Computers & Fluids 118 (2015) 114–130

Contents lists available at ScienceDirect

Computers & Fluids

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

New high-resolution-preserving sliding mesh techniques for higher-order finite volume schemes

Luis Ramírez [a], Charles Foulquié [b], Xesús Nogueira [a,*], Sofiane Khelladi [b], Jean-Camille Chassaing [c], Ignasi Colominas [a]

[a] Grupo de Métodos Numéricos en Ingeniería, GMNI, Universidade da Coruña, A Coruña, Spain
[b] Laboratoire de Dynamique des Fluides, Arts et Métiers ParisTech, 151 boulevard de l'Hôpital, 75013 Paris, France
[c] Sorbonne Universités, UPMC Univ Paris 06, CNRS, UMR7190, D'Alembert Institute, F-75005 Paris, France

ARTICLE INFO

Article history:
Received 4 August 2014
Received in revised form 24 May 2015
Accepted 8 June 2015
Available online 16 June 2015

Keywords:
Sliding mesh
Unstructured grid
High-order finite volume scheme
Moving least-squares

ABSTRACT

This paper presents a new sliding mesh technique for the computation of unsteady viscous flows in the presence of rotating bodies. The compressible Euler and incompressible Navier–Stokes equations are solved using a higher-order (>2) finite volume method on unstructured grids. A sliding mesh approach is employed at the interface between computational grids in relative motion. In order to prevent loss of accuracy, two distinct families of higher-order sliding mesh interfaces are developed. These approaches fit naturally in a high-order finite volume framework. To this end, Moving Least Squares (MLS) approximants are used for the transmission of the information from one grid to another. A particular attention is paid for the study of the accuracy and conservation properties of the numerical scheme for static and rotating grids. The capabilities of the present solver to compute complex unsteady vortical flow motions created by rotating geometries are illustrated on a cross-flow configuration.

1. Introduction

Flow unsteadiness and noise generation are currently among the most important limitations for the design of turbomachinery and renewable energy devices. These configurations involve complex unsteady flow patterns like vortex shedding, stalled flows, blade wake interactions which are, in general, due to the presence of moving or oscillating bodies. On one hand, one must employ high-order numerical methods to accurately compute both the unsteady flow field and the aeroacoustic field. On the other hand, dedicated techniques must be employed to carefully deal with the interface between static and moving grids in an unsteady flow framework. Such issue can be addressed using several numerical approaches, among others, the phase-lagged periodic boundary conditions for rotor–stator interaction in axial compressor [10,16,17], the body-fitted approach in an Arbitrary Lagrangian Eulerian (ALE) setting, Cartesian grid methods based on the immersed-boundary [40] or on the cut cell methods [3,54] and the non-boundary conforming sliding mesh approach. The later is attractive due to its ability to capture flow unsteadiness without requiring the use of a filtering procedure nor computationally expensive re-meshing strategies. The sliding mesh method was successfully employed by Rai [42,43] for the computation of rotor–stator interactions in a supersonic flow. This patched-grid technique allows relative sliding of one mesh adjacent to another static or moving mesh. A three steps explicit zonal scheme, which preserve flux conservation at the interface, is proposed in [41]. More recently, Gourdain et al. [15] employed the sliding mesh approach for the simulation of large-scale industrial flows in multistage compressors. In a comparative study between Chimera and sliding mesh techniques for unsteady simulations of counter rotating open-rotors, Francois et al. [14] shown that these methods give similar accuracy but the later requires much less memory than the Chimera approach. Note also that the sliding mesh algorithm was used by Steijl and Barakos [46] for the computational fluid dynamic analysis of helicopter rotor-fuselage aerodynamics.

Nowadays, sliding mesh techniques are commonly used to compute non-axisymmetrical unsteady flow fields and corresponding aerodynamic performances of cross-flow fans [33,49] and wind turbines [18,20,21,23,26,19,22,1]. In particular, McNaughton et al. [31] obtained a good agreement between coupled LES-sliding interfaces for thrust and power predictions of a tidal-stream turbine. As far as aeroacoustic computations are concerned, Moon et al. [33] developed a time-accurate viscous flow solver for the prediction of unsteady flow characteristics and the

* Corresponding author.
E-mail address: xnogueira@udc.es (X. Nogueira).

http://dx.doi.org/10.1016/j.compfluid.2015.06.008
0045-7930/

associated aeroacoustic blade tonal noise of a cross flow impeller. The sliding mesh approach, which was implemented in an unstructured finite-volume solver on triangular meshes, was able to correctly predict the oscillations of the eccentric vortex due to the mismatch of blade incidence angles and the recirculation bubbles around the blades.

However, most of the sliding mesh methods proposed in the literature belong to the family of low order interpolation schemes. Therefore they cannot be used in conjunction with higher-order numerical schemes without depreciating the overall accuracy of the numerical methods.

To the authors knowledge, few studies addressed such problem. A high order ($order \geqslant 3$) h/p Discontinuous Galerkin method with sliding mesh capabilities was recently proposed by Ferrer and Willden [12] for the computation of the unsteady incompressible flow field of a three bladed cross-flow turbine. They have successfully obtained spectral convergence rate when solving the incompressible Navier–Stokes equations on non-conformal grids. In [2] a mesh moving technique for sliding interfaces is presented for the numerical simulation of a wind turbine with a FEM-based ALE-VMS (variational multiscale formulation written in the arbitrary Lagrangian–Eulerian frame) formulation.

In this work, we intent to develop higher-order sliding mesh interface for the solution of transient flows on mixed rotating and static computational domains. To this end, we consider a high-resolution finite volume method based on Moving Least Squares (MLS) reconstructions.

The theoretical fundamentals of the used finite volume method (FV-MLS) were presented in [9,24,36,35,44] and references therein. A first application of FV-MLS for turbomachinery aeroacoustics was presented in [38]. In those works, artificial acoustic sources were propagated using the Linearized Euler Equations. Only stator blades and rotating sources into the propagating medium were considered. This first tentative permits to study the attenuation due to the acoustic screen effect of stator blades. The next step is to introduce the rotating part into the propagation medium by the use of sliding mesh method coupled to FV-MLS solver. In this work we present a sliding mesh model based on the use of Moving Least Squares (MLS) approximants [25]. It is used with a high-order (>2) finite volume method that computes the derivatives of the Taylor reconstruction inside each control volume using MLS approximants [9,24,36,35]. Thus, this new sliding mesh model fits naturally in a high-order finite volume framework for the computation of acoustic wave propagation into turbomachinery. We present two different approaches based on MLS approximants for the transmission of information from one grid to another. An interface-type sliding mesh approach, and a new methodology that does not require the computation of intersections.

The paper is organized as follows. In Section 2 the governing equations are written. In Section 3, the basic finite volume formulation is presented. Moving Least Squares (MLS) approximation and the FV-MLS method are briefly described in Section 4. The new MLS-based sliding-mesh technique is presented in Section 5. Then, Section 6 is devoted to numerical simulations. Finally, the conclusions are drawn.

2. Governing equations and numerical methods

In order to account the relative mesh motion of one mesh with respect to other, it is advantageous to write the two dimensional compressible Navier–Stokes equations in the Arbitrary Lagrangian–Eulerian (ALE) form,

$$\frac{\partial \boldsymbol{U}}{\partial t}+\frac{\partial\left(\boldsymbol{F}_x-\boldsymbol{F}_x^V\right)}{\partial x}+\frac{\partial\left(\boldsymbol{F}_y-\boldsymbol{F}_y^V\right)}{\partial y}=\boldsymbol{0} \tag{1}$$

where $\boldsymbol{U}$ is the vector of variables $\boldsymbol{F}=(\boldsymbol{F}_x,\boldsymbol{F}_y)$ is the inviscid flux vector and $\boldsymbol{F}^V=\left(\boldsymbol{F}_x^V,\boldsymbol{F}_y^V\right)$ is the viscous flux vector.

For compressible flows the conservatives variables are defined as

$$\boldsymbol{U}(\boldsymbol{x},t)=\begin{Bmatrix}\rho\\ \rho v_x\\ \rho v_y\\ \rho E\end{Bmatrix} \tag{2}$$

and the inviscid fluxes are given by

$$\boldsymbol{F}_x=\begin{pmatrix}\rho v_x-\rho v_x^{mesh}\\ \rho v_x^2+p-\rho v_x v_x^{mesh}\\ \rho v_x v_y-\rho v_y v_x^{mesh}\\ \rho v_x H-\rho E v_x^{mesh}\end{pmatrix}\qquad \boldsymbol{F}_y=\begin{pmatrix}\rho v_y-\rho v_y^{mesh}\\ \rho v_x v_y-\rho v_x v_y^{mesh}\\ \rho v_y^2+p-\rho v_y v_y^{mesh}\\ \rho v_y H-\rho E v_y^{mesh}\end{pmatrix} \tag{3}$$

where the (u_{mesh}, v_{mesh}) is the mesh velocity. The viscous fluxes $\boldsymbol{F}^V$ are given by the following expression,

$$\boldsymbol{F}_x^V=\begin{pmatrix}0\\ \tau_{xx}\\ \tau_{xy}\\ v_x\tau_{xx}+v_y\tau_{xy}-q_x\end{pmatrix}\quad \boldsymbol{F}_y^V=\begin{pmatrix}0\\ \tau_{xy}\\ \tau_{yy}\\ v_x\tau_{xy}+v_y\tau_{yy}-q_y\end{pmatrix} \tag{4}$$

The viscous stresses are modeled as

$$\begin{aligned}\tau_{xx}&=2\mu\frac{\partial v_x}{\partial x}-\frac{2}{3}\mu\left(\frac{\partial v_x}{\partial x}+\frac{\partial v_y}{\partial y}\right)\\ \tau_{yy}&=2\mu\frac{\partial v_y}{\partial y}-\frac{2}{3}\mu\left(\frac{\partial v_x}{\partial x}+\frac{\partial v_y}{\partial y}\right)\\ \tau_{xy}&=\mu\left(\frac{\partial v_x}{\partial y}+\frac{\partial v_y}{\partial x}\right)\end{aligned} \tag{5}$$

For incompressible flows, the assumption of incompressibility lead us to a system of equations with the following variables

$$\boldsymbol{U}(\boldsymbol{x},t)=\begin{Bmatrix}0\\ v_x\\ v_y\end{Bmatrix} \tag{6}$$

The inviscid fluxes are

$$\boldsymbol{F}_x=\begin{pmatrix}\rho v_x-\rho v_x^{mesh}\\ \rho v_x^2+p-\rho v_x v_x^{mesh}\\ \rho v_x v_y-\rho v_y v_x^{mesh}\end{pmatrix}\qquad \boldsymbol{F}_y=\begin{pmatrix}\rho v_y-\rho v_y^{mesh}\\ \rho v_x v_y-\rho v_x v_y^{mesh}\\ \rho v_y^2+p-\rho v_y v_y^{mesh}\end{pmatrix} \tag{7}$$

The viscous fluxes are given by

$$\boldsymbol{F}_x^V=\begin{pmatrix}0\\ \mu\frac{\partial v_x}{\partial x}\\ \mu\frac{\partial v_y}{\partial x}\end{pmatrix}\quad \boldsymbol{F}_y^V=\begin{pmatrix}0\\ \mu\frac{\partial v_x}{\partial y}\\ \mu\frac{\partial v_y}{\partial y}\end{pmatrix} \tag{8}$$

3. Basic finite volume formulation

The basic finite volume discretization stems from the integral form of Eq. (1) over a control volume Ω_I

$$\int_{\Omega_I}\frac{\partial \boldsymbol{U}}{\partial t}d\Omega+\int_{\Omega_I}\frac{\partial\left(\boldsymbol{F}_x-\boldsymbol{F}_x^V\right)}{\partial x}d\Omega+\int_{\Omega_I}\frac{\partial\left(\boldsymbol{F}_y-\boldsymbol{F}_y^V\right)}{\partial y}d\Omega=\boldsymbol{0} \tag{9}$$

Using the divergence theorem for the viscous and inviscid fluxes the following expression is obtained