Applied Mathematical Modelling 40 (2016) 1728–1740

Contents lists available at ScienceDirect

Applied Mathematical Modelling

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

A feature-based mesh adaptation for the unsteady high speed compressible flows in complex three-dimensional domains

Hoang-Huy Nguyen [a], Vinh-Tan Nguyen [a,*], Matthew A. Price [a], Oubay Hassan [b]

[a] Institute of High Performance Computing, 1 Fusionopolis Way, #16-16 Connexis 138632, Singapore
[b] College of Engineering, University of Wales, Swansea SA2 8PP, UK

ARTICLE INFO

Article history:
Received 7 December 2012
Revised 24 July 2015
Accepted 26 August 2015
Available online 30 September 2015

Keywords:
Mesh adaptation
Feature-based
Re-meshing
Unstructured grids
Boundary-conforming
Unsteady flows

ABSTRACT

We propose an unstructured mesh adaptation approach for unsteady high speed compressible Navier–Stokes applications involving blasts and explosions with the presence of strong shock waves propagating in three dimensional complex domains. The idea is to identify the locations of critical physics locally and then re-mesh these regions based on solution derived metrics. The approach ensures both geometry fidelity and mesh validity, especially for areas near complex geometries, a task that is always a challenge in mesh adaptation. The proposed adaptivity is applied for simulations of blast wave propagations and compared with available data in literature. The results show that the proposed method is fully robust and efficient for computational fluid dynamics (CFD) problems in complex three-dimensional domains.

1. Introduction

Modern CFD has the ability to explore problems that are more complex than ever before, partly because of more powerful computing resources. However, the recent evidence suggests that there is still significant unreliability in the numerical predictions made by current CFD codes for the same problem. A strong relation between solution quality and mesh topology has been shown, further indicating that current mesh design practices are not sufficient [6]. Due to the fundamental impact of mesh on the approximation of functions and PDE solutions, mesh adaptation was the focus of many researchers during the last two decades. There are four general approaches of mesh refinement methods. The first approach is *p*-adaptation, where the interpolation order is locally modified and does not require a new mesh to be generated.While *p*-adaptation can achieve excellent error convergence for smooth flows, difficulties arise near singularities or discontinuities. This contrasts with the other popular adaptation method, *h*-adaptation, where the local element size is modified from the current mesh. When combined with unstructured and anisotropic mesh generation capabilities, *h*-adaptation can improve mesh efficiency in boundary layers, wakes, shocks, etc. However, the disadvantage of *h*-adaptation is that it could experience large jumps in mesh size and require mesh regeneration. This potentially reduces the effectiveness and robustness of the approach. A related method, *r*-adaptation, is a simpler variation of *h*-adaptation. Instead of generating a new mesh, *r*-adaptation moves node locations without changing the mesh topology to improve the solution accuracy. The final approach is *hp*-adaptation, where adjustments in mesh size and interpolation order are combined. In this setting, *h*-adaptation is employed for non-smooth flow regions in the vicinity of singularities, and *p*-adaptation is used in smooth flow regions. Sometimes the choice of adaptation strategy in a particular element (*h* and/or *p*) is

* Corresponding author. Tel.: +65 64191591; fax: +65 6467 4350.
E-mail address: nvinhtan@hotmail.com, nguyenvt@ihpc.a-star.edu.sg (V.-T. Nguyen).

http://dx.doi.org/10.1016/j.apm.2015.08.006

unclear and criteria must be developed to aid that decision. It should be noted that for mesh refinement methods, it is difficult to guarantee the curvature of complex geometries. Refining and coarsening of regions near the domain boundaries usually generate problems [6].

In an effort to improve the robustness and automation of mesh adaptation, this paper proposes an adaptive *re-meshing* method that can be used for variety of problems in CFD and removes the meshing bottleneck which is common to boundary-conforming methods. It should be noted that this method is feature driven adaptivity and is different to mesh adaptation in transient flows involving moving boundaries as were proposed by Löhner [10]; Peraire et al. [14]; Morgan et al. [12]; Hassan et al. [8], etc. The present mesh adaptation is fundamentally different from goal-oriented approach used for unsteady flow simulations [2] in which optimal meshes are generated from a given output functional. Here an isotropic mesh adaptation based on solution features is proposed for unsteady simulations. Adopting isotropic mesh adaptation certainly makes the approach more robust than anisotropic ones in which boundary recovery phase may fail in the process of adaptation. Another difference of the proposed approach is that it does not use strategies such as edge split, edge collapse, edge swap, face swap, point move, etc as were employed in mesh modifications. Instead, it defines the holes locally based on an feature-based indicator and re-meshes these regions independently using surface and volume mesh generators. The surface meshes are performed using advancing front algorithm and volume meshes are based on Delaunay triangulation. The proposed re-meshing process is fully automatic without users intervention in run-time manner provided that initial surface and volume mesh can be generated from input geometries. In addition, it can perform mesh adaptation robustly and effectively in simulation run-time for complex configurations, especially for domains with curved boundaries. The numerical examples show that the use of the proposed adaptation strategy in three dimensions offers a great potential in having low cost CFD simulations with high quality mesh, resulting in more accurate solutions.

2. Problem statement

Considering unsteady inviscid compressible flows governed by the time-dependent, Euler equations on a three–dimensional Cartesian domain $\Omega \subset \mathbb{R}^3$, with surface $\partial\Omega$, it can be expressed in integral form as

$$\int_{\Omega} \frac{\partial \boldsymbol{U}}{\partial t} \mathrm{d}\boldsymbol{x} + \int_{\partial\Omega} \boldsymbol{F}_j n_j \mathrm{d}\boldsymbol{x} = \boldsymbol{0}, \tag{1}$$

where the conventional summation is employed and n_j is the outward unit normal vector to $\partial\Omega$. The unknown vector of the conservative variables, inviscid and viscous flux tensors are given by

$$\boldsymbol{U} = \begin{pmatrix} \rho \\ \rho u_1 \\ \rho u_2 \\ \rho u_3 \\ \rho\epsilon \end{pmatrix}, \quad \boldsymbol{F}_j = \begin{pmatrix} \rho u_j \\ \rho u_1 u_j + p\delta_{1j} \\ \rho u_2 u_j + p\delta_{2j} \\ \rho u_3 u_j + p\delta_{3j} \\ u_j(\rho\epsilon + p) \end{pmatrix}. \tag{2}$$

Here ρ denotes the fluid density, u_i the i'th componentof the velocity vector and ϵ the specific total energy. Fluid is considered as perfect gas with ideal equation of state $p = \rho RT$ and $\epsilon = c_v T + \frac{1}{2} u_k u_k$ where R is the real gas constant and $c_v = c_p - R$ is the specific heat at constant volume. In this expression, c_p is the specific heat at constant pressure. In this work, the ratio of the specific heats, $\gamma = \frac{c_p}{c_v}$ is set to $\gamma = 1.4$ for air at standard conditions.

Flow unsteady conditions are solved by first discretisation of the domain into a computational unstructured grid as a set of non overlapping tetrahedral elements. The governing equations are then solved on the discrete domain using second order cell based vertex centred finite volume approach with explicit time stepping scheme. For better capturing of flow features, solution based adaptivity is developed and employed to adjust computational grids. In subsequent sections, these techniques will be discussed in details.

3. Unstructured mesh generation

The computational domain Ω is subdivided into a set of non-overlapping tetrahedral elements using a unstructured mesh generation process. In this section, the methods for generating an unstructured grid are briefly summarised. In an unstructured mesh, the number of points and elements which are neighbours to an interior point is not kept constant throughout the domain. The mesh algorithm can handle arbitrary geometries in a fully automatic manner and provide control over the spatial mesh spacing throughout the domain. Therefore, the input data can be reduced to a geometric representation of the domain based on computer-aided design (CAD) defined geometries. The geometrical definition (or domain boundaries) contains curve and surface components. The curve components are the curvature continuous composite cubic splines. Surface components are represented by means of a rectangular network of points.

3.1. Background mesh and source distribution

Control over the mesh characteristics is obtained by the specification of a spatial distribution of mesh parameters. A background mesh as well as point, line and planar sources can be used to define the control function that specifies the distribution of the mesh spacings [16].