Chapter 1

Introduction

The approximation of physical phenomena with a numerical model in general, and finite element, volume or difference methods in particular, has been steadily gaining popularity. As an alternative means of investigation, computer simulations have proven to be a valuable complement to the established theoretical and experimental approaches. Fluid dynamics research in particular is benefitting greatly from the new methodology. Here the equations governing the behavior of the fluid, in spite of their relatively compact form, are extremely challenging from the mathematical point of view. Analytic solutions have been found only for a small subset of all possible fluid flow problems. Non-linear nature of the governing equations can produce extremely complex flow fields and multiple time-dependent solutions, even for simplest geometries. Before the advent of computational methods, many problems were possible to analyze only through experimentation. In the case of aerospace industry, this involves costly flight and wind tunnel testing.

The computational fluid dynamics (CFD) offers the potential of modeling the reality in a more economical manner, and opens the possibility of simulating phenomena which are impossible to reduce to an experimental scale, such as atmospheric or oceanic motions. The control over various parameters in numerical experiments is often greater and more convenient than what the real experimentation allows. Changing, e.g., the domain size with a few keystrokes obviously has its advantages over constructing a new experimental apparatus. This is not to say that a new artificial reality is devoid of a new set of difficulties. Numerical approximation of physical systems, and especially the necessary replacement of continuum with a finite number of variables, is rarely straightforward. The possibility of approximation errors obliterating the actual physical phenomena is a constant danger. Careful comparison with the experimental and analytic results is necessary to validate each new numerical method before its predictions can be trusted.

Nevertheless, with the interest in numerical methods fueled by the advances in computer technology, the understanding of the approximation process grew rapidly. Robust computational techniques were developed, especially in the field of structural mechanics and later in fluid dynamics itself. By now CFD has been adopted by many aerospace companies as an essential tool.

Inevitably, the range of problems facing the technique is also rapidly expanding. This thesis will discuss the application of finite element tools to several non-standard classes of problems in fluid mechanics. The first such class encompasses problems which involve moving boundaries and interfaces. The domain over which such problems have to be solved evolves in time, requiring modifications of the usual simulation techniques. We address this problem with a stabilized finite element formulation, which incorporates the motion of the domain automatically by virtue of deforming space-time elements. The second class of problems involves incompressible fluids that exhibit non-standard constitutive behavior. Specifically, flows of viscoelastic fluids are of interest here. In contrast to the Newtonian fluids, viscoelastic constitutive models usually cannot be incorporated by substituting an expression for the stress into the momentum equation, and demand a separate variational form corresponding to the constitutive equation. To this end, we will discuss a new mixed method, which is unique in allowing all combinations of interpolation functions for the stress, velocity and pressure fields.

Any discussion of numerical techniques cannot be easily separated from implementational issues. Practical application of finite element methods on scalar, and even vector, architectures has been extensively discussed in many textbooks and tutorials.

The advent of massively-parallel computers, while promising continuing advances in computational speed, rendered most of these classical finite element programming techniques obsolete. Both the data structure and the algorithm design have to be changed extensively to take advantage of the speedups offered by parallelism. An implementational framework suitable for unstructured finite element grids, yet resulting in good and scalable parallel performance, is discussed here as a relevant issue of extreme importance.

1.1 Overview

We will begin by introducing the equations of the fluid motion in Chapter 2, and by stating constitutive relations for both Newtonian and a particular class of non-Newtonian fluids. In Chapter 3, we open with a general discussion of numerical methods used to solve physical problems involving moving boundaries and interfaces. Subsequently, we present the space-time velocity-pressure formulation which takes such deformations into account in an elegant and straightforward manner. We describe the stabilization techniques which will endow the method with necessary robustness. Finally, we discuss in detail the applicability of the method to problems involving deforming domains, and related issues such as mesh movement options and interface effects.

In Chapter 4, we review previous finite element efforts to accurately simulate flows of viscoelastic fluids. Then we introduce a stabilized stress-velocity-pressure finite element formulation, which combines the ability to accommodate a wide range of constitutive relations with imperviousness to the choice of interpolation functions.

The formulations described in Chapters 3 and 4 give rise to large coupled linear systems of equations, which can be solved by using either a direct or an iterative solution technique. The merits and disadvantages of both approaches are discussed in Chapter 5. The iterative scheme under consideration is based on the preconditioned GMRES method. A brief section on preconditioning techniques is also included.

The applications of the methodology are presented in Chapter 6. Here, the space-time velocity-pressure formulation is used in a three-dimensional simulation of sloshing in a tank subjected to horizontal and vertical vibrations. The stress-velocity-pressure formulation, initially using a Newtonian constitutive model, is tested on flows past a circular cylinder at a range of Reynolds numbers. Then, the same formulation, but incorporating Maxwell-B and Oldroyd-B constitutive equations, is used to solve the test contraction flow problem.

Every chapter touching upon the implementational aspects of the finite element techniques in this thesis includes a section which should ease the application of the concepts included in that chapter on massively-parallel distributed-memory computers. Our experience here is restricted to the Single-Instruction-Multiple-Data (SIMD), or data parallel, style of programming, which is accessible on the Connection Machine CM-200 and CM-5 computers.