Introduction

Fluid–structure interaction (FSI) and, in a broader sense, multiphysics have become more and more the focus of computational engineering in the recent years. Fluid structure coupling can occur in many fields of engineering, and is a crucial consideration in the design of many engineering systems; for example, stability and response of aircraft wings in aerospace industry, flow of blood through arteries in biomedical applications, response of bridges and tall buildings to winds in civil engineering, and oscillation of heat exchangers in nuclear industry. These problems are often too complex to solve analytically and so they have to be analyzed by means of numerical simulations. Numerical simulations of coupled problems have been increasing for applications where experimental studies are very expensive and time consuming. These problems are computer time consuming and require new stable and accurate coupling algorithms to solve these problems. For the past decades, new development in coupling algorithms and the increased performance of computer have allowed solving some of these problems, besides solving some physical applications that were not accessible in the past. In the future this trend might continue to address more realistic problems. The use of numerical simulation can reduce the amount of time spent using experimental methods to assess a large number of design alternatives. A better understanding of the problem is obtained through a computational approach because

of increased amount of information gathered during computation. There is continuous research in the fields of computational fluid dynamics and computational solid mechanics, and it has reached a maturity level to help in solving large industrial and academic problems that were not accessible in the past. However, this level of achievement has not been reached to solve numerical simulation of fluid–structure interaction problems, and there are still several key questions in FSI regarding robustness, accuracy, stability, and computation time to be resolved satisfactorily. Various approaches have been investigated to model fluid–structure interaction problems. The added mass techniques have been extensively developed and used for problems involving small displacement and small deformation of structures coupled to potential flow. The objective of the added mass techniques is to investigate the dynamic behavior of the structure without computing the fluid motion, and consequently reduce the number of degrees of freedom in the problem and save computational time. Since the CFD techniques that solve the full Navier–Stokes equations to obtain the hydrodynamic force on the structure are extremely time consuming, for practical use the added mass techniques have been used by designers for predicting vibration levels and fatigue damage of the structure in a coupling. This approach is also used by different authors in solving flow-induced vibration problems, assuming small amplitude vibrations of the structure and potential flow for the fluid. For general hydrodynamic problems involving large deformation or moving structure, the full Navier–Stokes equations need to be solved in a moving mesh, since the fluid is described in a moving domain because of the structure motion. The most natural approach to solve these coupling problems is the ALE (Arbitrary Lagrangian–Eulerian) formulation for the fluid domain and the Lagrangian formulation for the structure domain. The ALE method has been intensively used for problems involving small and large structure displacements with no topological changes in the structure. In the presence of large structure displacements and complex geometries, the actual ALE remeshing algorithms are not general

and robust enough to handle complex meshes and topological changes in geometry, and fail for most complex three dimensional problems. For most three dimensional applications, automatic mesh generators are called on internally to create a new mesh with a new topology given the boundary segments. This method refers to a rezoning method; the dependent variables: velocity, pressure, internal energy, stress components, and plastic strain are updated on the new mesh by using a remap algorithm. Unlike a rezoning method, the topology of the mesh is fixed in an ALE method. The accuracy of an ALE calculation is often superior to the accuracy of a rezoned calculation because the algorithms used to remap the solution from distorted to undistorted mesh is second-order accurate for the ALE formulation when using second-order advection algorithms, while the algorithm for the remap in the rezoning is only first-order accurate. The purpose of this book is to address the challenges encountered in solving coupling problems and describe different algorithms and numerical methods being used to solve problems where fluid and structure can be weakly coupled using partitioned methods or strongly coupled using monolitical approach. In many applications, mainly industrial applications, partitioned methods are the most used techniques for solving coupled problems, because they allow use of specific designated fluid and structure codes and offer significant benefits in term of efficiency. Using the partitioned method, small and better conditioned systems are solved instead of a single solver.

Chapter 1 presents an introduction to ALE formulation, which is crucial to solve most of the fluid–structure interaction problems. For most FSI problems, one of the numerical challenges is the mesh motion and the ALE formulations that follow. Special care is needed while using ALE formulations, because classical time integration methods may lose their stability when used on moving mesh. The Lagrangian formulation where the mesh moves with the material, which is commonly used to solve problems in solid mechanics. This choice is very economical and resolves the material boundaries and

free surfaces very accurately. The main limitation of the Lagrangian formulation is that the deformations must be limited otherwise the distortion in the mesh will result in inaccuracies and numerical instability. The Eulerian formulation, where the mesh is fixed in space, is commonly used to solve problems in fluid mechanics; using a fixed mesh eliminates the concern of mesh distortion, but introduces additional complexity of the convective terms associated with the transport of the material through the mesh. Additional computation time and concerns need to be considered to take into account the advection in the Navier–Stokes equations. For problems involving fluid and structure, such as fluid–structure interaction, neither the Lagrangian nor the Eulerian formulation can be used for the entire domain; the ALE formulation permits the mesh to transition from being Eulerian for modeling the fluid flow to a nearly Lagrangian that follows the deformation of the structure. For general 3D applications, the remesh of the fluid domain is more complex due to the complexity of the 3D geometry of the computational domain.

Chapter 2 describes a survey related to numerical simulation of fluid–structure interaction problems for dynamic explicit problems. Dynamic fluid–structure interaction analysis is useful for a wide range of applications such as hydrodynamic impact, airbag inflation, fuel tank sloshing, and many other industrial applications. In recent years these problems have been considered to be of increased importance due to new development in industry to analyze the safety and integrity of the structure as a result of dynamic loading; in particular in the naval industry for high hydrodynamic impact analysis of deformable structure, in the automotive industry for airbag deployment and fuel tank sloshing, and in the aerospace industry for bird impact analysis and helicopter ditching. A demand for better design of the structure requires improved procedure analysis, and in many cases the safety and integrity of the structure cannot be estimated unless hydrodynamic forces for the structure loading are computed accurately by solving Navier–Stokes equations. In order to describe the algorithms

used for the ALE formulation and coupling, a detailed presentation of the ALE and coupling are given in this chapter. The coupling algorithm based on penalty coupling, spring damping system, is described in detail in this chapter. These algorithms allow us to solve coupling problems, where the structure is described in a Lagrangian formulation and the fluid in a Eulerian formulation. The mesh of the Lagrangian structure is allowed to deform with no mesh interface constraints inside a Eulerian fluid mesh. Coupling algorithms evaluate interaction forces connecting fluid and structure. To illustrate these methods, two academic problems, a piston problem, and a two-dimensional water impact problem are illustrated. Both applications carry analytical solutions that can be used for comparison and to evaluate the accuracy of the coupling algorithm. To extend the coupling algorithm to more complex problems, we consider two industrial examples. The simulation of the deployment of an airbag due to a high velocity gas flowing out of an inflator is described first. This problem is of high interest in the automotive industry and presents several complexities involving a contact algorithm between different airbag layers and the large deformation of flexible membrane elements of the airbag. The second example consists of a sloshing fluid tank under seismic loading. This application, used for earthquake response of liquid storage tanks, is of great interest in civil engineering.

Chapter 3 covers the second numerical challenge that concerns the numerical realization of the coupling mechanisms between structure and fluid. These mechanisms can be invoked at different levels within the numerical schemes, resulting either in more weakly or more strongly coupled procedures. The first and by far the simplest is the fully explicit partitioned coupling involving an alternating solution of solid and fluid problems with exchange of boundary conditions. Moreover, the partitioned approach allows us to solve the flow equations and the structural equations with different, possibly more efficient techniques that have been developed specifically for either flow equations or structural equations. This

approach is very flexible concerning the choice of the solvers for the individual fluid and solid subtasks, but it often suffers from poor stability and convergence properties, time-step restriction, and energy conservation at the fluid structure interface. In the past years, important improvements have been achieved in the partitioned coupling, which goes beyond data mapping, communication between different codes, and interpolation and projection techniques. In order to preserve the flexibility and modularity that are inherent in the partitioned coupling, strongly or implicit partitioned technique, using an iterative process, have been intensively developed and used to improve the properties of the partitioned coupling. After reviewing the basics of computational fluid and structural mechanics, the main aspects of fluid–structure interaction algorithms are developed, and numerical results for a variety of academic and industrial applications are described in this chapter.

Chapter 4 consists of the description of the monolithic approach involving the simultaneous solution of all unknowns. The equations governing the flow velocity and displacement structure are solved simultaneously with a single solver. In this case the convergence rate with respect to the coupling is usually optimal, but the full system is hard to solve and extensive modifications of the individual fluid and solid solvers are necessary. The iterative process used in the implicit partitioned coupling may be difficult to converge for problems where the fluid density is very close to the structure density. For such problems the monolithic approach is necessary to ensure numerical stability of the coupling problem. This difficulty does not occur for problems such as aeroelasticity where the fluid density is orders of magnitude smaller than the material structure density, but it becomes very important in biomechanical application where the arteries material and blood densities are of the same order.

Chapter 5 describes a monolithic approach to explain multiphase problems. In this approach different materials

described by different domains can be involved in the simulation, fluid, rigid or deformable solids. All materials are solved using a Eulerian mesh. The interaction between several phases involving liquid, gas, rigid solid and elastic material are considered in this chapter. In this approach, a monolithic coupling is performed to solve problems with different materials and their interaction, gas-solid or liquid-solid, by tracking the interface between materials using a level set function, which solves the material boundary and the free surface accurately. Interfaces between different domains are known implicitly through the values of a characteristic function defined on the whole computational domain and a specific solver associated with each domain. The main advantage of the multiphase formulation is that the mesh can be fixed. Since the coupling interface is a material interface and it is not defined geometrically through the mesh for accuracy of the multiphase problem, the mesh has to be refined at the interface. Level set technique used for interface tracking is described in this chapter. The numerical method used is based on the use of stable mixed formulation, which consists of continuous piecewise linear functions enriched with a bubble function for the velocity and piecewise linear functions for the pressure.

Mhamed SOULI