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, California 90089-1191, USA

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 Schraner et al. (2015) the equations and procedure for estimating the numerical dissipation rate and the numerical viscosity

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

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.

Keywords: LES, numerical dissipation, numerical viscosity, laminar separation bubble

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 (Pope, 2000; Sagaut, 2006; Garnier et al., 2009).

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 (Garnier et al., 2009). This approach was proposed by Boris et al. (1992) 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 (Galbraith and Visbal, 2008). For a thorough discussion on the ILES

approach one can refer to the ILES monograph of Grinstein et al. (2007). The MILES approach has been controversial and as such it has been the object of rigorous investigations (Garnier et al., 1999; Domaradzki et al., 2003). 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 (Garnier et al., 1999), 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 the high wave numbers (Domaradzki et al., 2003). The latter case can potentially be adjusted either by filtering or with the addition of an explicit SGS model (Tantikul and Domaradzki, 2010). 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. (2007), 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. (2009) 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 to its simplicity and good qualitative results for specific flows, even if often lacking a detailed comparison 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. (2006), 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 (Kravchenko and Moin, 1997) 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. (2003)). 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. (2004) 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 (2000).

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 detri-

mental to the performance of the code (Cadieux et al., 2014).

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. (2006); Wachtor et al. (2013); Gassner and Beck (2013)). 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 (1969) through the modified equation analysis. In the context of LES Ghosal (1996) 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 (1997)) that the aliasing error as well as the dissipation error can be of the same order as the SGS terms. Fureby and Grinstein (2002) 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. (2003). 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. (1998); Fureby and Grinstein (2002). The effective Reynolds number was formally defined by Aspden et al. (2009) and more recently used by Wachtor et al. (2013) and Zhou et al. (2014). Zhou et al. (2014) propose a method to estimate the effective Reynolds number in ILES. The proposed method relies on the existence of a 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 Schraner et al. (2015). Following the work of Domaradzki et al. (2003); Domaradzki and Radhakrishnan (2005), Schraner et al. (2015) 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

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

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

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

and the heat flux q_i is defined as

$$q_i = -k \frac{\partial T}{\partial x_i}, \quad (3)$$

where μ is the dynamic viscosity, k is the thermal conductivity, and T is the temperature. The definition of the total energy is

$$e = e^{in} + e^{kin}, \quad (4)$$

where e^{in} is the specific internal energy per unit mass and $e^{kin} = \frac{1}{2} u_i u_i$ is the kinetic energy. Following the procedure of Schraner et al. (2015) the

transport eq. (1) can be separated into the internal energy equation

$$\frac{\partial \rho e^{in}}{\partial t} + \frac{\partial \rho e^{in} u_j}{\partial x_j} = -p \frac{\partial u_j}{\partial x_j} + \tau_{ij} \frac{\partial u_i}{\partial x_j} - \frac{\partial q_j}{\partial x_j}, \quad (5)$$

and the kinetic energy equation

$$\frac{\partial \rho e^{kin}}{\partial t} + \frac{\partial \rho e^{kin} u_j}{\partial x_j} = -u_j \frac{\partial p}{\partial x_j} + u_i \frac{\partial \tau_{ij}}{\partial x_j}. \quad (6)$$

The viscous term in eq. (6) can be rewritten as

$$u_i \frac{\partial \tau_{ij}}{\partial x_j} = \frac{\partial u_i \tau_{ij}}{\partial x_j} - \tau_{ij} \frac{\partial u_i}{\partial x_j}, \quad (7)$$

where the first term on the right hand side represents the viscous work. Thus

eq. (6) becomes

$$\frac{\partial \rho e^{kin}}{\partial t} + \frac{\partial \rho e^{kin} u_j}{\partial x_j} = -u_j \frac{\partial p}{\partial x_j} + \frac{\partial u_i \tau_{ij}}{\partial x_j} - \tau_{ij} \frac{\partial u_i}{\partial x_j}. \quad (8)$$

For finite volume methods (FV) where the integral form of the N-S equations

is solved, it is convenient to rewrite the pressure term as

$$u_j \frac{\partial p}{\partial x_j} = \frac{\partial u_j p}{\partial x_j} - p \frac{\partial u_j}{\partial x_j}. \quad (9)$$

Therefore the integral form of eq. (8) can be written as

$$\begin{aligned} & \int_V \left(\frac{\partial}{\partial t} \rho e^{kin} + \frac{\partial \rho e^{kin} u_j}{\partial x_j} + \frac{\partial u_j p}{\partial x_j} - p \frac{\partial u_j}{\partial x_j} - \frac{\partial u_i \tau_{ij}}{\partial x_j} + \tau_{ij} \frac{\partial u_i}{\partial x_j} \right) dV = \\ & \frac{\partial}{\partial t} \int_V \rho e^{kin} dV + \int_A (\rho e^{kin} u_j + u_j p - u_i \tau_{ij}) n_j dA + \int_V \left(-p \frac{\partial u_j}{\partial x_j} + \tau_{ij} \frac{\partial u_i}{\partial x_j} \right) dV = \\ & \frac{\partial}{\partial t} E^{kin} + F^{kin} + F^{ac} - F^{vis} - W^p + \varepsilon^{vis} = 0, \end{aligned} \quad (10)$$

where F^{kin} , F^{ac} , F^{vis} are the kinetic energy, acoustic and viscous fluxes, n_j the outward unit vector normal to the bounding surface A , W^p the work due to pressure and ε^{vis} the viscous dissipation. We define a dissipation function ε as

$$\begin{aligned}\varepsilon &= \int_V \frac{1}{\nu} \tau_{ij} \frac{\partial u_i}{\partial x_j} dV \\ &= \int_V \rho \left(\frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \frac{\partial u_l}{\partial x_l} \right) dV,\end{aligned}\quad (11)$$

and the equation

$$\nu \varepsilon = \varepsilon^{vis}, \quad (12)$$

defines the averaged kinematic viscosity ν for the volume V .

2.2. Discretized form

If we assume a FV spatial discretization and a generic discretization in time then the discretized eq. (10) is contaminated by the truncation errors and a local residual

$$-\varepsilon_{(m)}^n = \frac{\Delta E_{(m)}^{kin}}{\Delta t} + F_{(m)}^{kin} + F_{(m)}^{ac} - F_{(m)}^{vis} - W_{(m)}^p + \varepsilon_{(m)}^{vis} \quad (13)$$

does not vanish. In the above equation the subscript $[\]_{(m)}$ refers to the m^{th} control volume. We call the residual ε^n the numerical dissipation rate because it has been shown that if integrated over a sufficiently large control

volume it has a predominantly dissipative character (Schraner et al., 2015; Domaradzki and Radhakrishnan, 2005). Eq. (12) allows for the definition of a numerical kinematic viscosity for the m^{th} control volume as

$$\nu_{(m)}^n = \frac{\varepsilon_{(m)}^n}{\varepsilon_{(m)}}. \quad (14)$$

The above definitions can be extended to a larger sub-domain or to the entire computational domain

$$\varepsilon_{sub}^n = \sum_m^M \varepsilon_{(m)}^n \quad ; \quad \nu_{sub}^n = \frac{\varepsilon_{sub}^n}{\varepsilon_{sub}}, \quad (15)$$

$$\varepsilon_{tot}^n = \sum_m^N \varepsilon_{(m)}^n \quad ; \quad \nu_{tot}^n = \frac{\varepsilon_{tot}^n}{\varepsilon_{tot}}, \quad (16)$$

where M is the total number of adjacent cells of a given sub-domain and N is the total number of cells of the simulation domain.

3. Procedure

This is a posteriori procedure where all quantities are computed using the numerical flow field. In principle this procedure does not require any information about the solver, all that is needed are temporal snapshots of the conserved variables (the density, components of the momentum/velocity, and the energy) and the corresponding mesh. In the present work, except for the

time discretization of the kinetic energy, all quantities appearing in eq. (13) have been calculated on-the-fly using the post-processing capabilities of the code itself. We can rely on the low-order post-processing tools embedded in the software since Schraner et al. (2015) showed that even with a low-order reconstruction one can obtain fair estimates of the terms in the kinetic energy balance. For the time discretization a second-order three-points finite difference formula is used, i.e.

$$\frac{\Delta E_{(m)}^{kin}}{\Delta t} = aE^{kin}(t - \Delta t^-)_{(m)} + bE^{kin}(t)_{(m)} + cE^{kin}(t + \Delta t^+)_{(m)}, \quad (17)$$

where the weights a , b and c are

$$a = -\frac{\Delta t^+}{\Delta t^-} \frac{1}{\Delta t^- + \Delta t^+}, \quad (18)$$

$$b = -a - c, \quad (19)$$

$$c = \frac{\Delta t^-}{\Delta t^+} \frac{1}{\Delta t^- + \Delta t^+}. \quad (20)$$

The kinetic energy, acoustic and viscous fluxes are calculated as follows

$$F_{(m)}^{kin} = \left(\sum_r (\rho e^{kin} u_j n_j)_{(r)} \Delta A_{(r)} \right)_{(m)}, \quad (21)$$

$$F_{(m)}^{ac} = \left(\sum_r (p u_j n_j)_{(r)} \Delta A_{(r)} \right)_{(m)}, \quad (22)$$

$$F_{(m)}^{vis} = \left(\sum_r (u_i \tau_{ij} n_j)_{(r)} \Delta A_{(r)} \right)_{(m)}, \quad (23)$$

where subscript $[\]_{(r)}$ represents a sum over the r^{th} face of the m^{th} control volume and $\Delta A_{(r)}$ is the area of the r^{th} face. Volume terms such as kinetic energy, pressure work and viscous dissipation rate are calculated in the following way

$$E_{(m)}^{kin} = \left(\frac{1}{2} \rho u_i u_i \right)_{(m)} \Delta V_{(m)}, \quad (24)$$

$$W_{(m)}^p = \left(p \frac{\partial u_i}{\partial x_i} \right)_{(m)} \Delta V_{(m)}, \quad (25)$$

$$\varepsilon_{(m)}^{vis} = \mu_{(m)} \left(\frac{\partial u_i}{\partial x_j} \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \frac{\partial u_i}{\partial x_j} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \frac{\partial u_l}{\partial x_l} \right)_{(m)} \Delta V_{(m)}, \quad (26)$$

where $\Delta V_{(m)}$ is the volume of the m^{th} control volume. These steps have been previously validated for Star-CCM+ by repeating the same test cases as in Schraner et al. (2015) for both incompressible and compressible solvers (Castiglioni and Domaradzki, 2015). Even though in this study a specific commercial software has been used, it is important to stress that the solver is treated as a black box and that all that is required to perform this analysis are temporal snapshots of the flow fields.

4. Numerical code

A low-order compressible N-S solver Star-CCM+ is employed. The use of a specific solver is intended as a representative example of a generic commercial code widely used in industry. Star-CCM+ solves the compressible

three-dimensional N-S equations in a conservative form. The equations are solved in a preconditioned dimensional form on an unstructured grid (CD-Adapco, 2013). The time is advanced through a dual time-stepping implicit scheme. The inviscid fluxes are evaluated by using the Weiss-Smith preconditioned Roe’s flux-difference splitting scheme. The scheme is formally at best second-order accurate. The viscous fluxes are evaluated by a standard central difference scheme. Two different SGS explicit models are available in the code: the dynamic Smagorinsky (Germano et al., 1991; Lilly, 1992) and the wall-adapting local eddy viscosity model (WALE) (Nicoud and Ducros, 1999), with the WALE constant set to $C_w = 0.544$. Both models have been used in the simulations presented in the study of Castiglioni and Domaradzki (2015). Additionally a UDNS simulation was performed and all subsequent analysis will be based on this no-model simulation.

5. Flow configuration

The methodology presented above has been used to analyze the results of a three-dimensional Laminar Separation Bubble (LSB) simulation over a NACA-0012 at $Re = 50,000$, 5° of angle of attack and $Ma = 0.4$. A sketch of the flow field is shown in Figure 1. The computational domain is a rectangular

prism with the following dimensions: $x = [-6, 6]$, $y = [-7, 7]$ and $z = [0, 0.2]$ where the x is the streamwise direction, y the normal direction and z the spanwise direction. The airfoil trailing edge is located at $(x, y) = (1, 0)$ and the chord has unit length. The total number of cells used was 1.6 million which is around 1% of the reference DNS resolution (Jones et al., 2008, 2010). The results from these simulations were presented by Castiglioni et al. (2014) to which the reader may refer for a more in depth description of the problem. The flow over the airfoil has regions of a laminar boundary layer, separated laminar shear layer, transitional separated shear layer, non-equilibrium attached turbulent boundary layer and equilibrium boundary layer. This is an interesting case to analyze because of the complex nature of the flow pattern. As quantitative metrics to evaluate the simulations, two quantities that are of great relevance for the design of airfoils were used: the pressure coefficient, $C_p = (p - P_0)/\frac{1}{2}\rho U_0^2$, and the skin friction coefficient, $C_f = \tau_w/\frac{1}{2}\rho U_0^2$ (where P_0 is the free stream pressure, U_0 is the free stream velocity, p is the pressure at the surface and τ_w is the wall shear stress).

As can be seen in Figure 2 the results obtained with very coarse UDNS agree surprisingly well with the reference DNS simulations of Jones et al. (2008) Furthermore it is interesting to notice that UDNS simulations and

Figure 1: Sub-domains location for NACA-0012.

simulations in which an explicit LES model is present have negligible differences, suggesting that the numerical dissipation provided by the numerics is dominant over the dissipation provided by the SGS models. The method for estimating the numerical dissipation described before should allow for the quantification of the above statement. For the reasons discussed above this simulation is the ideal candidate to test the method proposed by Schraner et al. (2015) in a realistic flow configuration.

6. Results

In the present work the procedure to quantify the numerical dissipation rate and viscosity is applied only to the UDNS case from Castiglioni et al. (2014) Initially three sub-domains have been selected to be analyzed, one in

(a) Pressure coefficient comparison

(b) Friction coefficient comparison

Figure 2: Three-dimensional Pressure and friction coefficients at the wall. DNS of Jones et al. (2008) (black dots). UDNS (solid red line), explicit LES with Dynamic Smagorinsky model (dashed dotted blue line), and explicit LES with WALE model (dashed green line) from Castiglioni et al. (2014).

the laminar region (SD1), one in the transitional region (SD2) and one in the fully turbulent region (SD3). The location and size of the sub-domains can be seen in Figure 1. They all have approximatively the same number of cells ($\sim 20,000$) and they all extend for the entire domain length in the spanwise direction. Starting from a converged statistically steady state solution the simulation has been run for an additional time unit, where a time unit is defined as the cord over the free stream velocity. The results for the numerical and viscous dissipation rate are shown in Figures 3(a)(c)(e) while the physical and total viscosity are shown in Figure 4(a)(c)(e). All three sub-domains behave as expected. The sub-domain SD1 encompasses a region where the boundary layer is fully laminar, therefore the flow is well resolved and the added numerical dissipation is very small, almost negligible (see Figure 4(a) and from now on refer to Table 1 for a summary of time averaged values). In SD1 the numerical viscosity is around 4% of the physical viscosity. Sub-domain SD2 is in the transitional region of the flow where there is a significant amount of turbulence. Here consequently the numerical dissipation is substantial (see Figure 4(c)), with the numerical dissipation being more than 1.5 times the physical one. Lastly sub-domain SD3 is in a fully turbulent region and we can observe a similar behavior as SD2 with an

even greater level of numerical dissipation (see Figure 4(e)). It is interesting to note that ε_{sub}^{vis} in the sub-domains SD1 and SD2 is considerably greater than in SD3. This behavior is due to the presence of the separated shear layer in the first two sub-domains leading to a higher viscous dissipation rate.

To date the methodology proposed by Schraner et al. (2015) has been applied only to a Taylor-Green vortex flow which is a flow that evolves in a box with periodic boundary conditions. In order to both confirm that the higher viscous dissipation is effectively coming from the shear layer and to investigate the effects of the presence of a no-slip boundary condition, another set of three sub-domains have been analyzed with approximatively the same sub-domain volumes. The new sub-domains that will be referred to as SD1a, SD2a and SD3a have the same position along the cord as the previous set and they are shifted away from the wall in the normal direction, the exact locations can be seen in Figure 1. The number of cells per sub-domain is around 12,000. The lower count of cells is due to the cell size increasing in the wall-normal direction moving away from the airfoil. It seems that indeed the high values of ε_{sub}^{vis} can be attributed to the presence of a shear layer, given that the value of the viscous dissipation rate in SD2a is similar to the values in SD3 and SD3a, see Figure 3. The procedure appears pretty robust

as for the new sub-domains it predicts values of numerical viscosity that are slightly higher for SD2a and SD3a with respect to the original sub-domains. This is consistent with the fact that at the wall turbulence is damped.

A separate more careful analysis is needed for the sub-domain SD1a. As can be seen from Figures 3(b) and 4(b) and Table 1, in this sub-domain the numerical dissipation rate is negative, furthermore the calculated numerical viscosity in absolute value is one order of magnitude higher than the physical viscosity. The method seems to fail in this region of the flow. The reason for this failure is that the region SD1a is laminar, away from the walls, where the velocity gradients ($\partial u_i / \partial x_j$) are negligible. Since from eq. (11) the dissipation function ε is a function of velocity gradients, in region SD1a ε (and the viscous dissipation) becomes negligible because of the absence of free-stream turbulence. Additionally, we note that the computed numerical dissipation rate in SD1a is two orders of magnitude smaller than in other regions considered. Therefore using eq. (14) involves a division of two small quantities which can lead to an unreliable result, in this case an excessive value of ν_n . What seems to be more troubling is that the computed numerical dissipation is negative. This can be explained by looking at the individual terms of the kinetic energy balance (see Figures 5 and 6). First we recall that the terms

in the balance are approximated with at best second-order reconstruction. With this in mind we can observe that the numerical dissipation rate order of magnitude is 10^{-7} while the kinetic energy flux, the acoustic flux, the work due to pressure and the viscous dissipation rate are on the order of 10^{-5} , thus a 1% error in any of the latter quantities could easily cause a sign switch in the numerical dissipation rate. Based on these observations, in the regions of laminar flows that do not contain strong velocity gradients (for instance in our simulations whenever the sub-domain of interest is outside the boundary layer) results produced by the method may be unreliable. However, this is because the numerical dissipation in such regions is very small, within error bounds of contributing terms, and thus of little significance in the simulations. In other words, in those regions the mesh resolution is adequate and the simulation results are not affected by the truncation errors.

As already mentioned the UDNS simulation performs surprisingly well in the prediction of the pressure and skin friction coefficient (see Figure 2). This seems to suggest that the numerical dissipation provided by the scheme is about the correct amount, i.e. it is about the amount that an explicit sub-grid scale model would provide. To quantify this hypothesis a sub-grid

(a) SD1

(b) SD1a

(c) SD2

(d) SD2a

(e) SD3

(f) SD3a

Figure 3: Time-evolution of numerical and viscous dissipation rates. ε_{sub}^n (blue solid line) and ε_{sub}^{vis} (red dashed line).

(a) SD1

(b) SD1a

(c) SD2

(d) SD2a

(e) SD3

(f) SD3a

Figure 4: Time-evolution of physical and numerical viscosities. Total viscosity $\nu + \nu^n$ (blue solid line), physical viscosity ν (red dashed line) and average total viscosity $\nu + \overline{\nu^n}$ (dot-dashed black line).

Figure 5: Time-evolution of single terms of the kinetic energy balance (eq. 13). Plotted are the values for sub-domain SD1 (solid blue line) against SD1a (red dashed line). (a) change in time of kinetic energy, (b) kinetic energy flux, (c) acoustic flux, (d) viscous flux.

(e) W^p

(f) ε^{vis}

(g) ε

(h) ε^n

Figure 6: Time-evolution of single terms of the kinetic energy balance (eq. 13). Plotted are the values for sub-domain SD1 (solid blue line) against SD1a (red dashed line). (e) pressure work, (f) viscous dissipation rate, (g) dissipation function and (h) numerical dissipation rate.

Sub-domain	$\varepsilon^{vis} \times 10^{-5}$	$\varepsilon^n \times 10^{-5}$	ε	$\nu \times 10^{-4}$	$\nu^n \times 10^{-4}$	$\nu^{tot} \times 10^{-4}$	ν^{tot}/ν
SD1	1.42	0.05	0.62	2.29	0.08	2.37	1.04
SD2	1.76	2.82	0.79	2.24	3.55	5.79	2.59
SD3	0.58	1.60	0.28	2.05	5.66	7.71	3.75
SD1a	0.00	-0.02	0.00	2.14	-30.2	-28.1	-13.1
SD2a	0.67	1.28	0.30	2.21	4.19	6.40	2.89
SD3a	0.34	1.22	0.17	2.05	7.24	9.29	4.53

Table 1: Summary of averaged in time dissipation rates and viscosities, where $\nu^{tot} = \nu + \nu^n$.

scale viscosity has been estimated via the classical Smagorinsky model

$$\nu_{(m)}^{SGS} = C_s^2 \Delta V_{(m)}^{2/3} \left(|S_{ij}| \right)_{(m)}, \quad (27)$$

where C_s is a constant and $|S_{ij}| = (2S_{ij}S_{ij})^{1/2}$ where S_{ij} is the strain tensor.

In literature the value of C_s can vary from 0.2 for fully isotropic turbulence to 0.1 for channel or shear flows, the reduction of C_s for the latter cases is justified by the fact the large scale non-zero field gradients add a contribution to $|S_{ij}|$ that has nothing to do with turbulence (Sagaut, 2006). $\nu_{(m)}^{SGS}$ is calculated for every cell in the sub-domain and volume averaged to obtain ν^{SGS} .

In Figure 7 the total viscosity provided by the code ($\nu + \nu^n$) and the total viscosity that should be predicted by the SGS model ($\nu + \nu^{SGS}$) are compared. For all sub-domains the SGS viscosity is provided for two C_s values, 0.1 as a reference and a value that allows the average SGS viscosity to match

the averaged numerical viscosity, 0.06, 0.24, and 0.36 for sub-domains SD1, SD2 and SD3 respectively. Sub-domain SD1 is in a laminar region therefore C_s should be zero. As expected we need a small value of the Smagorinsky constant in order to match the numerical viscosity with the sub-grid scale viscosity. For the sub-domains SD2 and SD3 the numerical dissipation provides a larger value of viscosity than the Smagorinsky model would predict. For the sub-domain SD3, in a region of fully developed turbulent boundary layer, a good match is achieved with a value of $C_s = 0.36$ which is 80% higher than what is typically found in literature for isotropic turbulence and 260% higher than for wall bounded flows. This suggests that the numerical dissipation does not provide the correct amount of sub-grid-scale dissipation. On the other hand, for this particular flow configuration, it seems that the excess of numerical dissipation, with respect to the sub-grid-scale dissipation, is not enough to be appreciable in the chosen metrics (Pressure and skin friction coefficients, separation and reattachment points). These findings also explain why there is a negligible difference in the results when an explicit LES model is added (see Figure 2) (Castiglioni et al., 2014). The numerical dissipation is dominant over the SGS dissipation, therefore adding an explicit model does not improve the results.

Figure 7: Time-evolution of physical, numerical and sub-grid-scale viscosities with different values of C_s . Total viscosity $\nu + \nu^n$ (blue line), physical viscosity ν (red dashed line), average total viscosity $\nu + \overline{\nu^n}$ (dot-dashed black line) and physical viscosity ν plus sub-grid scale viscosity ν^{SGS} (blue line with markers) .

7. Discussion and Conclusions

For the first time the method proposed by Schraner et al. (2015) has been applied to a compressible low-order Navier-Stokes solver with a realistic flow configuration. The method seems quite robust and self-consistent and it seems to be easily applicable to wall-bounded flows. A previously unknown limitation to the method has been found, specifically the method will fail to compute accurately the numerical viscosity for laminar flows that do not contain strong velocity gradients, for instance in our simulations whenever the sub-domain of interest is chosen outside the boundary layer. A posteriori this seems a quite obvious finding given that the numerical viscosity is computed from eq. (14) and the dissipation function tends to zero in absence of velocity gradients. The above described limitation of the method was not observed before because in TGV flows, even though at initial times the flow is fully laminar, strong gradients are always present in the flow due to initial conditions. This limitation does not diminish the utility of this tool since the method is designed for the estimation of numerical dissipation in turbulent flows.

The application of the method to this specific simulation enabled the quantification of previously reported qualitative findings (Castiglioni et al.,

2014). Two important observations have been quantified. First we have quantified the amount of numerical dissipation in different regions of the flow field in a coarse UDNS with a low-order scheme. Second we have been able to quantify the previously qualitative observation that the numerical dissipation dominates over the sub-grid-scale dissipation when an explicit model is added. The analyses confirmed that the addition of an explicit LES model to a coarse UDNS simulation with a low-order scheme is at best useless and can potentially even deteriorate the quality of a UDNS as previously reported (Kravchenko and Moin, 1997; Cadieux et al., 2014).

Acknowledgments

The co-authors were supported by a NSF grant (CBET-1233160). The computations for this work were performed at the University of Southern California Center for High-Performance Computing and Communications.

References

Aspden, A., Nikiforakis, N., Dalziel, S., Bell, J., 2009. Analysis of implicit LES methods 3 (1), 103–126.

- Boris, J., Grinstein, F., Oran, E., Kolbe, R., 1992. New insights into large eddy simulation. *Fluid Dyn. Res.* 10, 199–228.
- Cadieux, F., Domaradzki, J. A., Sayadi, T., Bose, T., 2014. DNS and LES of laminar separation bubbles at moderate Reynolds numbers. *ASME J. Fluids Eng.* 136, 061102.
- Camarri, S., Salvetti, M., Koobus, B., Dervieux, A., 2004. A low-diffusion MUSCL scheme for LES on unstructured grids. *Comp. & Fluids* 33 (9), 1101–1129.
- Castiglioni, G., Domaradzki, J., 2015. On the evaluation of numerical dissipation rate and viscosity in a commercial code. In: Submitted for the TSFP9 proceedings.
- Castiglioni, G., Domaradzki, J., Pasquariello, V., Hickel, S., Grilli, M., 2014. Numerical simulations of separated flows at moderate Reynolds numbers appropriate for turbine blades and unmanned aero vehicles. *Int. J. Heat and Fluid Flow* 49, 91–99.
- CD-Adapco, 2013. Star-CCM+ manual. Version 8.02.

- Colin, O., Rudgyard, M., 2000. Development of high-order Taylor-Galerkin schemes for LES. *J. Comp. Phys.* 162 (2), 338–371.
- Davidson, L., Cokljat, D., Fröhlich, J., Leschziner, M. A., Mellen, C., Rodi, W., 2003. LESFOIL: Large Eddy Simulation of Flow Around a High Lift Airfoil: Results of the Project LESFOIL Supported by the European Union 1998-2001. Vol. 83. Springer.
- Domaradzki, J., Radhakrishnan, S., 2005. Effective eddy viscosities in implicit large eddy simulations of decaying high Reynolds number turbulence with and without rotation. *Fluid Dyn. Res.* 36, 385–406.
- Domaradzki, J. A., Xiao, Z., Smolarkiewicz, P. K., 2003. Effective eddy viscosities in implicit large eddy simulations of turbulent flows. *Phys. Fluids* 15 (12), 3890–3893.
- Fureby, C., Grinstein, F. F., 2002. Large eddy simulation of high-Reynolds-number free and wall-bounded flows. *J. Comp. Phys.* 181 (1), 68–97.
- Galbraith, M., Visbal, M., 2008. Implicit large eddy simulation of low reynolds number flow past the SD7003 airfoil. In: 46th AIAA Aerospace Sciences Meeting and Exhibit. pp. 2008–225.

- Garnier, E., Adams, N., Sagaut, P., 2009. Large eddy simulation for compressible flows. Springer.
- Garnier, E., Mossi, M., Sagaut, P., Comte, P., Deville, M., 1999. On the use of shock-capturing schemes for large-eddy simulation. *J. Comp. Phys.* 153 (2), 273–311.
- Gassner, G. J., Beck, A. D., 2013. On the accuracy of high-order discretizations for underresolved turbulence simulations. *Theor. Comput. Fluid Dyn.* 27 (3-4), 221–237.
- Germano, M., Piomelli, U., Moin, P., Cabot, W. H., 1991. A dynamic subgrid-scale eddy viscosity model. *Phys. Fluids A* 3, 1760–1765.
- Ghosal, S., 1996. An analysis of numerical errors in Large-Eddy Simulations of turbulence. *J. Comp. Phys.* 125 (1), 187–206.
- Grinstein, F., Margolin, L., Rider, W., 2007. *Implicit Large Eddy Simulation: Computing Turbulent Flow*. Cambridge University Press.
- Hickel, S., Adams, N., Domaradzki, J., 2006. An adaptive local deconvolution method for implicit LES. *J. Comp. Phys.* 213, 413–436.

- Hirt, C., 1969. Computer studies of time-dependent turbulent flows 12 (12), II-219–II227.
- Jones, L., Sandberg, R., Sandham, N., 2008. Direct numerical simulations of forced and unforced separation bubbles on an airfoil of incidence. *J. Fluid Mech.* 602, 175–207.
- Jones, L., Sandberg, R., Sandham, N., 2010. Stability and receptivity characteristics of a laminar separation bubble on an airfoil. *J. Fluid Mech.* 648, 257–296.
- Kravchenko, A., Moin, P., 1997. On the effect of numerical errors in large eddy simulations of turbulent flows. *J. Comp. Phys.* 131 (2), 310–322.
- Lilly, D., 1992. A proposed modification of the Germano subgrid-scale closure method. *Phys. Fluids A* 4, 633–635.
- Nicoud, F., Ducros, F., 1999. Subgrid-scale stress modeling based on the square of the velocity gradient tensor. *Flow, Turbul. Combust.* 62 (3), 183–200.
- Pope, S. B., 2000. *Turbulent flows*. Cambridge University Press.

- Porter, D., Woodward, P. R., Pouquet, A., 1998. Inertial range structures in decaying compressible turbulent flows. *Phys. Fluids* 10 (1), 237–245.
- Sagaut, P., 2006. Large eddy simulation for incompressible flows: an introduction, 3rd Edition. Springer.
- Schranner, F., Domaradzki, J., Hickel, S., Adams, N., 2015. Assessing the numerical dissipation rate and viscosity in numerical simulations of fluid flows. *Comp. & Fluids* 114, 84–97.
- Tantikul, T., Domaradzki, J. A., 2010. Large eddy simulations using truncated Navier-Stokes equations with the automatic filtering criterion. *J. Turbul.* 11 (21), 1–24.
- Wachtor, A., Grinstein, F., DeVore, C., Ristorcelli, J., Margolin, L., 2013. Implicit large-eddy simulation of passive scalar mixing in statistically stationary isotropic turbulence. *Phys. Fluids* 25 (2), 025101.
- Zhou, Y., Grinstein, F. F., Wachtor, A. J., Haines, B. M., 2014. Estimating the effective Reynolds number in implicit large-eddy simulation. *Phys. Rev. E* 89 (1), 013303.