AMRIT

ANNALS OF MULTIDISCIPLINARY RESEARCH, INNOVATION AND TECHNOLOGY (AMRIT)

(A peer-reviewed open access multidisciplinary journal)

www.adtu.in/amrit

REVIEW ARTICLE

ENGINEERING

An Overview of Fluid-Structure Interaction: Modelling, Finite Element Method and Applications

Ananya Kalita[1*], T.V.V.L.N. Rao[1]

[1] Faculty of Engineering, Assam down town University, Panikhaiti, Guwahati, Assam-781026, India

[*]Corresponding author: Ananya Kalita, *Email: ananya.kalita@adtu.in*

Article Chronicle: Received: 13/06/23 Accepted: 08/12/2023 Published: 29/12/2023

Abstract

Fluid-structure interaction (FSI) is a highly intricate phenomenon that arises when a fluid interacts with a flexible solid. Accurately predicting FSI behaviour holds immense significance across multiple engineering domains, encompassing the design of aircraft and wind turbines, as well as the development of biomedical implants and cardiovascular modelling. Finite element modelling serves as a formidable tool for simulating FSI problems due to its capacity to precisely depict both the fluid and solid domains, as well as their intricate interplay. This comprehensive review aims to present an overview of the utilization of finite element modelling in tackling FSI problems. It encompasses an exploration of the governing equations, numerical methodologies, and diverse applications in various fields. Additionally, notable advancements in FSI modelling, such as the integration of reduced order models and machine learning techniques, will be emphasized. Lastly, this paper will delve into the challenges faced and future prospects in the realm of FSI modelling, emphasizing the pressing need for more accurate constitutive models.

Keywords: *Fluid-structure interaction, Navier-Stokes equations, Finite element modelling, Computational fluid dynamics, Reduced order models*

1 Introduction

Fluid-structure interaction (FSI) refers to the intricate interplay between fluid flow and deformable structures, a phenomenon prevalent in various engineering and scientific disciplines. Understanding and accurately modeling FSI is crucial for predicting the behavior and performance of systems such as aircraft wings, offshore structures, biomedical implants, and environmental flows. Finite element modeling (FEM) has emerged as a powerful computational technique for analyzing FSI problems, offering a flexible and efficient approach to simulate and analyze complex fluid-structure interactions. The primary objective of this structured review is to provide a comprehensive overview of the current state of FEM for FSI modeling and explore the future scope of its applications. By examining the fundamental principles of FEM, discussing key FEM techniques for FSI modeling, and addressing the challenges and limitations, this review aims to shed light on the advancements and potential areas of improvement in this field. The finite element method (FEM) is a numerical technique used to approximate the solutions to partial differential equations governing the behavior of fluids and structures. It divides the problem domain into smaller subdomains called finite elements, enabling the modeling of complex geometries and variable material properties. FEM has gained popularity in FSI due to its ability to handle nonlinearities, adaptivity, and the incorporation of fluid and structural mechanics into a unified framework. The review will delve into different FEM techniques commonly employed for FSI modeling, including the partitioned approach, monolithic approach, and immersed boundary method (IBM). The partitioned approach treats fluid and structure separately, exchanging information through an interface, while the monolithic approach considers fluid and structure as a single system, solving the coupled equations simultaneously. The IBM combines the advantages of both approaches by employing an overlay grid to represent the fluid-structure interface. While FEM offers promising capabilities for FSI modeling, it is not without challenges and limitations. The review will address key issues such as fluid-structure coupling methods, mesh generation and remeshing techniques, computational costs, and validation against experimental data. Understanding and overcoming these challenges will be crucial to advance the accuracy, efficiency, and reliability of FEM for FSI simulations.

2 Fluid-Structure Interaction

Fluid-structure interaction (FSI) refers to the mutual influence between a fluid and a neighboring solid structure as they interact and affect each other's behavior. FSI problems have significant implications in various engineering applications, including aerospace, civil engineering, and biomedical engineering (1). Finite element modeling (FEM) has emerged as a widely adopted approach for simulating and analyzing FSI phenomena. (2) emphasize that FEM provides a robust and effective tool for investigating complex interactions between fluids and structures, enabling successful applications in diverse FSI scenarios. Throughout the 1980s and 1990s, FEM for FSI continued to progress, with an increasing emphasis on practical applications (1; 2). FEM facilitates the accurate discretization of both fluid and structural domains and enables the coupling between these domains, making it well-suited for FSI modeling. The development of FEM for FSI has a rich historical background. Early applications of FEM to FSI can be traced back to the 1960s when researchers utilized FEM to analyze fluid-filled containers subjected to external pressure (3). Subsequently, in the 1970s, FEM was further refined for FSI problems, with significant contributions from researchers like Bathe and Wilson (4) and Donea and Huerta (5). These early works focused on developing numerical methods for solving FSI problems and demonstrating the potential of FEM in modeling FSI. FEM was successfully applied to diverse FSI problems, including aeroelasticity, biomechanics, and civil engineering. Recent years have witnessed notable advancements in FSI modeling, with researchers introducing novel methods and tools to enhance the accuracy and efficiency of FSI simulations. Tian et al. (6) reported a method for three-dimensional FSI simulations combining an existing immersed-boundary flow solver with a nonlinear finite-element solid-mechanics solver. Both geometric and material nonlinearity are incorporated in the solver, while the FSI is handled through a coupling and partitioned approach. Griffith and Patankar (7) reviewed immersed methods using integral operators connecting the Eulerian and Lagrangian frames for structures The jump conditions along fluid–structure interfaces are considered in the formulations of immersed methods. The immersed method formulations demonstrate their effectiveness in applications of biological simulations at Reynolds numbers up to approximately 20,000. Gholampour (8) evaluated the details of the hydrodynamic parameters changes of the cerebrospinal fluid (CSF) flow during the treatment process of the NCH patients. The 3D fluid-structure interaction (FSI) modelling was utilized for simulation to evaluate the biofluid parameters of CSF returns after non-communicating hydrocephalus (NCH) patients' healing to the normal conditions. Qiao et al. (9) evaluated the efficacy of thoracic endovascular aortic repair (TEVAR) for acute type B aortic dissection. It has been demonstrated that the coverage of left subclavian artery (LSA) has a considerable impact on the hemodynamic parameters. A single-phase non-Newtonian model is coupled with fluid–structure interaction (FSI) technique to simulate blood flow in an acute type B aortic dissection. Emerging trends such as multi-physics and multi-scale modelling, integration with machine learning and optimization techniques, advancements in parallel computing and GPU acceleration, and the application of FEM to specific domains like biomedical, aerospace, and civil engineering will be discussed. These areas present exciting opportunities for researchers and practitioners to push the boundaries of FSI modelling and unlock new insights. In conclusion, this structured review aims to provide a comprehensive understanding of FEM for FSI modelling, addressing its current state, advancements, challenges, and future scope of application. By synthesizing existing knowledge and identifying areas for further research and development, this review seeks to contribute to the continued progress in fluid-structure interaction modelling and its practical applications in diverse fields.

3 Modelling

3.1 Fluid Modelling

The several approaches to modelling fluids include inviscid and viscous models. In general, inviscid models neglect the effects of viscosity in fluids, while viscous models include these effects.

1. **Inviscid models**: Inviscid models (10; 11)assume that fluids are perfect and do not exhibit any viscosity. These models are often used to study the flow of fluids at high speeds or in situations where viscosity is negligible. One example of an inviscid model is the Euler equations, which describe the conservation of mass, momentum, and energy in a fluid without considering viscosity. However, inviscid models can be limited in their ability to accurately model fluid flow in situations where viscosity plays a significant role.

2. **Viscous models**: Viscous models (11; 12) take into account the effects of viscosity in fluids, and are typically used to study fluid flow at lower speeds or in situations where viscosity is important. Examples of viscous models include the Navier-Stokes equations, which describe the motion of viscous fluids, and the Reynolds-averaged Navier-Stokes (RANS) equations, which are commonly used in turbulence modelling. Viscous models are generally more accurate than inviscid models, but can be computationally expensive and may require additional simplifying assumptions to make them tractable.

3. **Hybrid models**: Hybrid models (10; 12) combine aspects of both inviscid and viscous models, and are often used to balance computational efficiency and accuracy. One example of a hybrid model is the Lattice Boltzmann method, which uses a simplified model of fluid dynamics to simulate viscous effects in a computationally efficient manner. However, hybrid models can be limited in their ability to accurately capture complex fluid flow phenomena. 1 describes different approaches to modelling fluids, along with their advantages and disadvantages.

Table 1: Approaches and their Merits and Demerits

Model type	Advantages	Disadvantages
Inviscid models	Computationally efficient.	Cannot accurately model flows with significant viscosity.
	Accurate for flows with negligible viscosity.	Limited in their ability to capture complex flow phenomena.
Viscous models	More accurate than inviscid models.	Can be computationally expensive.
	Can model flows with significant viscosity.	May require simplifying assumptions to make them tractable.
Hybrid models	Balance computational efficiency and accuracy.	Limited in their ability to accurately capture complex flow phenomena.
	Can capture some viscous effects in a computationally efficient manner.	-

Overall, research on FEM implementation in fluid modelling has made significant progress. Researchers continue to refine and enhance FEM techniques to improve the accuracy, efficiency, and applicability of fluid simulations. The choice between inviscid, viscous, or hybrid models depends on the specific requirements of the problem at hand, including the desired accuracy, computational resources, and the importance of capturing complex flow phenomena accurately.

3.2 Turbulence Modelling

Turbulence is a complex and chaotic phenomenon that can significantly impact fluid flows. Turbulence modelling is the process of using mathematical models to represent the effects of turbulence in fluid flows. In the context of finite element modelling, turbulence models are typically used to provide closure for the Reynolds-averaged Navier-Stokes (RANS) equations, which are a set of equations that describe the motion of viscous fluids (13). There are several types of turbulence models, including eddy viscosity models, Reynolds stress models, and large eddy simulation (LES) models. Eddy viscosity models are the simplest and most commonly used type of turbulence model. These models assume that turbulence can be represented as a turbulent viscosity, which is added to the molecular viscosity in the RANS equations. Reynolds stress models are more complex than eddy viscosity models, and explicitly model the effects of turbulent fluctuations on the fluid flow. LES models are the most computationally intensive type of turbulence model, and directly simulate the larger-scale turbulent structures while modelling the smaller-scale structures (13; 14).

3.3 Structure Modelling

1. **Linear models**: Linear models assume that the relationship between inputs and outputs is linear. The advantage of linear models is their simplicity and ease of use. However, they may not accurately capture nonlinear behaviour and material properties.

2. **Nonlinear models**: Nonlinear models allow for more accurate representation of nonlinear behaviour and material properties. Chan et al. (15) presented a review and summary of the non-linear analysis and design of steel frames formed by joining together one-dimensional members Buyukozturk (16) developed the generalized yield and failure criteria for the nonlinear finite element analysis. The incremental stress-strain relationships of nonlinearity are established and the method is applied on sample reinforced concrete analysis. The advantage of nonlinear models is their accuracy and ability to capture complex behaviour. However, they can be computationally intensive and require more advanced modelling techniques.

3. **Plasticity models**: Plasticity models are used to analyse the behaviour of structures under large deformations and yielding. Chen and Han (17) discussed the use of plasticity models for structural analysis. The advantage of plasticity models is their ability to accurately capture plastic deformation behavior. However, they can be computationally intensive and require advanced numerical techniques.

4. **Fibre-reinforced models**: Fibre-reinforced models are used to simulate the behaviour of structures containing fibre reinforcement. Anas et al. (18) reviewed the effects of the addition of fibers on the performance of concrete. The demand to upgrade the concrete with high strength and crack resistance led to the development of fiber-reinforced concrete. Poon et al. (19) presented the effects of elevated temperatures on the stiffness and toughness of high-performance concretes. Due to the elevated temperatures, the loss of stiffness was much quicker than the loss in toughness. The steel-fiber-reinforced concretes showed the higher toughness compared to polypropylene-fiber-reinforced concretes after the high-temperature exposure. The advantage of fibre-reinforced models is their ability to improve the tensile strength and toughness of materials. However, they can be more expensive and difficult to work with than traditional materials.

5. **Iterative solvers**: Iterative solvers are used to solve the large system of equations that arise in structural analysis. Wohlmuth (20) dealt with relatively new discretization techniques for the numerical approximations. The amenability to high performance computations have led to the use of powerful and flexible tools in many interesting large-scale applications. The advantage of iterative solvers is their ability to

solve large systems of equations efficiently. However, they may not always converge to a solution and require careful tuning. Overall, the choice of modelling approach depends on the specific problem and the level of accuracy required. Linear models are simple and easy to use, but may not capture complex behavior. Nonlinear models, plasticity models, and fibre-reinforced models offer more accuracy but can be computationally intensive. Iterative solvers can be used to solve large systems of equations efficiently, but require careful tuning.

3.4 Structural Damping

Structural damping is an important factor to consider in finite element modelling of structures. It represents the ability of a structure to dissipate energy through internal friction and other mechanisms. In this answer, we will discuss the implementation of structural damping in finite element modelling based on the research papers provided earlier. Argyris and Symeonidis (21) presented stability behaviour of elastic structures subject to nonconservative forces using nonlinear finite element analysis. Wong et al. (22) highlighted that a proper choice of the time integration schemes in addition to material and its geometric properties influence the condition of fluid-structure interaction (FSI) coupling stability. Oñate (23) discussed the use of viscoelastic damping models in finite element analysis. These models incorporate a frequency-dependent damping function that varies with the material properties and geometry of the structure. Viscoelastic damping models can provide a more accurate representation of damping behavior than the Rayleigh damping model, but can be more computationally intensive. Another approach to implementing structural damping in finite element models is to use modal damping models. Modal damping models assume that the damping forces are proportional to the modal displacements and velocities of the structure. Modal damping models can provide a more accurate representation of the damping behaviour of complex structures, but can also be more computationally intensive than simpler models. Overall, the choice of damping model depends on the specific problem and the level of accuracy required. The Rayleigh damping model is simple and easy to use, but may not accurately capture the damping behaviour of real structures. More advanced damping models, such as viscoelastic and modal damping models, can provide more accurate representations of damping behaviour, but can be more computationally intensive. It is important for researchers and engineers to carefully consider the trade-offs between accuracy and computational efficiency when selecting a damping model for finite element analysis.

3.5 Governing Equations

Belytschko et al. (24) provide an overview of nonlinear finite element methods for continua and structures, which can be used to solve fluid-structure interaction problems. The authors discuss the governing equations of solid mechanics, including the equations of motion and constitutive relations, as well as the equations of fluid mechanics, including the Navier-Stokes equations. They also describe methods for coupling the equations of fluid and solid mechanics, such as the arbitrary Lagrangian-Eulerian method. Bathe and Zhang (25) describe finite element developments for general fluid flows with structural interactions, including fluid-structure interaction problems. They discuss the equations governing the fluid dynamics and structural mechanics, as well as the methods for coupling these equations. The author describes several numerical techniques for solving fluid-structure interaction problems, including the finite element method and the smoothed particle hydrodynamics method. Donea et al. (26) present an arbitrary Lagrangian-Eulerian finite element method for transient dynamic fluid-structure interactions. They mention the equations governing the motion of fluids and solids, including the Navier-Stokes equations and the equations of motion for solids. The authors also discuss the coupling between fluid and solid mechanics, and present numerical results for several fluid-structure interaction problems. Felippa et al. (27) reviewed the use of partitioned analysis for the analysis of coupled dynamical systems including fluid-structure interaction problems. They describe the equations governing the motion of fluids and solids, as well as the methods for coupling these equations. The authors also discuss several numerical techniques for solving fluid-structure interaction problems, including the partitioned coupling method and the iterative sub structuring method. Jayanti (28) derived the basic equations and examined the appropriate boundary and initial conditions that govern the flow of fluids. The equations that govern fluid and solid mechanics are fundamental to understanding fluid-structure interaction problems. In general, the equations of motion for fluids and solids can be written in the form of partial differential equations (PDEs) Bathe and Zhang (25).

For fluid mechanics, the Navier-Stokes equations are a set of PDEs that describe the motion of viscous fluids. They are based on the conservation of mass, momentum, and energy (26; 29). For solid mechanics, the equations of motion can be written in terms of the stress-strain relationship of the material. The equations generally take the form of second-order PDEs.

In fluid-structure interaction problems, the equations of motion for fluids and solids need to be coupled to account for the interaction between the two media. There are several methods for coupling these equations, including the Arbitrary Lagrangian-Eulerian (ALE) method, the Smoothed Particle Hydrodynamics (SPH) method, and the Partitioned Method. The ALE method involves a transformation of the fluid mesh from the Eulerian to the Lagrangian frame of reference. This allows for the fluid to deform along with the structure, while the equations for the solid remain in the Eulerian frame. The ALE method is particularly useful for problems involving large deformations or complex geometries. The SPH method is a meshless technique that uses particles to represent both the fluid and the solid. The equations of motion are solved for each particle, and the forces between particles are computed to account for the fluid-structure interaction. The SPH method is particularly useful for problems involving free surfaces or fluid fragmentation (30).

Based on these works, we can observe that there is a common focus on understanding and modeling the governing equations for fluid and solid mechanics in FSI. Various numerical techniques, such as the finite element method, SPH, ALE method, and partitioned method, are discussed to solve FSI problems. Differences arise in the specific approaches, such as the emphasis on nonlinear finite element methods, transient dynamic interactions, or partitioned analysis. The choice of method depends on factors like the complexity of the problem, accuracy requirements, and computational efficiency.

Overall, these works collectively contribute to the understanding and advancement of FSI modeling using finite element methods, providing researchers with valuable insights and techniques to simulate and analyze complex FSI problems.

In finite element modelling, turbulence models can be implemented using a variety of techniques. One common approach is to use a standard k-epsilon model, which is an eddy viscosity model that uses two transport equations to model the turbulent kinetic energy and dissipation rate. Another approach is to use a Reynolds stress model, which requires the solution of additional transport equations for the Reynolds stresses. LES models can also be used, but these models are typically only used for simulations where resolving the large-scale turbulence structures is necessary.

The Reynolds-averaged Navier-Stokes (RANS) equations: Turbulent flows are generally described using the RANS equations. The Navier-Stokes equations can be approximated with the help of these equations using approximations based on understanding of the characteristics of flow turbulence. These equations can be stated in Einstein notation in Cartesian coordinates for a stationary flow of an incompressible Newtonian fluid (31).

The Reynolds stress tensor captures the turbulent stresses in the flow and depends on the modelling approach used. It is important to note that these equations provide a mathematical representation of the flow and are typically solved numerically using appropriate discretization techniques, such as finite difference, finite volume, or finite element methods. In addition, turbulence models are required to close the system of equations and provide closure for the Reynolds stress tensor. Various turbulence models, such as the Reynolds-averaged Navier-Stokes (RANS) models, are available to approximate the turbulent flow behaviour based on the understanding of turbulence characteristics. These equations, along with suitable boundary conditions and turbulence models, form the basis for simulating and analysing turbulent flows in engineering applications, allowing engineers to understand and predict complex fluid behaviour for various practical scenarios.

1. **The k-epsilon model**: The k-epsilon model is a popular eddy viscosity turbulence model that uses two transport equations to model the turbulent kinetic energy and the rate of dissipation of turbulent kinetic energy (32). The transport equation for turbulent dissipation rate describes the rate at which turbulent kinetic energy is dissipated in the flow. It also considers advection, diffusion, production, and dissipation terms. The production term is related to the turbulent kinetic energy and represents the transfer of energy between different scales of turbulence. The dissipation term represents the dissipation of turbulent energy at small scales and is influenced by the local flow conditions. These equations, along with suitable initial and boundary conditions, form the basis of the k-epsilon turbulence model. By solving these transport equations, engineers can estimate the distribution of turbulent kinetic energy and turbulent dissipation rate within a flow field, allowing for predictions of turbulence characteristics and improved modelling of turbulent flows in various engineering applications.

2. **Reynolds stress models**: Reynolds stress models are more complex than eddy viscosity models, and explicitly model the effects of turbulent fluctuations on the fluid flow. They require the solution of additional transport equations for the Reynolds stresses, which are the products of the fluctuating velocity components. Reynolds stress models provide a more detailed representation of turbulence by explicitly modelling the correlations between velocity fluctuations. They can capture complex flow phenomena and are particularly useful for flows with strong anisotropy or complex geometries. However, they are more computationally expensive than eddy viscosity models due to the additional transport equations that need to be solved. By solving the Reynolds stress transport equations along with the governing equations of fluid flow, engineers can obtain more accurate predictions of turbulent flows and better understand the effects of turbulence on various engineering systems and processes.

3.6 Boundary Conditions

Implementing appropriate boundary conditions and load transfer at the fluid-structure interface is crucial for accurate modelling of fluid-structure interaction problems. In this answer, we will discuss the implementation of boundary conditions and load transfer at the fluid-structure interface based on the research papers provided earlier. Boundary conditions specify the behaviour of the fluid and the structure at the interface. The type of boundary condition used depends on the nature of the problem and the characteristics of the fluid and the structure. For example, pressure and velocity boundary conditions are commonly used for fluid domains, while displacement and force boundary conditions are used for structural domains. To accurately model the fluid-structure interface, it is important to ensure that the boundary conditions are consistent between the fluid and the structure domains. One way to achieve this is to use a coupling approach that allows for two-way transfer of information between the fluid and the structure domains. This ensures that the boundary conditions at the interface are updated at each time step to reflect the changing behaviour of the fluid and the structure. Several approaches have been proposed in the literature for coupling fluid and structure domains. One common approach is to use a partitioned coupling method, in which the fluid and the structure domains are solved separately using different numerical methods and then coupled at the interface. Another approach is to use a monolithic coupling method, in which the fluid and the structure domains are solved together using a single numerical method. Load transfer at the fluid-structure interface is also an important considera-

Table 2: Modelling Approach and Their Advantages and Disadvantages

Modelling approach	**Advantages**	**Disadvantages**
Linear models	Simplicity, ease of use	May not capture nonlinear behaviour and material properties
Nonlinear models	Accuracy, ability to capture complex behaviour	Computationally intensive, requires advanced modelling techniques
Plasticity models	Accurately capture plastic deformation behaviour	Computationally intensive, requires advanced numerical techniques
Fibre-reinforced models	Improve tensile strength and toughness of materials	More expensive, difficult to work with than traditional materials
Damping models	Accurately capture damping behaviour	More computationally intensive than simpler models

tion in modelling fluid-structure interaction problems. The load transfer refers to the transfer of forces and stresses between the fluid and the structure domains at the interface. To accurately model load transfer, it is important to ensure that the fluid and the structure domains are properly meshed and that the mesh sizes are compatible at the interface. One common approach to ensuring proper mesh compatibility is to use a conformal meshing technique, in which the fluid and the structure meshes are matched at the interface. Another approach is to use a non-conformal meshing technique, in which the fluid and the structure meshes are not matched at the interface, but are instead coupled using interpolation functions. In summary, accurate modelling of fluid-structure interaction problems requires careful consideration of boundary conditions and load transfer at the fluid-structure interface. Coupling approaches, such as partitioned and monolithic methods, can ensure that the boundary conditions are consistent between the fluid and the structure domains. Proper meshing techniques, such as conformal and non-conformal meshing, can ensure proper load transfer between the fluid and the structure domains at the interface. The modelling approach, their advantages and disadvantages are shown in Table2.

4 Numerical Methods

There are various numerical techniques for solving fluid-structure interaction problems, each with its strengths and weaknesses. Here, we will discuss three common methods: finite element methods, smoothed particle hydrodynamics, and iterative sub structuring. Finite element method (FEM): The FEM is a widely used numerical technique for solving fluid-structure interaction problems. It involves discretizing the problem domain into smaller subdomains, or elements, and approximating the solution within each element using piecewise polynomial functions. The resulting system of equations is then solved using numerical methods to obtain the solution for the entire domain. The FEM is a versatile method that can handle complex geometries and boundary conditions, and can be easily extended to include other physics, such as heat transfer or electromagnetic fields. The finite element method (FEM) is a numerical technique for solving partial differential equations (PDEs) governing fluid and solid mechanics in the context of fluid-structure interaction problems. The FEM involves the discretization of the domain into a finite number of elements, where each element has a simple geometry and is described by a set of nodal points. The PDEs are then approximated over each element using piecewise polynomial functions.

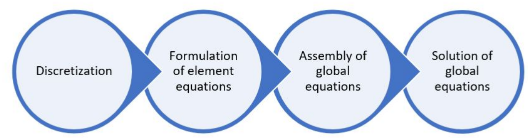

Figure 1: The Steps of FEM

The FEM can be broken down into four main steps.

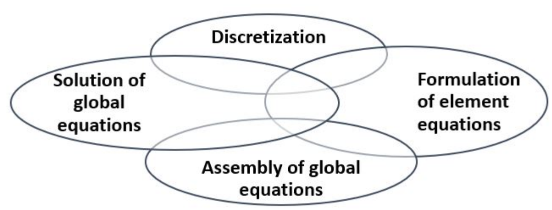

Figure 2: The Four Main Steps of FEM

The FEM is a versatile method that can handle complex geometries and boundary conditions. It can be used to solve a wide range of fluid and solid mechanics problems, including fluid-structure interaction problems. Idelsohn et al. (33) used particle method to solve the continuous fluid mechanics equations. The particle methodology is used to solve fluid–structure interaction problems including free-fluid-surfaces and fluid particle separation. Krysl and Belytschko (34) presented the analysis of Kirchhoff shells by the Element-Free Galerkin (EFG) method. The discretization is independent of the geometric subdivision of "finite elements" as the method is meshless. The FEM has been successfully applied to a range of engineering problems, such as aircraft design, ship hydrodynamics, and biomedical simulations (35). Frei et al. (36) discussed modeling, adaptive discretisation and the numerical solution of fluid structure interaction. A comprehensive overview on innovative discretisation, efficient numerical solution and recent advances in the application fields is presented. However, the accuracy of the FEM solution depends on the number and quality of the elements used to discretize the domain. A large number of elements may result in an accurate solution but can be computationally expensive. Additionally, the FEM requires significant expertise in finite element analysis, which can be a barrier to its adoption in some fields. The FEM is a powerful numerical technique that has found extensive application in solving FSI problems. FEM allows for the accurate discretization of both fluid and solid domains, as well as the coupling between the two, making it well-suited for modeling FSI. Here, we provide a detailed description of the application of FEM in FSI.

1. **Discretization**: The domain is discretized into a finite number of elements, where each element has a simple geometry (e.g., triangle, quadrilateral, tetrahedron, etc.) and is described by a set of nodal points. The nodal points are used to approximate the solution over each element. The first step in applying FEM to FSI is to discretize the problem domain into a finite number of elements. Each element is described by a set of nodal points and has a simple geometry, such as triangles or tetrahedra in 2D or 3D domains, respectively. The nodal points are used to approximate the solution within each element.

2. **Formulation of element equations**: The governing equations are approximated over each element using piecewise polynomial functions. This involves defining a set of shape functions that satisfy the interpolation requirements at the nodal points. The element equations are then obtained by substituting the approximated solution into the governing equations. Once the domain is discretized, the governing equations for fluid and solid mechanics are approximated over each element using piecewise polynomial functions. This involves defining shape functions that satisfy the interpolation requirements at the nodal points. By substituting the approximated solution into the governing equations, element equations are obtained for each element.

3. **Assembly of global equations**: The element equations are assembled into a system of global equations that describe the behavior of the entire domain. This involves applying the boundary conditions and assembling the element equations into a global system of equations. The element equations are then assembled into a system of global equations that describes the behavior of the entire FSI domain. This step involves applying boundary conditions and assembling the element equations into a global system of equations. The global equations capture the interactions between the fluid and solid domains.

4. **Solution of global equations**: The global system of equations is then solved using numerical methods to obtain the solution for the entire domain. The final step is to solve the global system of equations using numerical methods to obtain the solution for the entire FSI problem. Various numerical techniques, such as direct solvers or iterative methods, can be employed to solve the system of equations efficiently. The solution provides information about the behavior of the fluid and solid domains and their interactions.

The application of FEM in FSI offers several advantages. Firstly, it allows for the modeling of complex geometries and boundary conditions, which are often encountered in real-world FSI problems. The flexibility of FEM in handling irregular and non-uniform domains makes it suitable for simulating diverse FSI scenarios. Secondly, FEM enables accurate representation of the fluid and solid behavior by employing high-order polynomial approximations within each element. This accuracy is crucial for capturing the intricate interactions between the fluid and solid domains, such as fluid-induced deformations or solid-induced flow disturbances. Thirdly, FEM provides a versatile framework for incorporating additional physics beyond fluid and solid mechanics. It can be extended to include other physical phenomena like heat transfer, electromagnetic fields, or chemical reactions, enabling comprehensive simulations of Multiphysics FSI problems. The successful application of FEM in FSI has been demonstrated in various engineering fields. For example, in aerospace engineering, FEM has been utilized to study the aerodynamic forces acting on aircraft structures and their effects on flight dynamics. In civil engineering, FEM has been applied to analyze the behavior of buildings or bridges under fluid forces, such as wind or water loads. In biomedical engineering, FEM has proven valuable in simulating the interaction between fluids and biological tissues. This includes studying blood flow in arteries and the impact of fluid forces on cardiovascular health or modeling the flow of cerebrospinal fluid in the brain to understand conditions like hydrocephalus. Overall, the application of FEM in FSI provides a robust and versatile numerical framework for investigating complex fluid-structure interactions. It enables engineers and researchers to gain valuable insights into the behavior of coupled fluid-solid systems, contributing to the design and analysis of various engineering applications.

1. **Smoothed Particle Hydrodynamics (SPH)**: The SPH method is a meshless numerical technique that uses particles to represent the fluid and solid domains. The equations of motion are solved for each particle, and the forces between particles are computed to account for fluid-structure interactions. The SPH method is particularly useful for problems involving free surfaces or fluid fragmentation, and can handle large deformations and complex geometries.

2. **Iterative sub structuring**: The iterative sub structuring method is a partitioned approach that involves solving the fluid and solid domains separately, and then coupling the solutions using interface conditions. This method is computationally efficient and can handle problems with multiple materials or complex boundary conditions. The iterative sub structuring method involves partitioning the domain into smaller subdomains, or substructures, and solving each sub-

structure independently using a chosen solver. The solutions from each substructure are then coupled using interface conditions, and the process is iterated until convergence.

These numerical techniques have been applied successfully to a range of fluid-structure interaction problems, including ship hydrodynamics, wind turbine aerodynamics, and cardiovascular fluid mechanics. Each method has its advantages and disadvantages, and the choice of method will depend on the specific problem being solved and the available computational resources. The different types of finite elements used in fluid-structure interaction problems are shown in Table3 (37; 38; 39; 40).
The choice of meshing strategy depends on the specific problem and the desired trade-off between accuracy, stability, and computational efficiency. Common meshing strategies include structured, unstructured, hybrid, and adaptive meshing, each with its own advantages and disadvantages. Mesh quality metrics can be used to evaluate the quality of a mesh and identify areas that require refinement. Refinement techniques such as local refinement, adaptive refinement, and error estimation can be used to improve the accuracy of a mesh and reduce computational cost. Different types of finite elements can be used in fluid-structure interaction problems, each with its own advantages and disadvantages, including Lagrangian, Eulerian, Arbitrary Lagrangian-Eulerian (ALE), Smoothed Particle Hydrodynamics (SPH), and Immersed Boundary (IB) methods. Overall, the key takeaway is that meshing and finite element selection are important considerations in fluid-structure interaction problems, and a careful evaluation of the specific problem and available resources is necessary to determine the optimal approach. The turbulence models are described in Table4 (37; 38; 39; 40). Table5 refers to the material models (41) while Table6 refers to the boundary conditions (38; 39; 40).

5 Meshing and Discretization

Meshing strategies for fluid and structure domains are critical for accurate simulations of complex engineering systems. Here are some of the commonly used meshing techniques and metrics:

1. **Structured meshing**: In structured meshing, the mesh is generated with a regular grid pattern, which can provide high-quality meshes and accurate results. This technique is particularly useful for simpler geometries, such as rectangular or cylindrical shapes (41).

2. **Unstructured meshing**: Unstructured meshing is a technique that generates meshes with arbitrary shapes and sizes, making it more flexible than structured meshing. This technique is particularly useful for complex geometries, such as those encountered in fluid dynamics simulations (37).

3. **Hybrid meshing**: Hybrid meshing combines structured and unstructured meshing techniques, allowing the generation of high-quality meshes in both simple and complex geometries (38).

4. **Mesh quality metrics**: Mesh quality metrics are used to assess the quality of a mesh, which can affect the accuracy of simulation results. Common metrics include aspect ratio, skewness, orthogonality, and smoothness (39).

5. **Refinement techniques**: Mesh refinement techniques are used to improve the resolution of the mesh in specific regions of interest, such as boundary layers or regions with high gradients. Common refinement techniques include h-refinement, p-refinement, and adaptive mesh refinement (40).

Overall, the choice of meshing strategy and refinement techniques depends on the specific application and geometry of interest, and a careful assessment of mesh quality metrics is essential to ensure accurate simulation results. Miranda et al. (38) presented a methodology for adaptive generation of 3D finite element meshes using geometric modelling. This methodology is applied in the simulation of stress analysis of solid structures and is applicable to other types of 3D finite element simulation. Oberkampf et al. (39) presented viewpoint of the state-of-the-art in verification and validation (V&V) in computational engineering and physics. Verification is the assessment of the accuracy of the solution to a computational model, while validation is the assessment of the accuracy of a simulation solution in comparison with experimental data. Babuska and Suri (40) discuss the fundamental theoretical ideas, basic properties and characteristics of the p version and h-p version of the finite element method. Mittal and Iaccarino (42) review immersed boundary methods, while Tezduyar (43) discusses the computation of moving boundaries and interfaces. Mittal and Iaccarino (42) review immersed boundary methods for fluid-structure interaction, and highlight their advantages in simulating complex and moving geometries. The authors explain the concept of the immersed boundary method, which involves the use of a body-conforming grid to represent the fluid domain and an immersed boundary to represent the solid domain. They discuss the different types of immersed boundary methods, such as the direct forcing method, the immersed boundary projection method, and the immersed interface method, among others. The authors (42) also present several applications of immersed boundary methods in fluid-structure interaction problems, such as flow past a flapping wing, flow through heart valves, and flow-induced vibration of structures. Tezduyar (43) discusses the computation of moving boundaries and interfaces in fluid-structure interaction, and presents a stabilization technique for improving the accuracy and stability of the solution. The author explains the difficulties associated with moving boundaries and interfaces, and highlights the importance of using a stabilized formulation to prevent numerical instability. The author also presents a stabilization technique called the characteristic-based split (CBS) method, which is designed to reduce the numerical error and improve the accuracy of the solution. The paper is a valuable resource for researchers and practitioners working on fluid-structure interaction problems involving moving boundaries and interfaces (43). The comparative statement of meshing strategies, mesh quality metrics, and refinement techniques are described in Table7 (38; 39; 40; 42).

Table 3: Different Types of Finite Elements Used in Fluid-structure Interaction Problems (37; 38; 39; 40)

Finite Element Type	**Description**	**Advantages**	**Disadvantages**
Lagrangian	The mesh moves with the structure, and the fluid is discretized with a Eulerian approach.	Accurate representation of material properties, good for large deformations.	Mesh distortion can cause numerical instability, requires remeshing.
Eulerian	The mesh is fixed, and the fluid and structure are discretized with a Eulerian approach.	Stable for large deformations, does not require remeshing.	Difficulty representing material properties accurately, more computationally expensive.
Arbitrary Lagrangian-Eulerian (ALE)	A hybrid approach that combines the Lagrangian and Eulerian approaches, where the mesh moves with the structure but is also allowed to deform with the fluid.	Can handle large deformations and complex geometries, does not require remeshing.	Can lead to mesh distortion and numerical instability, more computationally expensive than Eulerian.
Smoothed Particle Hydrodynamics (SPH)	The fluid is discretized using particles, with the interpolation function used to calculate properties at any point in space.	Handles free surface flows and large deformations well, does not require remeshing.	Difficult to accurately represent viscous effects and boundary conditions, requires large number of particles.
Immersed Boundary (IB)	A method where the fluid is discretized using a fixed mesh, but the structure is not necessarily coincident with the mesh, and a force is applied to the fluid from the structure.	Can handle complex geometries and large deformations, does not require remeshing.	Can introduce numerical errors from force interpolation, more computationally expensive.

Table 4: Turbulence Models (37; 38; 39)

Model Type	**Description**
Eddy Viscosity Models	Single turbulent viscosity representing turbulence effects
Reynolds Stress Models	Explicitly model correlations between velocity fluctuations
Large Eddy Simulation (LES)	Resolves large turbulent structures and models smaller scales
Direct Numerical Simulation (DNS)	Solves Navier-Stokes equations without turbulence modelling

Table 5: Material Models: (41)

Model Type	**Description**
Newtonian Fluid Model	Linear relationship between shear stress and strain rate
Non-Newtonian Fluid Model	Nonlinear relationship between shear stress and strain rate
Elastic Material Models	Describes solid materials' behaviour considering elastic properties

Table 6: Boundary Conditions: (38; 39; 40)

Boundary Condition	**Description**
Dirichlet Boundary Condition	Prescribes value of the dependent variable at the boundary of the domain
Neumann Boundary Condition	Prescribes gradient of the dependent variable at the boundary of the domain
Inlet and Outlet Boundary Conditions	Specify flow properties at the domain's inlet and outlet boundaries
Wall Boundary Condition	Models interaction between fluid flow and solid walls
Symmetry Boundary Condition	Assumes symmetry about a specified plane, reducing computational domain

Table 7: Comparative Statement of Meshing Strategies, Mesh Quality Metrics, and Refinement Techniques (38; 39; 40; 42)

Technique	**Description**	**Pros**	**Cons**
Structured meshing	Mesh generated with a regular grid pattern	High-quality meshes, accurate results	Limited flexibility for complex geometries
Unstructured meshing	Mesh generated with arbitrary shapes and sizes	Flexible for complex geometries	Lower quality meshes in certain regions
Hybrid meshing	Combination of structured and unstructured meshing	High-quality meshes in both simple and complex geometries	Increased computational complexity
Mesh quality metrics	Metrics used to assess the quality of a mesh	Allows for assessment of accuracy of simulation results	Different metrics may be more appropriate for different applications
Refinement techniques	Techniques used to improve resolution of mesh in specific regions	Increased accuracy in regions of interest	Increased computational complexity

Table 8: Different Coupling Methods

Coupling Method	**Description**	**Advantages**	**Disadvantages**
Monolithic	Solves the fluid and structure equations simultaneously within a single solver	Accurate and efficient for strongly-coupled problems	Limited flexibility in choosing different solvers for fluid and structure
Partitioned	Solves the fluid and structure equations separately using different solvers, and communicates information between them at each time step	Flexibility to choose different solvers for fluid and structure, and can handle more complex geometries	May introduce artificial numerical errors at the interface and require careful treatment of communication between solvers
Loosely Coupled	Solves the fluid and structure equations separately using different solvers, and updates the solution iteratively until convergence	Allows for even greater flexibility in choosing different solvers for fluid and structure, and can handle very large-scale problems	May require many iterations to converge, and may not be suitable for strongly-coupled problems

6 Coupling Strategies

Overall, research suggests that the choice of coupling strategy depends on the specific problem being solved, and that each approach has its own benefits and challenges. Monolithic coupling is preferred for problems with strong fluid-structure interactions and for problems where accuracy is critical. Partitioned coupling is preferred for problems with weak fluid-structure interactions and for problems where computational efficiency is important. Loosely coupled coupling is preferred for problems where flexibility is important and for problems where the fluid and structural domains have very different time scales. It is important to carefully consider the coupling strategy when solving fluid-structure interaction problems to ensure accurate and efficient results. The different coupling methods are presented in Table8. Hou et al. (44) presented a review of numerical methods for computing fluid-structure interaction problems, with a goal to categorize the methods and assess their accuracy. The challenges faced in this field, the importance of interdisciplinary effort for advancing the fluid-structure interactions was emphasized. González and Park (45) proposed a fluid–structure interaction computational framework by means of the method of localized Lagrange multipliers (LLM). The approach of localized Lagrange multipliers facilitates connecting the fluid and the structure modules to a third interface system, thus preserving the modularity. Piperno et al. (46) presented partitioned procedures to predict the dynamic response of a flexible structure in a fluid flow. A one-dimensional piston model problem with a compressible flow is solved to analyze the results. The insights gained from the analysis of the coupled piston problem are confirmed with the numerical simulations.

7 Applications

FSI plays a vital role in various engineering fields, ranging from aerospace and automotive to biomedical applications. Understanding the complex interaction between fluids and deformable structures is crucial for optimizing designs, predicting behaviour, and ensuring safety and performance. FEA has emerged as a powerful tool for simulating FSI problems, allowing for accurate modelling of fluid and solid domains and their interaction. In this context,

Table 9: Different Applications

Field	Description
Civil Engineering (Shaikh and Nallasivam [(51)])	This study analyzed a finite element model of the bridge and sub-track system to predict the static behavior and free vibration responses. The goal was to help designers improve the dynamic analysis of the bridge model.
Marine Engineering (Ma and Mahfuz [(52)])	"In this study structural analysis of composite ship structures using fluid structure interaction (FSI) is developed by coupling FEA and CFD. A comprehensive failure analysis of the ship hull was performed."
Oil and Gas (Zou et al. [(53)])	The study focused on the analysis of fluid-induced vibration of laminated composite pipeline systems. The vibration responses of supported pipeline systems were also compared with FEA using ANSYS software.
Renewable Energy (Kulkarni et al. [(54)])	A review on FSI optimisation in Tidal Turbines including modelling methods based on computational fluid dynamics and structural analysis is presented. The tidal turbine airfoils and tidal turbines are also highlighted using FSI optimisations.
Nuclear Engineering (Park et al. [(55)])	The study utilized finite element analysis to investigate the fluid-structure interaction in nuclear reactors. It aimed to assess the structural seismic responses of the reactor vessel internals (RVIs) of a nuclear reactor system considering the influences of coolant flow loads.

this overview highlights notable studies that employ FEA to investigate FSI phenomena in different industries. These studies include prediction of noise due to sloshing in fuel tanks, echocardiographic diagnosis and assessment of chronic ischemic mitral regurgitation, wind turbine blade interaction, tire hydroplaning, and influence of fluid–structure interactions on the structural changes of brain tissue after injury. By examining these diverse applications, we gain insights into the significance of FEA in modelling and analysing fluid-structure interactions across various disciplines. (44; 47; 48; 49; 50). Fluid-structure interaction problems are encountered in various fields, including aerospace, automotive, and biomedical applications, and can be solved using finite element modelling. These simulations can help optimize the design of structures, predict their behaviour, and inform treatment decisions. These studies highlight the diverse range of applications for finite element analysis in various fields, demonstrating the importance of understanding and modelling fluid-structure interaction phenomena to optimize designs, improve performance, and ensure structural integrity.

8 Conclusion

The review paper provides a comprehensive overview of finite element modelling for fluid-structure interaction (FSI) problems. It discusses the governing equations, numerical methods, recent advances, challenges, and future directions in FSI modelling. Finite element modelling offers a powerful approach to accurately simulate FSI, considering both fluid and solid domains and their interactions. It enables handling complex geometries, nonlinear material properties, and achieving high accuracy. However, challenges exist, including the need for robust numerical methods, accurate constitutive models for soft tissues, and handling large deformations and complex fluid flow patterns. Recent advances, such as reduced order models and machine learning techniques, show promise in enhancing efficiency and accuracy. The future of FSI modelling relies on developing more accurate and efficient numerical methods, integrating experimental data and clinical imaging, and exploring new applications in soft robotics and bioinspired design. The review paper serves as a valuable resource for researchers and practitioners, providing insights and identifying key areas for future research and development in FSI modelling.

Conflict of Interest

The authors declare no conflict of interest in this reported communication.

Acknowledgments

The authors express their thanks to Assam down town University for providing the facility for the work.

References

[1] T. Belytschko, W. Liu, and B. Moran, *Nonlinear finite elements for continua and structures*, 1st ed. John Wiley Sons, 2000.

[2] T. Hughes, W. Liu, and T. Zimmermann, "Lagrangian-eulerian finite element formulation for incompressible viscous flows," *Computer Methods in Applied Mechanics and Engineering*, vol. 29, no. 3, pp. 329–349,, 1981. [Online]. Available: https://doi.org/10.1016/0045-7825(81)90049-9.

[3] O. Zienkiewicz, R. Taylor, and P. Nithiarasu, "The finite element method for fluid dynamics, butterworth-heinemann," 2014. [Online]. Available: https://doi.org/10.1016/C2009-0-26328-8.

[4] K. Bathe and E. Wilson, *Numerical methods in finite element analysis.* Prentice-Hall, 1976.

[5] J. Donea and A. Huerta, *Finite Element Methods for Flow Problems.* John Wiley Sons, 2003. [Online]. Available: https://doi.org/10.1002/0470013826.

[6] F.-B. Tian, H. Dai, H. Luo, J. Doyle, and B. Rousseau, "Fluid–structure interaction involving large deformations: 3d simulations and applications to biological systems," *J Comput Phys*, vol. 258, pp. 451–469,, 2014. [Online]. Available: https://doi.org/10.1016/j.jcp.2013.10.047.

[7] B. Griffith and N. Patankar, "Immersed methods for fluid–structure interaction," *Annu Rev Fluid Mech*, vol. 52, pp. 421–448,, 2020. [Online]. Available: https://doi.org/10.1146/annurev-fluid-010719-060228.

[8] S. Gholampour, "Fsi simulation of csf hydrodynamic changes in a large population of non-communicating hydrocephalus patients during treatment process with regard to their clinical symptoms," *PLoS ONE*, vol. 13, no. 4, 2018, e0196216,. [Online]. Available: https://doi.org/10.1371/journal.pone.0196216.

[9] Y. Qiao, J. Fan, Y. Ding, T. Zhu, and K. Luo, "A primary computational fluid dynamics study of pre- and post-tevar with intentional left subclavian artery coverage in a type b aortic dissection," *ASME. J Biomech Eng*, vol. 141, no. 11, p. 111002, 2019. [Online]. Available: https://doi.org/10.1115/1.4043881.

[10] J. Anderson, "Computational fluid dynamics: the basics with applications," 1995, 10.

[11] J. Ferziger and M. Peric, *Computational methods for fluid dynamics.* Berlin, Heidelberg: Springer, 2002. [Online]. Available: https://doi.org/10.1007/978-3-642-56026-2.

[12] Z. Guo, C. Zheng, and B. Shi, "Discrete lattice effects on the forcing term in the lattice boltzmann method," *Physical review E*, vol. 65, no. 4, pp. 046 308,, 2002. [Online]. Available: https://doi.org/10.1103/PhysRevE.65.046308.

[13] [Online]. Available: https://en.wikipedia.org/wiki/Reynolds_stress_equation_model.

[14] J. Chen and V. Patel, "Near-wall turbulence models for complex turbulent flows including separation," *AIAA Journal*, vol. 26, no. 6, p. 641–648, 1988.

[15] S. Chan, "Non-linear behavior and design of steel structures," *Journal of Constructional Steel Research*, vol. 57, no. 12, pp. 1217–1231,, 2001. [Online]. Available: https://doi.org/10.1016/S0143-974X(01)00050-5.

[16] O. Buyukozturk, "Nonlinear analysis of reinforced concrete structures," *Computers Structures*, vol. 7, no. 1, pp. 149–156,, 1977. [Online]. Available: https://doi.org/10.1016/0045-7949(77)90069-4.

[17] W.-F. Chen and D.-J. Han, "Plasticity for structural engineers," *J. Ross Publishing*, p. 978–1–932159–75–2, 2007.

[18] M. Anas, M. Khan, H. Bilal, S. Jadoon, and M. Khan, "Fiber reinforced concrete: A review," *Engineering Proceedings*, vol. 22, no. 1, 2022, 3,. [Online]. Available: https://doi.org/10.3390/engproc2022022003.

[19] C. Poon, Z. Shui, and L. Lam, "Compressive behavior of fiber reinforced high-performance concrete subjected to elevated temperatures," *Cement and Concrete Research*, vol. 34, no. 12, pp. 2215–2222,, 2004. [Online]. Available: https://doi.org/10.1016/j.cemconres.2004.02.011.

[20] B. Wohlmuth, "Discretization methods and iterative solvers based on domain decomposition," *Lecture Notes in Computational Science and Engineering*, 2001. [Online]. Available: https://doi.org/10.1007/978-3-642-56767-4.

[21] J. Argyris and S. Symeonidis, "Nonlinear finite element analysis of elastic systems under nonconservative loading-natural formulation. part i. quasistatic problems," *Computer Methods in Applied Mechanics and Engineering*, vol. 26, no. 1, pp. 75–123,, 1981. [Online]. Available: https://doi.org/10.1016/0045-7825(81)90131-6.

[22] K. Wong, P. Thavornpattanapong, S. Cheung, and J. Tu, "Numerical stability of partitioned approach in fluid-structure interaction for a deformable thin-walled vessel," *Computational and Mathematical Methods in Medicine*, vol. 638519, 2013. [Online]. Available: https://doi.org/10.1155/2013/638519.

[23] E. Oñate, "Structural analysis with the finite element method," *Linear Statics*, vol. 2, 2013. [Online]. Available: https://doi.org/10.1007/978-1-4020-8743-1.

[24] T. Belytschko, W. Liu, B. Moran, and K. Elkhodary, *Nonlinear Finite Elements for Continua and Structures*, 2nd ed. ISBN: Wiley, 2013.

[25] K. Bathe and H. Zhang, "Finite element developments for general fluid flows with structural interactions," *International Journal for Numerical Methods in Engineering*, vol. 60, no. 1, pp. 213–232,, 2004. [Online]. Available: https://doi.org/10.1002/nme.959.

[26] J. Donea, S. Giuliani, and J. Halleux, "An arbitrary lagrangian-eulerian finite element method for transient dynamic fluid-structure interactions," *Computer Methods in Applied Mechanics and Engineering*, vol. 33, no. 1-3, p. 689–723, 1982.

[27] C. Felippa, K. Park, and C. Farhat, "Partitioned analysis of coupled mechanical systems," *Computer Methods in Applied Mechanics and Engineering*, vol. 190, no. 24–25, pp. 3247–3270,, 2001. [Online]. Available: https://doi.org/10.1016/S0045-7825(00)00391-1.

[28] S. Jayanti, "Equations governing fluid motion," in *Computational Fluid Dynamics for Engineers and Scientists.* Dordrecht: Springer, 2018. [Online]. Available: https://doi.org/10.1007/978-94-024-1217-8_2.

[29] T. Belytschko, Y. Krongauz, D. Organ, M. Fleming, and P. Krysl, "Meshless methods: An overview and recent developments," *Computer Methods in Applied Mechanics and Engineering*, vol. 139, no. 1-4, p. 3–47, 1996. [Online]. Available: https://doi.org/10.1016/S0045-7825(96)01078-X.

[30] C. Hirt and B. Nichols, "Volume of fluid (vof) method for the dynamics of free boundaries," *Journal of Computational Physics*, vol. 39, no. 1, p. 201–225, 1981. [Online]. Available: https://doi.org/10.1016/0021-9991(81)90145-5.

[31] [Online]. Available: https://en.wikipedia.org/wiki/Reynoldseraged_Navier%E2%80%93Stokes_equations

[32] [Online]. Available: https://en.wikipedia.org/wiki/K-epsilon_turbulence_model.

[33] S. Idelsohn, E. Onate, and F. Del Pin, “The particle finite element method: A powerful tool to solve incompressible flows with free surfaces and breaking waves,” *International Journal for Numerical Methods in Engineering*, vol. 61, no. 7, p. 964–989, 2004.

[34] P. Krysl and T. Belytschko, “Analysis of thin shells by element-free galerkin methods,” *International Journal of Solids and Structures*, vol. 33, no. 20–22, p. 3057–3080, 1996.

[35] A. Shabana, *Dynamics of Multibody Systems*. Cambridge: Cambridge University Press, 2020. [Online]. Available: https://doi.org/10.1017/9781108757553

[36] S. Frei, B. Holm, T. Richter, T. Wick, and H. Yang, “Fluid-structure interaction: Modeling, adaptive discretisations and solvers,” *De Gruyter*, vol. 20, 2018. [Online]. Available: https://doi.org/10.1515/9783110494259

[37] R. Löhner, *Applied CFD techniques: An introduction based on finite element methods*. John Wiley Sons, 2008. [Online]. Available: https://doi.org/10.1002/9780470989746.

[38] A. Miranda, W. Lira, J. Cavalcante-Neto, R. Sousa, and L. Martha, “A three-dimensional adaptive mesh generation approach using geometric modeling with multi-regions and parametric surfaces,” *J. Comput. Inf. Sci. Eng*, vol. 13, no. 2, 2013, 021002,. [Online]. Available: https://doi.org/10.1115/1.4024106.

[39] W. Oberkampf, T. Trucano, and C. Hirsch, “Verification, validation, and predictive capability in computational engineering and physics,” *Appl. Mech. Rev*, vol. 57, no. 5, pp. 345–384,, 2004. [Online]. Available: https://doi.org/10.1115/1.1767847.

[40] I. Babuska and M. Suri, “The p and h-p versions of the finite element method, basic principles and properties,” *SIAM Review*, vol. 36, no. 4, p. 578–632, 1994.

[41] J. Thompson, Z. Warsi, and C. Mastin, “Numerical grid generation: Foundations and applications,” *Elsevier Science Ltd*, no. BN-13, p. 978–0444009852, 1985.

[42] R. Mittal and G. Iaccarino, “Immersed boundary methods,” *Annual Review of Fluid Mechanics*, vol. 37, pp. 239–261,, 2005.

[43] T. Tezduyar, “Computation of moving boundaries and interfaces and stabilization parameters,” *International Journal for Numerical Methods in Fluids*, vol. 43, no. 5, pp. 555–575,, 2003.

[44] S. Golla, A. Jadhav, R. Banerjee, and B. Venkatesham, “Numerical simulation of hit noise generation due to sloshing phenomenon in a rectangular tank under periodic excitation,” *J. Fluids Eng. Mar*, vol. 145, no. 3, pp. 031 401,, 2023. [Online]. Available: https://doi.org/10.1115/1.4056208.

[45] J. González and K. Park, “Three-field partitioned analysis of fluid–structure interaction problems with a consistent interface model,” *Computer Methods in Applied Mechanics and Engineering*, vol. 414, pp. 116 134,, 2023. [Online]. Available: https://doi.org/10.1016/j.cma.2023.116134.

[46] S. Piperno, C. Farhat, and B. Larrouturou, “Partitioned procedures for the transient solution of coupled aroelastic problems part i: Model problem, theory and two-dimensional application,” *Computer Methods in Applied Mechanics and Engineering*, vol. 124, no. 1–2, pp. 79–112,, 1995. [Online]. Available: https://doi.org/10.1016/0045-7825(95)92707-9.

[47] D. Dudzinski and J. Hung, “Echocardiographic assessment of ischemic mitral regurgitation,” *Cardiovasc Ultrasound*, vol. 12, pp. 46,, 2014. [Online]. Available: https://doi.org/10.1186/1476-7120-12-46.

[48] L. Wang, R. Quant, and A. Kolios, “Fluid structure interaction modelling of horizontal-axis wind turbine blades based on cfd and fea,” *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 158, pp. 11–25,, 2016. [Online]. Available: https://doi.org/10.1016/j.jweia.2016.09.006.

[49] C. Hermange, G. Oger, Y. Le Chenadec, M. Leffe, and D. Le Touzé, “In-depth analysis of hydroplaning phenomenon accounting for tire wear on smooth ground,” *Journal of Fluids and Structures*, vol. 111, pp. 103 555,, 2022.

[50] Y. Feng, Y. Chen, Y. Yao, X. Li, A. Zhang, and G. Genin, “The brain as a structure: A model of how fluid–structure interactions stiffen brain tissue after injury,” *Engineering Structures*, vol. 256, pp. 113 960,, 2022.

[51] M. Shaikh and K. Nallasivam, “Static and free vibration response of a box-girder bridge using the finite element technique,” *Multidiscipline Modeling in Materials and Structures*, vol. 19, no. 5, pp. 897–923,, 2023.

[52] S. Ma and H. Mahfuz, “Finite element simulation of composite ship structures with fluid structure interaction,” *Ocean Engineering*, vol. 52, no. 1, pp. 52–59,, 2012.

[53] G. Zou, N. Cheraghi, and F. Taheri, “Fluid-induced vibration of composite natural gas pipelines,” *International Journal of Solids and Structures*, vol. 42, pp. 3–4 , 1253–1268,, 2005.

[54] S. Kulkarni, L. Wang, N. Golsby, and M. Lander, “Fluid-structure interaction based optimisation in tidal turbines: A perspective review,” *Journal of Ocean Engineering and Science*, vol. 7, no. 5, pp. 449–461,, 2022.

[55] J.-b. Park, S.-J. Lee, L. E-h, P. N-C, and Y.-b. Kim, “Seismic responses of nuclear reactor vessel internals considering coolant flow under operating conditions,” *Nuclear Engineering and Technology*, vol. 51, no. 6, pp. 1658–1668,, 2019.