Computers & Fluids 119 (2015) 37–46

Contents lists available at ScienceDirect

Computers & Fluids

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

A numerical dissipation rate and viscosity in flow simulations with realistic geometry using low-order compressible Navier–Stokes solvers

G. Castiglioni *, J.A. Domaradzki

Department of Aerospace and Mechanical Engineering, University of Southern California, Los Angeles, CA 90089-1191, USA

ARTICLE INFO

Article history:
Received 10 January 2015
Received in revised form 18 May 2015
Accepted 3 July 2015
Available online 6 July 2015

Keywords:
LES
Numerical dissipation
Numerical viscosity
Laminar separation bubble

ABSTRACT

Recently it has become increasingly clear that the role of numerical dissipation, originating from the discretization of governing equations of fluid dynamics, rarely can be ignored regardless of the formal order of accuracy of a numerical scheme used in either explicit or implicit Large Eddy Simulations (LES). The numerical dissipation inhibits the predictive capabilities of LES whenever it is of the same order of magnitude or larger than the sub-grid-scale (SGS) dissipation. The need to estimate the numerical dissipation is most pressing for low-order methods employed by commercial CFD codes. Following the recent work of Schranner et al. (2015) the equations and procedure for estimating the numerical dissipation rate and the numerical viscosity in a commercial code are presented. The method allows to compute the numerical dissipation rate and numerical viscosity in the physical space for arbitrary sub-domains in a self-consistent way, using only information provided by the code in question. The procedure has been previously tested for a three-dimensional Taylor–Green vortex flow in a simple cubic domain and compared with benchmark results obtained using an accurate, incompressible spectral solver. In the present work the procedure is applied for the first time to a realistic flow configuration, specifically to a laminar separation bubble flow over a NACA 0012 airfoil at $Ma = 0.4$ and $Re = 50,000$. The method appears to be quite robust and its application reveals that for the code and the flow in question the numerical dissipation can be significantly larger than the viscous dissipation or the dissipation of the classical Smagorinsky SGS model.

1. Introduction

Direct Numerical Simulations (DNS) of turbulent flows are excessively computationally expensive for complex geometries and/or high Reynolds number flows due to the wide separation of physical scales that need to be resolved. A relatively successful way to reproduce the dynamics of Navier–Stokes (N–S) equations while reducing the number of degrees of freedom is the Large Eddy Simulations (LES) approach. In LES the number of degrees of freedom is reduced by means of a spatial filter that suppresses the effects of small scales at the cost of introducing sub-grid scale (SGS) unknowns (i.e. for the incompressible N–S the SGS stress tensor) which must be explicitly modeled [27,29,14].

An alternative approach is to use the numerical dissipation originating from the discretization of the N–S equations as an implicit LES (ILES) model. The strategy of using the numerical dissipation as an implicit model relies on the hypothesis that the effect of the SGS terms on the resolved scales is primarily dissipative [14]. This approach was proposed by Boris et al. [2] who utilized a Flux-Corrected Transport (FCT) scheme. The FCT method assures the monotonicity of the solution, which is why the original approach was dubbed monotonically integrated LES (MILES). This approach has been expanded to other monotonic and non-monotonic schemes and has been more generally renamed as ILES. An example of a non-monotonic ILES is the compact finite difference scheme stabilized by filters [13]. For a thorough discussion on the ILES approach one can refer to the ILES monograph of Grinstein et al. [19]. The MILES approach has been controversial and as such it has been the object of rigorous investigations [15,11]. These studies have not been particularly encouraging. Even when MILES appears to reproduce qualitatively the dynamics of N–S equations, a more in-depth, quantitative investigation has shown that this is not the case. Broadly speaking, these studies present two scenarios. Either the numerical dissipation is excessive with respect to the correct SGS dissipation leading to poor results both in ILES and explicit LES (ELES) configurations [15], or the scheme is under-dissipative (with respect to the correct SGS dissipation) leading to good results for short time integrations and poor results for long time integrations due to accumulation of energy in

* Corresponding author. Tel.: +1 213 400 2514.
E-mail address: castigli@usc.edu (G. Castiglioni).

http://dx.doi.org/10.1016/j.compfluid.2015.07.004
0045-7930/

the high wave numbers [11]. The latter case can potentially be adjusted either by filtering or with the addition of an explicit SGS model [31]. These examples show that care is needed to design a scheme that would produce accurate results with the ILES approach, since there must be a mechanism embedded in the numerics to ensure the correct amount of SGS dissipation. In Chapter 5 of the ILES monograph Grinstein et al. [19], Rider and Margolin suggest that not all non-oscillatory schemes make good ILES models. Through the modified equation analysis of several non-oscillatory schemes, Rider and Margolin show that a good ILES must have a truncation error that exhibits a scaling consistent with the scaling dictated by the physics of turbulence. In particular the poor performance observed by Garnier et al. [14] is attributed to the specific implementation of the scheme chosen for the simulations. At any rate, the ILES approach has been gaining more popularity due its simplicity and good qualitative results for specific flows, even if often lacking a detailed comparisons with DNS results. In ILES the numerics are rarely designed in such a way that the numerical dissipation matches, a priori, the physical SGS dissipation. For a properly designed ILES such a matching should be attempted at least for canonical turbulent flows, e.g., isotropic turbulence or the turbulent channel flow. Whenever the numerics are not constrained to reproduce the correct amount of numerical dissipation, ILES may not provide even basic quantities correctly such as the log-law of the wall or the skin friction. An example of a proper ILES implementation is the adaptive local deconvolution method (ALDM) of Hickel et al. [20], where the discretization is based on a solution-adaptive deconvolution operator which allows for control of the truncation error so that the numerical viscosity matches the values predicted for isotropic turbulence by turbulence theories. Early ELES studies pointed out that low-order numerical methods are not suitable in the ELES framework as the interaction between numerical dissipation and the SGS dissipation [24] would negatively affect the results, while for high-order/spectral methods the leading source of error is aliasing of the non-linear term. In industrial applications the use of solvers based on unstructured grid is almost compulsory, such solvers are typically second-order (for specific examples refer to the collection of solvers used in the LESFOIL project, Davidson et al. [9]). Unstructured high-order solvers are usually prohibitively computationally expensive. With this in mind, efforts have been made in order to reduce the numerical dissipation in low-order schemes. For instance Camarri et al. [4] utilize a low-diffusion MUSCL-type scheme on a finite volume tetrahedral grid stabilized by a numerical diffusion term based on sixth order derivatives. Another example is the low-dissipative third-order Taylor–Galerkin scheme of Colin and Rudgyard [8].

Recently it has been shown that at very coarse resolutions even formally high-order methods can suffer from the interaction between numerical dissipation and SGS dissipation making the addition of an ELES model detrimental to the performance of the code [3].

Despite the issues mentioned above, flow simulations without explicit LES models are becoming more and more popular. The reason for this trend and the attractiveness of the approach are due to several reasons: its simplicity; a resulting qualitative behavior that mimics the dynamics of N–S equations; and the lack of a universal explicit SGS model that could guarantee clearly superior results in all situations. ILES results are often validated with experimental results which may suffer from a high degree of uncertainty for many complex flows of interest in engineering. Nonetheless, recent comparisons between ILES, LES and DNS for Taylor–Green Vortex simulations have shown that ILES approaches can outperform explicit LES models (see Hickel et al. [20], Wachtor et al. [32], Gassner and Beck [16]). Due to the increasing popularity of the ILES approach it is prudent to make a formal distinction between ILES schemes that are designed to provide the correct amount of SGS dissipation and those that are simply run without an explicit model, with no constraints imposed on the numerical dissipation. We will refer to ILES schemes in the former case and to under-resolved DNS (UDNS) in the latter.

Early attempts to quantify a priori the numerical dissipation of a given numerical scheme were made by Hirt [21] through the modified equation analysis. In the context of LES Ghosal [18] introduced a kinematic analysis based on the 'joint-normal hypothesis' to estimate the numerical error in the non-linear LES equations. This analysis suggested (as later confirmed by Kravchenko and Moin [24]) that the aliasing error as well as the dissipation error can be of the same order as the SGS terms. Fureby and Grinstein [12] applied the modified equation analysis to draw a formal parallel between ILES and ELES. The first method to quantify a posteriori the numerical dissipation was proposed by Domaradzki et al. [11]. This method relies on the use of a reference spectral-Fourier solver and it is therefore limited to periodic domains. The idea of effective, resolution dependent, Reynolds number has been used by several authors in the past Porter et al. [28], Fureby and Grinstein [12]. The effective Reynolds number was formally defined by Aspden et al. [35] and more recently used by Wachtor et al. [32] and Zhou et al. [33]. Zhou et al. [33] propose a method to estimate the effective Reynolds number in ILES. The proposed method relies on the existence of an inertial range and the idea that the energy flux is the sole connection between the large scales governing the flow and the small dissipative scales. The dissipation energy is then estimated by computing the energy flux at the resolved energy containing scales which are not influenced by the specifics of the numerical scheme. The need to assess the numerical dissipation in non-periodic domains has been addressed by Schranner et al. [30]. Following the work of Domaradzki et al. [11], Domaradzki and Radhakrishnan [10], Schranner et al. [30] developed a methodology that allows for the quantification of the numerical dissipation for an arbitrary CFD code in a self consistent way, i.e., using only information about the flow field provided by the code being analyzed, viz., the solver can be treated as a black box without the need to know the details of the implementation. This numerical dissipation quantification tool provides a rigorous method to judge a posteriori the quality of a given simulation allowing for an impartial assessment of the impact of the numerical dissipation. In the work reported in this paper we apply this latter methodology for the first time to a non-trivial flow configuration solved using a commercial, low-order compressible CFD code.

2. Equations

2.1. Analytical form

The transport energy equation for the compressible Navier–Stokes (N–S) equations is

$$\frac{\partial \rho e}{\partial t} + \frac{\partial}{\partial x_j}\left[(\rho e + p)u_j\right] = \frac{\partial u_i \tau_{ij}}{\partial x_j} - \frac{\partial q_j}{\partial x_j}, \tag{1}$$

where u_i are the components of the velocity vector, p the pressure, ρ the density and e the total energy per unit mass. The constitutive relation between stress and strain rate for a Newtonian fluid is

$$\tau_{ij} = \mu\left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right], \tag{2}$$

and the heat flux q_i is defined as

$$q_i = -k\frac{\partial T}{\partial x_i}, \tag{3}$$