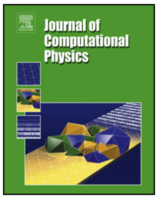
Journal of Computational Physics 330 (2017) 810–827

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp



Time efficient aeroelastic simulations based on radial basis functions

Wen Liu [*], ChengDe Huang, Guowei Yang

Institute of Mechanics, Chinese Academy of Sciences, Beijing 100190, China

ARTICLE INFO

Article history:
Received 8 October 2015
Received in revised form 19 October 2016
Accepted 27 October 2016
Available online 2 November 2016

Keywords:
Aeroelastics
Fluid–structure interpolation
Mesh deformation
Radial basis functions

ABSTRACT

Aeroelasticity studies the interaction between aerodynamic forces and structural responses, and is one of the fundamental problems to be considered in the design of modern aircraft. The fluid–structure interpolation (FSI) and mesh deformation are two key issues in the CFD–CSD coupling approach (the partitioned approach), which is the mainstream numerical strategy in aeroelastic simulations. In this paper, a time efficient coupling scheme is developed based on the radial basis function interpolations. During the FSI process, the positive definite system of linear equations is constructed with the introduction of pseudo structural forces. The acting forces on the structural nodes can be calculated more efficiently via the solution of the linear system, avoiding the costly computations of the aerodynamic/structural coupling matrix. The multi-layer sequential mesh motion algorithm (MSM) is proposed to improve the efficiency of the volume mesh deformations, which is adequate for large-scale time dependent applications with frequent mesh updates. Two-dimensional mesh motion cases show that the MSM algorithm can reduce the computing cost significantly compared to the standard RBF-based method. The computations of the AGARD 445.6 wing flutter and the static deflections of the three-dimensional high-aspect-ratio aircraft demonstrate that the developed coupling scheme is applicable to both dynamic and static aeroelastic problems.

© 2016 Elsevier Inc. All rights reserved.

1. Introduction

As a typical fluid–structure interaction phenomenon, aeroelasticity is one the fundamental problems to be considered in the design of modern aircraft. The numerical simulations of aeroelastic problems can be classified broadly under two categories: the monolithic approach that solves the aerodynamic forces and structural responses simultaneously using the integrated aero-structural solver; and the partitioned approach (coupling approach) that solves the aerodynamic forces and structural motions in a separate manner with additional interfacing technique to communicate between different solvers. The monolithic approach usually requires the solutions of integrated aero-structural equations in both Eulerian and Lagrangian systems [1–3]. This leads to the matrices being orders of magnitudes stiffer for structure system than fluid system, which makes it virtually impossible to solve the equations for large-scale problems. The partitioned approach, however, allows the use of existing tools for computational fluid dynamics (CFD) and computational structure dynamics (CSD). This flexibility of choosing different solvers maintains the independence of the fluid and structure system, and thereby is favored in current computational aeroelastic researches.

---

[*] Corresponding author.
E-mail address: liuwen@imech.ac.cn (W. Liu).

http://dx.doi.org/10.1016/j.jcp.2016.10.063
0021-9991/© 2016 Elsevier Inc. All rights reserved.

One issue for the partitioned approach is the data communication between fluid and structure systems. Because both the aerodynamic and structural domains are discretized in a physically different manner, the two meshes will generally not coincide at the fluid–structure interface. In order for the calculation to proceed, it is necessary to transfer the aerodynamic loads from the aerodynamic surface to the structural nodes so that the deflections can be computed by the CSD method. These deflections then need to be transferred back to deform the aerodynamic surface consistently for the CFD computation. Numerous approaches have been investigated for this CFD–CSD interpolation, including the methods of Infinite Plate Spline (IPS) [4], Inverse Isoparametric Mapping (IIM) [5,6], Constant Volume Tetrahedra (CVT) [7,8], and Boundary Element (BEM) [9,10], etc. The partitioned approach typically involves motions of fluid–structure interface due to the deflection of the structure. To be able to perform the CFD computations accurately with the moving boundaries, it is usually necessary to adapt the fluid volume mesh based on the deformation of flow boundaries. It is a natural choice to regenerate the entire CFD mesh according to changes of the fluid domain. However, the generation of a complex grid is usually nontrivial and computationally expensive, especially in the case that the mesh update needs to be performed each time step in an unsteady flow computation. Furthermore, the grid topology is often not preserved by generating new mesh, which may introduce more uncertainties for the numerical errors. Mesh deformation is another choice to update the CFD grid, which could potentially be a more convenient and efficient approach with mesh topology preserved. Many ways to deform the volume mesh have been investigated, such as Transfinite Interpolation (TFI) [11], Spring Analogy (SA) [12–14], PDE solution method [15,16], etc. However, all the mesh type dependent methods require the knowledge of the grid connectivity. The displacement of interior grids usually needs to be solved by a system of equations including all the points in the domain, and therefore can be computationally expensive. Hence, for large meshes or time-dependent simulations with frequent mesh updates, effective methods requiring no connectivity information are preferable. Liu et al. [17] has developed a fast approach that utilizes the Delaunay mapping to interpolate deformations with a simple meshless interpolation algorithm. Although this scheme is quite effective, it makes no attempt to preserve mesh orthogonality near the deforming boundary, and large boundary movements may cause some points move across each other which results in negative coefficients for the interpolation.

Recently, Rendall and Allen [18] presented a multivariate interpolation scheme using RBF, which leads to a unified formulation for the fluid–structure interpolation (FSI) and mesh motion problems. The proposed RBF-based method was demonstrated to be a generic approach applicable to arbitrary mesh type. The displacement of the targeted point (aero-surface node in the FSI or volume node in the mesh deformation) can be interpolated, with no connectivity information required, by the control points at the boundary (structure node in the FSI or aero-surface node in the mesh deformation). A small system of equations, only involving the boundary points, has to be solved to determine the interpolation function, and no elaborate computations are required to evaluate the movements of targeted points except simple matrix–vector multiplications. The computational effort associated with the volume mesh motion scales with $N_{\mathrm{vp}} \times N_{\mathrm{sp}}$ (N_{vp} is the total number of volume points, and N_{sp} denotes the number of control points at the boundary). Rendall and Allen [19] improved the performance of the RBF interpolation method by the implementation of the 'greedy' algorithm to reduce the size of the control points at the boundaries, sacrificing the accuracy of the deformation at the surface boundary with an acceptable error. In time-dependent simulations with large meshes, the volume mesh update may still consume significant computing resources, since the evaluations usually contain millions of multiplication and summation operations at each time step where the number of volume grid N_{vp} is massive. Therefore, it is necessary to further reduce the size of volume mesh in the interpolation to improve the efficiency. Michler [20] presented a confinement technique, which locally restricts the mesh deformation to the vicinity of the moving components enclosed by the control surface. However, the confinement technique was initially designed for mesh deformations with prescribed local component deflections, and thereby was highly configuration dependent.

In the present paper, a time efficient coupling scheme is developed based on the RBF interpolations. During the FSI process, the positive definite system of linear equations is constructed with the introduction of pseudo structural forces. The acting forces on the structural node can be calculated more efficiently via the solution of the linear system, avoiding the costly computations of the aerodynamic/structural coupling matrix in the previous study [18]. For mesh motions, the multilayer sequential mesh motion (MSM) algorithm is proposed to improve the efficiency of the volume mesh deformations in large mesh applications. In MSM, the evaluation of volume mesh motions can be split into sequential sub-layers, each of which has its specific interpolation relation. The total computational effort equals to the summation of the local cost in each sub-layer where both surface control points and volume points can be reduced considerably, and therefore the performance of the mesh deformation can be effectively improved. The MSM algorithm has great advantages in deforming the mesh of multi-element geometry with local component deflections (e.g. the flap deflection in the high-lift wing system). By confining the mesh deformation in the vicinity of the moving component surface, the use of MSM could effectively reduce the overall computational complexities, and more importantly, maintain the grid consistency in the majority part of the domain which makes the deformed mesh more robust for the time-dependent simulations. Moreover, the MSM algorithm can be implemented easily with no requirement of additional boundary conditions, and has the potential to become a generic mesh deformation method.

The organization of present paper is as follows. Section 2 describes the partitioned strategy for the fluid–structure interaction simulations. Section 3 discusses the fluid–structure interpolation using RBF, and a time efficient solution strategy is derived to calculate the structural forces. The MSM algorithm for volume mesh deformations is proposed in Section 4. The

Download English Version:

https://daneshyari.com/en/article/4967779

Download Persian Version:

https://daneshyari.com/article/4967779

Daneshyari.com