Advances in Applied Mathematics and Mechanics
Adv. Appl. Math. Mech., Vol. **5**, No. 2, pp. 194-211

DOI: 10.4208/aamm.11-m1186
April 2013

A Pseudo-Stokes Mesh Motion Algorithm

Luciano Gonçalves Noleto[2,*], Manuel N. D. Barcelos Jr.[2] and Antonio C. P. Brasil Jr.[1]

[1] *Universidade de Brasília, Faculdade de Tecnologia, Departamento de Engenharia Mecânica, Laboratório de Energia e Ambiente. Asa Norte. Brasília-DF, Brasil, 70.910-900*
[2] *Universidade de Brasília no Gama. Complexo de Educação, Cultura, Esporte e Lazer. Área Especial de Indústria 1, Setor Leste, Faculdade Lote 1. Gama-DF, Brasil, 72.444-210*

Received 3 May 2011; Accepted (in revised version) 10 October 2012

Available online 22 February 2013

Abstract. This work presents a moving mesh methodology based on the solution of a pseudo flow problem. The mesh motion is modeled as a pseudo Stokes problem solved by an explicit finite element projection method. The mesh quality requirements are satisfied by employing a null divergent velocity condition. This methodology is applied to triangular unstructured meshes and compared to well known approaches such as the ones based on diffusion and pseudo structural problems. One of the test cases is an airfoil with a fully meshed domain. A specific rotation velocity is imposed as the airfoil boundary condition. The other test is a set of two cylinders that move toward each other. A mesh quality criteria is employed to identify critically distorted elements and to evaluate the performance of each mesh motion approach. The results obtained for each test case show that the pseudo-flow methodology produces satisfactory meshes during the moving process.

AMS subject classifications: 76M10, 65M50, 35R37

Key words: Moving boundaries, mesh motion strategy, finite element projection methods.

1 Introduction

Nowadays for several CFD problems, the fluid domain encloses or is enclosed by moving boundaries. Thus, when the moving boundary is subjected to a large amplitude motion, a moving mesh strategy is necessary. Therefore, a mesh motion algorithm has to be employed throughout a simulation. This algorithm is basically a numerical solver that

*Corresponding author.
Email: lucianonoleto@unb.br (L. G. Noleto), manuelbarcelos@unb.br (M. N. D. Barcelos Jr.), brasiljr@unb.br (A. C. P. Brasil Jr.)

computes and updates mesh node displacements and velocities, evaluated at any given time step.

The mesh motion algorithm must fulfill three conditions to be efficient and reliable [10, 11]:

- The algorithm must be compatible with the Geometric Conservation Law;
- The mesh after moving must have good quality;
- The computational effort for mesh motion must be small.

The geometric conservation law states that any change in the element control area or volume between two consecutive time steps must be equal to the area or volume swept by the cell boundary during these time steps [17, 19]. As a consequence, the geometric quantities associated to the moving mesh must be computed in a way that the integration preserves a uniform flow field. Literature shows that the geometric conservation law can be represented by a null mesh velocity divergent constraint [21]. Also, there are cases where the non-fulfillment of this constraint leads to numerical instabilities and as a consequence to wrong flow solutions or spurious results [7, 17].

The first numerical schemes [2, 16] to solve the mesh motion problem modeled the mesh as a fictitious structure where linear springs were placed between the nodes. The stiffness of every linear spring is inversely proportional to the distance between the nodes. The total stiffness of the pseudo structure has the contribution of each linear spring. Due to the model characteristics, the finite element method (FEM) was a natural choice for solving the problem. However, this procedure was limited to problems with small mesh deformations between consecutive time steps because large deformations may lead to element collapsing.

Other mesh motion approaches compute the mesh displacement field by solving a diffusion problem. The displacements are associated to nodal velocity field. The problem formulation is based on the Laplacian of the nodal velocity field. Hence, the Laplace equations are solved, and the mesh displacements are recovered by multiplying the resulting velocity field by a predefined time step. The main drawback of this method is the choice of the right diffusivity coefficient to avoid collapsed elements in the moving mesh process.

Robust mesh motion approaches based on the solution of pseudo structure problems use the superposition of linear and torsional springs [9]. From the beginning, the linear and torsional springs method was developed for two-dimensional triangular elements. Adapting this technique to three-dimensional elements such as tetrahedrons required a repositioning of the torsional spring action plane which avoids the element face and volume collapse [6]. For large mesh deformation, distorted element might still be produced and as a consequence this can lead to numerical stiffness and wrong flow solutions.

In the last few years several works in the literature proposed new mesh motion approaches employing geometrical function interpolation and mapping to tackle large mesh deformation problems. For instance, [28] presented a mesh moving algorithm for dynamic meshes. A Delaunay graph is generated for the domain to represent the mov-