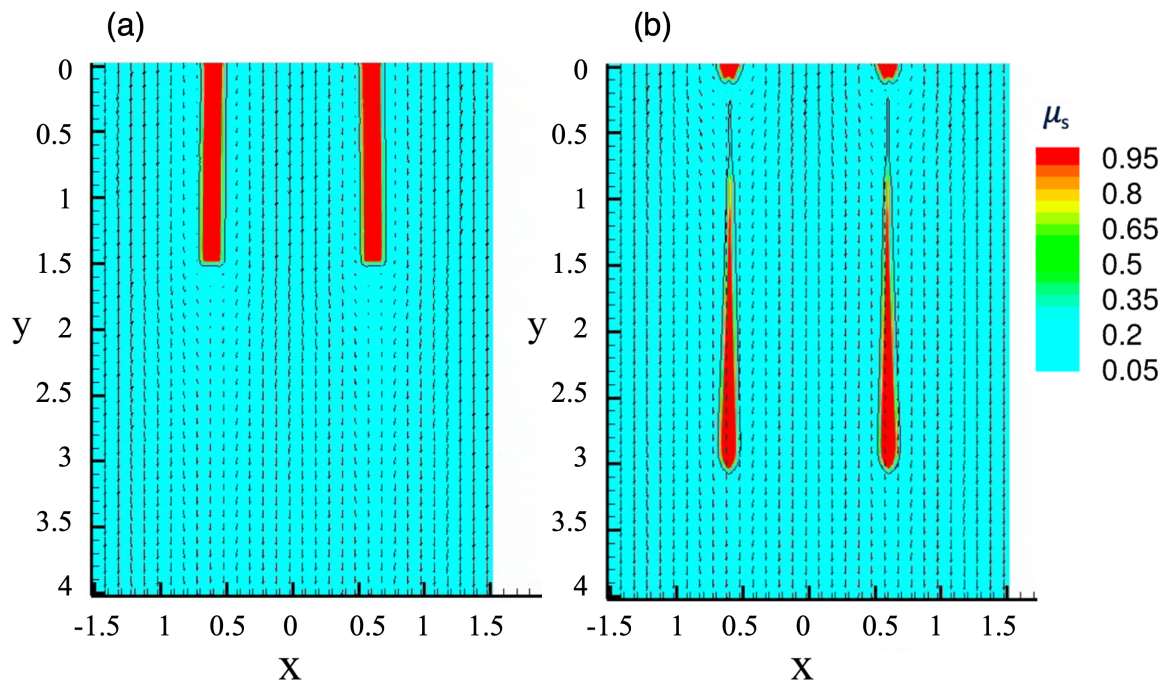


Figure 4.18: Structural response of the solid layer subjected to steady flow, comparing a highly rigid solid with a stiffness parameter of $S = 20$ (a) against a highly compliant solid with $S = 0.2$ (b) on a 300×400 grid.

tain the imposed shear stress, leading to extreme elongation and ultimate structural rupture. Conversely, with a substantially higher shear modulus ($S = 20$), the solid successfully withstands the hydrodynamic loading, bending gracefully until the fluid traction equilibrates with the internal elastic stress, allowing the layer to maintain a steady, finite deformation.

Building upon the steady-state observations, the structural response of the solid layer is subsequently investigated under an oscillatory fluid driving force. As shown in Fig. 4.19, a fully developed, time-varying velocity profile is prescribed in the channel between the solid films, with the maximum inlet velocity fluctuating periodically to induce alternating shear cycles. For this dynamic simulation, the fluid viscosity is set to a Reynolds number of $Re = 100$ and the solid shear modulus is fixed at $S = 10$.

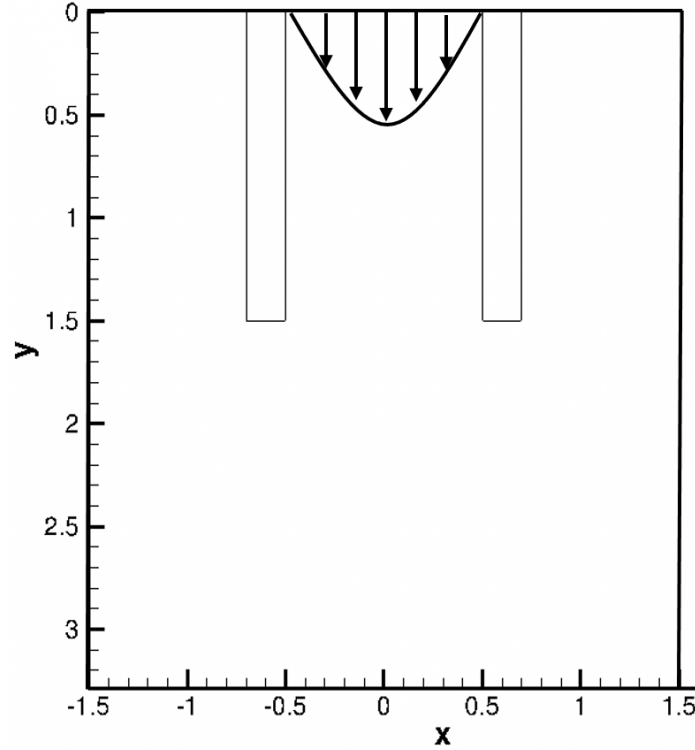


Figure 4.19: Boundary conditions for the dynamic simulation of a solid film deforming under a fully developed, periodically oscillating velocity profile prescribed between the solid boundaries.

Beyond analyzing the periodic deformation dynamics, we critically evaluate the effects

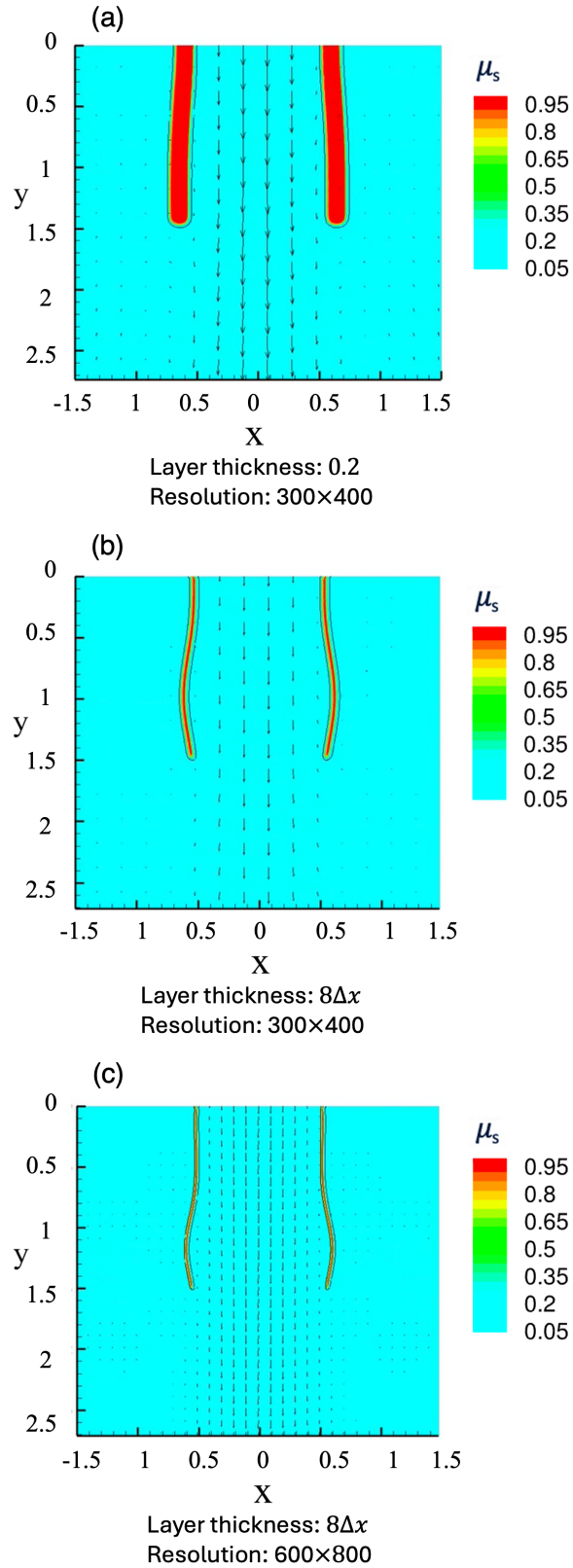


Figure 4.20: Effects of physical layer thickness and underlying grid resolution on the deformation of the solid layer .

of grid resolution on the finite-thickness representation of the solid layer. Because the numerical formulation relies on a fixed number of grid cells to span the solid film region, altering the global grid resolution inherently changes the physical thickness of the simulated structure, as visualized in Fig. 4.20.

Because this geometric thickness artificially alters the bending stiffness and apparent structural rigidity of the layer, we propose mathematically scaling the shear modulus as a function of the local layer thickness. This calibration ensures that the modeled solid layer exhibits a consistent macroscopic stiffness, rendering the fluid-structure interaction behavior practically independent of the underlying grid resolution. A preliminary investigation into this thickness-induced scaling is conducted by dynamically tracking a material midpoint at the bottom boundary of the solid layer (indicated in Fig. 4.20).

4.11 Deformation of a pre-deformed solid film

This case examines the capability of the formulation to simulate the relaxation dynamics of a pre-deformed elastic structure interacting with multiple surrounding fluids. The deformation is achieved by first using a uniform velocity inflow $u = 1$ to deform the film for a time $t = 0.56$ to a deformed state, where the tracked tip position deviate for a distance of 0.1 from the stress-free state. This deformed shape was used as an initial condition to start the simulation with 0 velocity everywhere inside the domain. The boundaries on the top and bottom are open, allowing the flow to get in and out of the domain. By initializing the solid film with stored elastic energy and allowing it to evolve freely without external forcing, the test isolates the interplay between elastic restoring forces and viscous fluid stresses. Varying the density ratio (DR) and viscosity ratio (VR) between the two fluid layers provides a systematic way to evaluate how inertial and dissipative effects influence the oscillatory relaxation of the structure. The results demonstrate that the nu-

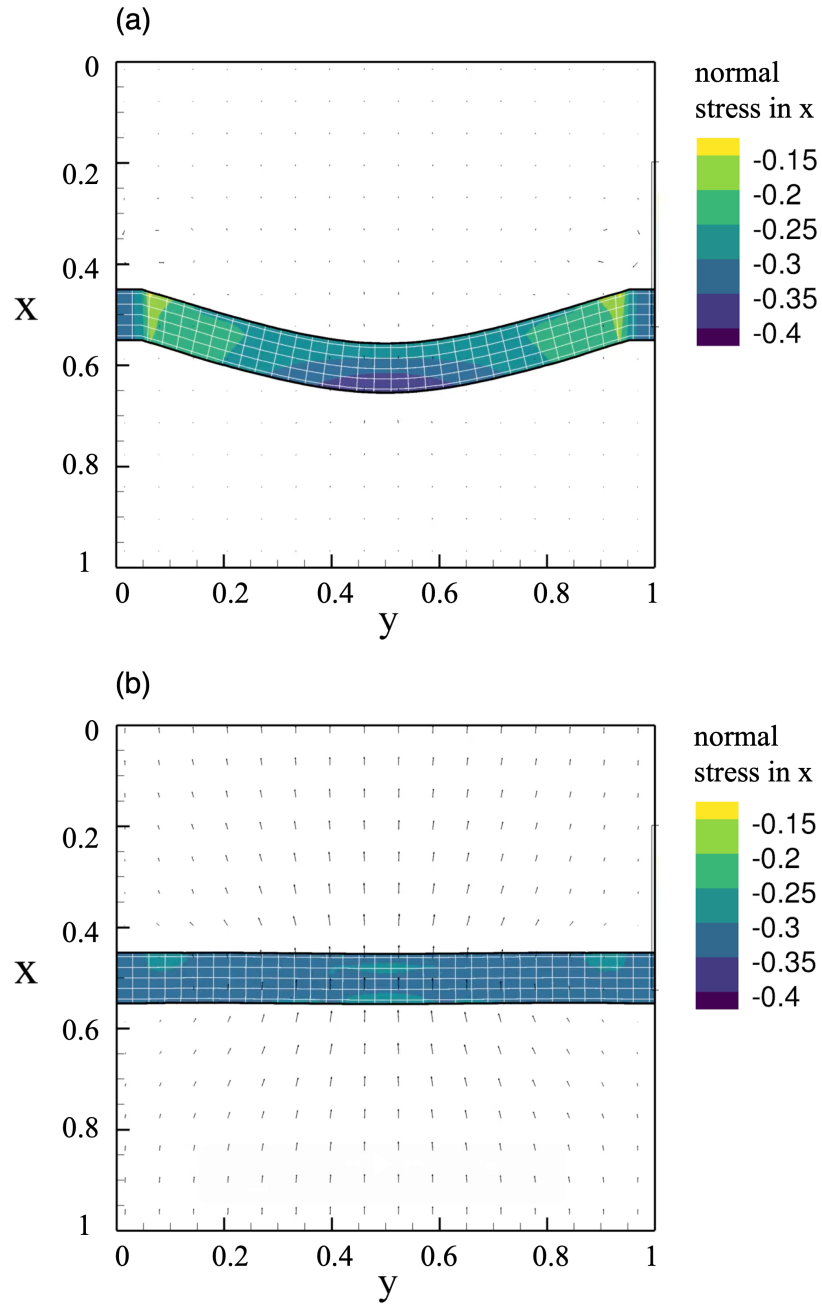


Figure 4.21: Oscillation of a pre-deformed solid film with a 128×128 grid. $Re = 100$, $S = 0.1$ (part 1). The white line shows the instantaneous shape of the dynamic grid. Stress contour and velocity vectors are also plotted.

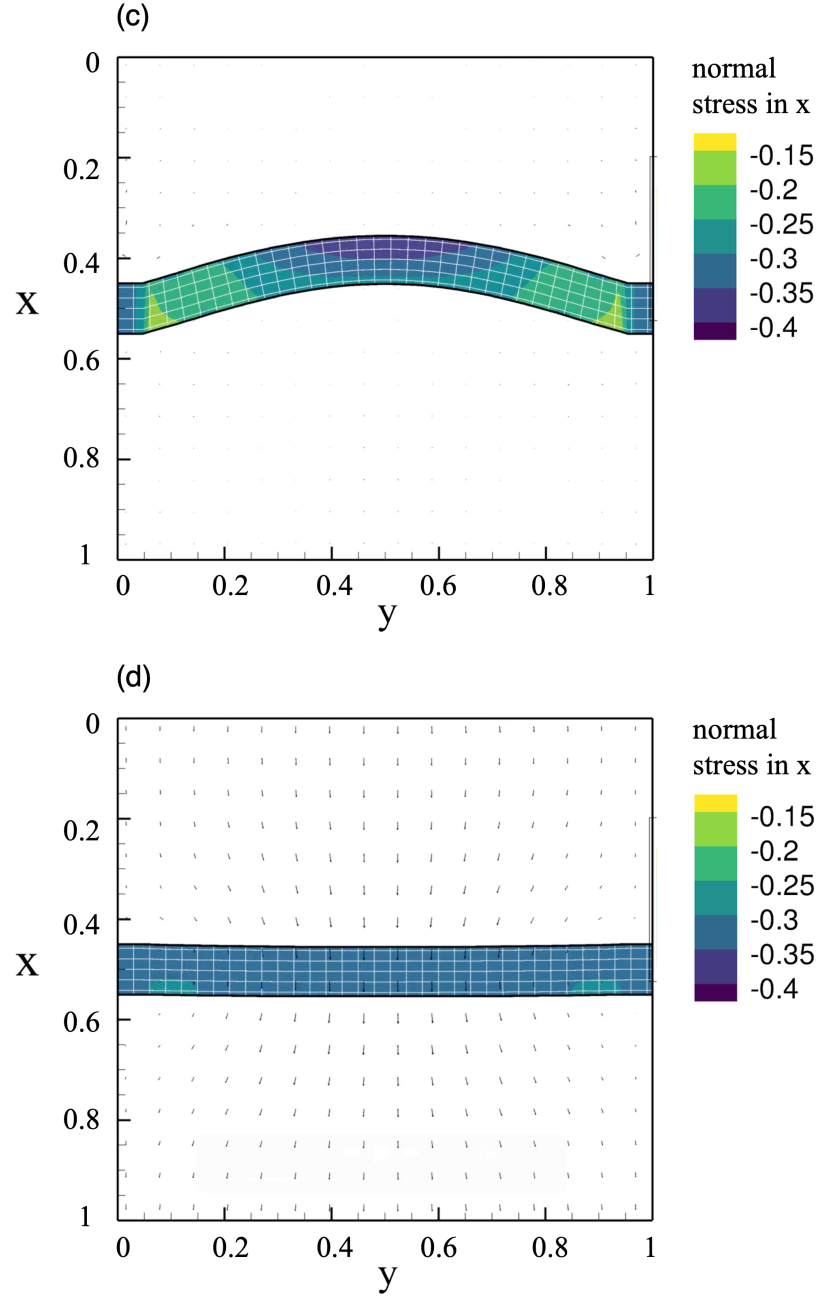


Figure 4.22: Oscillation of a pre-deformed solid film with a 128×128 grid. $Re = 100$, $S = 0.1$ (continued). The white line shows the instantaneous shape of the dynamic grid. Stress contour and velocity vectors are also plotted.

merical method can accurately capture the coupled dynamics of multiphase fluid–structure interaction, including inertia-controlled oscillation timescales and viscosity-driven damping behavior. The computational domain is a two-dimensional rectangular region defined over $0 \leq x \leq 1$ and $0 \leq y \leq 1$ with a 128×128 grid. The solid film is initially located near the center of the domain and separates the upper and lower fluids, denoted as Fluid 1 (initially $x < 0.45$) and Fluid 2 (initially $x > 0.55$), respectively. The film is pinned at both the left and right boundaries, enforcing fixed end conditions that prevent rigid body motion. Consequently, all observed deformation arises from internal elastic forces and hydrodynamic interactions.

At $t = 0$, the solid is prescribed with a pre-deformed configuration, introducing stored elastic energy into the system. Upon release, the film evolves freely under the combined effects of elastic restoring forces and viscous stresses exerted by the surrounding fluids. No external forcing is applied, and the system undergoes a natural relaxation process.

The density of the solid is set equal to that of Fluid 2 to eliminate buoyancy effects and isolate the role of viscous and inertial interactions. The flow is characterized by a Reynolds number $Re = 100$ and a non-dimensional stiffness parameter $S = 0.1$, indicating a moderately compliant structure. To investigate the effect of fluid properties, simulations are performed for different density ratios (DR) and viscosity ratios (VR), defined as the ratios of Fluid 2 to Fluid 1.

Fig. 4.21 and Fig. 4.22 present the spatial distribution of the solid deformation, along with a contour plot of a representative stress component within the film. The deformation remains smooth and continuous, with the largest deflection occurring near the center of the domain, consistent with the fixed boundary conditions at both ends. The stress distribution indicates higher magnitudes near regions of maximum curvature, reflecting the concentration of elastic stresses associated with bending. The temporal evolution of the film deformation is quantified by tracking the vertical displacement of the midpoint of the

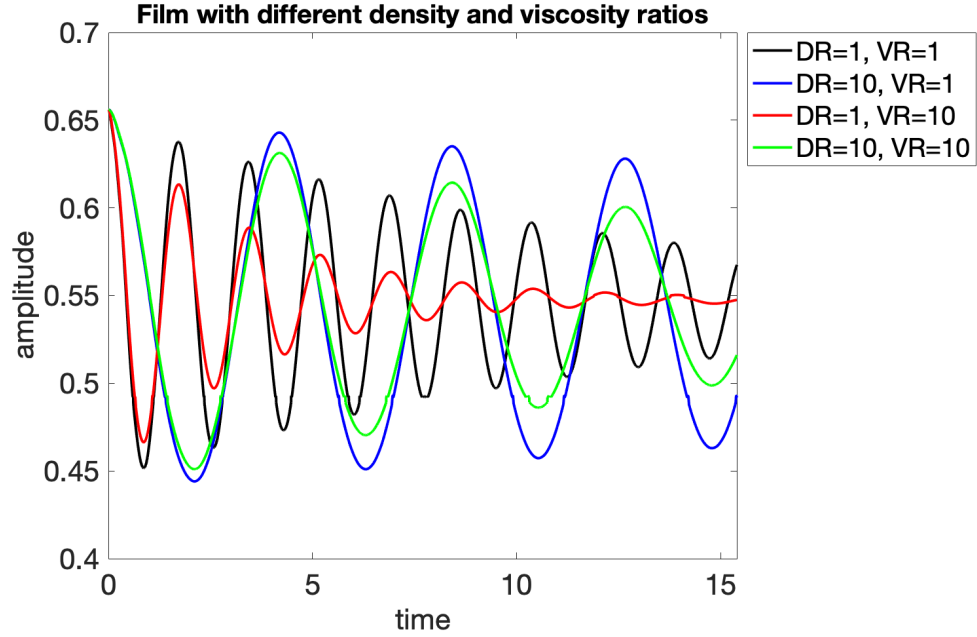


Figure 4.23: Amplitude of the midpoint over time for film with different DR and VR

film. The resulting amplitude histories for different combinations of density and viscosity ratios are shown in Figures (a)–(d). In all cases, the system exhibits an oscillatory relaxation behavior, where the initial deformation leads to a series of damped oscillations as the film approaches its equilibrium configuration.

To systematically investigate the influence of fluid properties on the deformation dynamics of the pre-deformed solid film, a series of simulations are performed with varying DR and VR. The evolution of the film is quantified by tracking the vertical displacement of its midpoint, providing a measure of oscillation amplitude and damping behavior.

The tip position on the film was tracked and the results in Fig. 4.23 reveal that the deformation dynamics are strongly governed by the interplay between inertial and viscous effects in the surrounding fluids. For the baseline case with $DR = 1$ and $VR = 1$, the system exhibits a symmetric and moderately damped oscillatory response. The initial elastic energy stored in the pre-deformed configuration is gradually dissipated through viscous effects, resulting in a smooth decay of oscillation amplitude over time.

When the density ratio is increased while keeping the viscosity ratio fixed, a clear reduction in oscillation frequency is observed. This behavior is attributed to the increased effective inertia of the fluid–structure system, which slows down the dynamic response of the film. For example, cases with $DR = 10$ exhibit longer oscillation periods compared to the baseline case. The amplitude of oscillation may also be slightly enhanced during the initial cycles due to the greater resistance of the heavier fluid to rapid acceleration.

In contrast, variations in the viscosity ratio primarily affect the rate of energy dissipation. Increasing the viscosity ratio leads to stronger damping, as viscous stresses more effectively dissipate the kinetic energy generated during oscillation. As a result, cases with $VR = 10$ show a rapid decay in amplitude and a faster transition toward equilibrium. Conversely, lower viscosity ratios allow oscillations to persist for a longer duration, with slower attenuation of amplitude.

The combined effects of density and viscosity ratios further illustrate the competing roles of inertia and dissipation. For cases with both high density and high viscosity ratios $DR = 10, VR = 10$, the system exhibits low-frequency, heavily damped oscillations. The increased inertia prolongs the oscillation period, while the elevated viscosity accelerates energy dissipation, resulting in fewer observable oscillation cycles before the system reaches equilibrium.

Additionally, subtle asymmetries in the deformation profiles can arise when density and viscosity contrasts are large, reflecting differences in momentum transfer across the fluid–solid interfaces. These effects highlight the sensitivity of the coupled system to property mismatches between the two fluid layers.

Overall, the parametric study demonstrates that density ratio primarily controls the timescale of oscillation (inertial effect), while viscosity ratio governs the rate of energy dissipation (damping effect). The numerical results consistently capture these trends, confirming the capability of the method to resolve the complex interplay between fluid inertia,

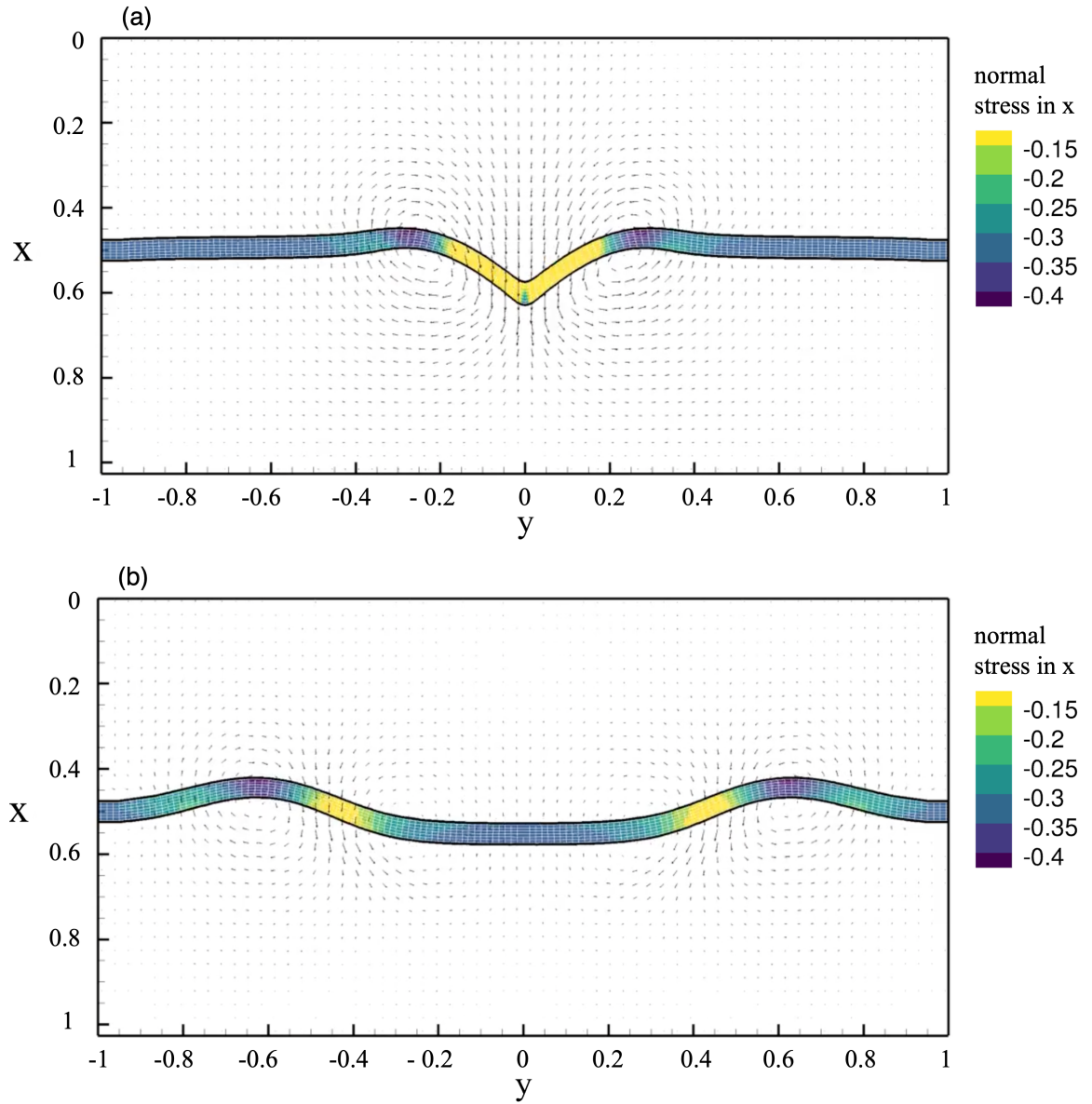


Figure 4.24: Oscillation of a solid film with a thickness of 0.05 under a point force with a 256×128 grid with $S=10$, $\text{pfrt}=0.05$, $\text{pfrd}=0.2$ and $\text{pfma}=150$ (part 1). The white line shows the instantaneous shape of the dynamic grid. Stress contour and velocity vectors are also plotted.

viscous stresses, and structural elasticity in multiphase fluid–structure interaction problems.

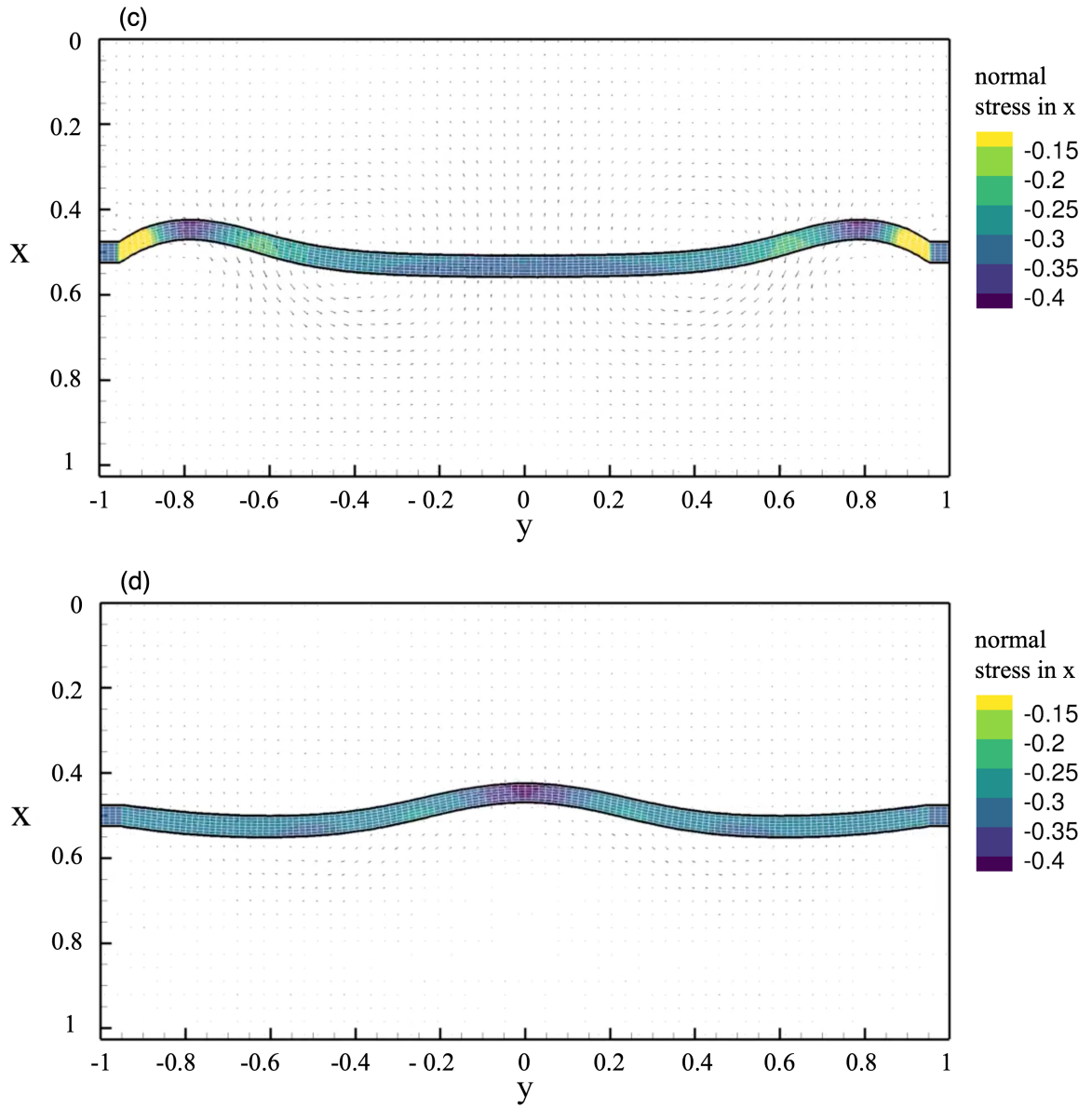


Figure 4.25: Oscillation of a solid film with a thickness of 0.05 under a point force with a 256×128 grid with $S=10$, $\text{pfrt}=0.05$, $\text{pfrd}=0.2$ and $\text{pfma}=150$ (continued). The white line shows the instantaneous shape of the dynamic grid. Stress contour and velocity vectors are also plotted.

4.12 Deformation of a solid film under a point force

This case evaluates the ability of the formulation to capture the dynamic response of a deformable structure subjected to localized forcing. A point force was implemented by the following procedure: We define a few constants: point force ramp up time (pf_{rt}), a point force ramp down time (pf_{rd}), and a point force maximum amplitude (pf_{ma}). The force amplitude $f(t)$ is applied using a smoothed error function:

$$f(t) = \begin{cases} \text{pfma} \cdot \text{erf}\left(\frac{2t}{\text{pfrt}}\right), & t \leq \text{pfrd}, \\ \text{pfma} \cdot \left[1 - \text{erf}\left(\frac{2(t - \text{pfrd})}{\text{pfrt}}\right)\right], & t > \text{pfrd}. \end{cases} \quad (4.8)$$

The application of a point force at the midpoint of the solid film generates a highly localized deformation and initiates elastic waves that propagate along the structure, providing a stringent test of the numerical treatment of stress transmission and structural dynamics. By varying the stiffness parameter S , the simulations examine how material rigidity influences the amplitude, frequency, and damping characteristics of the oscillatory response. This test therefore demonstrates the capability of the method to resolve localized loads, elastic wave propagation, and the coupled effects of fluid inertia and viscous dissipation in deformable structures. In this section, we examine the dynamic response of an elastic solid film subjected to a localized point force. This case is designed to investigate the transient deformation, wave propagation, and damping behavior of the film under concentrated loading in a fluid–structure interaction setting.

The computational domain consists of a two-dimensional rectangular region, within which a thin elastic solid film with a thickness of 0.05 is immersed between two fluid layers. The point force is defined in a circular region at center of the film with radius of 0.05 by a point force level set function $\text{pfls}(x, y, z)$. The forcing magnitude $F(x, y, z)$ is

defined by:

$$F(x, y, z) = \frac{f(t)}{2} \left(1 + \operatorname{erf} \left(-\frac{\text{pfls}(x, y, z)}{2dx} \right) \right) \quad (4.9)$$

In this test, $\text{pfrt}=0.05$, $\text{pfrd}=0.2$ and $\text{pfma}=150$ were used. The 2×1 domain is discretized into a 256×128 grid. The film is initially flat and horizontally aligned, separating the upper and lower fluids. Both ends of the film are fixed at the left and right boundaries, enforcing clamped boundary conditions that prevent rigid body motion and ensure that all deformation is induced by the applied force and subsequent fluid–structure interaction.

A localized point force is applied at the midpoint of the film in the vertical direction. This forcing introduces a concentrated deformation that perturbs the equilibrium configuration and generates elastic waves that propagate along the film. The surrounding fluids interact with the deforming structure, providing additional inertial and viscous effects that influence the overall dynamics.

The flow is characterized by a fixed Reynolds number in the simulations ($Re = 100$), and the solid behavior is governed by the non-dimensional stiffness parameter S . To investigate the role of structural rigidity, simulations are performed for a range of stiffness values, specifically $S = 2, 5, 10, 20, 50$, corresponding to increasingly resistant films.

Fig. 4.24 and Fig. 4.25 illustrates the instantaneous deformation of the film and the corresponding distribution of normal stress in the streamwise direction at selected time instances. Immediately after the application of the point force, a pronounced localized deflection is observed at the midpoint of the film. This deformation generates strong stress concentrations in the vicinity of the forcing location, as indicated by the contour plots. As time progresses, the initial disturbance propagates outward along the film in the form of elastic waves, leading to a redistribution of stress and a gradual smoothing of the deformation profile. The stress distribution becomes more uniform, and the film gradually approaches its equilibrium configuration.

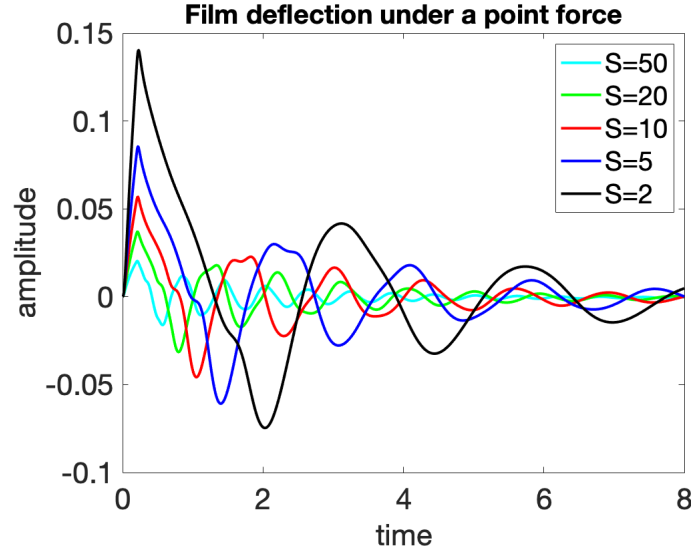


Figure 4.26: Amplitude of the midpoint over time for film with different stiffness S

The temporal evolution of the deformation is quantified in Fig. 4.26, which presents the amplitude of the midpoint displacement as a function of time for different stiffness parameters. All cases exhibit an oscillatory response, characteristic of an underdamped system where elastic restoring forces and inertia dominate the initial dynamics. However, the stiffness parameter significantly influences both the amplitude and frequency of oscillation.

For lower stiffness values (e.g. $S = 2$), the film is more compliant and exhibits larger initial deflections and lower oscillation frequencies. The reduced stiffness leads to a slower restoring force, allowing the deformation to persist over a longer time scale. In contrast, higher stiffness values (e.g. $S = 50$) result in smaller deformation amplitudes and higher oscillation frequencies, as the stronger elastic forces rapidly counteract the applied disturbance.

Additionally, the rate of amplitude decay is affected by the stiffness of the film. More compliant films tend to exhibit stronger damping, as larger deformations enhance viscous dissipation in the surrounding fluid. Conversely, stiffer films experience weaker damping and maintain oscillatory motion for a longer duration, although with smaller amplitudes.

Overall, the results demonstrate that the deformation dynamics of the solid film under a point force are governed by a balance between elastic restoring forces, fluid inertia, and viscous dissipation. The stiffness parameter plays a critical role in controlling the amplitude, frequency, and decay rate of the oscillations. These findings highlight the capability of the numerical method to accurately capture localized forcing, wave propagation, and coupled fluid–structure interaction effects in deformable systems.

4.13 Deformation solid filaments with periodic inflow

This case examines the capability of the formulation to simulate the collective dynamics of multiple deformable structures subjected to unsteady flow forcing. The oscillatory inlet velocity generates periodic hydrodynamic loading on the anchored filaments, producing cyclic bending, vortex formation, and complex fluid–structure coupling. Compared with previous single-structure cases, the presence of multiple filaments introduces wake interactions and shielding effects, providing a more demanding test of the solver’s ability to capture multi-body FSI and flow-mediated structural coupling. The results demonstrate that the method can accurately resolve the phase lag between the oscillatory flow and structural response, as well as the formation of secondary vortical structures and vertical momentum transport induced by the deforming filaments. The oscillatory inlet velocity u defined by:

$$u = \sin(2\pi t) \quad (4.10)$$

where $u = 1$ in this case. The computational domain consists of a two-dimensional rectangular region of non-dimensional length $L = 2$ and height $H = 1$ with a grid resolution of 256×128 . Three identical elastic solid filaments, each with a height of 0.8 and thickness of 0.1, are immersed within the fluid and vertically aligned along the bottom boundary. The

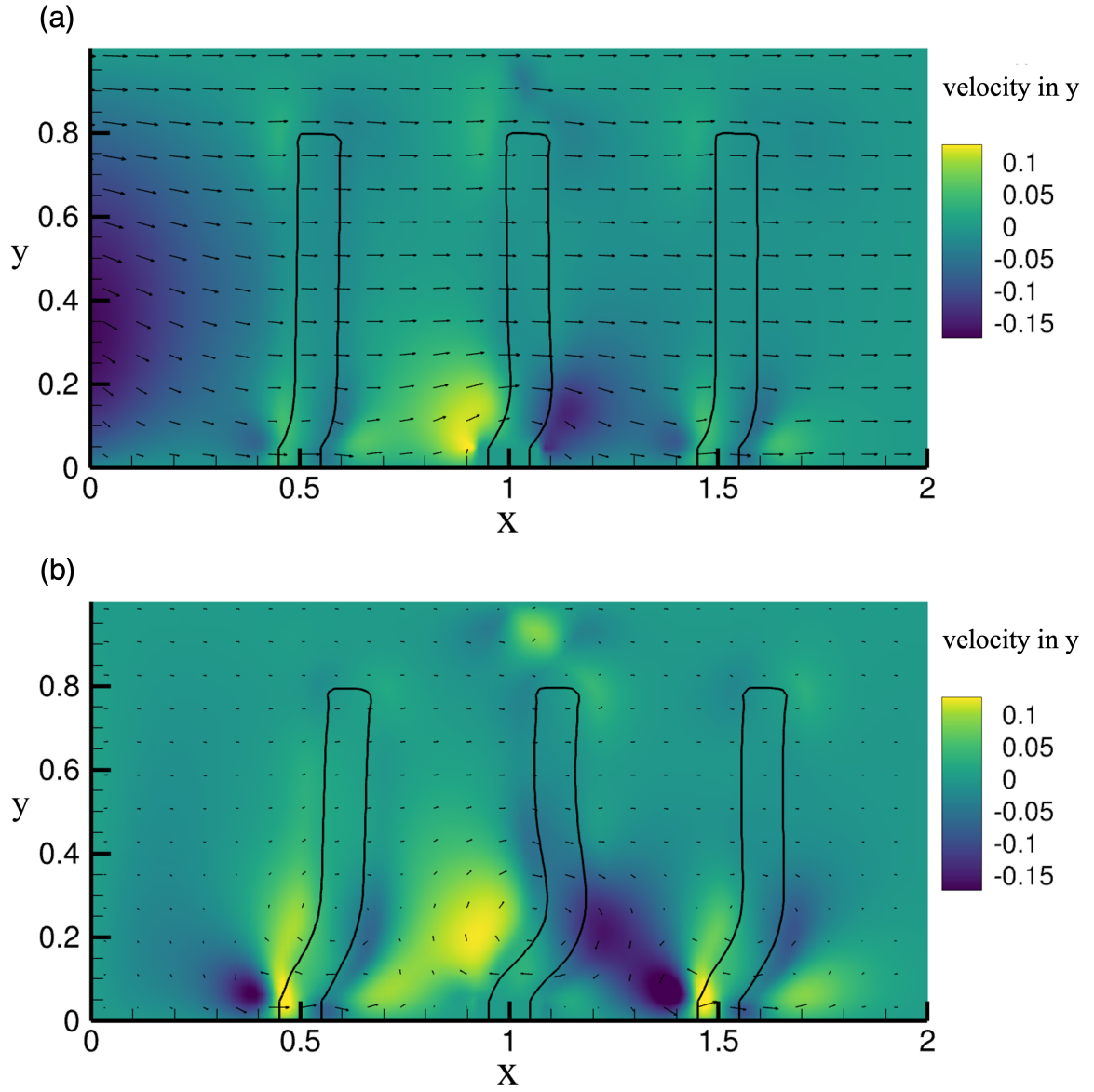


Figure 4.27: Movement of 3 solid filaments at the bottom of a channel with $Re = 100$ and $S = 1$ deforming by an oscillatory velocity inlet (part 1). Shape evolution and y velocity contours are plotted.

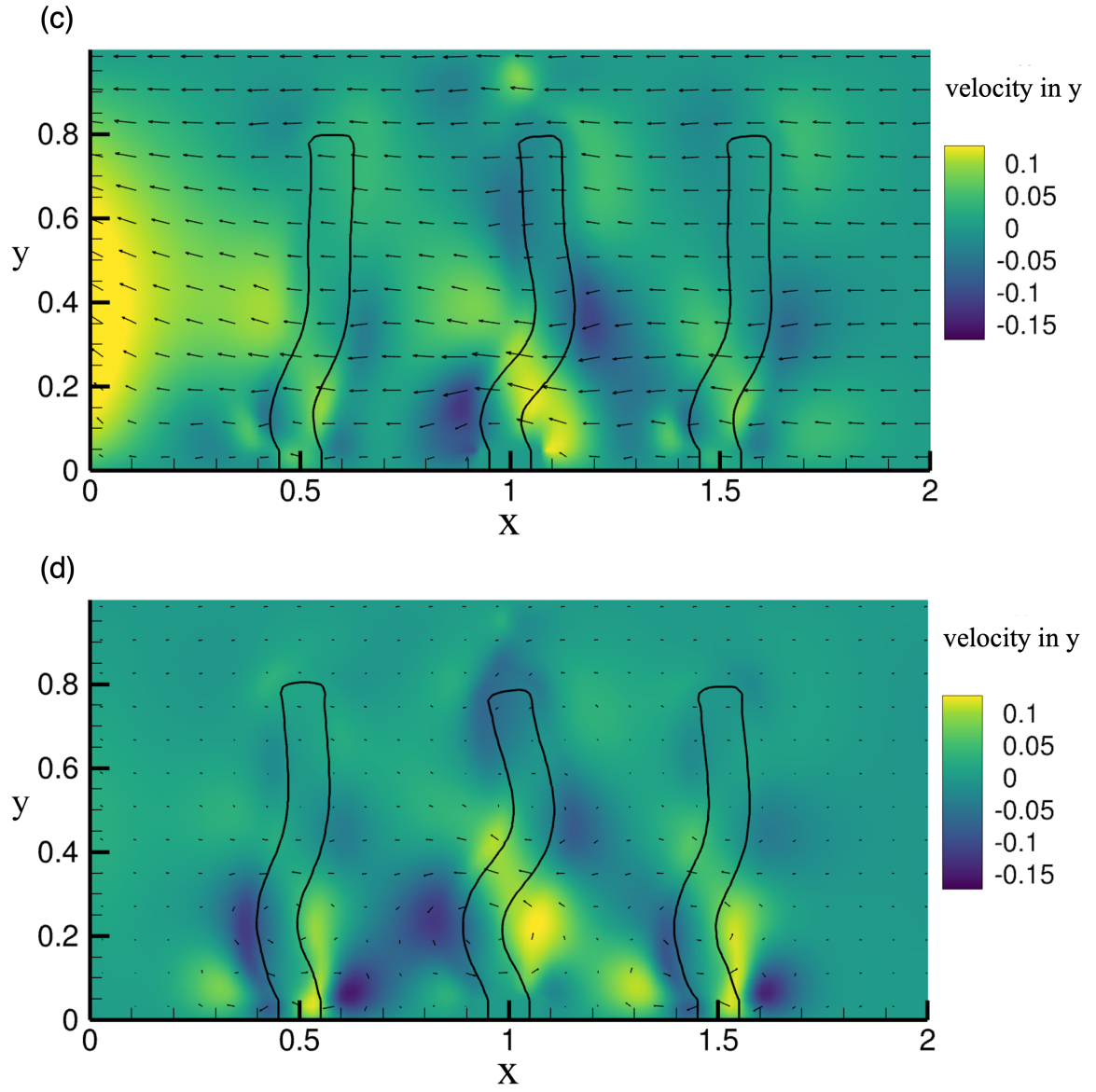


Figure 4.28: Movement of solid filaments at the bottom of a channel with $Re = 100$ and $S = 1$ deforming by an oscillatory velocity inlet (Continued). Shape evolution and y velocity contours are plotted.

filaments are initially straight and anchored at $y = 0$ using clamped boundary conditions, while their remaining surfaces are free to deform in response to the surrounding flow. The flow is driven by an oscillatory velocity inlet at $x = 0$, prescribed to simulate a reversing pressure gradient that periodically perturbs the structures. This forcing introduces a cyclic bending motion that generates complex secondary flows within the channel. The surrounding fluid interacts with the deforming filaments, providing inertial and viscous effects that govern the phase relationship between the flow and the structural response. The simulations are characterized by $Re = 100$ and $S = 1$, which dictates the resistance of the filaments to the hydrodynamic loads. Fig. 4.27 and Fig. 4.28 illustrate the instantaneous deformation of the filaments and the corresponding distribution of the vertical velocity component (v) at selected time instances. Immediately following the acceleration of the inlet flow, a pronounced localized deflection is observed as the filaments bend in the streamwise direction. This deformation generates strong vertical velocity concentrations, as indicated by the yellow (positive v) and purple (negative v) contour plots. As the cycle progresses, the initial bending evolves into higher-order modes, leading to a redistribution of the velocity field and the formation of recirculating zones between the structures. The vertical velocity distribution becomes particularly intense during the mid-cycle phases, where the "S-shaped" profiles of the filaments indicate a transition in the bending regime. The temporal evolution of the flow field reveals that the filaments act as dynamic obstacles that "pump" fluid vertically. For the observed stiffness and flow parameters, the system exhibits a clear phase lag; the maximum structural deflection does not coincide instantaneously with the peak inlet velocity, a characteristic of underdamped fluid–structure coupling where structural inertia and fluid damping play a competitive role. Furthermore, the proximity of the filaments influences the collective dynamics. The leading filament at the upstream position encounters the undisturbed oscillatory flow, while the subsequent filaments are subjected to the wake and shed vortices of their predecessors. This interaction leads to a modulation of the stress dis-

tribution along the filament surfaces and a variation in the amplitude of the vertical velocity pockets. Overall, these results demonstrate that the deformation dynamics of solid filaments in an oscillatory flow are governed by a complex balance between elastic restoring forces, fluid-induced drag, and the formation of secondary vortical structures. The filaments effectively convert horizontal fluid motion into vertical transport, highlighting the capability of the numerical method to capture the intricate coupling between multi-body structural deformations and unsteady fluid mechanics.

Chapter 5: Fluid–Elastic Solid Interaction in 2–D –Axisymmetric

Building upon the two-dimensional Cartesian validations presented in the previous chapter, this chapter extends the fully Eulerian fluid–structure interaction (FSI) framework to two-dimensional axisymmetric coordinate systems. This extension demonstrates the modularity of the numerical method and its capability to efficiently model problems using appropriate boundary conditions and cylindrical geometries, which are highly relevant to the tubular runner and gating systems frequently encountered in industrial metal casting. Specifically, the chapter investigates the dynamic response of deformable elastic boundaries subjected to periodic fluid forcing in two primary configurations: flow through an axisymmetric vessel and flow through an axisymmetric nozzle. By analyzing the resulting strain fields, structural deformations, and complex velocity profiles within these confined geometries, this chapter serves as a critical bridge between the foundational 2-D simulations and the fully three-dimensional implementations explored later in the dissertation.

5.1 Periodic flow through an axisymmetric vessel

This case evaluates the capability of the formulation to simulate unsteady fluid–structure interaction in an axisymmetric geometry. The imposed sinusoidal inflow generates a pulsatile flow field that produces time-dependent fluid stresses on the compliant vessel wall, leading to periodic structural deformation. This configuration provides a stringent test of the solver’s ability to capture the coupled dynamics between pulsatile fluid motion and

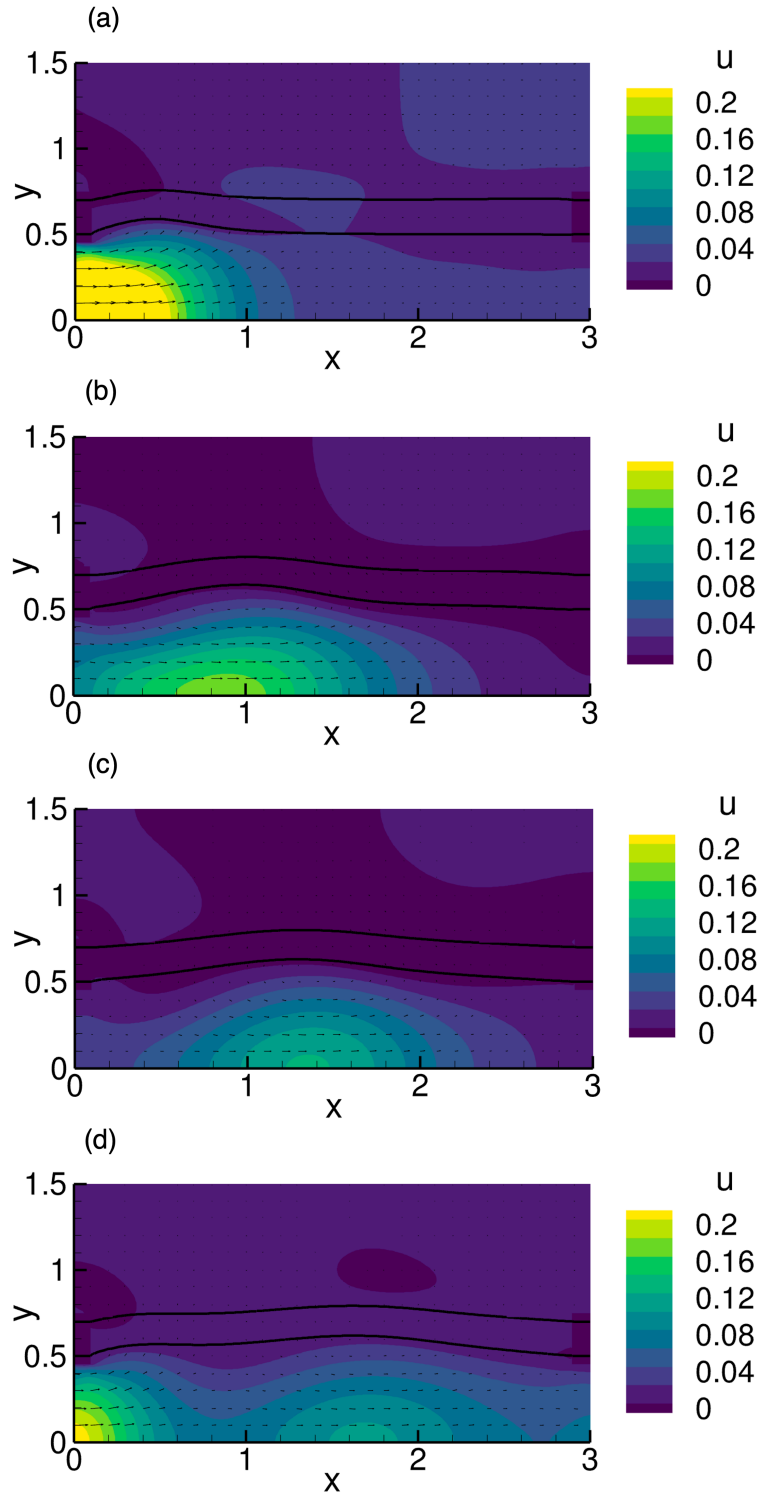


Figure 5.1: Axisymmetric flow through a vessel. $Re = 100$ and $S = 1$. Figure (a) to (d) are at different time instances within one oscillation cycle of the inflow

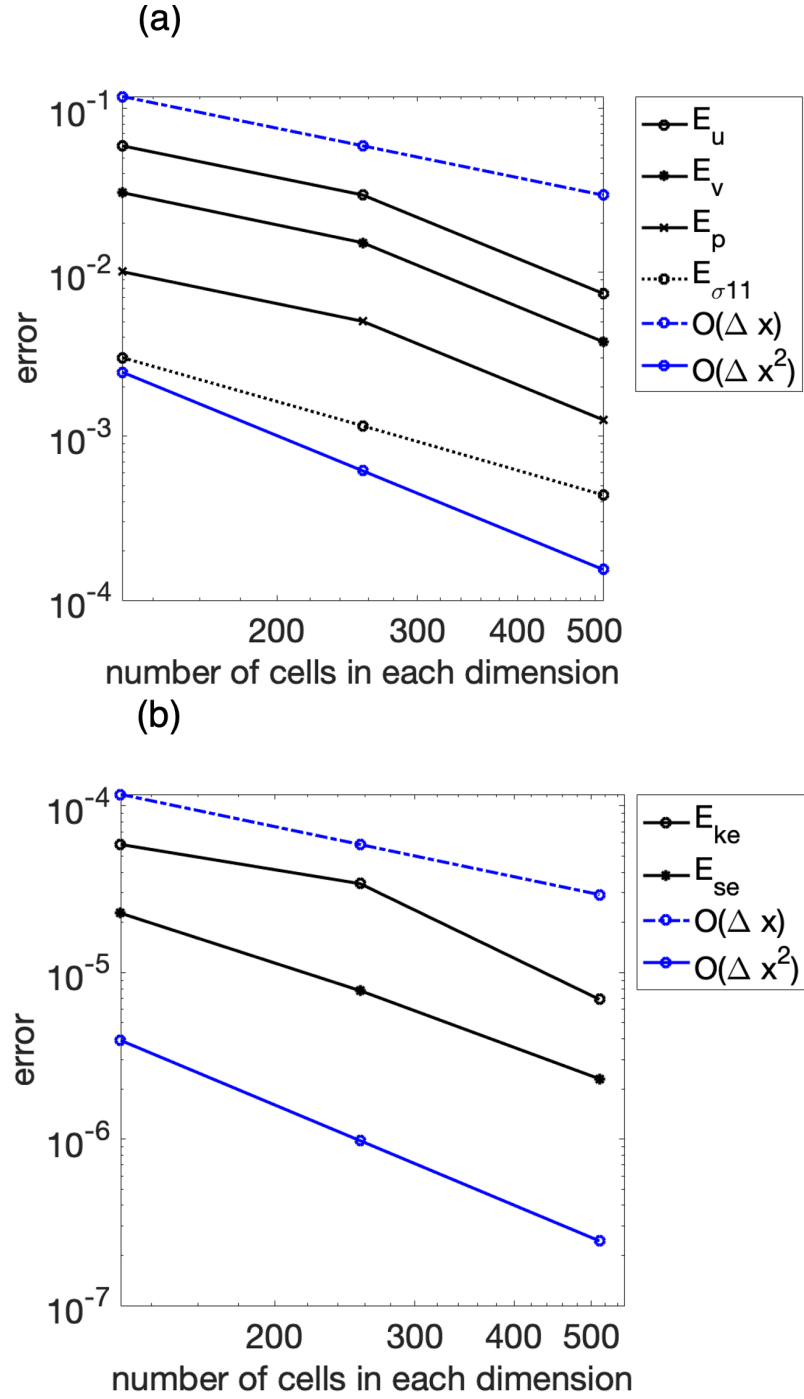


Figure 5.2: Convergence rate evaluated by the rms errors for u, v, p and σ_{11} (left), and the errors for ke and se (right) using the equations described in section 4.3 at $t = 2$. The reduction in error approaches $O(\Delta x^2)$ on the finer mesh.

elastic structural response while preserving the symmetry constraints of the axisymmetric formulation. The results demonstrate that the method can accurately resolve the propagation of oscillatory flow, the corresponding vessel wall deformation, and the convergence behavior of the numerical scheme in an axisymmetric fluid–structure interaction problem. Fig. 5.1 presents the simulation of axisymmetric flow through a deformable vessel, highlighting the coupled dynamics between a time-dependent inflow and the structural response of the vessel wall. The computational domain extends from $x=0$ to $x=3$ and from $y=0$ to $y=1.5$, where the bottom boundary ($y=0$) represents the axis of symmetry. The grid resolution is 256×128 . The left boundary serves as the inlet, where a time-varying sinusoidal velocity profile is prescribed by

$$u = 1.0 + 0.9 \sin(2\pi t) \quad (5.1)$$

with an average velocity of 1.0, while the top and right boundaries are treated as pressure outlets.

The inflow condition introduces an unsteady forcing characterized by a sinusoidal variation in time with a frequency of $\pi/4$. The flow operates at a Reynolds number $Re = 100$, corresponding to a laminar regime in which viscous effects remain significant. The solid structure is characterized by a stiffness parameter $S = 1$, indicating a relatively stiff vessel wall that undergoes moderate deformation under fluid loading.

The contour plots in Fig. 5.1 show the streamwise velocity component, with superimposed velocity vectors illustrating the flow direction and local magnitude. The vessel wall is represented by solid boundaries, which deform in response to the fluid stresses. Near the inlet, the oscillatory inflow generates a periodically accelerating jet that induces strong velocity gradients and shear near the lower wall. As the flow propagates downstream, viscous diffusion causes the jet to spread, resulting in a smoother velocity distribution across

the cross-section.

The deformation of the vessel wall is closely correlated with the temporal variation of the inflow. During phases of increased inflow velocity, the wall experiences higher fluid stresses, leading to enhanced outward deformation. Conversely, as the inflow weakens, elastic restoring forces dominate, and the wall relaxes toward its original configuration. This interaction produces a time-dependent deformation pattern that follows the imposed oscillatory forcing.

Panels (a)–(d) represent selected time instances within one oscillation cycle of the inflow. The results show that the deformation remains smooth and continuous, with no evidence of spurious numerical oscillations. The symmetry condition along the bottom boundary ensures consistency with the axisymmetric formulation, while the pressure outlet boundaries allow for natural outflow without artificial reflections.

Overall, the simulation demonstrates the capability of the numerical method to accurately capture unsteady fluid–structure interaction in a confined, axisymmetric geometry. The results highlight the combined effects of pulsatile inflow, viscous transport, and structural elasticity, which are key features in applications such as flow in compliant vessels and biological transport systems.

Fig. 5.2 shows the convergence rate calculated from Eq.(4.1-4.5) start to be around $O(\Delta x)$ but approaches $O(\Delta x^2)$ as the grid resolution increases. This is inline with the previous 2D results.

5.2 Periodic flow through an axisymmetric nozzle

This case further evaluates the capability of the formulation to simulate unsteady fluid–structure interaction in an axisymmetric configuration involving a flexible nozzle. The oscillatory inflow generates a pulsatile jet that interacts with the compliant nozzle wall, producing

time-dependent deformation that is particularly pronounced near the nozzle tip. By examining the influence of forcing frequency, Reynolds number, structural stiffness, and nozzle thickness, this case systematically investigates how fluid inertia, viscous effects, and elastic restoring forces govern the coupled dynamics. The results demonstrate that the numerical method can accurately capture the phase relationship between pulsatile flow and structural response, as well as the sensitivity of deformation amplitude to both flow and structural parameters. In addition, the convergence study confirms that the solver maintains near second-order spatial accuracy for this complex unsteady axisymmetric fluid–structure interaction problem. The parameters in this case and computational domain along with the grid resolution are the same as section 5.1 except that the nozzle has a length of 1.5. In this simulation, the baseline case is of $Re = 100$ and $S = 0.5$. The inflow boundary condition is prescribed as a sinusoidal velocity profile, introducing unsteady forcing into the system by:

$$u = 1.0 + 0.9 \sin(1/4\pi t) \quad (5.2)$$

The deformation of the nozzle is quantified by tracking the displacement amplitude of the tip at the downstream end of the structure.

Fig. 5.3 presents instantaneous snapshots of the flow field using contours of the stream-wise velocity component, along with velocity vectors and the deformed configuration of the nozzle. The results show that the oscillatory inflow generates a pulsatile jet that interacts strongly with the flexible wall. During phases of high inflow velocity, the jet penetrates further downstream and induces pronounced deformation of the nozzle, particularly near the tip region. As the inflow weakens, the jet decelerates and spreads, and the structure partially relaxes due to elastic restoring forces. The deformation remains smooth and continuous throughout the cycle, indicating stable coupling between the fluid and solid solvers.

The temporal evolution of the nozzle deformation is further quantified in Fig. 5.4, where

the tip displacement amplitude is plotted as a function of time under different forcing frequencies and structural stiffnesses. In Fig. 5.4(a), increasing the inflow frequency leads to a reduction in deformation amplitude and a noticeable phase lag between the forcing and structural response. At lower frequencies, the nozzle has sufficient time to respond to the fluid forcing, resulting in larger and more regular oscillations. In contrast, higher-frequency forcing limits the structural response, producing attenuated oscillations with reduced amplitude.

Fig. 5.4(b) illustrates the effect of the stiffness parameter on the nozzle dynamics. As the stiffness increases, the amplitude of the tip displacement decreases, reflecting the increased resistance of the structure to fluid-induced deformation. Softer configurations exhibit larger oscillations and more pronounced temporal variation, while stiffer cases respond more weakly and exhibit nearly harmonic behavior. This trend is consistent with the balance between fluid forcing and elastic restoring forces.

The influence of Reynolds number and structural thickness is examined in Fig. 5.5. In Fig. 5.5(a), increasing the Reynolds number results in larger deformation amplitudes and more complex oscillatory behavior. At higher Reynolds numbers, inertial effects dominate, leading to stronger fluid forcing and enhanced structural response. The waveform of the tip displacement also becomes less smooth, indicating increased nonlinear interaction between the flow and the structure.

Fig. 5.5(b) shows the effect of the structural thickness t on the nozzle deformation. Thinner structures exhibit larger displacement amplitudes due to reduced bending stiffness, while thicker structures display more damped responses. The results also indicate that thinner nozzles are more sensitive to the imposed oscillatory forcing, exhibiting larger variations in both amplitude and phase over time.

Finally, Fig. 5.6 presents a convergence study calculated from Eq.(4.1-4.5) for the coupled system, showing the root-mean-square errors for velocity, pressure, and stress compo-

nents, as well as global quantities such as kinetic and strain energy. The results demonstrate that the numerical method achieves close to second-order accuracy on refined meshes, confirming the consistency and reliability of the approach for resolving unsteady fluid–structure interaction problems.

Overall, the simulations demonstrate that the response of the flexible nozzle is strongly dependent on the inflow frequency, Reynolds number, structural stiffness, and thickness. The numerical framework successfully captures these coupled effects and provides physically consistent predictions of the deformation and flow dynamics in an axisymmetric configuration.

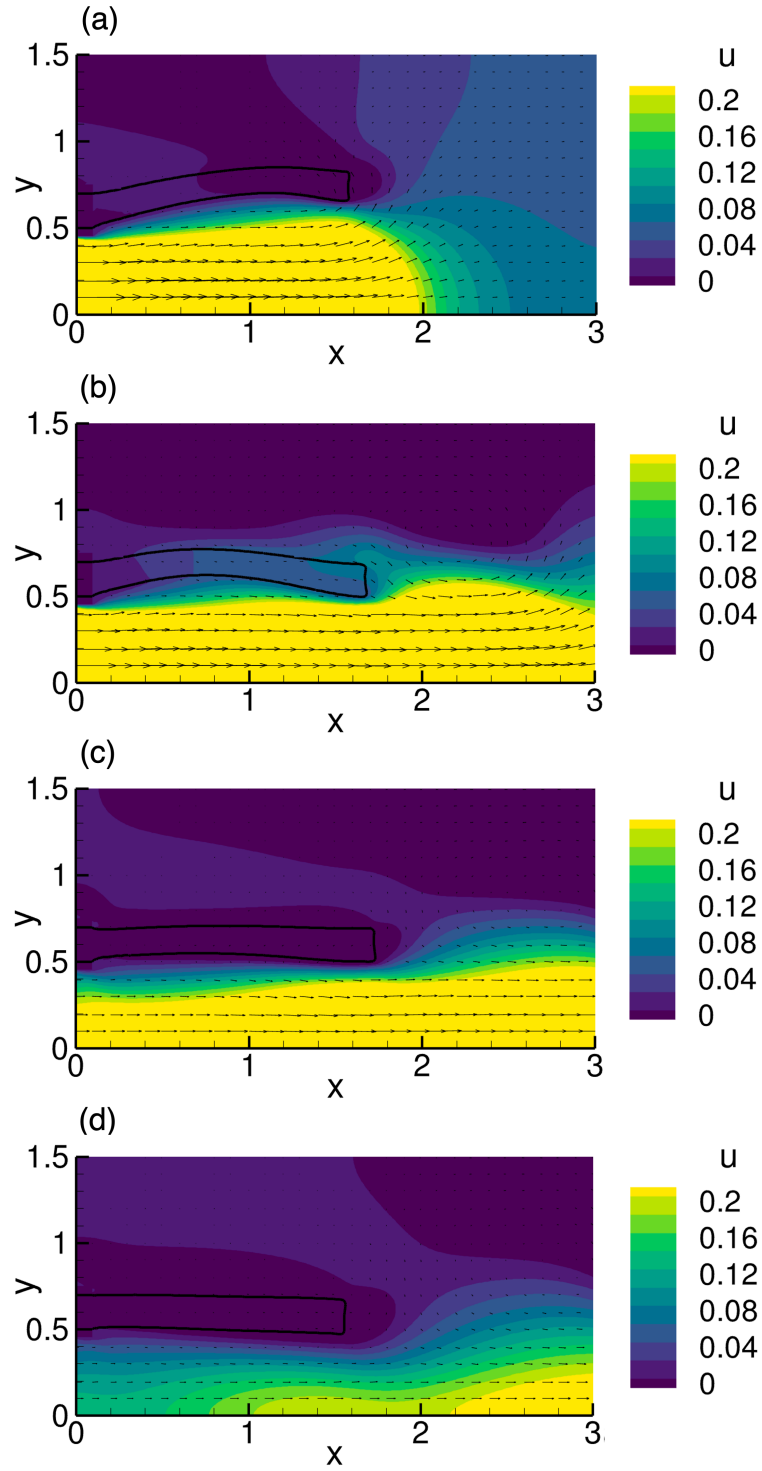


Figure 5.3: Axisymmetric flow through a nozzle. $Re = 100$ and $S = 0.5$. Figure (a) to (d) are at different time instances within one oscillation cycle of the inflow

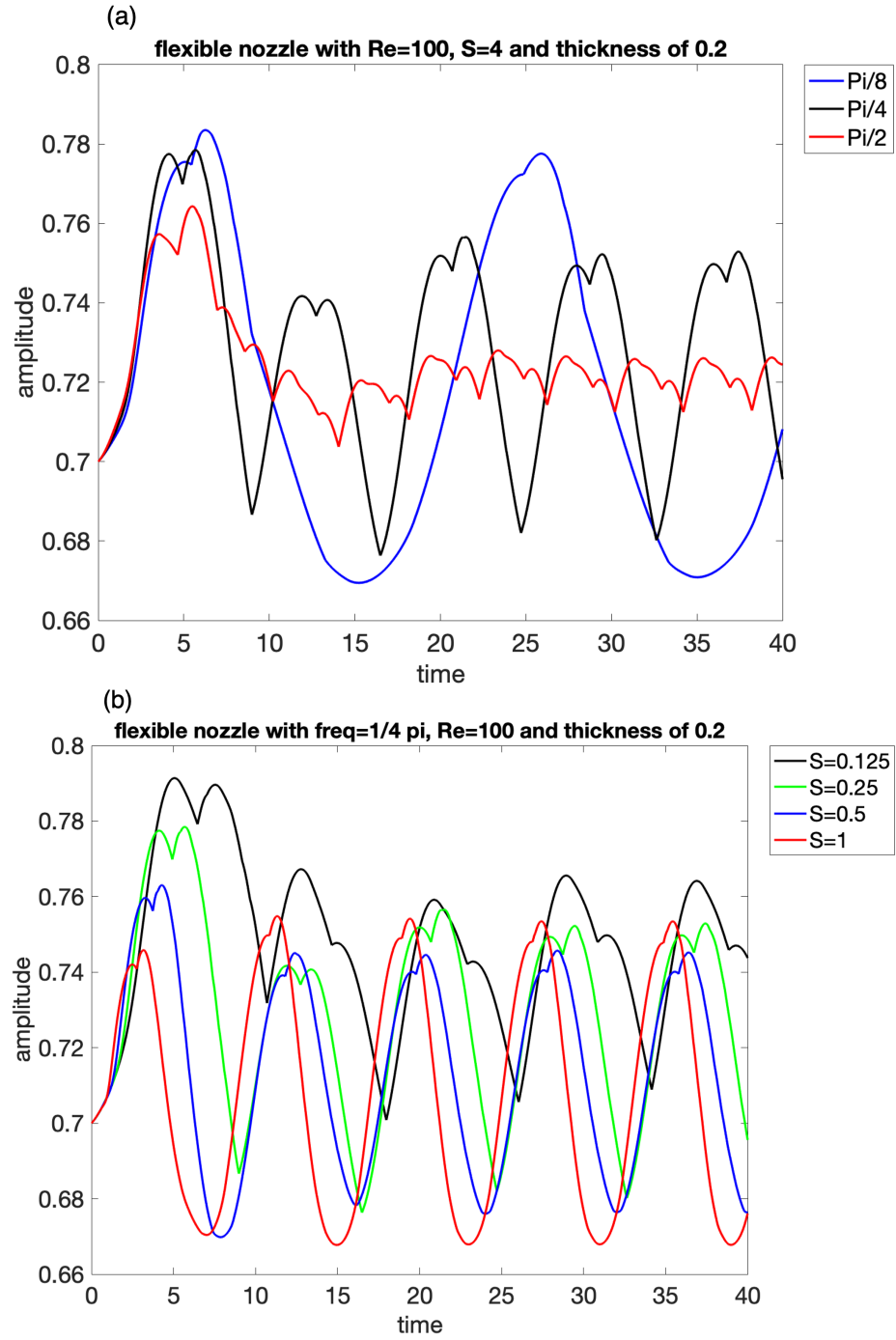


Figure 5.4: Parametric study for different inlet frequency (a) and solid stiffness parameter (b)

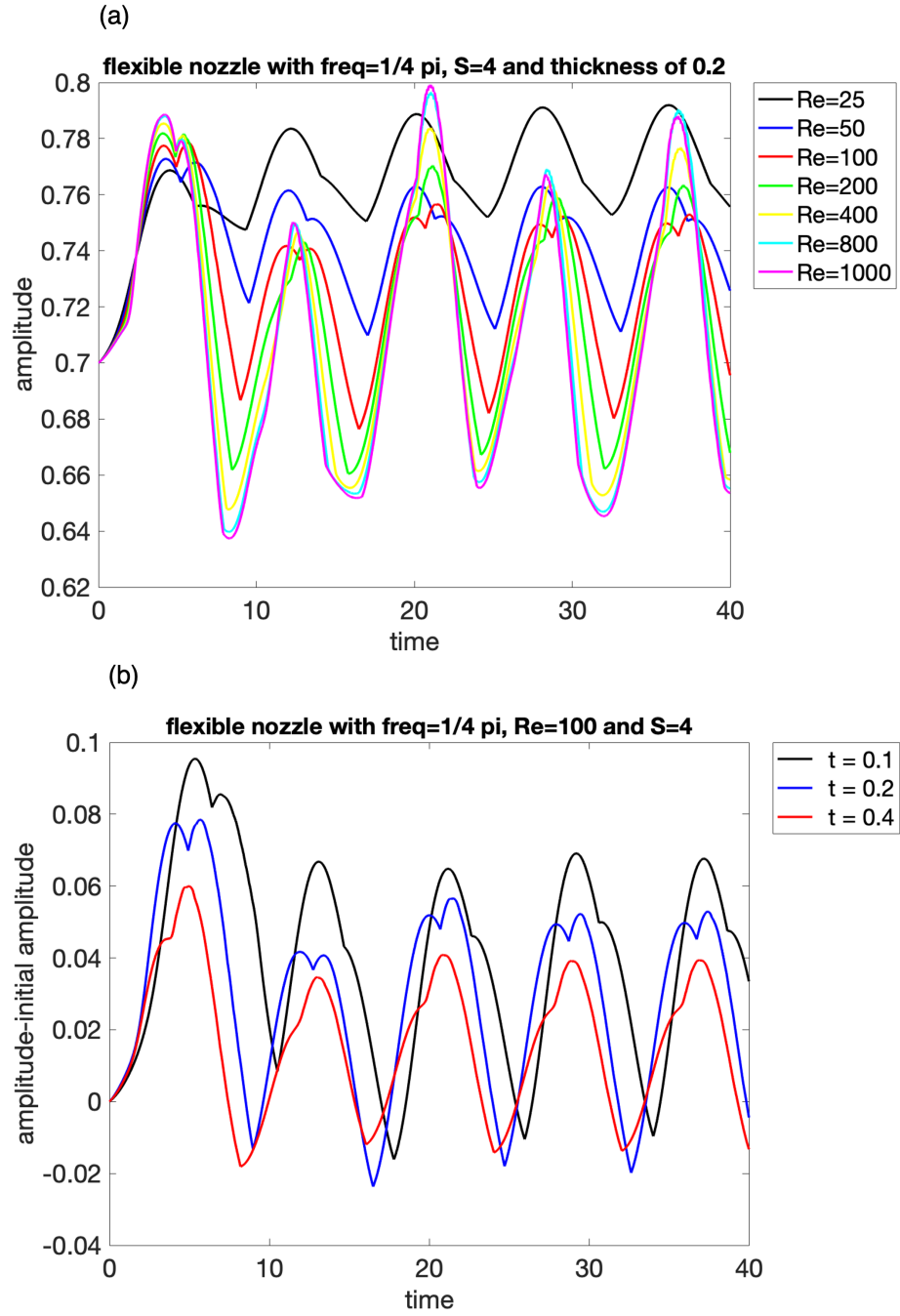


Figure 5.5: Parametric study for different fluid Reynolds number (a) and solid thickness (b)

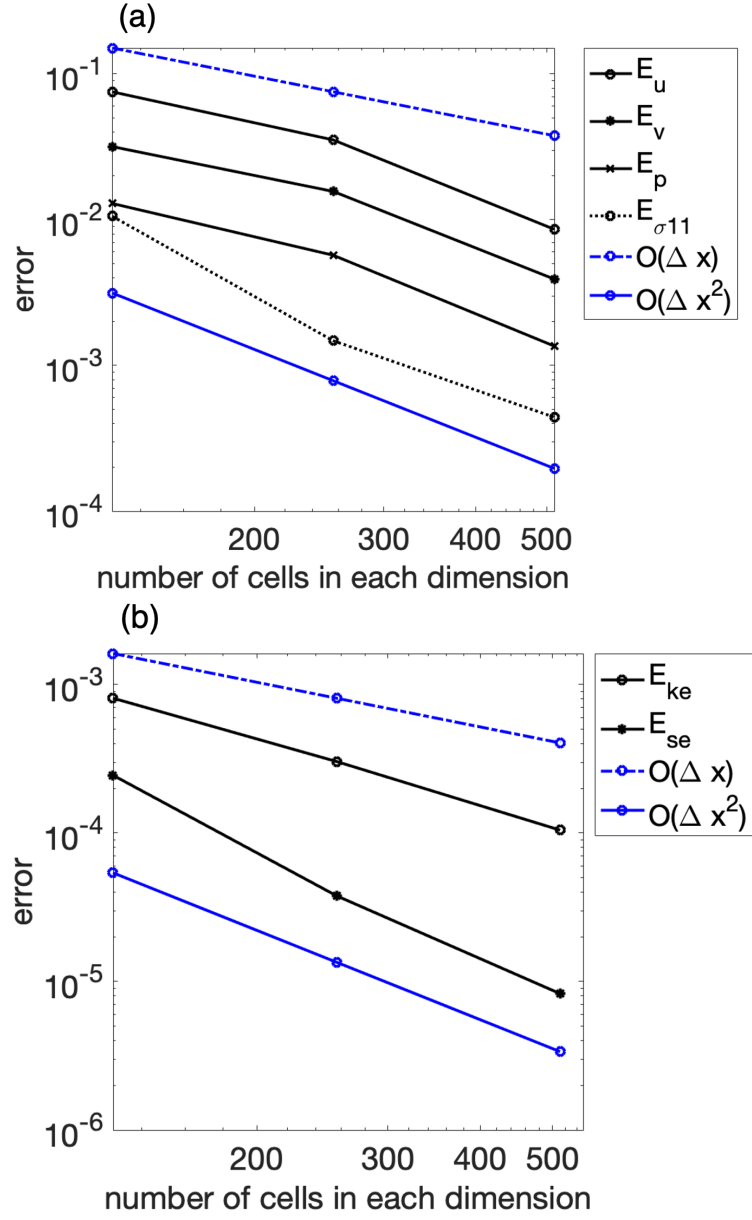


Figure 5.6: Convergence rate evaluated by the rms errors for u, v, p and σ_{11} (left), and the errors for ke and se (right) using the equations described in section 4.3 at $t = 2$. The reduction in error approaches $O(\Delta x^2)$ on the finer mesh.

Chapter 6: Fluid–Elastic Solid Interaction in 3–D

Having established and validated the Eulerian fluid–structure interaction (FSI) framework in two-dimensional and axisymmetric domains, this chapter presents the culmination of the numerical implementation: the extension to fully three-dimensional space. Transitioning to 3-D is essential for capturing the complex, out-of-plane deformations and multi-directional strain fields inherent to real-world industrial applications, such as the spatial dynamics of oxide films in liquid metal casting. This chapter demonstrates the robustness, accuracy, and scalability of the unified numerical approach by first simulating canonical 3-D benchmark problems, specifically an immersed oscillating sphere and an immersed oscillating cylinder. Building on these foundational validations, the chapter concludes with the highly complex simulation of a fully three-dimensional deformable solid film interacting with the surrounding fluid flow. Together, these cases illustrate the framework’s capability to efficiently resolve fully coupled, multi-dimensional hydrodynamics, paving the way for the realistic industrial scenarios discussed in the subsequent chapter.

6.1 Immersed oscillating sphere

The present Eulerian–Lagrangian framework can be straightforwardly extend to fully three-dimensional fluid–structure interaction problems without additional algorithmic complexity. This case demonstrates the capability to solve fully three-dimensional fluid–structure interaction problems. The Taylor–Green vortex provides a smooth vortical flow field with

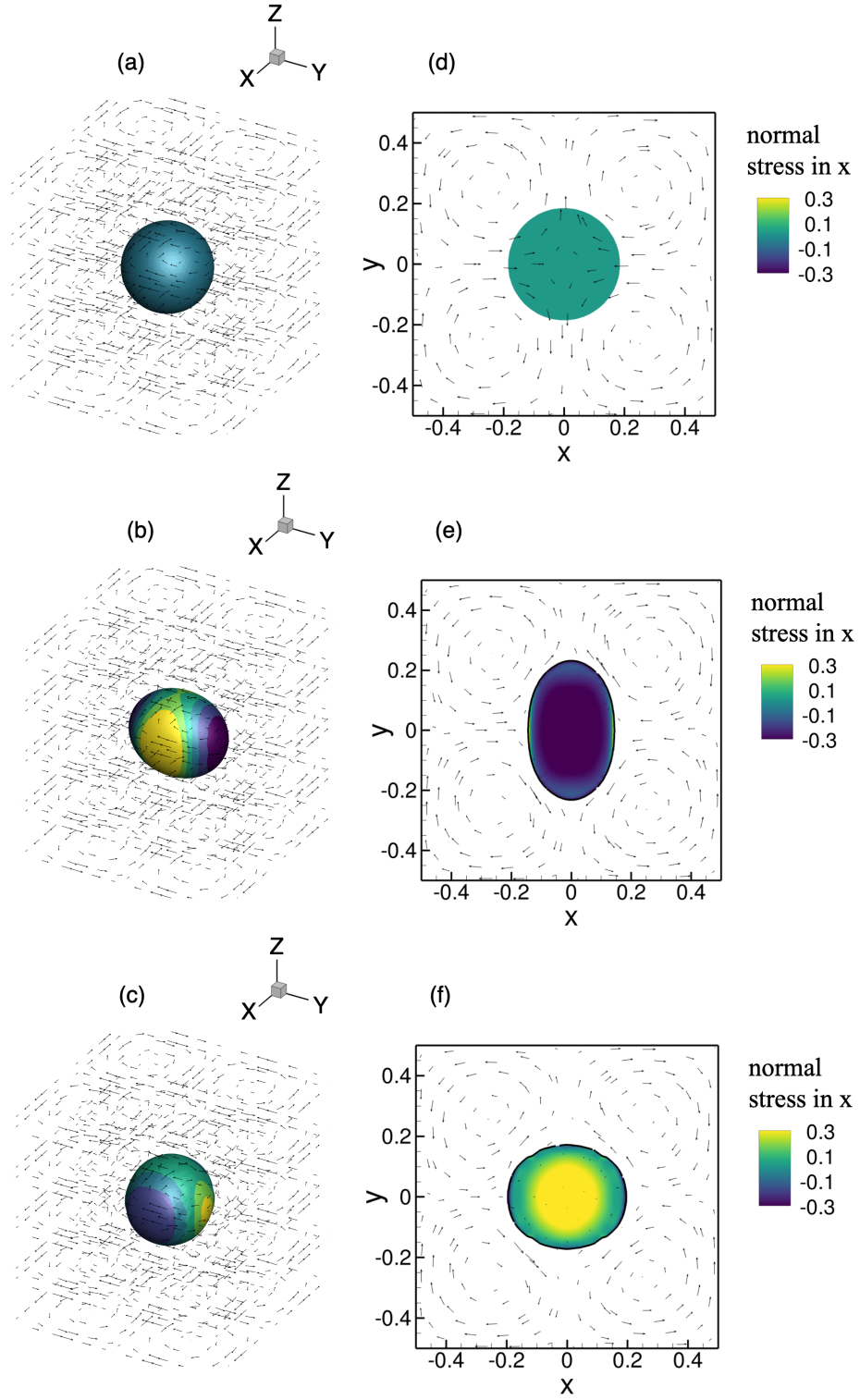


Figure 6.1: Figure (a)–(c) show the deformation of the solid sphere with prescribed Taylor-Green vortex field. $Re = 1000$ and $S = 1.0$. The normal solid stress in x direction is plotted on the surface of the solid. Figure (d)–(f) show stress contours and velocity vectors at the x-y plane in the center of the domain ($z=0$) for the corresponding sphere with $Re = 1000$ and $S = 1.0$.

spatially varying strain and rotation, offering a well-defined environment to test the structural response of an immersed elastic body under complex three-dimensional loading. The relatively high Reynolds number ($Re = 1000$) introduces significant inertial effects, creating strong velocity gradients that induce shear and extensional stresses within the solid. This configuration therefore provides a stringent test of the solver's ability to capture three-dimensional deformation, stress distribution, and two-way coupling between the fluid and the elastic structure while maintaining numerical stability and interface regularity.

The flow is initialized using the Taylor–Green vortex, which provides a smooth, spatially periodic velocity field characterized by well-defined vortical structures and alternating regions of strain and rotation described by:

$$\begin{aligned} u &= 0.1\pi \sin(2\pi x) \cos(2\pi y) \\ v &= -0.1\pi \sin(2\pi y) \cos(2\pi x) \\ w &= 0 \end{aligned} \tag{6.1}$$

The Reynolds number is fixed at $Re = 1000$, corresponding to a moderately inertial regime where both advection and viscous effects are important. The solid stiffness parameter is set to $S = 1$, representing a deformable solid that maintains structural coherence while allowing noticeable deformation.

The sphere is initially located at the center of the computational domain, where the symmetry of the flow field ensures balanced but spatially varying loading conditions. As the flow evolves, the imposed velocity gradients generate internal stresses within the solid, driven primarily by shear and extensional components of the surrounding fluid motion. These stresses lead to periodic deformation and oscillatory motion of the sphere.

The deformation of the sphere exhibits smooth and continuous variations over time, indicating that the numerical method preserves interface regularity and avoids spurious

numerical artifacts. The distribution of normal solid stress in the x -direction reveals clear correlations with the underlying flow structures. Regions aligned with extensional flow exhibit elevated tensile stresses, while compressive stresses arise in regions associated with flow convergence. This behavior reflects the direct coupling between the kinematics of the fluid and the elastic response of the solid.

Cross-sectional analysis at the mid-plane of the domain ($z = 0$) further highlights the coupled dynamics. The velocity field shows localized perturbations near the fluid–solid interface, demonstrating two-way interaction between the phases. Inside the solid, the stress field reflects a balance between viscous forcing imposed by the fluid and elastic restoring forces within the material. This interplay results in sustained oscillatory behavior without numerical instability, confirming the robustness of the method in three-dimensional settings.

u, v and w are corresponding x, y and z component of the velocity vectors. Solid sphere is located at the center of the domain. Deformation of the sphere and the stress contours are shown in Fig. 6.1 (a)–(c). Contours for normal solid stress in x direction are plotted at the cross sections normal to z direction in the middle of the solid. Results for stress and velocity are shown in Fig. 6.1 (d)–(f).

6.2 Immersed oscillating cylinder

This case further evaluates the robustness of the formulation by considering a different three-dimensional geometry subjected to the same Taylor–Green vortex field. Because the imposed velocity field is uniform in the spanwise (z) direction, the forcing conditions remain identical across all cross-sections of the cylinder, providing a useful test of the method’s ability to preserve physical symmetry in three-dimensional simulations. Unlike the sphere, which undergoes fully three-dimensional deformation, the cylindrical geometry

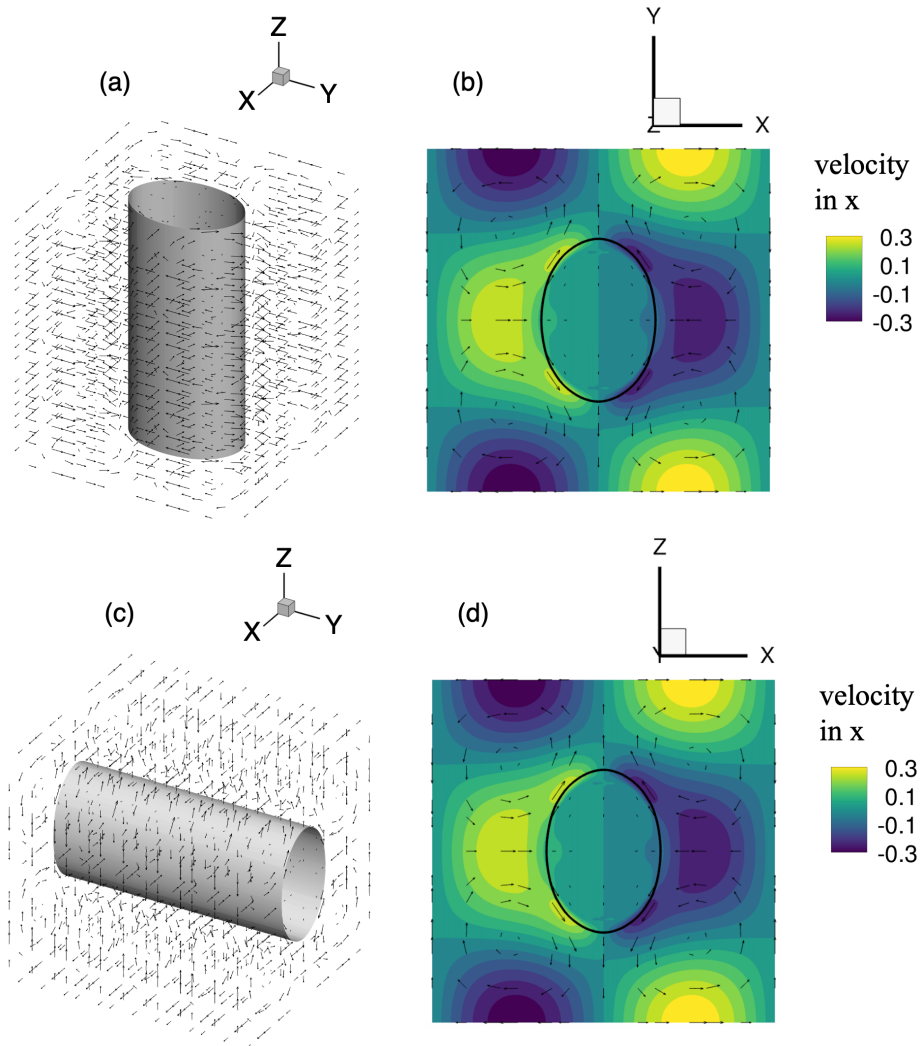


Figure 6.2: Figure (a) and (c) show velocity vectors and the deformation of the solid cylinder with prescribed Taylor-Green vortex field. $Re = 1000$ and $S = 1.0$. Figure (b) and (d) show u velocity contour and velocity vectors at the x - y plane and x - z plane respectively

results in a predominantly in-plane deformation response that is nearly invariant along the axial direction. This configuration therefore verifies that the numerical framework can accurately capture geometry-dependent deformation while maintaining consistent structural response along the spanwise direction. The results confirm the capability of the method to simulate three-dimensional fluid–structure interaction for bodies with different geometrical characteristics while preserving the underlying symmetry of the flow field.

The Taylor–Green vortex field for this case is the same as the one in Eq. 6.1. Despite the spanwise uniformity of the forcing, the geometric characteristics of the cylinder lead to deformation behavior that differs from that of the sphere. The cylinder experiences spatially varying loading within each cross-sectional (x – y) plane due to the non-uniform distribution of velocity gradients in the Taylor–Green vortex field. Consequently, the deformation is primarily governed by in-plane stretching and compression, while remaining nearly invariant along the axial direction.

Compared to the sphere, which undergoes fully three-dimensional and symmetric deformation, the cylinder exhibits a quasi-two-dimensional response with consistent deformation patterns along its length. This behavior reflects the underlying symmetry of the flow and confirms that the numerical method preserves physical invariance in the spanwise direction.

Overall, the results demonstrate that the proposed method accurately captures fluid–structure interaction under spatially uniform forcing while maintaining consistency across multiple cross-sections. The uniformity of the deformation along the axial direction further validates the robustness and reliability of the method for three-dimensional problems with inherent symmetry.

Deformation of the cylinder with the velocity vectors are shown in Fig. 6.2.

6.3 3D solid film

This case further demonstrates the capability of the numerical framework to simulate fully three-dimensional fluid–structure interaction involving localized jet forcing. The impinging jet generates strong spatially varying fluid stresses that induce complex deformation of the solid film, providing a demanding test of the solver’s ability to capture coupled flow and structural dynamics in three dimensions. Comparison with the corresponding two-dimensional configuration reveals that the 3D film exhibits a comparatively stiffer response, reflecting the influence of additional geometric constraints and out-of-plane stress contributions present in three dimensions. By varying the stiffness parameter S , the simulations also highlight the sensitivity of the deformation to material rigidity and demonstrate the robustness of the method for modeling highly deformable thin structures subjected to localized fluid loading. As shown in Fig. 6.3. The objective of this study is to investigate the dynamic response of the solid film under fluid forcing and to compare its behavior with corresponding two-dimensional configurations.

The results reveal notable differences between the 2D and 3D cases. In particular, the 3D solid film exhibits a comparatively stiffer response, even when characterized by a smaller stiffness. This increased apparent stiffness can be attributed to the additional geometric constraints and out-of-plane stress contributions present in three dimensions, which enhance the overall structural resistance to deformation.

Fig. 6.3 illustrates the deformation of the solid film under different stiffness parameters. Fig. 6.3(a), with a higher stiffness parameter ($S = 100$), shows a more localized and restrained deformation, indicating a stronger resistance to fluid-induced stresses. In contrast, in Fig. 6.3(b), corresponding to a relatively soft material ($S = 1.0$), the film undergoes significant deformation due to the impinging jet, with pronounced elongation and shear near the impact region. The differences between these cases highlight the sensitivity of the

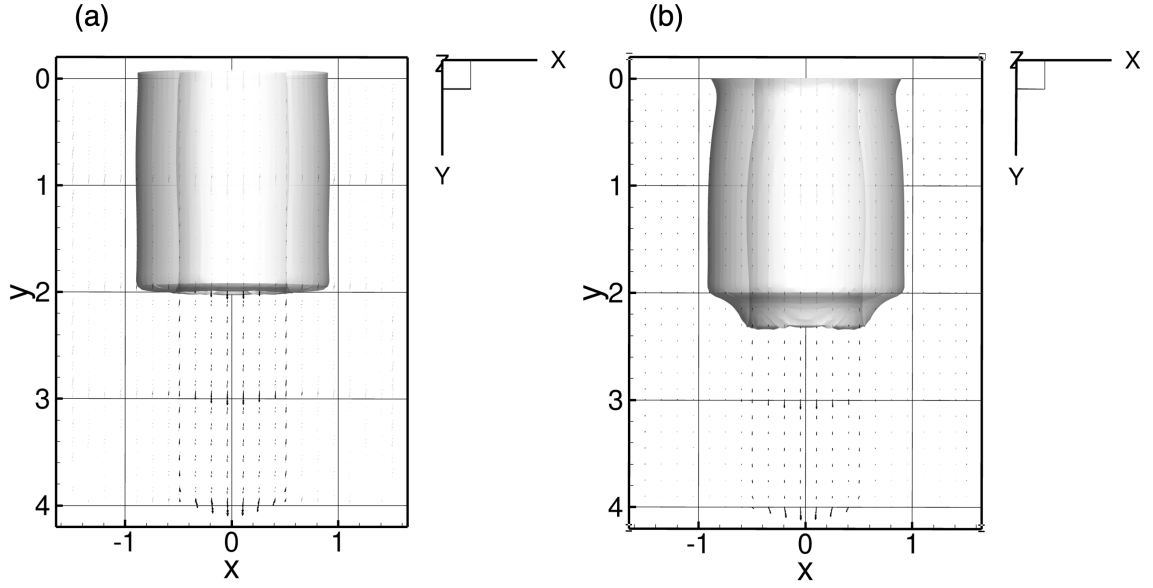


Figure 6.3: Shearing of the solid film of different solid stiffness. Figure (a) is the solid with $S = 100$. Figure (b) is the solid with $S = 1.0$.

film dynamics to material properties.

These simulations are part of the study aimed at characterizing the behavior of 3D solid films under varying flow and material conditions. In particular, we are systematically exploring the effects of inlet velocity amplitude, oscillation frequency, solid shear modulus, and film thickness on the deformation and stability of the structure. This parametric analysis will provide deeper insight into the coupled fluid–structure dynamics and help establish scaling relationships governing the response of deformable films in three-dimensional flow environments.

Chapter 7: Industrial Application and Solver Capability Evaluation: Internship Work at Dassault Systèmes

Building upon the fundamental numerical developments presented in previous chapters, this chapter transitions the focus toward the evaluation and application of fluid-structure interaction and multiphase flow solvers within a large-scale industrial setting. The research detailed herein was conducted during an industrial internship at Dassault Systèmes, specifically focusing on validating their state-of-the-art PowerFLOW and XFlow solvers against key industrial and academic benchmarks. The overarching motivation was to assess the capability of existing commercial methods to handle complex multiphase phenomena (forced oscillation, rotational movement, lubrication and wetting) and to explore complementary FSI numerical schemes. This work serves as a validation of the expertise developed during this doctoral research and provides an experience to integrate and validate newly developed numerical schemes, like the novel fully Eulerian method presented earlier, placing it within the context of established industry standards. This chapter is structured to cover three primary areas of evaluation: multiphase validation of commercial solvers, wetting and contact angle studies, and the development and verification of an Immersed Boundary Scheme using the Lattice Boltzmann Method (LBM).

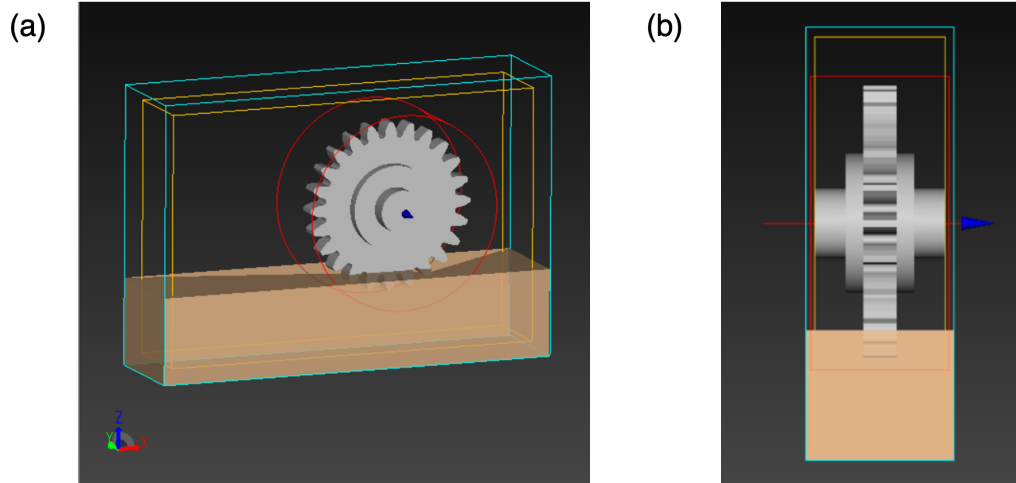


Figure 7.1: Setup for the single FZG gear lubrication simulation. Different oil viscosities and gear tip speeds are used.

7.1 Multiphase Solver Validation for Industrial Lubrication Systems

Understanding oil distribution and torque prediction in gearboxes is critical for optimizing cooling and lubrication of the gear and reducing energy loss. A key objective was to evaluate the lubrication capability of established commercial solvers against an industrial benchmark, the FZG (Forschungsstelle für Zahnräder und Getriebebau) gear test rig. Two primary multiphase simulation methods were validated: a Phase-field method implemented within the PowerFLOW solver and a particle-based method implemented within the XFlow solver.

7.1.1 FZG Gear Lubrication Simulation Setup

As shown in Fig. 7.1, the FZG gear lubrication case provides a challenging benchmark involving high-speed rotation, complex free-surface dynamics, and strong fluid-solid interactions (the rotating gears acts as moving boundaries). Simulations were performed to predict key operational parameters, specifically focusing on oil distribution within the

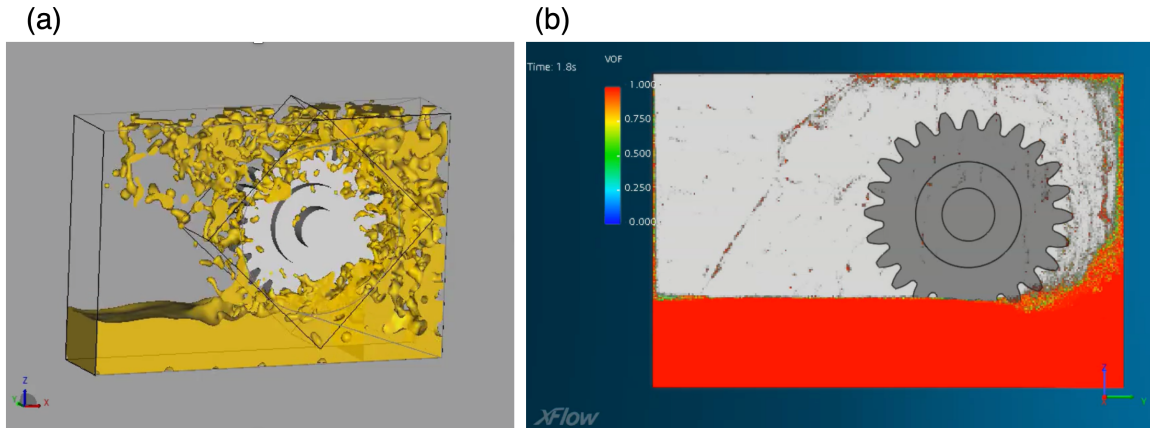


Figure 7.2: Oil distribution in the for the gear tip speed of 5m/s with FVA3 oil. The results in (a) are from PowerFLOW and the results in (b) are from XFlow.

gear case and the resulting torque on the input shaft. Precise modeling of the gas-liquid interface and oil-wall interaction are required to capture these parameters accurately. This work directly addresses the fundamental challenges of modeling a real world problem by providing a multiphase validation study.

7.1.2 Torque Prediction and Oil Distribution Results

The validation effort focused on comparing the numerical predictions of the two solvers against experimental data from the FZG rig. Key findings are summarized as follows:

- **Phase-Field Method (PowerFLOW):** The phase-field formulation demonstrated robust handling of the complex, topological free-surface changes inherent in the churning oil. However, visual analyses in Fig. 7.2 (a) confirmed that the predicted oil distribution and lubrication patterns on the gear teeth does not match the expected experimental dynamics. A consistent over-splashing has been observed in all the simulations especially in this high rotating speed condition. In addition, the predictions for torque is not accurate due to the physics not being captured appropriately.

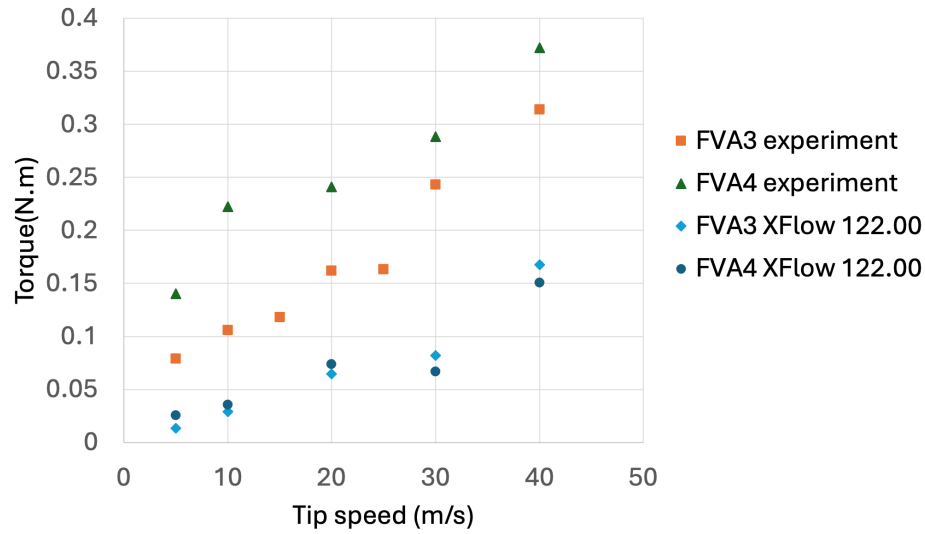


Figure 7.3: Experimental and simulation results (from XFlow solver version 122.00) for torque at different tip speed for FVA3 and FVA4 oil.

- Particle-Based Method (XFlow):** As a complementary approach, the particle-based method provided another avenue for validating the FZG case. The method, known for its ability to handle extremely complex geometries and topological changes with ease, was evaluated for fluid visualization and torque prediction. Results in Fig. 7.2 (b) shows good capability for capturing the dynamic free-surface features, resulting in good agreement in oil distribution to the experiments. As for the torque predictions, the XFlow solver has consistently under-predict the torque which can be found in Fig. 7.3.

In summary, this extensive validation of two diverse commercial multiphase methods shows that both solvers have potential of handling high speed rotating problems but currently find it challenging to predict oil distribution and accurate torque correctly. These tests provided some insights for future developments.

7.2 Validation of Sloshing Dynamics: Tuned Liquid Damper

The accurate prediction of violent free-surface sloshing is highly relevant to many applications including gear lubrication and the metal casting. For example, during the pouring process in metal casting, liquid metal often undergoes severe sloshing and wave breaking within the crucible or runner system, which is a primary driver of oxide film entrainment and dross formation. The PowerFLOW solver's capability to capture these highly dynamic multiphase phenomena has been tested using a Tuned Liquid Damper (TLD) benchmark.

7.2.1 Computational Setup and Methodological Variations

The TLD case involves a partially filled rectangular tank subjected to horizontal harmonic excitations. To establish a baseline for the sloshing dynamics, the fundamental natural frequency of the water sloshing motion, f_w , was first predicted using linear water wave theory:

$$f_w = \frac{1}{2\pi} \sqrt{\frac{\pi g}{L} \tanh\left(\frac{\pi h_0}{L}\right)} = 0.458$$

where g is the acceleration due to gravity, L is the tank length, and h_0 is the initial fluid depth.

Rather than utilizing a moving mesh approach to physically translate the container walls, the shaking table vibration was efficiently modeled by adding an oscillation force as a body force directly into the momentum equations. This oscillation acceleration, a_{osc} , is defined as:

$$a_{osc} = (2\pi f_e)^2 A \sin(2\pi f_e t)$$

where f_e is the excitation frequency and A is the excitation amplitude.

This benchmark rigorously tests a solver’s ability to track rapid topological changes at the gas-liquid interface and accurately compute the resulting time-varying sloshing forces on the container walls.

To assess numerical stability and accuracy, two distinct advection schemes (registries) for the phase-field variable were compared: a standard momentum-based registry and a newly implemented Velocity-based registry. The system was simulated across a sweep of excitation force frequency ratios—specifically $f_r = 0.9, 1.0, 1.18$, and 1.35 —at a constant baseline amplitude to capture both sub-resonant and super-resonant sloshing behaviors.

7.2.2 Results and Discussion

The numerical predictions were validated against experimental time-history data for both the non-dimensional water surface elevation at the left wall and the horizontal sloshing force (F_x).

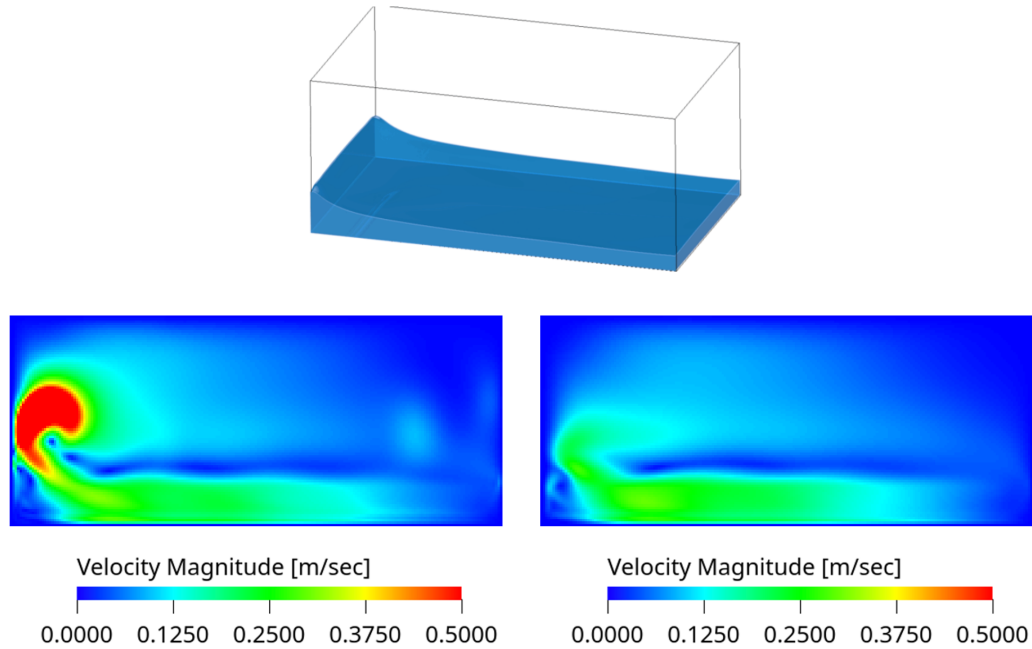


Figure 7.4: Water elevation and velocity magnitude with momentum-based (left contour) and velocity-based scheme (right contour).

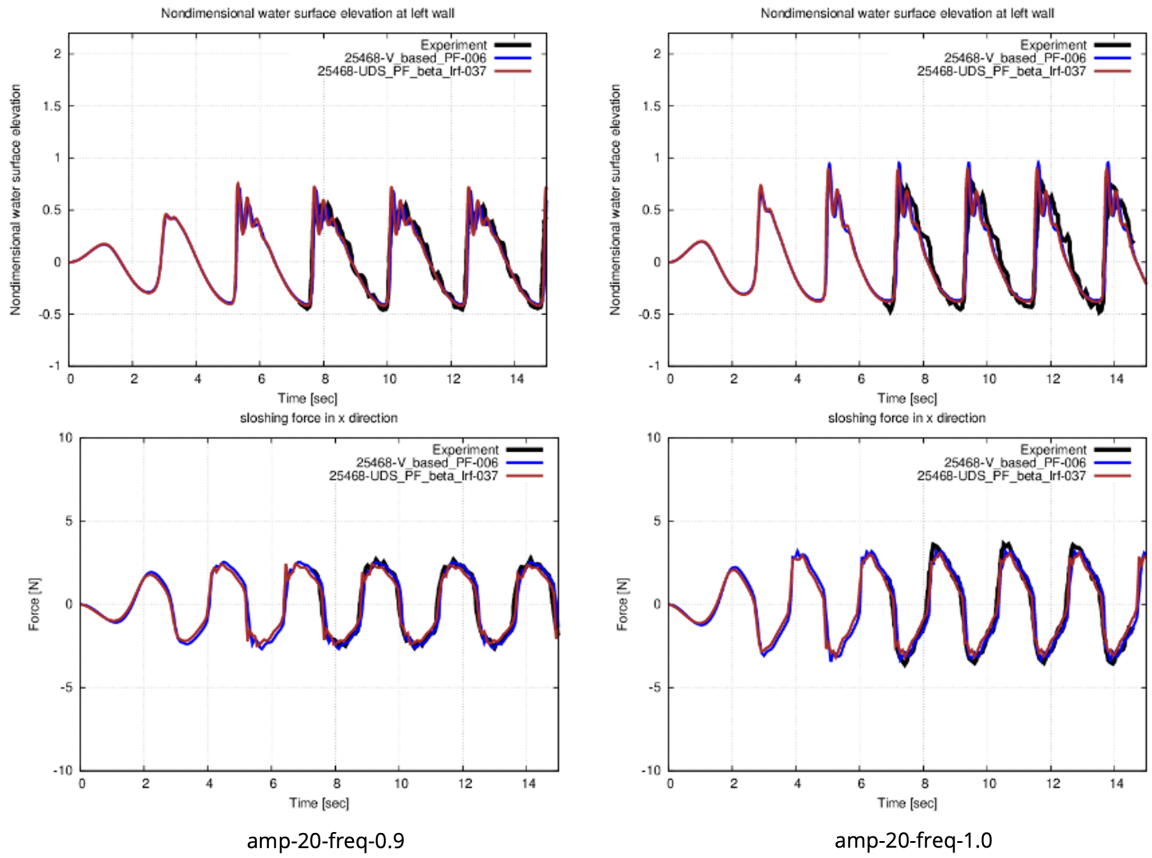


Figure 7.5: Comparison of experimental and numerical results for surface elevation and sloshing force across various frequency ratios (f_r) of 0.9 and 1.0.

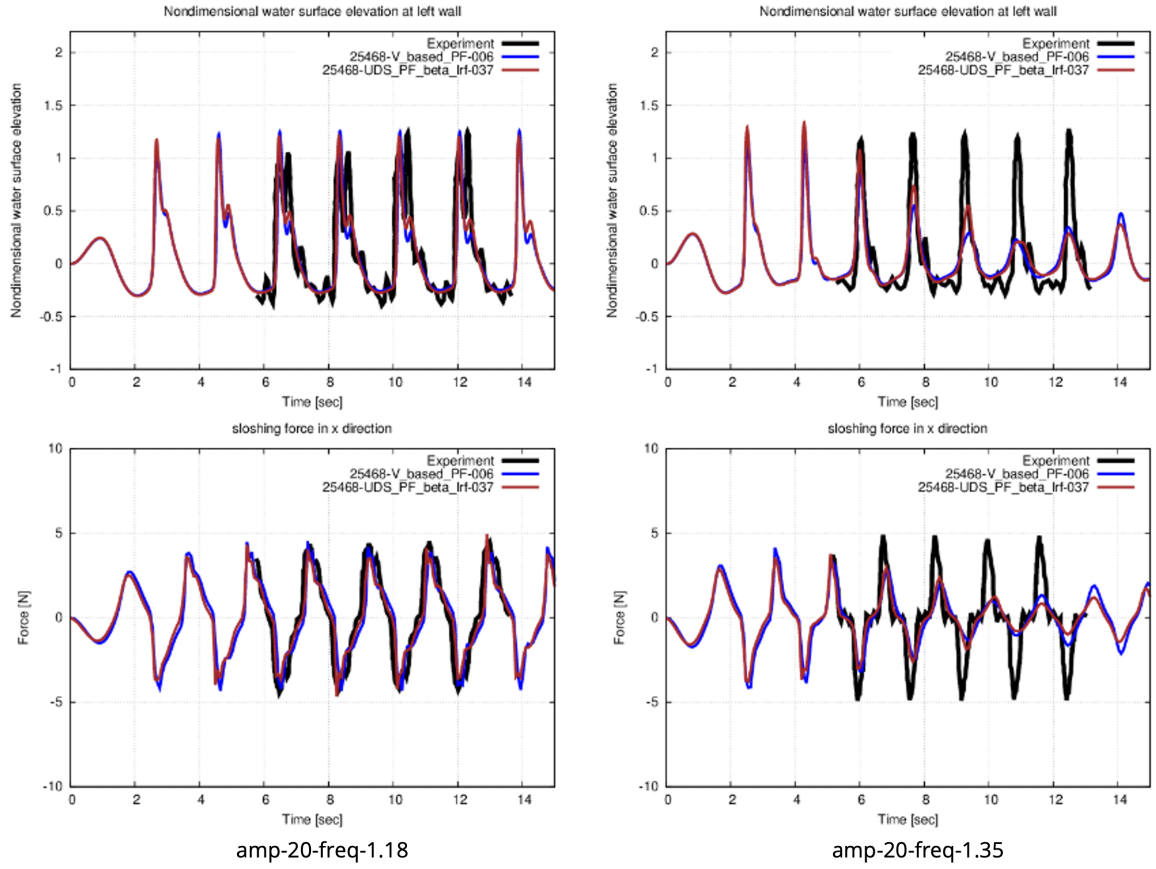


Figure 7.6: Comparison of experimental and numerical results for surface elevation and sloshing force across various frequency ratios (f_r) of 1.18 and 1.35.

Qualitative Flow Field Improvements:

An initial analysis of the flow field revealed that the choice of the phase-field registry significantly impacts local numerical stability. The baseline momentum-based scheme (the contour on the left) occasionally generated an unphysical velocity hotspot near the free surface during violent wave impacts. The implementation of the velocity-based registry (the contour on the right) successfully removed this artifact, yielding a smoother and more physically consistent velocity field without destabilizing the interface.

Quantitative Validation:

Macroscopically, both the momentum-based and velocity-based schemes (registries) produced similar time-history profiles for surface elevation and sloshing force. The accuracy of the solver was found to be highly dependent on the excitation frequency ratio:

- **Low Frequency Ratios ($f_r \leq 1.0$):** At frequency ratios of 0.9 and 1.0, the numerical model demonstrated an excellent match with the experimental data. The solver accurately captured the phase, amplitude, and non-linear wave deformations characteristic of the resonant sloshing regime.
- **High Frequency Ratios ($f_r > 1.0$):** As the frequency ratio increased to 1.18 and 1.35, the wave dynamics became increasingly chaotic. While the solver successfully tracked the fundamental frequency of the sloshing force, the simulated surface elevation and peak forces exhibited numerically damped behavior compared to the sharp peaks observed in the experiment.

Overall, the TLD validation study confirmed that the Phase-Field method is highly capable of predicting macro-scale sloshing forces and tracking highly deformable free surfaces, particularly near the fundamental resonance frequencies most critical to industrial container design. The identification of numerical damping at higher frequencies also pro-

vides valuable insight into the resolution requirements for modeling highly turbulent, high-frequency surface disruptions in liquid metal pouring.

7.3 Development and Validation of an Immersed Boundary Lattice Boltzmann Method (IBM-LBM) Scheme

Complementing the validation of commercial tools, a parallel study focused on fundamental numerical scheme development. This involved the implementation of an Immersed Boundary Method from Balaras [88] within a Lattice Boltzmann Method (LBM) framework using a custom C++ code. It is a direct forcing approach, distinct from the fully Eulerian scheme developed earlier in this dissertation. Validating an IBM-LBM scheme provides a powerful benchmark and contrasting methodology for the fully unified approach developed for this thesis.

7.3.1 Development of a Custom IBM-LBM Solver

The development of the custom Immersed Boundary Lattice Boltzmann Method (IBM-LBM) solver relies on an explicit numerical formulation that couples the macroscopic boundary forces with the mesoscopic lattice interactions. The solver was developed from first principles, governing the fluid domain via the Lattice Boltzmann equation with a standard D2Q9 Lattice augmented with an Immersed Boundary (IB) forcing term:

$$f_i(x + e_i \delta t, t + \delta t) - f_i(x, t) = -\frac{1}{\tau} \left[f_i(x, t) - f_i^{(eq)}(x, t) \right] + F_i \delta t$$

where f_i is the particle distribution function, e_i is the discrete lattice velocity, τ is the relaxation time, $f_i^{(eq)}$ is the equilibrium distribution, and F_i is the discrete forcing term. The discrete forcing term is evaluated as:

$$F_i = \left(1 - \frac{1}{2\tau}\right) w_i \left[\frac{e_i - u}{C_s^2} + \frac{e_i \cdot u}{C_s^4} e_i \right] \cdot F$$

where w_i are the lattice weights, C_s is the lattice speed of sound, u is the macroscopic velocity, and F is the applied IB body force. The macroscopic fluid velocity is thus defined by:

$$u = \frac{1}{\rho} \sum_i e_i f_i + \frac{1}{2\rho} F \delta t$$

Following the velocity correction method proposed by Wu and Shu [89], an intermediate velocity u^* and a velocity correction term δu are introduced into the formulation:

$$u^* = \frac{1}{\rho} \sum_i e_i f_i$$

$$\delta u = \frac{1}{2\rho} F \delta t$$

Consequently, the forcing term F necessary to strictly enforce the no-slip boundary condition at the immersed solid boundary can be explicitly calculated by:

$$F = \frac{2\rho(u - u^*)}{\delta t}$$

To correctly evaluate this force at the solid interface and apply it to the surrounding fluid nodes, a sharp-interface reconstruction approach is employed, introduced by the work of Balaras [88]. The numerical algorithm at the boundary follows a 7-step procedure:

1. **Search:** Identify the Immersed Boundary (IB) points (x_{ib}, y_{ib}) within the local 3×3 grid adjacent to the interface.
2. **Normal Calculation:** At each IB point, utilize the level-set function to calculate the surface normal components n_x and n_y .

3. **Intersection Location:** Solve for the intersecting point (x_s, y_s) of the line $y = ax + b$ (where constants a and b are derived from the coordinates of the IB point and the calculated normal) and the solid interface curve, defined by $s(x, y) = 0$.
4. **Virtual Point Identification:** Find the location of a virtual point (x_v, y_v) in the fluid domain using the coordinates of the solid interface (x_s, y_s) and the IB point (x_{ib}, y_{ib}) .
5. **Fluid Velocity Interpolation:** Find the surrounding fluid nodes of (x_v, y_v) and perform interpolation to obtain the intermediate velocity $u^*(x_v, y_v)$ at that location.
6. **IB Point Velocity Update:** Interpolate using the virtual point velocity $u^*(x_v, y_v)$ and the prescribed solid boundary velocity $u(x_s, y_s)$ to get the target velocity $u(x_{ib}, y_{ib})$ at the IB point.
7. **Forcing Term Calculation:** Calculate the final IB forcing term using the equation $F = \frac{2\rho(u-u^*)}{\delta t}$ described above, completing the fluid-solid momentum coupling.

7.3.2 Validation: 2D Uniform Flow Over a Solid Cylinder

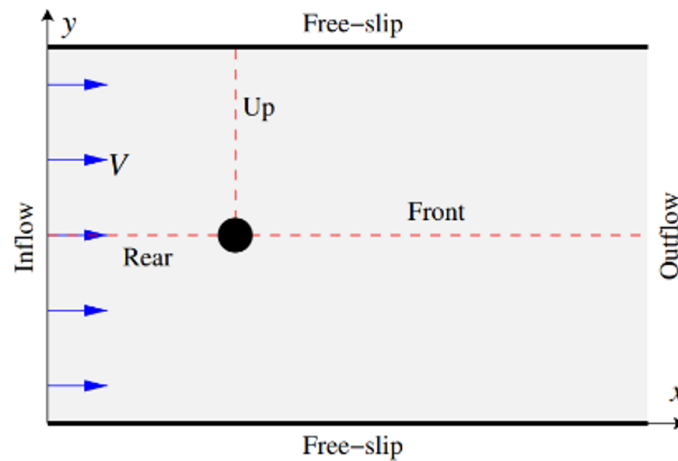


Figure 7.7: Schematic of the problem.

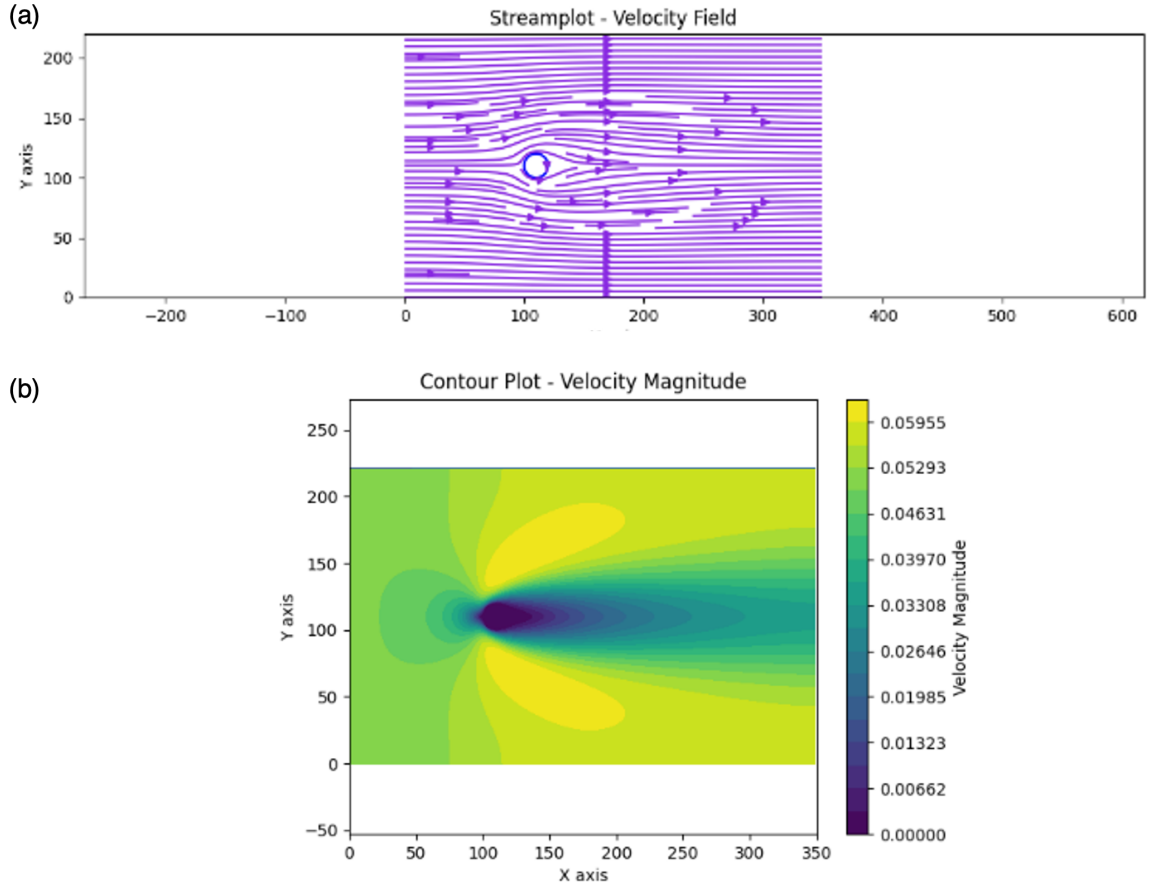


Figure 7.8: Streamlines with velocity vectors (a) and velocity magnitude contour (b) for flow around a cylinder for $Re = 10$.

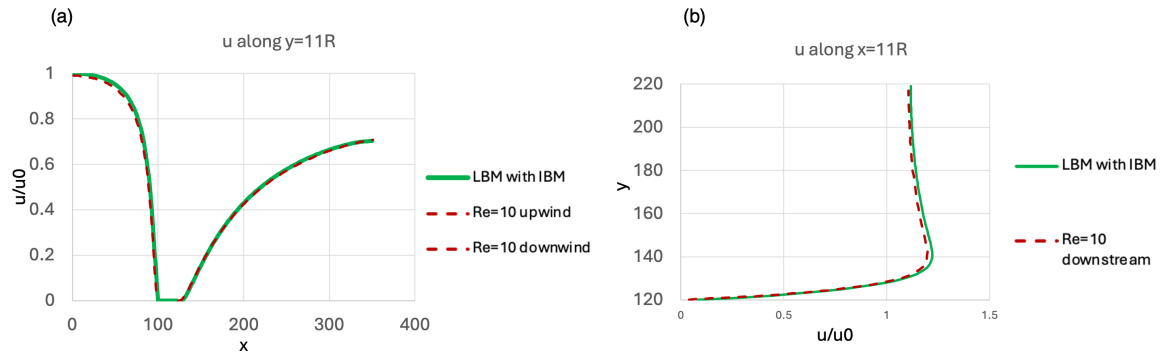


Figure 7.9: Velocity in x-direction along the x and y centerlines of the cylinder. The solid green line is the results from the implmentation in this study. The red-dashed line shows the results from De Rosis et al. [90].

To rigorously validate the newly developed C++ Immersed Boundary Lattice Boltzmann Method (IBM-LBM) solver, the canonical benchmark of a 2D uniform laminar flow over a stationary solid cylinder was simulated.

As depicted in the problem schematic in Fig. 7.7, the computational domain features a uniform continuous inflow velocity (V) from the left boundary, an outflow condition on the right, and free-slip boundary conditions on the top and bottom walls to minimize boundary confinement effects. The simulation was conducted at a Reynolds number of $Re = 10$, representing a steady, laminar flow regime.

The qualitative performance of the solver is illustrated through the macroscopic velocity field visualizations in Fig. 7.8. The streamplot of the velocity field confirms that the fluid smoothly navigates around the immersed solid boundary, demonstrating that the discrete forcing term successfully mimics the presence of the obstacle. Furthermore, the velocity magnitude contours correctly capture the expected fluid dynamics, clearly exhibiting distinct stagnation zones (characterized by near-zero velocity) at the front and rear surfaces of the cylinder, alongside symmetric flow acceleration around the upper and lower cylinder edges.

For a strict quantitative verification, the local velocity profiles were extracted and analyzed along the domain centerlines in Fig. 7.9. Specifically, the normalized horizontal velocity component (u/u_0) was plotted along the horizontal line intersecting the cylinder ($y = 11R$) and the vertical line passing through its wake ($x = 11R$). The results generated by the custom IBM-LBM implementation (represented by solid green lines) were directly compared against established benchmark data from De Rosis et al. (represented by red-dashed lines).

The plotted comparisons demonstrate exceptional agreement, with the custom solver's predictions precisely overlaying the literature data across both profiles. This strong alignment mathematically verifies the accuracy of the implemented multi-step boundary recon-

struction and forcing algorithm, confirming its capability to rigorously resolve no-slip fluid-solid interactions at the mesoscopic scale without requiring complex body-fitted meshing.

7.4 Summary and Relationship to Main Dissertation Work

The research conducted during the industrial internship at Dassault Systèmes provides a critical bridge between the theoretical numerical formulations developed in the earlier chapters of this dissertation and the stringent demands of large-scale industrial applications. By systematically evaluating commercial solvers and developing independent mesoscopic codes, this chapter establishes a robust context for the dissertation’s primary fully Eulerian fluid-structure interaction (FSI) framework.

First, the extensive validation of multiphase solvers—specifically the Phase-Field method in PowerFLOW and the particle-based method in XFlow—against complex industrial benchmarks like the FZG gear lubrication and the Tuned Liquid Damper (TLD) rigorously demonstrated the capabilities of state-of-the-art computational fluid dynamics to handle violent free-surface dynamics. The accurate prediction of topological changes and resonant sloshing forces directly parallels the fundamental physical challenges associated with liquid metal pouring and dross formation, which served as the primary motivation for this thesis (Chapter 1). Verifying that advanced phase-tracking methods can successfully capture these macroscopic phenomena validates the fundamental pursuit of high-fidelity multiphase models for casting applications.

Furthermore, the independent development and validation of the C++ Immersed Boundary Lattice Boltzmann Method (IBM-LBM) solver served as an invaluable contrasting study to the main methodology of this dissertation. While chapters 2 through 6 focus on a completely unified, fully Eulerian continuous approach using Level Set and Neo-Hookean solid representations, the IBM-LBM scheme provides a sharply different, mesoscopic per-

spective. It relies on explicit boundary tracking and discrete force coupling to represent the solid domain. Successfully implementing this algorithm and validating it against canonical aerodynamic benchmarks (such as the 2D flow over a cylinder) broadened the methodological scope of this research, proving competency in both implicit (Eulerian) and explicit (Immersed Boundary) FSI coupling techniques.

Ultimately, this industrial validation and supplementary code development reinforce the core objectives of the dissertation. They confirm the necessity of robust interface-tracking and FSI coupling mechanisms for industrial multiphase flows, thereby validating the relevance and necessity of the novel Eulerian framework developed, analyzed, and scaled to 3-D in the preceding chapters.

Chapter 8: Conclusions

8.1 Contributions

In this dissertation, a comprehensive and high-fidelity direct numerical simulation (DNS) framework was developed to model the dynamic fluid–structure interaction (FSI) of deformable elastic solids immersed in incompressible fluids. This research was fundamentally motivated by the critical need in the metal casting industry to systematically predict and control dross formation during the pouring of highly oxidative liquid metals, such as steel and aluminum alloys. By transitioning away from costly empirical trial-and-error methods, this work establishes a rigorous, physics-based computational tool capable of resolving the complex dynamics of oxide films and their influence on surrounding melt flows.

The core methodological contribution of this study, detailed in Chapters 2 and 3, is the development of a two-way, pressure-coupled FSI approach formulated entirely within a fully Eulerian framework. In the present work, we used fully Eulerian level set formulations to track both the fluid–solid interfaces and the strain history of the deformable solid. We successfully addressed the stability issues reported in He et al. (2011) [15] by using the linear extrapolation method [81] to re-initialize the dynamic grid. By tracking the strain field via this dynamic grid of reference level set functions, the traditional challenges of mesh distortion inherent to Lagrangian methods were circumvented. The deformation gradient tensor was coupled with a hyper-elastic Neo-Hookean constitutive model to re-

late strain to solid stress. A key achievement of this formulation is its unified nature; by applying a finite-width treatment of the solid–fluid interface to capture interfacial normal stress jumps, the entire domain’s pressure and velocity fields can be resolved by solving a single Poisson equation. Embedded within the FLASH 4 solver framework and utilizing the HYPRE algebraic multigrid solver, this method is robust, relatively easy to implement, and computationally inexpensive compared to conventional Lagrangian methods.

The robustness and accuracy of this numerical implementation were extensively validated by comparing our solver to literature results and exploring it across multiple dimensions:

- **2-D FSI Dynamics (Chapter 4):** A wide array of benchmark and novel scenarios were simulated, including immersed oscillating bodies, deformable structures in lid-driven cavities, and thin elastic films subjected to point forces and localized shearing. Notably, the investigation of solid filaments under periodic inflow highlighted the framework’s capability to accurately capture complex secondary flows, wave propagation, and structural phase lags characteristic of underdamped, coupled systems.
- **Axisymmetric and 3-D Extensions (Chapters 5 and 6):** The modularity of the Eulerian formulation allowed for a seamless and robust extension into multi-dimensional spaces. The simulations of periodic flows through axisymmetric vessels, as well as fully 3-D immersed oscillating spheres, cylinders, and solid films, demonstrated that the method retains its accuracy and computational efficiency when scaled to highly complex spatial domains.

8.2 Future Work

While the present multidimensional DNS framework provides a powerful tool for modeling fluid–elastic solid interactions, several avenues for future research exist to further extend its capabilities toward complete, industrial-scale predictions of dross formation:

1. **Incorporation of Plasticity and Fracture Mechanics:** The current structural model assumes a hyper-elastic, Neo-Hookean material behavior. While accurate for continuous deformation, oxide films in liquid metal casting often undergo yield, tearing, and mechanical failure when subjected to high turbulent shear. Building upon the framework’s potential to handle highly deformed solids, future work should incorporate elastoplastic material models and failure criteria into the Eulerian reference level set framework to simulate the actual breaking, rupture, and entrainment of dross particles into the melt.
2. **Complex Industrial Geometries:** With the established scalability of the FLASH 4 implementation, future studies should apply the 3-D solver to realistic casting runner and gating system geometries. The solid film is thin so development of a sub-grid scale model is necessary in order to resolve the oxide film and conduct full-scale simulations for the pouring processes. These simulations will provide direct insights into optimizing physical gating designs to minimize surface turbulence and film entrainment.
3. **Thermal Coupling and Solidification:** Metal casting is fundamentally a non-isothermal process. Coupling the current momentum transport framework with the energy equation would enable the study of temperature-dependent material properties. Furthermore, implementing phase-change models would allow the

tool to simulate how the formation and deformation of the solid film interact with the cooling and solidification of the underlying liquid metal.

Bibliography

- [1] J. Campbell. Entrainment defects. *Materials Science and Technology*, 22(2):127–145, February 2006.
- [2] X Cao and J Campbell. Oxide inclusion defects in Al-Si-Mg cast alloys. *Canadian Metallurgical Quarterly*, 44(4):14.
- [3] Shan-Guang Liu, Fu-Yang Cao, Xin-Yi Zhao, Yan-Dong Jia, Zhi-Liang Ning, and Jian-Fei Sun. Characteristics of mold filling and entrainment of oxide film in low pressure casting of A356 alloy. *Materials Science and Engineering: A*, 626:159–164, February 2015.
- [4] Shi-ping Wu, Chang-yun Li, Jing-jie Guo, Yan-qing Su, Xiu-qiao Lei, and Heng-zhi Fu. Numerical simulation and experimental investigation of two filling methods in vertical centrifugal casting. *Transactions of Nonferrous Metals Society of China*, 16(5):1035–1040, October 2006.
- [5] X. Yang, X. Huang, X. Dai, J. Campbell, and J. Tatler. Numerical modelling of entrainment of oxide film defects in filling of aluminium alloy castings. *International Journal of Cast Metals Research*, 17(6):321–331, December 2004.
- [6] Liu Cao, Fei Sun, Tao Chen, Yulong Tang, and Dunming Liao. Quantitative prediction of oxide inclusion defects inside the casting and on the walls during cast-filling processes. *International Journal of Heat and Mass Transfer*, 119:614–623, April 2018.
- [7] C. Reilly, N. R. Green, M. R. Jolly, and J. C. Gebelin. The modelling of oxide film entrainment in casting systems using computational modelling. *Applied Mathematical Modelling*, 37(18):8451–8466, October 2013.
- [8] Charles S Peskin. Flow patterns around heart valves: A numerical method. *Journal of Computational Physics*, 10(2):252–271, October 1972.

- [9] Rajat Mittal and Gianluca Iaccarino. Immersed Boundary Methods. *Annual Review of Fluid Mechanics*, 37(1):239–261, 2005.
- [10] Charles S. Peskin. The immersed boundary method. *Acta Numerica*, 11:479–517, January 2002.
- [11] Amneet Pal Singh Bhalla, Rahul Bale, Boyce E. Griffith, and Neelesh A. Patankar. A unified mathematical framework and an adaptive numerical method for fluid–structure interaction with rigid, deforming, and elastic bodies. *Journal of Computational Physics*, 250:446–476, October 2013.
- [12] Gianluca Iaccarino and Roberto Verzicco. Immersed boundary technique for turbulent flow simulations. *Appl. Mech. Rev*, 56(3):331–347, May 2003. Publisher: American Society of Mechanical Engineers Digital Collection.
- [13] Jianming Yang and Elias Balaras. An embedded-boundary formulation for large-eddy simulation of turbulent flows interacting with moving boundaries. *Journal of Computational Physics*, 215(1):12–40, June 2006.
- [14] Hong Zhao, Jonathan B. Freund, and Robert D. Moser. A fixed-mesh method for incompressible flow–structure systems with finite solid deformations. *Journal of Computational Physics*, 227(6):3114–3140, March 2008.
- [15] Ping He and Rui Qiao. A full-Eulerian solid level set method for simulation of fluid–structure interactions. *Microfluid Nanofluid*, 11(5):557, May 2011.
- [16] Boris Valkov, Chris H. Rycroft, and Ken Kamrin. Eulerian Method for Multi-phase Interactions of Soft Solid Bodies in Fluids. *J. Appl. Mech*, 82(4), April 2015.
- [17] Suhas S. Jain, Ken Kamrin, and Ali Mani. A conservative and non-dissipative Eulerian formulation for the simulation of soft solids in fluids. *Journal of Computational Physics*, 399:108922, December 2019.
- [18] Ryan J. Larsen, Michael D. Dickey, George M. Whitesides, and David A. Weitz. Viscoelastic properties of oxide-coated liquid metals. *Journal of Rheology*, 53(6):1305–1326, November 2009.
- [19] S. W. Gong and E. Klaseboer. Interaction between collapsing bubble and viscoelastic solid: Numerical modelling and simulation. *Applied Mathematical Modelling*, 40(7):4746–4764, April 2016.
- [20] G. Ryskin and L. G. Leal. Numerical solution of free-boundary problems in fluid mechanics. Part 1. The finite-difference technique. *J. Fluid Mech.*, 148:1–17, November 1984.

- [21] G. Ryskin and L. G. Leal. Numerical solution of free-boundary problems in fluid mechanics. Part 2. Buoyancy-driven motion of a gas bubble through a quiescent liquid. *J. Fluid Mech.*, 148:19–35, November 1984.
- [22] J. Feng, H. H. Hu, and D. D. Joseph. Direct simulation of initial value problems for the motion of solid bodies in a Newtonian fluid Part 1. Sedimentation. *J. Fluid Mech.*, 261:95–134, February 1994.
- [23] J. Feng, H. H. Hu, and D. D. Joseph. Direct simulation of initial value problems for the motion of solid bodies in a Newtonian fluid. Part 2. Couette and Poiseuille flows. *J. Fluid Mech.*, 277:271–301, October 1994.
- [24] B.D. Nichols, C.W. Hirt, R.S. Hotchkiss, and NM (USA) Los Alamos Scientific Lab. Sola-vof: a solution algorithm for transient fluid flow with multiple free boundaries. 1980.
- [25] David Youngs. *Time-Dependent Multi-material Flow with Large Fluid Distortion*, volume 24, pages 273–285. 01 1982.
- [26] Denis Gueyffier, Jie Li, Ali Nadim, Ruben Scardovelli, and Stéphane Zaleski. Volume-of-Fluid Interface Tracking with Smoothed Surface Stress Methods for Three-Dimensional Flows. *Journal of Computational Physics*, 152(2):423–456, July 1999.
- [27] G Weymouth, D G Dommermuth, and K Hendrickson. Advances in Cartesian-grid methods for computational ship hydrodynamics.
- [28] G.D. Weymouth and Dick K.-P. Yue. Conservative Volume-of-Fluid method for free-surface simulations on Cartesian-grids. *Journal of Computational Physics*, 229(8):2853–2865, April 2010.
- [29] James Edward Pilliod and Elbridge Gerry Puckett. Second-order accurate volume-of-fluid algorithms for tracking material interfaces. *Journal of Computational Physics*, 199(2):465–502, September 2004.
- [30] Stéphane Popinet. An accurate adaptive solver for surface-tension-driven interfacial flows. *Journal of Computational Physics*, 228(16):5838–5866, September 2009.
- [31] Stanley Osher and James A Sethian. Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations. *Journal of Computational Physics*, 79(1):12–49, November 1988.
- [32] Stanley Osher and R. Fedkiw. *The Level Set Methods and Dynamic Implicit Surfaces*, volume 57. 05 2004.

- [33] Myungjoo Kang, Ronald P. Fedkiw, and Xu-Dong Liu. A Boundary Condition Capturing Method for Multiphase Incompressible Flow. *Journal of Scientific Computing*, 15(3):323–360, September 2000.
- [34] Guang-Shan Jiang and Danping Peng. Weighted ENO Schemes for Hamilton–Jacobi Equations. *SIAM J. Sci. Comput.*, 21(6):2126–2143, January 2000. Publisher: Society for Industrial and Applied Mathematics.
- [35] Mark Sussman and Emad Fatemi. An efficient, interface-preserving level set re-distancing algorithm and its application to interfacial incompressible fluid flow. *SIAM Journal on Scientific Computing*, 20(4):1165–1191, 1999.
- [36] Giovanni Russo and Peter Smereka. A Remark on Computing Distance Functions. *Journal of Computational Physics*, 163(1):51–67, September 2000.
- [37] Antoine Du Ch  n  , Chohong Min, and Fr  d  ric Gibou. Second-Order Accurate Computation of Curvatures in a Level Set Framework Using Novel High-Order Reinitialization Schemes. *J Sci Comput*, 35(2-3):114–131, June 2008.
- [38] Zhipeng Qin, Keegan Delaney, Amir Riaz, and Elias Balaras. Topology preserving advection of implicit interfaces on Cartesian grids. *Journal of Computational Physics*, 290:219–238, June 2015.
- [39] Mark Sussman and Elbridge Gerry Puckett. A Coupled Level Set and Volume-of-Fluid Method for Computing 3D and Axisymmetric Incompressible Two-Phase Flows. *Journal of Computational Physics*, 162(2):301–337, August 2000.
- [40] C. Peskin. Numerical analysis of blood flow in the heart. *Journal Of Computational Physics*, 25:220–252, 1977.
- [41] C. Peskin and D. McQueen. A three-dimensional computational method for blood flow in the heart i. immersed elastic fibers in a viscous incompressible fluid. *Journal Of Computational Physics*, 81:372–405, 1989.
- [42] D. McQueen and C. Peskin. A three-dimensional computational method for blood flow in the heart. ii. contractile fibers. *Journal Of Computational Physics*, 82:289–297, 1989.
- [43] C. Peskin and B. Printz. Improved volume conservation in the computation of flows with immersed elastic boundaries. *Journal Of Computational Physics*, 105:33–46, 1993.
- [44] P. Crosetto, P. Reymond, S. Deparis, D. Kontaxakis, N. Stergiopoulos, and A. Quarteroni. Fluid–structure interaction simulation of aortic blood flow. *Computers & Fluids*, 43:46–57, 2011.

- [45] J. Freund. Numerical simulation of flowing blood cells. *Annual Review Of Fluid Mechanics*, 46:67–95, 2014.
- [46] R. Torii, M. Oshima, T. Kobayashi, K. Takagi, and T. Tezduyar. Influence of wall elasticity on image-based blood flow simulations. *Nippon Kikai Gakkai Ronbunshu, A Hen/Transactions Of The Japan Society Of Mechanical Engineers, Part A*, 70:1224–1231, 2004.
- [47] L. Zhu and C. Peskin. Interaction of two flapping filaments in a flowing soap film. *Physics Of Fluids*, 15:1954–1960, 2003.
- [48] G. Constantinescu and K. Squires. Les and des investigations of turbulent flow over a sphere at $re = 10,000$. *Flow, Turbulence And Combustion*, 70:267–298, 2003.
- [49] G. Kalitzin and G. Iaccarino. Turbulence modeling in an immersed-boundary rans method. *CTR Annual Briefs*, pages 415–426, 2002.
- [50] A. Korobenko, M. Hsu, I. Akkerman, J. Tippmann, and Y. Bazilevs. Structural mechanics modeling and fsi simulation of wind turbines. *Mathematical Models And Methods In Applied Sciences*, 23:249–272, 2013.
- [51] P. Martínez-Ferrer, L. Qian, Z. Ma, D. Causon, and C. Mingham. An efficient finite-volume method to study the interaction of two-phase fluid flows with elastic structures. *Journal Of Fluids And Structures*, 83:54–71, 2018.
- [52] M. Gazzola, W. Van Rees, and P. Koumoutsakos. C-start: Optimal start of larval fish. *Journal Of Fluid Mechanics*, 698:5–18, 2012.
- [53] R. Bale, I. Neveln, A. Bhalla, M. MacIver, and N. Patankar. Convergent evolution of mechanically optimal locomotion in aquatic invertebrates and vertebrates. *PLoS Biology*, 13:e1002123, 2015.
- [54] B. Sprinkle, R. Bale, A. Bhalla, M. MacIver, and N. Patankar. Hydrodynamic optimality of balistiform and gymnotiform locomotion. *European Journal Of Computational Mechanics*, 26:31–43, 2017.
- [55] A. Hoover, B. Griffith, and L. Miller. Quantifying performance in the medusan mechanospace with an actively swimming three-dimensional jellyfish model. *Journal Of Fluid Mechanics*, 813:1112–1155, 2017.
- [56] A. Hoover, A. Porras, and L. Miller. Pump or coast: the role of resonance and passive energy recapture in medusan swimming performance. *Journal Of Fluid Mechanics*, 863:1031–1061, 2019.
- [57] B. E. Griffith and N. A. Patankar. Immersed methods for fluid–structure interaction. *Annual Review Of Fluid Mechanics*, 52:421–448, 2020.

- [58] Y. Bao, A. Donev, B. Griffith, D. McQueen, and C. Peskin. An immersed boundary method with divergence-free velocity interpolation and force spreading. *Journal Of Computational Physics*, 347:183–206, 2017.
- [59] O. Maxian, A. Kassen, and W. Strychalski. A continuous energy-based immersed boundary method for elastic shells. *Journal Of Computational Physics*, 371:333–362, 2018.
- [60] L. Ménez, P. Parnaudeau, M. Beringhier, and E. Da Silva. Assessment of volume penalization and immersed boundary methods for compressible flows with various thermal boundary conditions. *Journal Of Computational Physics*, 493:112465, 2023.
- [61] I. Lee and H. Choi. A discrete-forcing immersed boundary method for the fluid–structure interaction of an elastic slender body. *Journal Of Computational Physics*, 280:529–546, 2015.
- [62] Lucy Zhang, Axel Gerstenberger, Xiaodong Wang, and Wing Kam Liu. Immersed finite element method. *Computer Methods in Applied Mechanics and Engineering*, 193(21-22):2051–2067, 2004.
- [63] Xingshi Wang and Lucy T. Zhang. Modified immersed finite element method for fully-coupled fluid–structure interactions. *Computer Methods in Applied Mechanics and Engineering*, 267:150–169, 2013.
- [64] Wing Kam Liu, Do Wan Kim, and Shaoqiang Tang. Mathematical foundations of the immersed finite element method. *Computational Mechanics*, 39(3):211–222, 2006.
- [65] R Glowinski. A distributed Lagrange multiplier/fictitious domain method for particulate flows. *International Journal of Multiphase Flow*, 1999.
- [66] N A Patankar, P Singh, D D Joseph, and R Glowinski. A new formulation of the distributed Lagrange multiplier/ fictitious domain method for particulate flows. *International Journal of Multiphase Flow*, 2000.
- [67] Zhaosheng Yu. A DLM/FD method for fluid/flexible-body interactions. *Journal of Computational Physics*, 207(1):1–27, 2005.
- [68] T. Dunne. An eulerian approach to fluid–structure interaction and goal-oriented mesh adaptation. *International Journal For Numerical Methods In Fluids*, 51:1017–1039, 2006.
- [69] T. Richter. A fully eulerian formulation for fluid–structure-interaction problems. *Journal Of Computational Physics*, 233:227–240, 2013.

- [70] T. Wick. Fully eulerian fluid–structure interaction for time-dependent problems. *Computer Methods In Applied Mechanics And Engineering*, 255:14–26, 2013.
- [71] Ken Kamrin, Chris H. Rycroft, and Jean-Christophe Nave. Reference map technique for finite-strain elasticity and fluid–solid interaction. *Journal of the Mechanics and Physics of Solids*, 60(11):1952–1969, November 2012.
- [72] Chris H. Rycroft, Chen-Hung Wu, Yue Yu, and Ken Kamrin. Reference map technique for incompressible fluid–structure interaction. *J. Fluid Mech.*, 898:A9, September 2020.
- [73] Y. Lin, N. Derr, and C. H. Rycroft. Eulerian simulation of complex suspensions and biolocomotion in three dimensions. *Proceedings Of The National Academy Of Sciences*, 119:e2105338118, 2022.
- [74] Kazuyasu Sugiyama, Satoshi Ii, Shintaro Takeuchi, Shu Takagi, and Yoichiro Matsumoto. A full Eulerian finite difference approach for solving fluid–structure coupling problems. *Journal of Computational Physics*, 230(3):596–627, February 2011.
- [75] M. Rosti and L. Brandt. Numerical simulation of turbulent channel flow over a viscous hyper-elastic wall. *Journal Of Fluid Mechanics*, 830:708–735, 2017.
- [76] X. Mao and R. Jaiman. An interface and geometry preserving phase-field method for fully eulerian fluid-structure interaction. *Journal Of Computational Physics*, 476:111903, 2023.
- [77] B. Rath, X. Mao, and R. Jaiman. An interface preserving and residual-based adaptivity for phase-field modeling of fully eulerian fluid-structure interaction. *Journal Of Computational Physics*, 488:112188, 2023.
- [78] Paola Causin, Jean-Frédéric Gerbeau, and Fabio Nobile. Added-mass effect in the design of partitioned algorithms for fluid–structure problems. *Computer Methods in Applied Mechanics and Engineering*, 194(42-44):4506–4527, 2005.
- [79] John N. Newman. *Marine Hydrodynamics*. MIT Press, Cambridge, MA, 1977.
- [80] Christian Förster, Wolfgang A Wall, and Ekkehard Ramm. Artificial added mass instabilities in sequential staggered coupling of nonlinear fluid-structure interaction problems. *Computer Methods in Applied Mechanics and Engineering*, 196(7-8):1278–1293, 2007.
- [81] Tariq D. Aslam. A partial differential equation approach to multidimensional extrapolation. *Journal of Computational Physics*, 193(1):349–355, January 2004.

- [82] S. Osher, R. Fedkiw, and K. Piechor. Level set methods and dynamic implicit surfaces. *Appl. Mech. Rev.*, 57:B15–B15, 2004.
- [83] Moon Soo Lee, Amir Riaz, and Vikrant Aute. Direct numerical simulation of incompressible multiphase flow with phase change. *Journal of Computational Physics*, 344:381–418, September 2017.
- [84] Akash Dhruv, Elias Balaras, Amir Riaz, and Jungho Kim. A formulation for high-fidelity simulations of pool boiling in low gravity. *International Journal of Multiphase Flow*, 120:103099, November 2019.
- [85] K. Delaney. *An adaptive mesh refinement solver for multiphase incompressible flows with large density ratios*. PhD thesis, 2014.
- [86] G. Holzapfel. *Nonlinear Solid Mechanics: A Continuum Approach for Engineering*. John Wiley & Sons, Ltd., West Sussex, England, 2000.
- [87] R. Ogden. *Non-linear elastic deformations*. Courier Corporation, 1997.
- [88] Elias Balaras. Modeling complex boundaries using an external force field on fixed Cartesian grids in large-eddy simulations. *Computers & Fluids*, 33(3):375–404, March 2004.
- [89] J. Wu and C. Shu. Implicit velocity correction-based immersed boundary-lattice boltzmann method and its applications. *Journal of Computational Physics*, 228(6):1963–1979, 2009.
- [90] Alessandro De Rosis, Stefano Ubertini, and Francesco Ubertini. A comparison between the interpolated bounce-back scheme and the immersed boundary method to treat solid boundary conditions for laminar flows in the lattice boltzmann framework. *J. Sci. Comput.*, 61(3):477–489, December 2014.