Comput. Methods Appl. Mech. Engrg. 233–236 (2012) 109–122

Contents lists available at SciVerse ScienceDirect

Comput. Methods Appl. Mech. Engrg.

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

A fully coupled RANS Spalart–Allmaras SUPG formulation for turbulent compressible flows on stretched-unstructured grids

C. Wervaecke [a], H. Beaugendre [b,c,d,*], B. Nkonga [e]

[a] ONERA, Meudon, Département d'Aérodynamique Appliquée, France
[b] Univ. Bordeaux, IMB, UMR 5251, F-33400 Talence, France
[c] CNRS, IMB, UMR 5251, F-33400 Talence, France
[d] INRIA, Equipe projet BACCHUS, F-33400 Talence, France
[e] Université de Nice, INRIA Sophia Antipolis Equipe-projet PUMAS, France

A R T I C L E I N F O

Article history:
Received 7 September 2011
Received in revised form 19 March 2012
Accepted 4 April 2012
Available online 19 April 2012

Keywords:
Turbulence
Stabilized finite element method
SUPG
Unstructured grids
High aspect ratio
High order scheme

A B S T R A C T

When high Reynolds turbulent flows are combined with complex and large size geometries, computers are no longer enough powerful to deal with Direct Numerical Simulation (DNS) and with the resolution of all the scales of turbulence motion. Therefore, the RANS approaches solve averaged equations and use a model to simulate these scales. This model contains dissipation processes that should not be polluted by the numerical diffusion needed to stabilized approximations for convection-dominated flows. In this paper, we proposed a strongly coupled numerical formulation for the Spalart–Allmaras model, in the framework of stabilized finite element methods. Computations are performed for compressible Newtonian fluids (2D and 3D) on unstructured grids of high aspect ratio. Results are compared with experimental data and also with solutions obtained by different numerical strategies.

1. Introduction

One of the most important challenges in Computational Fluid Dynamics (CFD) is the simulation of high Reynolds turbulent flows around complex geometries of large size. Indeed, most of validated production tools used in the industrial context aeronautics, turbomachinery, and combustion engines always assume simplified strategies where turbulent scales are modeled by additional transport equation(s) coupled with the Navier–Stokes equations. There is an important variety of reduced models for the eddy viscosity as a parameter or a variable. Among all these models, the single eddy viscosity transport-type equation of [43] (SA), is one of the simplest and most popular strategy, especially for aerodynamic flows. The SA model, and its improved versions, provides some calibrated mechanisms for eddy production, dissipation and destruction that can be managed in a large range of applications. However, the numerical scheme used to discretize the model should be carefully designed in order to avoid the annihilation of the subscale turbulence model mechanism by the numerical dissipation. Indeed, for convection dominated flows under consideration, there is a need of numerical dissipation mechanism in order to stabilized the numerical scheme.

For finite volumes/finite differences approaches, numerical stabilization is achieved through an upwinding that is for example embedded in the Godunov-type fluxes. However the associated Riemann problems are solved in the directions of the normals to the mesh faces. As a consequence the numerical diffusion is highly dependent on the mesh topology even for high order MUSCL versions. This is very damaging for eddy viscosity transport, especially since anisotropic meshes are often used for simulations of turbulent flows around obstacles. To overcome this difficulty, it is strongly recommended to use structured grids in the boundary layer. Indeed, these meshes provide the correct directions for the Riemann fluxes and reduce crosswind diffusion. This recommendation must face the difficult problem of structured grids generation for simulations over complex geometries of aeronautics design. Other alternatives, in the context of finite volumes/finite difference schemes, is to either use techniques for the control of crosswind diffusion, as in [30,3,12] or the use of genuinely upwind solvers, as in [13,1]. This research topic is still in progress. When the standard Galerkin finite element method based on $\mathcal{C}^0$ polynomial approximation is applied to convection-dominated flows, unphysical oscillations related to

* Corresponding author at: Univ. Bordeaux, IMB, UMR 5251, F-33400 Talence, France.
E-mail address: heloise.beaugendre@math.u-bordeaux1.fr (H. Beaugendre).

0045-7825/$ - see front matter © 2012 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.cma.2012.04.003

the absence of upwinding are obtained. Christie et al. [11] for one-dimensional case, [19] for two-dimensional case, and [20] for one-dimensional quadratic elements, proposed stable schemes for the steady-state advection–diffusion equation. In this context, stabilization is achieved by the use of Petrov–Galerkin formulation where test functions are different to trial functions. The test functions are locally defined so as to send more information downstream. Exact nodal values can even be obtained in one-dimensional case. This formulation also suffers of arbitrary crosswind diffusion. In the beginning of the 1980s, [23] pointed out that to derive stables schemes, it is sufficient to add an artificial diffusion term only in the direction of streamlines. They observed that such a streamline diffusion term can be introduced in the standard Galerkin method without modifying the original governing equation. The method consists in perturbing the weighting functions with their derivatives, thus resulting in a high-order accurate method with good stability properties. The Streamline Upwind Petrov Galerkin (SUPG) stabilized finite element method, provides a general framework to define the numerical dissipation, by proper definition of the stabilization tensor. It allows to obtain a stable numerical scheme any damaging mechanism for unresolved turbulence scales. SUPG is a residual-based upwinding approach proposed in [9] for advection–diffusion and incompressible Navier–Stokes equations. SUPG has been later extended to the compressible Navier–Stokes equations (see [24,45]) in the context of conservative variables. Since the first developments of this method in the eighties, the SUPG scheme has been widely used and improved. The formulation was rewritten using entropy variables plus a shock-capturing term added to handle discontinuities in [25]. It has been shown that the formulation written using conservative variables and supplemented with a shock-capturing term is as accurate as the formulation using entropy variables [5]. Later on the SUPG scheme has been applied to the simulation of turbulent flows [31,42,7]. In such problems, a finite element approach where dissipation can be controlled is a promising method. It was also shown by Hughes [22] that stabilized methods could be derived within the general framework of variational multiscale formulation (VMS). For a given set of equations, the VMS framework provides attractive guidelines for the development of stabilized schemes. VMS also provides tools for the Large-Eddy-Simulations (LES) and has been intensively used in this field as in [26] and in [33]. When the goal is to stabilize the resolved scales, the main issue in VMS is to derive an approximation (algebraic) of fine scales and the impact on coarse scales. This stabilization process was earlier defined by a parameter and recently new ways of defining it based on element-level matrices and vectors were introduced. The amount of additional diffusion introduced in SUPG formulations is tuned by a tensor parameter $\underline{\tau}$ that must be chosen in a suitable way. In the context of compressible Euler and Navier–Stokes equations, according to a lot of numerical tests, several recipes have been proposed for the choice of $\underline{\tau}$ (see for example [42,46]). The need for a suitable convincing argument to guide the choice of $\underline{\tau}$ is still considered as a major drawback of SUPG methods. In presence of strong shocks, an additional discontinuity-capturing term is added to the SUPG scheme, [28], to remove specific oscillations produced in this context and improve the robustness of the numerical approach.

This paper is concerned with a strongly coupled and accurate numerical approximation of the SA turbulence model within the framework of stabilized finite element method for unstructured anisotropic grids. The additional transport equations for subscale model are often numerically weakly coupled to Navier–Stokes equations through operator splitting. These variables are strongly coupled for the transport process within a stabilized finite element formulation. The stabilization tensor is defined, such as to reduce mesh dependencies and to still be consistent at the asymptotic of highly anisotropic meshes. Indeed, this tensor involves a measure of the local length scale h which should be carefully defined in the case of a stretched element. In this work, the local length scale is implicitly given by the inverse of the absolute flux jacobian matrix as proposed in [4] and more recently in [2]. The stabilized finite element strategy is also suitable for complex geometries and the resulting schemes have a compact stencil which we exploit for efficient parallel strategies combining domain decomposition and message passing tools (MPI). The paper is organized as follows: in Section 2 the description of the Spalart–Allmaras model fully coupled with the Navier–Stokes equations is given. Section 3 deals with the SUPG formulation for this type of turbulent compressible flows, and Section 4 is devoted to the numerical results. Finally, concluding remarks are given in Section 5.

2. Spalart–Allmaras turbulence model for compressible flows

2.1. Navier–Stokes equations

We consider a turbulent flow described by compressible Navier–Stokes equations coupled with an eddy viscosity transport equation. The conservative form of the Reynolds Average Navier–Stokes equations can be written as follows (Gravity effects are assumed negligible):

$$\begin{aligned}
&\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial \boldsymbol{x}} \cdot (\rho \boldsymbol{u}) = 0 \\
&\frac{\partial}{\partial t}(\rho \boldsymbol{u}) + \frac{\partial}{\partial \boldsymbol{x}} \cdot (\rho \boldsymbol{u} \otimes \boldsymbol{u}) + \frac{\partial p}{\partial \boldsymbol{x}} = \frac{\partial}{\partial \boldsymbol{x}} \cdot \underline{\boldsymbol{\pi}} \\
&\frac{\partial}{\partial t}(\rho e) + \frac{\partial}{\partial \boldsymbol{x}} \cdot ((\rho e + p)\boldsymbol{u}) = \frac{\partial}{\partial \boldsymbol{x}} \cdot (\underline{\boldsymbol{\pi}}\boldsymbol{u} + \boldsymbol{q})
\end{aligned} \tag{1}$$

where ρ is the density, $\boldsymbol{u}$ the velocity, e the specific total energy. The pressure p and the heat flux $\boldsymbol{q}$ are respectively defined by the perfect gas equation of state and the Fourier law:

$$p = (\gamma - 1)\rho T \quad \text{and} \quad \boldsymbol{q} = -\frac{c_p(\mu + \mu_t)}{Pr}\frac{\partial T}{\partial \boldsymbol{x}} \quad \text{with} \quad T = e - \frac{1}{2}|\boldsymbol{u}|^2.$$

Pr is the laminar Prandtl number, c_p is the heat capacity at constant pressure, T is the temperature. For newtonian compressible fluids the stress tensor is given by

$$\underline{\boldsymbol{\pi}} = (\mu + \mu_t)\left(\left[\frac{\partial \boldsymbol{u}}{\partial \boldsymbol{x}}\right] + \left[\frac{\partial \boldsymbol{u}}{\partial \boldsymbol{x}}\right]^T - \frac{2}{3}\left[\frac{\partial}{\partial \boldsymbol{x}} \cdot \boldsymbol{u}\right]\underline{\boldsymbol{Id}}\right)$$

where μ and μ_t are respectively the laminar and the turbulent viscosities. In order to close these equations we need either an algebraic formula or an additional system of equations to define μ_t.

2.2. Turbulence closure model

The Spalart–Allmaras (SA) model provides a single equation for the evolution of the turbulent kinematic viscosity $\nu_t = \mu_t/\rho$. This is an empirical and powerful equation that models production, transport, diffusion and destruction of the turbulent viscosity. In [43], the method is described for incompressible flows. Although there are different approaches to adapt the model for compressible flows, we consider in the sequel the following extension

$$\frac{\partial \rho \tilde{\nu}}{\partial t} + \frac{\partial}{\partial \boldsymbol{x}} \cdot (\rho \tilde{\nu} \boldsymbol{u}) = M(\tilde{\nu})\tilde{\nu} + P(\tilde{\nu})\tilde{\nu} - D(\tilde{\nu})\tilde{\nu} \tag{2}$$

where $M(\tilde{\nu})\tilde{\nu}$ represents the diffusion term, $P(\tilde{\nu})\tilde{\nu}$ the production source term and $D(\tilde{\nu})\tilde{\nu}$ the wall destruction source term. The eddy viscosity is obtained from $\tilde{\nu}$ via

$$\nu_t = \tilde{\nu} f_{v1}, \quad f_{v1} = \frac{\chi^3}{\chi^3 + c_{v1}^3}, \quad \chi = \frac{\tilde{\nu}}{\nu} \tag{3}$$

where ν is the molecular viscosity. The production source term is given by