Journal of Computational Physics 346 (2017) 91–130

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Fully-coupled pressure-based finite-volume framework for the simulation of fluid flows at all speeds in complex geometries

Cheng-Nian Xiao, Fabian Denner, Berend G.M. van Wachem *

Department of Mechanical Engineering, Imperial College London, Exhibition Road, London, SW7 2AZ, United Kingdom

A R T I C L E   I N F O

Article history:
Received 25 January 2017
Received in revised form 24 May 2017
Accepted 5 June 2017
Available online 9 June 2017

Keywords:
All-speeds flows
Fully-coupled framework
Unstructured grids
Shock-capturing schemes
Transient accuracy

A B S T R A C T

A generalized finite-volume framework for the solution of fluid flows at all speeds in complex geometries and on unstructured meshes is presented. Starting from an existing pressure-based and fully-coupled formulation for the solution of incompressible flow equations, the additional implementation of pressure–density–energy coupling as well as shock-capturing leads to a novel solver framework which is capable of handling flows at all speeds, including quasi-incompressible, subsonic, transonic and supersonic flows. The proposed numerical framework features an implicit coupling of pressure and velocity, which improves the numerical stability in the presence of complex sources and/or equations of state, as well as an energy equation discretized in conservative form that ensures an accurate prediction of temperature and Mach number across strong shocks. The framework is verified and validated by a large number of test cases, demonstrating the accurate and robust prediction of steady-state and transient flows in the quasi-incompressible as well as subsonic, transonic and supersonic speed regimes on structured and unstructured meshes as well as in complex domains.

1. Introduction

Since the widespread increase and availability of computing power and resources, computational fluid dynamics (CFD) has become a mainstay tool in academic research as well as commercial development for problems related to fluid dynamics. Thus, the demand for numerical frameworks able to simulate various complex flows has also become more pronounced. One well-known flow configuration that has proved to be problematic for most state-of-the-art numerical frameworks arises in the simultaneous presence of multiple flow speed regimes, ranging from the incompressible limit to supersonic flows. A variety of numerical frameworks for specific flow regimes, dealing with either high-speed or nearly incompressible flows, have been previously proposed in the literature [1–4]. However, each of them has its shortcomings in the flow speed regime for which they are not designed.

In the quasi-incompressible speed regime, *i.e.* for Mach numbers $M = |\boldsymbol{u}|/|\boldsymbol{c}| \ll 0.1$ throughout, where $\boldsymbol{u}$ is the local flow velocity and $\boldsymbol{c}$ is the local speed of sound, the flow can be considered as (nearly) incompressible, *i.e.* the density, ρ of each fluid particle remains (almost) constant, $D\rho/Dt = 0$ [5]. Thus, the fluid density ρ is either decoupled or, at best, weakly coupled to velocity and pressure. The most common numerical frameworks specialized to these types of flows reflect this property by solving the incompressible flow equations in a decoupled manner [1,4]. These methods generally apply a segregated pressure-correction approach, where the velocity is first predicted by the momentum equations using a prelim-

* Corresponding author.
E-mail address: berend.van.wachem@gmail.com (B.G.M. van Wachem).

http://dx.doi.org/10.1016/j.jcp.2017.06.009

inary estimate of the pressure field, then both pressure and velocity are corrected to ensure the divergence-free condition as dictated by the continuity equation. The correction step is accomplished by the solution of a reformulated continuity equation as a Poisson equation for the pressure correction [1,6]. Different variants of this approach are the marker-and-cell (MAC) method, projection or fractional step methods, and the family of semi-implicit method for pressure linked equations (SIMPLE) [1,4,7]. Recently, a number of studies reported coupled numerical frameworks, notably [8–12], where the primary variables (velocity and pressure) are solved in a single linear system of equations. The implicit coupling of pressure and velocity is typically associated with an increased robustness and numerical stability for complex flow configurations, for instance flows with large density and viscosity discontinuities, flows in complex domains or in the presence of large source terms.

At high flow speeds with $M > 0.2$ the compressible nature of the flow becomes noticeable, since convective effects of the fluid flow can effectively compete with acoustic waves at the same time scale. Thus, the fluid motion itself has a significant effect on the distribution of density and other thermodynamic variables inside the flow field [13]. The most established algorithms for high speed flows are density-based, *i.e.* they are based on the algebraic equivalents of the governing equations in which the fluid density ρ is one of the unknowns to be solved for [1]. The governing equations for this approach are cast in form of a coupled nonlinear system of conservation laws, where the unknown variables are the density, momentum components as well as total energy or enthalpy. The main strengths of this approach derives from the large body of knowledge in applied mathematics relating to hyperbolic equation systems which encompass the governing equations for inviscid flows [14,15]. Some of the most well-known representatives of density-based numerical frameworks for compressible flows are the *MacCormack* scheme, the *Beam–Warming* scheme, and the *Jameson–Schmidt–Turkel* (JST) scheme [2]. The interested reader is referred to the textbooks of Anderson et al. [1] and Wesseling [2] for a detailed account of these and other density-based frameworks. However, the major drawback of the density-based formulation lies in the requirement of a strong coupling between density ρ and pressure p. This condition is violated for the simulation of flows at low Mach numbers ($M < 0.1$) if the chosen numerical time-step is adapted to the convective time scale of the flow and thus too large to resolve acoustic waves. Hence, such cases become numerically ill-conditioned for density-based frameworks [1,2].

More recently, a number of numerical frameworks that are suitable for all flow speeds have been proposed by extending solution methods originally designed for incompressible or compressible flows [16–18]. The most often used strategy to stabilize density-based solvers at the low-speed limit is preconditioning [19] or artificial compressibility [20]. These methods have, however, major drawbacks. Firstly, they require an additional numerical parameter which has to be readjusted for each individual flow case and, secondly, they are not able or too inefficient to reproduce correct transient results. Even with the addition of the most sophisticated techniques, density-based frameworks tend to be less efficient than pressure-based frameworks for (nearly) incompressible flows [2]. More recently, an adapted pressure-correction strategy based on the governing equations for compressible flows has been proposed by Bijl and Wesseling [18], which appears to be promising at addressing the aforementioned issues. However, it is predicated on the use of a non-conservative version of the energy equation as well as the segregated solution sequence which decouples momentum equations from the continuity equation.

Extensions of pressure-based frameworks designed for incompressible flows to the compressible regime is typically based on a variant of the SIMPLE or PISO (Pressure Implicit with Splitting of Operators) family of algorithms which solve a pressure-correction equation transformed from the continuity equation separately from the momentum equations [7,16–18,21,22]. Demirdžić et al. [16] proposed a form of pressure–density coupling in the continuity equation that is applied at the pressure-correction step, which was later improved and extended in several other studies [17,23]. Although the extension of pressure-based schemes are more efficient at low-speed flows than the density-based methods, the segregated solution algorithm as well as the use of pressure and velocity instead of density and momentum as the conserved quantities are less amenable to the well-established theory of hyperbolic equation systems that is typically the basis for fully-coupled density-based frameworks. As a consequence, it is not straightforward to formally derive accurate shock-capturing schemes required at supersonic speeds within pressure-based frameworks. Wesseling and co-workers [2,18] presented simulation results for the shock tube problem using an all-speeds pressure-correction framework, which however uses a non-conservative form of the energy equation. This appears to cause inherent density and Mach number overshoots in the presence of strong shocks which do not vanish with mesh refinement [2]. Indeed, non-conservative formulations of the energy equation in terms of thermal energy, enthalpy or temperature appear to be commonly used for transient, pressure-based all-speeds frameworks [2,16–18,24]. Hence, it is to be expected that these numerical frameworks suffer from the same drawbacks as reported in [2] where significant overshoots have been observed when predicting density or related thermodynamic variables at strong shocks. The extension of the fully-coupled framework, which solves for the static pressure simultaneously with other flow variables, has been proposed by Chen and Przekwas [10] and Darwish and Moukalled [11].

However, none of the frameworks applicable to all speed regimes as presented above have demonstrated the capability to correctly predict transient flow behaviour at all Mach numbers. In fact, simulation results of pressure-based solvers for transient compressible flows are much more rare to find when compared to steady-state flow predictions used as benchmark cases in almost all published works. In some cases, transient flows have not been studied at all as the governing equations are formulated and solved in steady-state form, see for instance [7,23,25,26]. Transient simulation results for subsonic flows have been presented by Issa et al. [27] and Chen and Pletcher [28], whereas simulations of acoustic propagation via pressure-based frameworks have been reported by Moguen et al. [29]. Simulation results for shock tube problems using pressure-based formulations can be found in [2,18]. More recently, solver algorithms combining fully coupled density-based schemes with pressure-correction strategies derived from pressure-based frameworks have been developed and successfully