Available online at www.sciencedirect.com
ScienceDirect
Comput. Methods Appl. Mech. Engrg. ▮ (▮▮▮▮) ▮▮▮–▮▮▮
Computer methods in applied mechanics and engineering
www.elsevier.com/locate/cma

Solution of low Mach number aeroacoustic flows using a Variational Multi-Scale finite element formulation of the compressible Navier–Stokes equations written in primitive variables

Camilo Bayona[a,b], Joan Baiges[b,*], Ramon Codina[a,b]

[a] *Centre Internacional de Mètodes Numèrics en Enginyeria (CIMNE), Edifici C1, Campus Nord UPC, C/Gran Capità, S/N, 08034 Barcelona, Spain*
[b] *Universitat Politècnica de Catalunya, Jordi Girona 1-3, Edifici C1, 08034 Barcelona, Spain*

Received 28 July 2017; received in revised form 19 January 2018; accepted 22 January 2018
Available online xxxxx

Abstract

In this work we solve the compressible Navier–Stokes equations written in primitive variables in order to simulate low Mach number aeroacoustic flows. We develop a Variational Multi-Scale formulation to stabilize the finite element discretization by including the orthogonal, dynamic and non-linear subscales, together with an implicit scheme for advancing in time. Three additional features define the proposed numerical scheme: the splitting of the pressure and temperature variables into a relative and a reference part, the definition of the matrix of stabilization parameters in terms of a modified velocity that accounts for the local compressibility, and the approximation of the dynamic stabilization matrix for the time dependent subscales. We also include a weak imposition of implicit non-reflecting boundary conditions in order to overcome the challenges that arise in the aeroacoustic simulations at low compressibility regimes. The order of accuracy of the method is verified for two- and three-dimensional linear and quadratic elements using steady manufactured solutions. Several benchmark flow problems are studied, including transient examples and aeroacoustic applications.

Keywords: Compressible flow; Aeroacoustics; Low Mach numbers; Non-reflecting boundary conditions; Variational Multi-Scale (VMS)

1. Introduction

The increasing amount of computing resources available in scientific and industrial research has motivated the solution of realistic computational fluid dynamics applications. This is the case of aerodynamics and aeroacoustics; both described by a solution of the compressible Navier–Stokes equations that spans a wide range of scales. An illustrative application of this problem is the simulation of the human voice production inside the mouth, in which the solution of the relevant (small) scales of turbulence and sound waves must be achieved at low compressibility

* Corresponding author.
E-mail addresses: cbayona@cimne.upc.edu (C. Bayona), joan.baiges@upc.edu (J. Baiges), ramon.codina@upc.edu (R. Codina).

https://doi.org/10.1016/j.cma.2018.01.040

conditions. Indeed, compressible turbulent flows and sound waves propagation require an accurate computational method: the small fluctuating scales of sound need to be described, either by higher precision numerical schemes, or by small mesh and time step sizes. In any case, the number of operations required by the computational simulation is very demanding.

Another important requirement for such aeroacoustic problem is that the numerical approximation of the compressible Navier–Stokes equations at low compressibility conditions must be accurate, but many times those methods are not designed to work in these conditions. Most of the compressible flow solvers found in the literature exhibit a degradation in the solution accuracy when the free stream Mach number is progressively reduced. As explained in [1], most of the compressible flow solvers suffer a mismatch between the numerical and the continuous fluxes, and this is principally attributed to the broad difference in length and time scales of the solution. A possible approach for simulating aeroacoustic problems is to solve an aeroacoustic model together with the incompressible Navier–Stokes equations (e.g. reference [2]), but this leads directly to discarding the important physics related to the sound wave production, and consequently, this possibility is discarded in the present article. The type of methods that are intended to be suitable for either compressible or incompressible flows are the so-called *unified methods*. Among the most remarkable physical models, some modify the incompressible Navier–Stokes equations including extra term (like in the Boussinesq or Low Mach formulations described in [3], and references therein). There are also numerical approximations like [4–8], that aim to be valid at any compressibility regime. These unified approaches lead to more or less accurate descriptions of the fluid flow, but again, they cannot be used in wave propagation problems.

A formulation of the compressible Navier–Stokes equations that can be suitable for the low Mach number limit (or both for compressible and incompressible flows) is important for many reasons. One is that very low Mach numbers can coexist with regions where the flow compressibility becomes considerable. But also, several applications that are traditionally solved with incompressible formulations can be successfully handled by suitable compressible formulations. In this sense, it is important to remark that conservative variables (namely density, momentum, and total energy) are typically used in the compressible formulation, while primitive variables (pressure, velocity, and temperature) are used in the incompressible equations. Including density as a variable in the compressible problem (like in the conservative formulation), results in singularities for the convective terms in the low Mach number limit. Possibilities restrain to primitive and entropic unknowns, which for certain regularity conditions can be shown to satisfy the incompressible equations in the zero Mach number limit. The entropy variable formulation assure a global entropy stability condition, but it is subject to the definition of the entropy function. For more details on entropy variable formulations we refer to [9,10]. In the case of the compressible Navier–Stokes equations written in primitive variables, the formulation simplifies to the incompressible equations when the incompressible constraint is included (as explained in [11]), and consequently, it is well defined in the low Mach number limit.

Another particular aspect of the nearly incompressible aeroacoustic flows is that computational boundaries may cause artificial wave reflections related to the ingoing part of the sound waves. Ingoing waves may not only interfere with the acoustic signal relevant to the problem, but they can also produce numerical instabilities if the numerical method is not able to provide enough dissipation [12]. Several approaches to counteract spurious reflections have emerged in the computational aeroacoustics field, among the most popular are: the explicit damping of the compressible equations, the addition of an artificial counter signal, and the solution of non-reflecting boundaries. Damping some terms of the compressible Navier–Stokes equations may be a robust approach that filters the solution in some local regions (called buffer zones), but the computational effort that is required to solve those extra terms is large, and it has led to other approaches, one of which has been to remove the computational domain that is not of prior interest and to solve a non-reflecting boundary condition. The recent literature on non-reflecting boundary conditions is too vast to survey here, but we mention the early work in [13] using asymptotic sets of non-reflecting conditions with pseudo-differential operators, the heuristic radiation arguments in [14], or the asymptotic expansion matching the linearized Euler solution in [15], and refer to the review article [16] for a deeper understanding. Most of non-reflecting boundary methods attempt to damp the incident wave coming from the interior part of the domain.

As commented before, the small fluctuating scales of compressible turbulent flows and sound waves propagation require higher precision numerical schemes in both the temporal and the spatial discretization. This article is focused on the Finite Element Method (FEM) discretization. Among its advantages are the capability of using unstructured meshes to discretize complex geometries, and the possibility of developing efficient high order methods, which are two features highly desired in the Computational Fluid Dynamics (CFD) community. Nonetheless, when the Galerkin method is used to approximate this problem, which possesses non-symmetric operators, an unstable behavior of the

Please cite this article in press as: C. Bayona, et al., Solution of low Mach number aeroacoustic flows using a Variational Multi-Scale finite element formulation of the compressible Navier–Stokes equations written in primitive variables, Comput. Methods Appl. Mech. Engrg. (2018), https://doi.org/10.1016/j.cma.2018.01.040.

solution might appear generated, for instance, by unresolved boundary layers. These local instabilities may arise when convection is dominant, and also due to the inf–sup stability condition that restrain the interpolation compatibility of the different variables. Stabilized numerical methods like the Petrov Galerkin Streamline Upwind (SUPG), and the methods that can be framed in the Variational Multi-Scale (VMS) concept, add a stabilization term to the Galerkin formulation, and eliminate both the inf–sup and the convection restrictions. Stabilized finite element methods have been widely applied to both compressible and incompressible formulations, as shown in [17,18], and references therein. See also [19] for a review of VMS methods in CFD. Because only subsonic flows are considered in this study, and more specifically, we restrict the attention to low Mach number flows, the extra control over the gradients of the solution given by discontinuity capturing operators is not required, and consequently, shock capturing techniques are not surveyed here.

In this context, we center the review to the well established VMS stabilization method. This approach splits the unknowns of the problem into a coarse-scale that belongs to the finite element space and a subgrid scale or subscale, which is the remainder. The main idea of the VMS formulation is to include the effect of the unresolved scales in order to stabilize the finite element equations. As a consequence, the correct design of the stabilization operator used to represent the subscales in terms of the resolved scales is crucial for the accuracy and stability of the discretization. Classical stabilized compressible flow formulations, including the compressible Euler equations, give inaccurate results near the low Mach number limit mainly because the standard definitions of the stabilization operator fail to provide adequate numerical diffusion. In order to overcome this difficulty, non diagonal definitions have been proposed (e.g. by [10]). Nonetheless, the diagonal definition can be used with few modifications: this has been the approach taken by [1,20], in which dimensional analysis is used to design a compressible stabilization matrix that is suitable for both compressible and incompressible flows.

In the present work, the primitive variables compressible formulation is used because of its correspondence with the incompressible Navier–Stokes equations in the incompressible limit. Some other improvements are included in the formulation. One important aspect is the use of an implicit time integration scheme in order to avoid the inefficiency that arises from the aeroacoustic propagation in the low Mach number limit. In this sense, and depending on the system of units used for the solution of the problem, the transient problem can include very large quantities in the right-hand-side (RHS) of the continuity and energy equations, which degrade the iterative solution of the discrete linear system. From our perspective, this lack of scaling of the discrete linear system can be healed by splitting the primitive pressure and temperature unknowns into a relative and a reference part. The idea presented here is to change the primitive unknowns to the relative ones, but taking care for including the original unknowns in the non-linear terms of the problem. From the numerical point of view, this transformation not only scales the problem, but also allows to implement iterative linear system solvers in the case of transient problems. We also incorporate some particular features of the VMS framework that we have applied in other flow problems (like in [21,22], for example). Since there are different ways to define the subscales, three different attributes are studied in this work. The first attribute is the definition of the space of the subscales, which can be either the space of finite element residuals, or the space orthogonal to the finite element space [23]. The second attribute is the inclusion of the transient term of the subscale equation [24]. The third attribute is the inclusion of the subscales in all the non-linear terms of the problem [25]. In addition to these definitions, we design the stabilization operator in terms of the local compressibility of the flow, which, in the low Mach number limit, converges to the usual incompressible stabilization operator definition for the continuity and momentum equations. The proposed stabilization operator includes a physical parameter that defines the transition between the compressible and incompressible flow, hence, it combines the features of both stabilized formulations and is capable of describing the range of subsonic Mach numbers.

The focus of the present work is also to investigate the nearly incompressible applications of the compressible flow solver, which involve aeroacoustic applications. For doing so, we implement the characteristic decomposition of waves as a non-reflecting method over the boundaries and prescribe an implicit solution of this method in a weak sense by means of introducing some penalization terms to the compressible formulation. This decomposition of waves, introduced by Thompson in [26], together with the non-reflecting boundary conditions for the Navier–Stokes equations given by Poinsot and Lele [27], is able to obtain a damped solution at the boundary using a simplification of the conservation equations that are solved inside the computational domain. Instead of solving the compressible Navier–Stokes equation over the boundary, the method annihilates the in-going sound wave by exploiting the characteristics of the Euler equations, where viscous and reaction terms are neglected. Research on annihilating methods for spurious sound wave reflections of the computational boundaries is not the original

Please cite this article in press as: C. Bayona, et al., Solution of low Mach number aeroacoustic flows using a Variational Multi-Scale finite element formulation of the compressible Navier–Stokes equations written in primitive variables, Comput. Methods Appl. Mech. Engrg. (2018), https://doi.org/10.1016/j.cma.2018.01.040.

motivation for the present study. More details about the non-reflecting boundary conditions applied to aeroacoustic flow solvers may be found in articles such as [28,29].

The article is organized as follows. We recall the primitive variable formulation of the compressible Navier–Stokes equations in the first part of Section 2. Then we describe the transformation of the primitive variables into a relative and a reference part. Later in that section we present the governing equations for the non-reflecting boundary conditions. In Section 3, we apply the VMS framework in order to discretize in space; we focus our attention to the construction of the stabilization operator in order to properly reach the low Mach number limit. The implicit scheme for advancing in time, and the weak imposition of the non-reflecting conditions are also included in that section. In Section 4, we validate the numerical method for two- and three-dimensional applications by using steady manufactured solutions. We also simulate the differential heated cavity and the flow past a cylinder unsteady problems. Two numerical examples are solved as aeroacoustic applications: the Aeolian tones generated by the flow past a cylinder, and the flow past an open cavity. Finally, we address some conclusions in Section 5.

2. Governing equations

In this section we present the compressible Navier–Stokes equations written in primitive variables. For doing this we depart from the conservative formulation, and express the system in terms of the primitive vector of unknowns. The Jacobian and diffusive matrices are therefore written, and the specific formulation for the ideal gas law is included. Then we implement a change of variables, which allows us to scale the discrete linear system. In the last part we describe the governing equations for the non-reflecting boundary conditions.

2.1. Compressible Navier–Stokes equations written in conservation form

The problem we consider consists in the compressible Navier–Stokes equations posed in a time interval $\left(0, t_f\right)$ and in a domain $\Omega \subset \mathbb{R}^d$, being d the number of space dimensions ($d = 2$ or 3). Let $t \in \left(0, t_f\right)$ be a given time instant in the temporal domain, and $\boldsymbol{x} \in \Omega$ a given point in the spatial domain. Let Γ be the boundary of the domain Ω, and $\boldsymbol{n}$ the geometric unit outward normal vector on Γ. We split Γ into two sets: the *Dirichlet* boundary denoted as Γ_G, and the *Neumann* boundary denoted as Γ_N. The strong form of the initial and boundary-value problem consists of finding the solution vector $\boldsymbol{U} : \Omega \times (0, t_f) \to \mathbb{R}^{d+2}$, where $d+2$ is the number of unknowns (the same as equations of the system), such that for the given Dirichlet boundary conditions $\boldsymbol{U}_G : \Gamma_G \times (0, t_f) \to \mathbb{R}^{d+2}$ and the Neumann boundary conditions $\boldsymbol{H} : \Gamma_N \times (0, t_f) \to \mathbb{R}^{d+2}$, the following compressible Navier–Stokes equations are satisfied:

$$\partial_t \boldsymbol{U} + \partial_j \boldsymbol{E}_j(\boldsymbol{U}) + \partial_j \boldsymbol{G}_j(\boldsymbol{U}) = \boldsymbol{F} \qquad \text{in} \quad \Omega \subset \mathbb{R}^d, \quad t \in \left(0, t_f\right), \tag{1}$$

$$\boldsymbol{D}(\boldsymbol{U}) = \boldsymbol{D}(\boldsymbol{U}_G) \qquad \text{on} \quad \Gamma_G, \quad t \in \left(0, t_f\right), \tag{2}$$

$$\boldsymbol{B}(\boldsymbol{U}) = \boldsymbol{H} \qquad \text{on} \quad \Gamma_N, \quad t \in \left(0, t_f\right), \tag{3}$$

$$\boldsymbol{U} = \boldsymbol{U}^0(\boldsymbol{x}) \qquad \text{in} \quad \Omega \subset \mathbb{R}^d, \quad t = 0. \tag{4}$$

Here ∂_t and ∂_j are short notations that indicate the Eulerian time derivative and $\partial/\partial x_j$, respectively. The usual summation convention is implied in the equation presented before, with indices running from 1 to d. The Dirichlet boundary operator $\boldsymbol{D}(\cdot)$ is used to impose the components of $\boldsymbol{U}$ on different parts of Γ. The Neumann boundary conditions are given by the operator $\boldsymbol{B}(\cdot)$. Note that with our notation Γ_G and Γ_N may overlap.

The vector of conservative variables $\boldsymbol{U}$, the *convective* flux in the jth-direction $\boldsymbol{E}_j$, the *diffusive* flux in the jth-direction $\boldsymbol{G}_j$, and the vector of forces $\boldsymbol{F}$ are defined as:

$$\boldsymbol{U} = \begin{bmatrix} \rho \\ \rho u_1 \\ \rho u_2 \\ \rho u_3 \\ e_{\text{tot}} \end{bmatrix}, \qquad \boldsymbol{E}_j(\boldsymbol{U}) = \begin{bmatrix} \rho u_j \\ \rho u_j u_1 + p\delta_{1j} \\ \rho u_j u_2 + p\delta_{2j} \\ \rho u_j u_3 + p\delta_{3j} \\ u_j(e_{\text{tot}} + p) \end{bmatrix}, \tag{5}$$

$$\boldsymbol{G}_j(\boldsymbol{U}) = \begin{bmatrix} 0 \\ -\tau_{j1} \\ -\tau_{j2} \\ -\tau_{j3} \\ -u_i\tau_{ij} + q_j \end{bmatrix}, \qquad \boldsymbol{F} = \begin{bmatrix} 0 \\ \rho \boldsymbol{f} \\ \rho \boldsymbol{f} \cdot \boldsymbol{u} + \rho r \end{bmatrix}.$$

In these relations ρ is the density, p is the pressure, $\boldsymbol{u}$ is the velocity, e_{tot} is the total energy, defined as $e_{\text{tot}} = \rho\left(e + \boldsymbol{u}\cdot\boldsymbol{u}/2\right)$, e being the internal energy, $\boldsymbol{f}$ is a body force vector, r is a heat source/sink term, $\boldsymbol{I} = \left[\delta_{ij}\right]$ is the identity or *Kronecker* tensor, $\boldsymbol{\tau}$ is the viscous stress tensor, which for a Newtonian fluid is defined as

$$\tau_{ij}(\boldsymbol{u}) = \mu\left(\partial_j u_i + \partial_i u_j\right) - \frac{2\mu}{3}\left(\partial_l u_l\right)\delta_{ij}, \tag{6}$$

and $\boldsymbol{q}$ is the heat flux vector, that we calculate using Fourier's law as

$$q_i(T) = -\lambda\partial_i T, \tag{7}$$

where μ is the viscosity, λ is the thermal conductivity, and T is the temperature of the fluid. The caloric equation $e = c_v(T)\,T$ and the ideal law for gases $p = \rho RT$ are used to calculate the pressure and the acoustic speed c. In these relations the specific heat at constant volume $c_v(T)$ and the specific heat at constant pressure $c_p(T)$ are thermodynamic properties of the fluid. We also define $\gamma = c_p/c_v$ for the ratio between the specific heats, and $R = c_p - c_v$ for the specific gas constant. The non-dimensional Mach number M$= |\boldsymbol{u}|/c$ is used to calculate the compressibility regime.

2.2. Compressible Navier–Stokes equations written in primitive variables

In order to formulate the equations using primitive variables, we express the conservative unknowns $\boldsymbol{U}$ as a function of the primitive variables $\boldsymbol{Y} = (p, \boldsymbol{u}, T)^{\top}$. Hereafter, let us denote the transpose operation by the superscript $\top$. Hence, Eq. (1) can be written as

$$\partial_t\boldsymbol{U}(\boldsymbol{Y}) + \partial_j\boldsymbol{E}_j(\boldsymbol{U}(\boldsymbol{Y})) + \partial_j\boldsymbol{G}_j(\boldsymbol{U}(\boldsymbol{Y})) = \boldsymbol{F}. \tag{8}$$

If $\boldsymbol{Y}$ is sufficiently smooth, it also satisfies the quasi-linear form

$$\boldsymbol{A}_0(\boldsymbol{Y})\,\partial_t\boldsymbol{Y} + \boldsymbol{A}_j(\boldsymbol{Y})\,\partial_j\boldsymbol{Y} - \partial_k\left(\boldsymbol{K}_{kj}(\boldsymbol{Y})\,\partial_j\boldsymbol{Y}\right) = \boldsymbol{F}. \tag{9}$$

Here we have written the divergence of the fluxes in Eq. (8) in a more convenient manner by defining the Jacobian matrices $\boldsymbol{A}_0(\boldsymbol{Y}) = \partial\boldsymbol{U}/\partial\boldsymbol{Y}, \boldsymbol{A}_j(\boldsymbol{Y}) = \partial\boldsymbol{E}_j(\boldsymbol{U}(\boldsymbol{Y}))/\partial\boldsymbol{Y}$, and the diffusivity matrix $\boldsymbol{K}(\boldsymbol{Y}) = \left[\boldsymbol{K}_{kj}(\boldsymbol{Y})\right]$, such that $\partial_j\boldsymbol{G}_j(\boldsymbol{U}(\boldsymbol{Y})) = -\partial_k\left(\boldsymbol{K}_{kj}(\boldsymbol{Y})\,\partial_j\boldsymbol{Y}\right)$.

In order to express all the previous relations in terms of the primitive variables alone, we make the supposition that the fluid is divariant and account for the ideal gas law. This makes possible to formulate density as a function of the pressure and the temperature, and to rearrange the Navier–Stokes equations (see [11,30]). Hence, the Jacobian matrices $\boldsymbol{A}_j(\boldsymbol{Y}),\ j = 0,\ldots,d$, can be formulated as functions of primitive variables and thermodynamic coefficients:

$$\boldsymbol{A}_0(\boldsymbol{Y}) = \begin{bmatrix} \rho\beta_t & \boldsymbol{0}^{\top} & -\rho\alpha_p \\ \rho\beta_t\boldsymbol{u} & \rho\boldsymbol{I} & -\rho\alpha_p\boldsymbol{u} \\ \rho\beta_t a_1 - \alpha_p T & \rho\boldsymbol{u}^{\top} & -\rho\alpha_p a_1 + \rho c_p \end{bmatrix},$$

$$\boldsymbol{A}_j(\boldsymbol{Y}) = \begin{bmatrix} \rho\beta_t u_j & \rho\boldsymbol{e}_j^{\top} & -\rho\alpha_p u_j \\ \rho\beta_t\boldsymbol{u}u_j + \boldsymbol{e}_j & \rho\boldsymbol{I}u_j + \rho\boldsymbol{u}\otimes\boldsymbol{e}_j & -\rho\alpha_p\boldsymbol{u}u_j \\ \left(\rho\beta_t a_1 - \alpha_p T + 1\right)u_j & \rho\boldsymbol{u}^{\top}u_j + \rho a_1\boldsymbol{e}_j^{\top} & \left(-\rho\alpha_p a_1 + \rho c_p\right)u_j \end{bmatrix},$$

for $j = 1,\ldots,d$,. Hereafter $\boldsymbol{0}$ is the vector of $\mathbb{R}^d$ with zero in all its components. The thermodynamic relation a_1 stands for $a_1 = c_v T + p/\rho + |\boldsymbol{u}|^2/2$, α_p is the volume expansivity, and β_t is the isothermal compressibility, given by

$$\alpha_p = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)_p \quad \text{and} \quad \beta_t = \frac{1}{\rho}\left(\frac{\partial\rho}{\partial p}\right)_T, \tag{10}$$

with $(\cdot)_p$ and $(\cdot)_T$ defining a constant pressure and temperature constrain, respectively. Additionally, the diffusive matrix $\boldsymbol{K}_{kj}(\boldsymbol{Y})$ is constructed as

$$\boldsymbol{K}_{jj}(\boldsymbol{Y}) = \begin{bmatrix} 0 & \boldsymbol{0}^{\top} & 0 \\ \boldsymbol{0} & \mu\boldsymbol{I} + \frac{1}{3}\mu\boldsymbol{e}_j\otimes\boldsymbol{e}_j & \boldsymbol{0} \\ 0 & \mu\boldsymbol{u}^{\top} + \frac{1}{3}\mu u_j\boldsymbol{e}_j^{\top} & \lambda \end{bmatrix} \quad \text{for } j = 1,\ldots,d, \text{ and}$$