ARTICLE IN PRESS

Computers and Mathematics with Applications ▮ (▮▮▮▮) ▮▮▮–▮▮▮

Contents lists available at ScienceDirect

Computers and Mathematics with Applications

journal homepage: www.elsevier.com/locate/camwa

Computation of compressible and incompressible flows with a space–time stabilized finite element method

Deepak Garg *, Antonella Longo, Paolo Papale

Istituto Nazionale di Geofisica e Vulcanologia, Sezione di Pisa, Via Uguccione della Faggiola 32, I-56126 Pisa, Italy

ARTICLE INFO

Article history:
Received 9 September 2017
Received in revised form 7 March 2018
Accepted 15 March 2018
Available online xxxx

Keywords:
Discontinuous Galerkin
Stabilization
Shock
Flow separation

ABSTRACT

This paper presents the numerical results of compressible and incompressible flow problems through a unified approach based on a stabilized space–time finite element method. The numerical approach is continuous in space and discontinuous in time. The proposed method starts by the use of a stabilized space–time variational formulation, which allows the use of the same order interpolation functions for all solution variables. The numerical technique is tested through comparison with standard compressible and incompressible flow benchmarks. Compressible flow cases include 1D and 2D shock problems. Incompressible flow cases include lid-driven cavity flow and flow over a backward-facing step computed over a range of *Re* numbers. The results demonstrate high stability and accuracy of the numerical technique over a wide range of flow regimes, suggesting straightforward extension to many flow cases not yet investigated.

1. Introduction

The finite element method provides several stabilizers for handling complex problems in super/subsonic compressible/incompressible flows [1,2]. In the context of the Galerkin finite element method, time-dependent fluid flow is solved either with the semi-discrete approach or with the space–time formulation. In the semi-discrete approach the solution is advanced in time by discretizing the time domain by the finite difference method. In the space–time finite element method both space and time are discretized simultaneously, generating space–time slabs. For the basics of the space–time formulation the reader is referred to [3,4]. The space–time finite element formulation has been successfully used for various flow problems [5–12].

The computational methods for compressible and incompressible flows have been developed separately considering the issues related to numerical stability and the choice of variables associated with the flow problems. Various researchers have proposed ideas for a unified approach to compressible and incompressible flows. Weiss and Smith [13] proposed a unified time marching scheme by combining the ideas of low Mach number preconditioning and artificial compressibility. Karimian and Schneider [14] presented a collocated pressure-based method that works in both compressible and incompressible regimes. Xiao [15] employed the multi-integrated moments unified approach. Zienkiewicz et al. [16] introduced the characteristic-based split algorithm applicable to both compressible and incompressible flows. Many other works have also been done, see for example [17–19].

Hauke et al. [20] presented a unified finite element formulation for solving the compressible Navier–Stokes equations with different sets of variables. The numerical scheme uses the Galerkin least-squares (GLS) [21] and discontinuity capturing (DC) [22] stabilization operators to attain a stable solution. The numerical technique was tested in the incompressibility limit.

* Corresponding author.
E-mail address: deepak.garg@ingv.it (D. Garg).

https://doi.org/10.1016/j.camwa.2018.03.028

It was shown that the sets of variables including density do not achieve finite values in the flux matrices. The entropy and pressure primitive variables were shown to be suitable for both compressible and incompressible flows. Furthermore, it was demonstrated that pressure variables provide better results than entropy variables for incompressible flows at high Reynolds number while at the same time being easier to specify as boundary conditions.

In this article we investigate compressible and incompressible flows for a variety of problems of compressible low/high Mach/Reynolds number flows. In [20] the driven cavity flow was simulated up to *Re* 400. In this work we analyse the same case for *Re* up to 10,000. We also study the backward facing step reaching *Re* of 800, and several shock tube problems. The computations are carried out with pressure primitive variables. The numerical code is written with meta-template C++ programming using OpenMPI. In Section 2 we present the strong form of the initial–boundary value problem. Equations are written in general form for any independent set of variables that can be obtained by transformation from the conservative ones. In Section 3 we focus on the space–time variational formulation for the Navier–Stokes equations in pressure primitive variables with GLS stabilization followed by discretization and solution of the set of algebraic equations. Benchmarks for compressible and incompressible flows are presented in Section 4. The method is demonstrated to be effective and efficient over a wide range of conditions from incompressible to compressible, and for low to high *Re* numbers, leading us to conclude, in Section 5, that the numerical approach and code can be successfully used in a wide variety of flow problems having scientific and industrial interest.

2. Governing equations and boundary conditions

The mathematical model describing compressible viscous flows consists of the conservative system of Navier–Stokes equations. The system of equations in conservative variables with applied boundary conditions can be written in compact form as

$$\boldsymbol{U}_{,t} + \boldsymbol{F}^a_{i,i} = \boldsymbol{F}^d_{i,i} + \boldsymbol{S} \qquad \text{on } \Omega \subset \mathbb{R}^n,\ t > 0, \tag{1}$$

$$\boldsymbol{U} = \boldsymbol{U}_g \qquad \text{on } \partial\Omega^g,\ t > 0, \tag{2}$$

$$(\boldsymbol{F}^a_i + \boldsymbol{F}^d_i)\cdot \boldsymbol{n}_i = \boldsymbol{h} \qquad \text{on } \partial\Omega^h,\ t > 0, \tag{3}$$

$$\boldsymbol{U}(t=0) = \boldsymbol{U}_0 \qquad \text{on } \Omega_0 \subset \mathbb{R}^n, \tag{4}$$

where $()_{,t}$ denotes the Eulerian time derivative, Ω is the spatial domain in n dimensions and $\boldsymbol{U} = [\rho, \rho\boldsymbol{v}, \rho E]'$ is the vector of conservation variables. $\partial\Omega^g$ and $\partial\Omega^h$ are the Dirichlet and Neumann boundaries, respectively. $\boldsymbol{U}_g$ and $\boldsymbol{h}$ are the vectors of Dirichlet and Neumann data, respectively. $\boldsymbol{U}_0$ is the initial data vector. $\hat{\boldsymbol{n}}$ is the unit outward normal vector. $\boldsymbol{F}^a_i$ and $\boldsymbol{F}^d_i$ are the advective and diffusive flux vectors, respectively, in ith direction, and $\boldsymbol{S}$ is the source vector. The vectors are given by

$$\boldsymbol{F}^a_i = \begin{bmatrix} \rho v_i \\ \rho v_i \boldsymbol{v} + \boldsymbol{\delta_i} p \\ \rho v_i E + v_i p \end{bmatrix} \qquad \boldsymbol{F}^d_i = \begin{bmatrix} 0 \\ \boldsymbol{\tau_i} \\ \tau_{ij} v_j - q_i \end{bmatrix} \qquad \boldsymbol{S} = \begin{bmatrix} 0 \\ \rho\boldsymbol{b} \\ \rho(b_i v_i + r) \end{bmatrix}. \tag{5}$$

The system of equations (1) represent the conservation of mass, momentum and energy; ρ is the density; $\boldsymbol{v} = [v_i]'$ is the velocity vector; E is the total energy density; p is the pressure; $\boldsymbol{\tau} = [\tau_{ij}]$ is the viscous-stress tensor; $\boldsymbol{\delta}_i = \boldsymbol{\delta}\boldsymbol{e}_i$ and $\boldsymbol{\tau}_i = \boldsymbol{\tau}\boldsymbol{e}_i$, where $\boldsymbol{e}_i$ is the unit basis vector in the ith direction and $\boldsymbol{\delta} = [\delta_{ij}]$ is the Kronecker delta; $\boldsymbol{q} = [q_i]'$ is the heat-flux vector; $\boldsymbol{b} = [b_i]'$ is the body force vector per unit mass; r is the heat supply per unit mass; $[\,]'$ refers to the transpose of the vector; and the summation convention is assumed throughout. The total energy density is defined as $E = c_v T + |\boldsymbol{v}|^2/2$. The heat flux is defined as $\boldsymbol{q}_i = -\kappa T_{,i}$. The system is closed by an equation of state. In the compressible flow applications presented below we assume the ideal gas equation $p = \rho R T$, where $R = c_p - c_v$ is the gas constant. c_p and c_v are the specific heats at constant pressure and volume, respectively. For incompressible flows the equation of state corresponds to constant density. The compressible behaviour is characterized by means of two quantities corresponding to isobaric expansion coefficient α_p and isothermal coefficient of compressibility β_T, defined as

$$\alpha_p = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)_p \qquad \beta_T = \frac{1}{\rho}\left(\frac{\partial\rho}{\partial p}\right)_T.$$

For an ideal gas the coefficients take the values of $\alpha_p = 1/T$ and $\beta_T = 1/p$. For incompressible flow both coefficients are zero.

Eq. (1) can be rewritten for any independent set of variables $\boldsymbol{Y}$ as

$$\boldsymbol{A}_0 \boldsymbol{Y}_{,t} + \boldsymbol{A}_i \boldsymbol{Y}_{,i} = (\boldsymbol{K}_{ij}\boldsymbol{Y}_{,j})_{,i} + \boldsymbol{S}, \tag{6}$$

where $\boldsymbol{A}_0 = \boldsymbol{U}_{,\boldsymbol{Y}}$, $\boldsymbol{A}_i = \boldsymbol{F}^a_{i,\boldsymbol{Y}}$ is the ith Euler Jacobian matrix and $\boldsymbol{K} = [\boldsymbol{K}_{ij}]$ is the diffusivity matrix with $\boldsymbol{K}_{ij}\boldsymbol{Y}_{,j} = \boldsymbol{F}^d_i$.

3. Space–time variational formulation

In the space–time finite element method both space and time are discretized simultaneously by taking a tensor product of basis functions for the spatial domain and a one-dimensional basis function in the time direction. Let $Q = \Omega \times (0, t_{tot})$ be