Computers & Fluids 118 (2015) 245–254

Contents lists available at ScienceDirect

Computers & Fluids

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

Towards robust unstructured turbomachinery large eddy simulation

R.A. Watson [a,*], P.G. Tucker [a], Z.-N. Wang [b], X. Yuan [b]

[a] Department of Engineering, University of Cambridge, CB2 1PZ, United Kingdom
[b] Laboratory for Thermal Science, Tsinghua University, Beijing 100084, China

ARTICLE INFO

Article history:
Received 20 June 2014
Received in revised form 19 March 2015
Accepted 12 June 2015
Available online 18 June 2015

Keywords:
Large eddy simulation
Kinetic energy preserving
Roe
Industrial
Unstructured
Low Mach

ABSTRACT

Industrial legacy codes usually have had long pedigrees within companies, and are deeply embedded into design processes. As the affordability and availability of computing power has increased, these codes have found themselves pushed into service as large eddy simulation solvers. The approximate Riemann solver of Roe, which is frequently used as the core method in such legacy codes, is shown to need much user care when adopted as the discretisation scheme for large eddy simulation. A kinetic energy preserving (KEP) scheme—which retains the same advantageous stencil and communications halo as the original Roe scheme—is instead implemented and tested. The adaptations of code required to switch between the two schemes were found to be extremely straightforward. As the KEP scheme intrinsically bounds the growth of the kinetic energy, it is significantly more stable than the classical non-dissipative schemes. This means that the expensive smoothing terms of the Roe scheme are not always necessary. Instead, an explicit subgrid scale turbulence model can be sensibly applied. As such, a range of mixed linear–non-linear turbulence models are tested. The performance of the KEP scheme is then tested against that of the Roe for canonical flows and engine-realistic turbine blade cutback trailing edge cases. The new KEP scheme is found to perform better than the original in all cases. A range of mesh topologies: hexahedral; prismatic; and tetrahedral; are also tested with both schemes, and the KEP scheme is again found to perform significantly better on all mesh types for these flows.

1. Introduction

In many industrial codes, the emphasis is placed on the ability to conform to complex geometries. This necessitates the use of unstructured formulations, and frequently results in second order algorithms being employed. Many of these industrial codes have long pedigrees stretching back many years, and were originally designed to perform Reynolds-Averaged Navier–Stokes (RANS) calculations. The extreme advances in computational power and corresponding decrease in hardware costs have brought the significantly more computationally intensive—but higher fidelity—Large Eddy Simulation (LES) and hybrid RANS/LES methods in from the realm of academic and intellectual investigation into the purview of industrial design calculations.

From the point of view of both the end-user and the support team, it is attractive to reuse as much of the RANS code as possible in constructing an LES solver, as much of the structural framework already exists, requiring little or no adaptation to file formats, boundary conditions, parallelisation libraries, and substantial code optimisation has already been conducted. To take advantage of this, an industrial legacy RANS code is here applied to LES simulations, its performance assessed, and any adaptations or improvements which are found to be necessary are made.

The code used in this exercise is HYDRA, a staple of the Rolls-Royce design process for many years. Variants of the scheme used in HYDRA have, in the past, been used to successfully perform a range of large eddy simulation calculations. Typically, these have either involved relatively high Mach number—very compressible—flows, or codes which have been heavily, and often on a problem-by-problem basis, modified. Ciardi et al. [1] developed a low dissipation version of the Roe scheme based on suppressing the appearance of dispersive "wiggles" in the local flow solution. A set of user-supplied constants were used to control the trade-off between minimising excessive dissipation and ensuring solution stability.

Other attempts to reduce any inherent dissipation have been made over the years. Page and McGuirk [19] and Li et al. [11] chose to calculate and use a Ducros et al. switch [3] on the smoothing terms to try and keep the diffusion under control—the subgrid effects themselves were then separately modelled with more explicitly added terms. O'Mahoney et al. [18] were also compelled to reduce the strength of the stabilising terms to avoid the

* Corresponding author. Tel.: +44 1223 332600.
E-mail address: rob.watson@eng.cam.ac.uk (R.A. Watson).

http://dx.doi.org/10.1016/j.compfluid.2015.06.017
0045-7930/

Nomenclature

$\mathcal{B}^C$	continuous Burgers' operator	x	streamwise direction
C_*	constant	Δ	filter width
$\mathcal{D}$	transport discretisation operator	Δx	one dimensional filter width
$\mathcal{D}^K$	KEP transport discretisation operator	$\Delta x^+,\ \Delta y^+,\ \Delta z^+$	non-dimensional wall distance
$\mathcal{D}^R$	Roe's transport discretisation operator	ε_2	smoothing constant
$\mathcal{F}$	flux vector	ε_3	smoothing constant
E_k	total kinetic energy	η	film cooling effectiveness
k	kinetic energy	ϕ	convected variable
k	wavenumber	ρ	density
M	blowing ratio	τ_{ij}	stress tensor
Ma	Mach number	$\bar{\phi}$	local mean value
p	static pressure	$\overline{\phi}$	time averaged value
$\mathcal{Q}$	conserved variable vector	ϕ^I	inviscid value
Re	Reynolds number	ϕ^S	smoothed value
t	time	ϕ^V	viscous value
u	velocity		

suppression of turbulence when modelling ingestion by rim seals. The dangers of excessive smoothing have been extensively discussed by Tristanto et al. [23].

Examples of some successful simulations in more specific applications include the transonic Mach number jets flows of Eastwood et al. [5], and the chevroned nozzles of Xia and Tucker [25]. In both of these cases, the smoothing constant was geometrically sculpted *a priori*, to act as a numerical turbulence model in areas of interest, and to provide stability in the far field. This has proved a successful approach, as long as the general nature of the flow is understood before the sculpting is carried out.

Despite these successes and advances, the results from the use of the Roe scheme as a discretisation for an LES solver for wider problems—or problems involving a range of flow conditions—have been disappointing at times, particularly when flows have been dominated by regions with low Mach number. In this paper, it is hoped that the problems HYDRA encounters as a legacy RANS solver applied to LES calculations can be explained, quantified, and mitigated as painlessly as possible.

The Euler equations are given by:

$$\frac{\partial \mathcal{Q}}{\partial t} + \frac{\partial \mathcal{F}^I(\mathcal{Q})}{\partial x} = 0 \tag{1}$$

This equation can be discretised in many ways, but for these purposes, discretisations are restricted to two-point edge based schemes, for ease of development, efficiency of parallelisation, and simplicity of application to unstructured problems. Here, the established Roe scheme is compared to a kinetic energy preserving (KEP) "Jameson" formulation. To solve the full Navier–Stokes equations, Eq. (1) must be augmented with a viscous flux term, $\mathcal{F}^V$. When these equations have been filtered onto a grid, residual stress tensors emerge, necessitating a model for the subgrid scale effects of turbulence.

This paper first discusses the two different numerical approaches, after which the various test cases are introduced, before the numerical results are presented, before finally discussing what these findings suggest for industrial large eddy simulation.

2. Numerical methods

The solver used in this work, in its original RANS formulation, is built around the approximate Riemann solver of Roe [21]. In smooth regions of the flow, away from sharp gradients such as shocks, the inviscid flux calculation takes on the form of a central difference of the end points of each node, smoothed by some function of the Laplacian of the conserved variable vector.

$$\overrightarrow{\mathcal{F}^{I,S}}_{ij} = \frac{1}{2}(\mathcal{F}(\mathcal{Q}_L) + \mathcal{F}(\mathcal{Q}_R)) - \frac{1}{2}\left|\mathbf{A}_{ij}\right|\varepsilon_2\left(\mathcal{L}(\mathcal{Q})_R - \mathcal{L}(\mathcal{Q})_L\right) \tag{2}$$

Near to sharp gradients, the inviscid flux calculation decomposes to first order, and takes the form of a central difference smoothed by some function of the conserved variable vector itself.

$$\overrightarrow{\mathcal{F}^{I,S}}_{ij} = \frac{1}{2}(\mathcal{F}(\mathcal{Q}_L) + \mathcal{F}(\mathcal{Q}_R)) - \frac{1}{2}\left|\mathbf{A}_{ij}\right|\varepsilon_3(\mathcal{Q}_R - \mathcal{Q}_L) \tag{3}$$

The weighting of these two functions is given by a local pressure switch, intended to ensure the full second order accuracy of the former in areas of smooth flow, and to allow this accuracy to degrade for stability in the region of strong gradients. The matrix $\mathbf{A}_{ij}$ is the inviscid flux Jacobian, $\mathbf{A}_{ij} = \partial F^I/\partial Q$. The variables ε_2 and ε_3 represent user supplied constants which control the smoothing levels. These variables, which are constants in the classical Roe scheme, were modified by Eastwood [4] and Ciardi et al. [1] to vary in space, and to vary in both time and space, respectively. An explicit 5-stage Runge–Kutta (RK) method is used to integrate the Roe scheme in time.

The unsmoothed fluxes themselves are given by the average of the flux each side of the control volume face.

$$\mathcal{F}^I_{ij} = \frac{1}{2}\{\rho_L u_L \phi_L + \rho_R u_R \phi_R\} \tag{4}$$

This scheme brings a number of advantages to the RANS solution process. It offers considerable stability, whilst retaining second order accuracy in smooth regions. On industrial grids, a Courant number of 2.00 is found to be sufficient for stability (but not, for LES, for accuracy in time), even with the smoothing and viscous flux calculations updated only every other RK sub-step. This leads to a considerable acceleration in steady convergence, whilst maintaining second order accuracy away from shocks.

2.1. Conservation of kinetic energy

It is understood that as a bounding quantity of incompressible flow, the global growth rate of kinetic energy will dictate the stability of the scheme—with a global kinetic energy growth rate of zero, the scheme will be stable. For compressible flows, conservation of kinetic energy does not formally guarantee the stability of a solution, but evidence suggests that it does impart some resilience to schemes. Global kinetic energy is given by: