Implicit Large Eddy Simulation for Unsteady Turbulent Flows

Marco Hahn and Dimitris Drikakis
Department of Aerospace Sciences, School of Engineering
Cranfield University, Cranfield
Bedfordshire MK43 0AL
UK

m.hahn@cranfield.ac.uk, d.drikakis@cranfield.ac.uk

A numerical investigation of high-resolution schemes for solving the compressible Navier-Stokes equations in the context of Implicit Large Eddy Simulation (ILES) is presented. Computations have been performed using a hybrid total variation-diminishing (TVD) scheme. The hybrid TVD scheme combines third-order Godunov-type fluxes with first-order (dissipative) schemes using the flux limiters approach. Results are presented from numerical experiments of turbulent flow past an open cavity.

1   INTRODUCTION

It has been observed for more than a decade that high-resolution methods have characteristics that mimic the effects of finite viscosity and thermal conductivity and lead to a 'dissipation' which is similar to that achieved by adequately resolved numerics involving finite Reynolds and Prandtl numbers [1], [2], [3], [4], [5], [6]. The idea to use these methods as an implicit way to numerically model complex turbulent flows, e.g., flows dominated by vorticity leading to turbulence, flows featuring shock waves and turbulence, and the mixing of materials, is an evolving area of research referred to as Implicit Large Eddy Simulation (ILES) or Monotonically Integrated LES (MILES). Such flows are extremely difficult to practically obtain stably and accurately in under-resolved conditions (with respect to grid resolution) using classical linear, both second and higher-order accurate schemes.

The success of high-resolution methods to compute turbulent flows seems to depend on a delicate balance of truncation errors due to wave-speed-dependent terms (chiefly responsible for numerical dissipation) in the case of Godunov-type methods and hyperbolic part of the flux. It is the essence of this balance that needs to be understood in order to regularise the flow and allow shock propagation to be captured physically realistically even if it is not fully resolved on the computational mesh. Nonlinear limiters in high-resolution methods guard the methods against catastrophic failures due to nonlinear wave-steepening or unresolved features by triggering entropy producing mechanisms that safeguard the calculation when the need arises. The two key questions are: (i) what criteria should be used to design the nonlinear limiter that triggers the entropy production, and (ii) to what extent numerical dissipation accounts for turbulent flow effects.

Results from the implementation of high-resolution methods in wall-bounded flows [3], [6] show that in principle there is nothing that prevents the use of these methods in near wall flows even without using an *explicit* turbulence model. Both ILES and traditional LES, based on an explicit subgrid scale (SGS) turbulence modelling, pose substantial challenges in high-Reynolds, near-wall flows, especially in the presence of separation from gently curved surfaces, where resolution and thus computing-cost issues are critical.

In the present study, we investigate the ILES approach in the context of under-resolved simulations, using a hybrid TVD scheme that combines third-order Godunov-type fluxes with first-order (dissipative) fluxes

through the implementation of limiters. Results are presented from a numerical experiment of turbulent flow past an open cavity.

2   NUMERICAL FRAMEWORK

The physics of (Newtonian) fluid flow is governed by the Navier-Stokes equations. These equations consist of the continuity, momentum and energy laws which can be written in a coupled generalised conservation form

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0,$$

$$\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla \cdot \mathbf{P},$$

$$\frac{\partial e}{\partial t} + \nabla \cdot (e\mathbf{u}) = -\nabla \cdot (\mathbf{u} \cdot \mathbf{P}) - \nabla \cdot \mathbf{q},$$

where $\mathbf{u}$, ρ, e, and $\mathbf{q}$ stand for the velocity components, density, total energy per unit volume, and heat flux, respectively. The tensor $\mathbf{P}$ for a Newtonian fluid is defined by

$$\mathbf{P} = \mathrm{p}(\rho, T)\mathbf{I} + \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I} - \mu[(\nabla \mathbf{u}) + (\nabla \mathbf{u})^{\mathbf{T}}] \ ,$$

where $\mathrm{p}(\rho, T)$ is the scalar pressure, $\mathbf{I}$ is the identity tensor, T is the temperature, and μ is the dynamic viscosity coefficient. The above system is completed by an equation of state. For a perfect gas the equation of state is given by $p = \rho R T$, where R is the gas constant. For the solution of the Navier-Stokes equations, we have employed an explicit, third-order TVD Runge-Kutta scheme [7], central differences for the viscous terms and a high-resolution scheme for the discretisation of the advective terms. This scheme is briefly described below (for further details see [8], [9]) for the one-dimensional, inviscid counterpart of the equations in matrix form

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{E}}{\partial x} = 0 \ ,$$

where $\mathbf{U}$ is the array of the unknown variables and $\mathbf{E}$ is the flux associated with the terms in x-direction. The advective flux derivative $\partial \mathbf{E} / \partial x$ (similarly, for the other advective flux derivatives) is discretised at the centre of the control volume (*i*) using the values of the inter-cell fluxes, i.e.,

$$\frac{\partial \mathbf{E}}{\partial x} = \frac{\mathbf{E}_{i+1/2} - \mathbf{E}_{i-1/2}}{\Delta x} \ .$$

The high-resolution method employed here for the calculation of the convective fluxes is a hybrid TVD scheme. This calculates the flux as

$$\mathbf{E}_{i+1/2} = \psi_{i+1/2}\mathbf{E}_{i+1/2}^{SW-FVS} + (1 - \psi_{i+1/2})\mathbf{E}_{i+1/2}^{CB} \ ,$$

where $\mathbf{E}_{i+1/2}^{SW-FVS}$ and $\mathbf{E}_{i+1/2}^{CB}$ are the inter-cell fluxes according to a Steger-Warming flux vector splitting (SW-FVS) scheme and a Godunov-type, characteristics-based (CB) scheme, respectively. The term $\psi_{i+1/2}$

is a limiter function [8] defined by the square of the (local) Mach number differences across cell faces. Second- and third-order of accuracy is achieved in discontinuities and smooth flow regions, respectively.

The Godunov-type, characteristics-based scheme [10] uses up to third-order (non-oscillatory) interpolation [8] to compute the conservative variables along the characteristics. The scheme is third-order accurate in both space and time when it is used in conjunction with a third-order TVD Runge-Kutta scheme for the time integration.

The SW-FVS scheme defines the inter-cell flux as

$$\mathbf{E}_{i+1/2}^{SW-FVS} = \mathbf{E}_{i+1/2}^{+}(\mathbf{U}_L) + \mathbf{E}_{i+1/2}^{-}(\mathbf{U}_R) \ ,$$

where the left, $\mathbf{U}_L$, and right, $\mathbf{U}_R$, states of the conservative variables can be obtained by first-, second- or higher-order accurate interpolation schemes. Here, the positive and negative fluxes are computed by an improved version of the Steger-Warming flux vector splitting (SW-FVS) scheme [8], [9] in conjunction with first-order interpolation for $\mathbf{U}_L$ and $\mathbf{U}_R$. The first order-interpolation was selected in order to provide a lower-order (first-order) flux in the TVD scheme.

Limiters are the general nonlinear mechanism that distinguishes modern methods from classical linear schemes. Their role is to act as a nonlinear switch between more than one underlying linear methods thus adapting the choice of numerical method based upon the behaviour of the local solution. Limiters result in nonlinear methods even for linear equations in order to achieve second-order accuracy simultaneously with monotonicity. Numerical flux limiters can act like dynamic, self-adjusting models, modifying the numerical viscosity to produce a nonlinear eddy viscosity [4], [6].

3   RESULTS

Compressible, turbulent flow past an open cavity features a variety of flow phenomena including large and small vortical structures, free shear layers, transitional flow, flow separation and flow re-laminarisation, shock and rarefaction waves.

Under-resolved simulations at Mach number 0.8 and Reynolds number 2500 have been carried out, showing the development of an oscillating unsteady pressure field, see Figures 1-3. It is self-sustained and coherent but shifted in phase between the cavity's leading edge and trailing edge. The arising acoustic radiation is shown in Figure 4. Acoustic waves emanating from the trailing edge propagate upstream and trigger vortex shedding. This feedback mechanism is responsible for the self-sustained oscillating pressure field.

Furthermore, the ILES results for the dominant Strouhal number were found in good agreement with DNS of [11] for computations at different Mach number conditions, e.g., at Mach number 0.8 ILES and DNS predicted values of 0.635 and 0.65, respectively, while at Mach number 0.6 the values of the Strouhal number were computed as 0.683 and 0.7, respectively. The comparison of dominant frequencies shows the applicability of ILES for open cavity flows. Simulations for the transonic cases using a SGS (Smagorinsky-type) model have shown no further improvement of the results. This agrees with the conclusion of [5] that when no explicit subgrid scale modelling is added, a high-resolution method will not add any unnecessary diffusion.
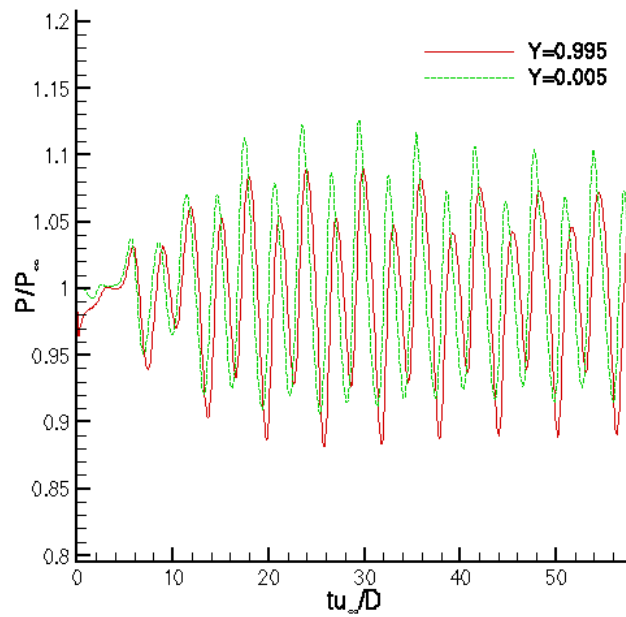
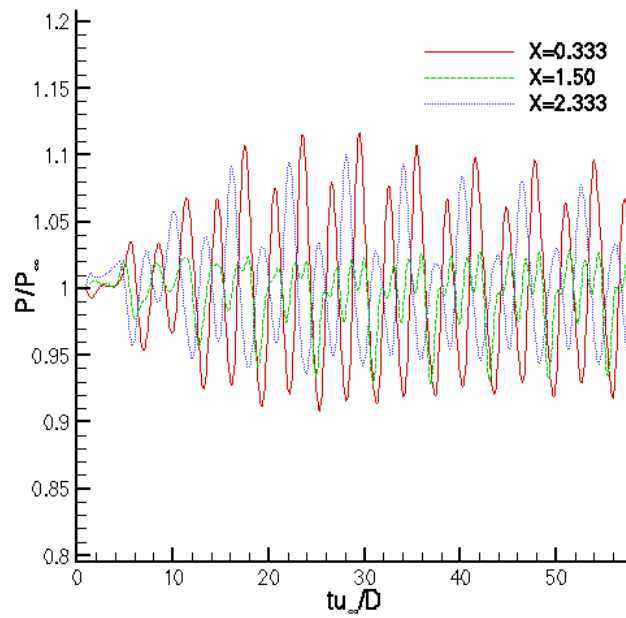**Figure 1: Pressure at the upstream face of the cavity.**

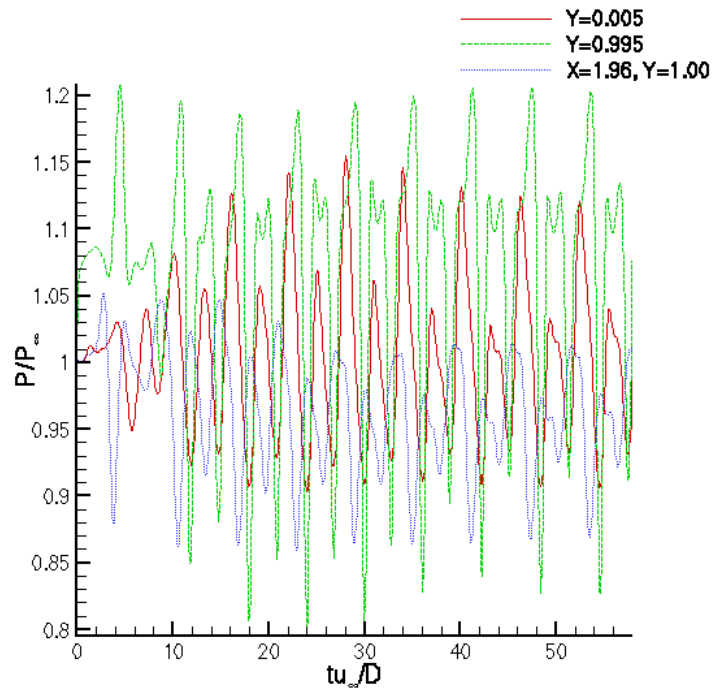**Figure 2: Pressure at the bottom of the cavity.**

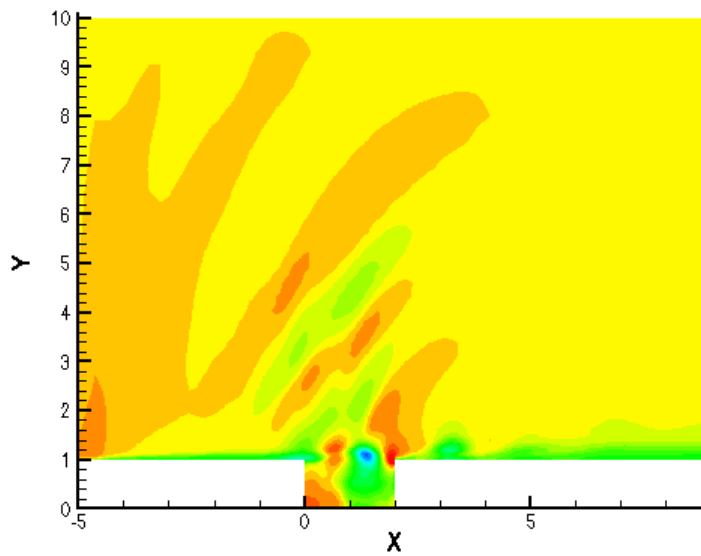**Figure 3: Pressure at the downstream face of the cavity and in the shear layer (X=1.96, Y=1.00).**

Figure 4: Instantaneous density contours in the computational domain.

REFERENCE

[1]   Oran ES, Boris JP. Numerical simulation of reactive flow. Cambridge University Press, 2000.

[2]   Youngs DL. Application of MILES to Rayleigh-Taylor and Richtmyer-Meshkov mixing. AIAA 2003; Paper 2003–4102.

[3]   Fureby C, Grinstein FF. Large eddy simulation of high Reynolds number free and wall bounded flows. Journal of Computational Physics 2002; 181:68–97.

[4]   Margolin LG, Rider WJ. A rationale for implicit turbulence modeling. International Journal for Numerical Methods in Fluids 2001; 39:821–841.

[5]   Margolin LG, Smolarkiewicz PK, Wyszogrodzki AA. Implicit turbulence modelling for high Reynolds number flows. Journal of Fluids Engineering 2002; 124:862–867.

[6]   Drikakis D. Advances in turbulent flow computations using high-resolution methods. Progress in Aerospace Sciences 2003; 39:405–424.

[7]   Shu CW, Osher S. Efficient implementation of essentially non-oscillatory shock-capturing schemes. Journal of Computational Physics 1988; 77:439–471.

[8]   Z´oltak J, Drikakis D. Hybrid upwind methods for the simulation of unsteady shockwave diffraction over a cylinder. Computer Methods in Applied Mechanics and Engineering 1998; 162:165–185.

[9]   Bagabir A, Drikakis D. Numerical experiments using high-resolution schemes for unsteady, inviscid, compressible flows. Computer Methods in Applied Mechanics and Engineering 2004; accepted, to appear.

[10] Eberle A. Characteristic flux averaging approach to the solution of Euler's equations. VKI Lecture Series, Computational Fluid Dynamics 1987; Report 1987–04.

[11] Rowley CW. Modeling, Simulation, and Control of Cavity Flow Oscillations. PhD thesis, California Institute of Technology, 2002.

SYMPOSIA DISCUSSION

REFERENCE AND/OR TITLE OF THE PAPER: 34

DISCUSSOR'S NAME: E. Dowell
AUTHOR'S NAME: M. Hahn

QUESTION:
If the grid is refined indefinitely does the LES solution approach the exact solution of the Navier-Stokes equations?

AUTHOR'S REPLY:
Not necessarily, we still have a certain amount of numerical dissipation through the truncation terms. The general idea of LES is however not to replace DNS or go to very fine grids since this is not feasible in the foreseeable future due to limited computing power.