Computers and Fluids 166 (2018) 46–56

Contents lists available at ScienceDirect

Computers and Fluids

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

Review

Acceleration of Euler and RANS solvers via Selective Frequency Damping

Frédéric Plante*, Éric Laurendeau

Polytechnique Montréal, Montréal, H3C3A7, Canada

ARTICLE INFO

Article history:
Received 5 September 2017
Revised 19 January 2018
Accepted 23 January 2018

Keywords:
Euler equations
Navier–Stokes equations
Selective frequency damping
Steady-state
Solution acceleration

ABSTRACT

This paper presents the application of a Selective Frequency Damping (SFD) algorithm as an acceleration technique for the solution of the Euler and Navier–Stokes equations. The SFD method is implemented in a segregated way in a cell-centered finite volume code with added artificial dissipation. Its effect is also analyzed in combination with state of the art acceleration techniques, such as implicit residual smoothing, W-cycle multigrid and local time stepping. The proposed approach relies on the addition of a proportional feedback control and a low-pass filter to the system of equations to damp out targeted frequencies. This method was originally developed for the computation of the steady-state solution of unstable flows. It is here applied to stable cases specifically to enhance the convergence rate to steady state. The method is also applied to overset grid systems for which effective multigrid remains challenging. Significant improvements in the residual convergence rates are found with and without using conventional acceleration techniques. The paper presents the implementation of the SFD algorithm. A verification of the method on a documented application case from the literature is performed. Then, the convergence acceleration provided by the SFD is shown for Euler and Navier-Stokes equations on classic one-to-one grid and overset grid systems.

1. Introduction

The choice of a numerical model in the field of computational fluid dynamics (CFD) is directed by two criteria: fidelity and computational cost. The variety of models extend from potential (inviscid, incompressible and irrotational) to direct numerical simulation in which the turbulence is fully resolved. Intermediate models include Euler (inviscid), Reynolds-Averaged Navier–Stokes (RANS), Large-Eddy Simulation (LES) and hybrid RANS-LES like Detached-Eddy Simulation (DES). The cost of CFD simulations grows rapidly as turbulence scales are added to the model, as discussed by Spalart and Venkatakrishnan [1]. Moreover, the resolution of turbulent scales implies a time accurate solution. For RANS simulations the turbulent fluctuations are averaged and modeled, allowing for steady-state solutions, which makes them even faster to compute. These solutions are attractive for many engineering applications where only mean values are required. All these factors contribute to the popularity of RANS models in the aerospace industry.

A well-established technique to solve the Euler and RANS equations is to integrate the equations in time until the time derivative terms tend to zero. For this purpose, the temporal accuracy of the solver is not required and a wide variety of explicit and implicit time integration schemes as well as convergence acceleration methods were devised. An extensive review of the numerical methods for the solution of the Euler and Navier-Stokes equations is presented by Witherden et al. [2].

Explicit schemes, like the multi-stage Runge–Kutta schemes, are easy to implement and require low memory. However, their stability is limited to time steps which satisfy the CFL condition. These schemes can be optimized to maximize the propagation and the damping of the transient error. To do so, hybrid multistage schemes are often used [3,4]. The implicit residual smoothing method [3,5–7] can also be added, allowing a higher maximum CFL number by a factor 2 or 3 [2]. Another approach is the enthalpy damping [8] which uses the fact that the enthalpy is constant for steady solution of the Euler equations. Thus this method is only applicable to inviscid flows. The convergence rate can be further enhanced with local time stepping by using the maximum time step allowable by the CFL condition in every cell of the computational domain. The local time step can be seen as a scalar preconditioner [9] that mitigates the stiffness caused by variation of the cell size and spectral radius in the domain. In the same way, a ma-

* Corresponding author.
E-mail addresses: frederic.plante@polymtl.ca (F. Plante), eric.laurendeau@polymtl.ca (É. Laurendeau).

https://doi.org/10.1016/j.compfluid.2018.01.027
0045-7930/

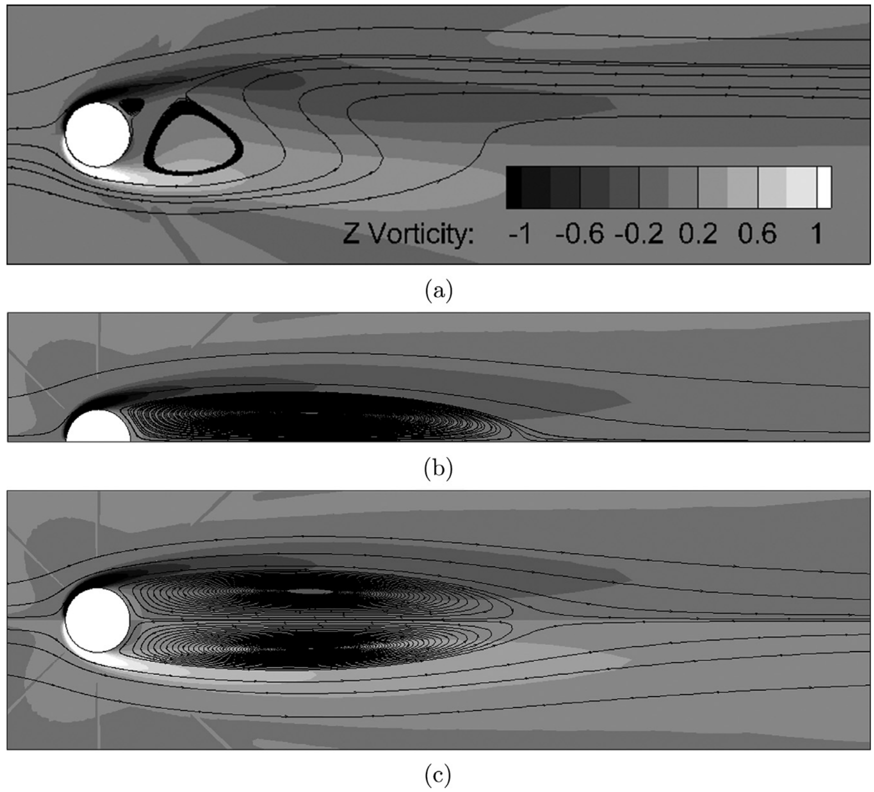

Fig. 1. Vorticity contours and streamlines for (a) the steady solution (not converged), (b) the half domain case and (c) the full domain SFD case for the flow past a cylinder at Reynolds 150.

Table 1
Grid convergence of the force coefficients for the inviscid NACA0012 at Mach 0.5 and $\alpha = 1.25^\circ$.

Number of cells	C_L	C_D	C_M
32×32	0.17163756	0.01309135	-0.00386408
64×64	0.17456452	0.00333578	-0.00210174
128×128	0.17750765	0.00063206	-0.00203024
256×256	0.17883743	$\boxed{0.00010354}$	-0.00216000
512×512	$\boxed{0.17926809}$	$\boxed{0.00002635}$	$\boxed{-0.00222334}$
1024×1024	$\boxed{0.17939665}$	$\boxed{0.00001876}$	$\boxed{-0.00224615}$
2048×2048	$\boxed{0.17943635}$	0.00001884	$\boxed{-0.00225400}$
Continuum(C_X^*)	0.17945409	0.00001793	-0.00225811
Order p	1.695	3.345	1.540
FLO82 order p (39)	2.107	1.805	2.130
OVERFLOW order p (39)	1.162	0.820	0.967
CFL3Dv6 order p (39)	1.154	2.061	0.876

trix preconditioner such as a point-implicit block-Jacobi [9–11] can be used. These preconditioners scale the time step of each characteristic equation with their corresponding eigenvalue, allowing the time steps to be closer to the stability limit for each equation. A matrix preconditioner is especially suitable for the integration of the Navier–Stokes equations [9].

On the other hand, implicit schemes theoretically offer unconditional stability allowing for infinite CFL numbers. In practice the direct inversion [12] of the implicit scheme is not used and approximate factorization were devised. Popular approaches include ADI [13,14] and LU-SGS/LU-SSOR [11,15,16]. Newton–Krylov methods such as GMRES solvers [17] can also be used, as discussed by Witherden et al. [2]. For sake of simplicity, explicit schemes are used in this paper.

One of the most efficient acceleration techniques for Euler [18] and Navier–Stokes [3,7] equations is multigrid. This method increases the convergence rate by employing coarser grid derived from the fine grid. Two mechanisms are exploited to enhance the convergence rate. First, larger time steps can be used on the coarse grids, while meeting the CFL condition. Second, the low frequency error on a fine grid is transferred into a high frequency one on the coarse grid and can be damped out by the time integration scheme. To generate the coarse grids from a structured mesh, one can coarsen the grid in all the topology directions, use a directional