8th. World Congress on Computational Mechanics (WCCM8)
5th European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS 2008)
June 30 –July 5, 2008
Venice, Italy

CONSERVATIVE TREATMENT OF 3D MULTI-BLOCK UNSTRUCTURED MESH INTERFACES FOR FINITE VOLUMES COMPUTATIONS OF FLUID FLOWS WITH MOVING BOUNDARIES

D. Vigneron[1] and J.-A. Essers[2]

[1] Assistant
A&M Department, University of Liège
Chemin des chevreuils, 1, Bât B52/3
B-4000 Liège, Belgium
Didier.Vigneron@ulg.ac.be

[2] Professor
A&M Department, University of Liège
Chemin des chevreuils, 1, Bât B52/3
B-4000 Liège, Belgium
JA.Essers@ulg.ac.be

Key Words: *Unsteady Fluid Flows, Finite Volumes Methods, Multi-block Mesh, Mesh Deformation.*

ABSTRACT

Computations of unsteady fluid flows with moving boundaries must be made on moving grids. Mesh deformation techniques, often based on elasticity models, have been designed to adapt the mesh to a new computation domain geometry at each time step. Nevertheless, for large movements of boundaries or immersed bodies, these techniques will be unable to maintain a sufficiently good mesh quality. To avoid a necessary remeshing step, one solution is to use a multi-block mesh, composed of a number of meshes that can be independently deformed, instead of one single block mesh.

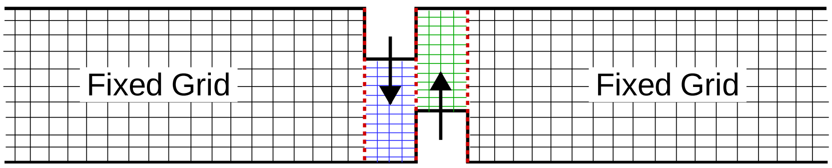
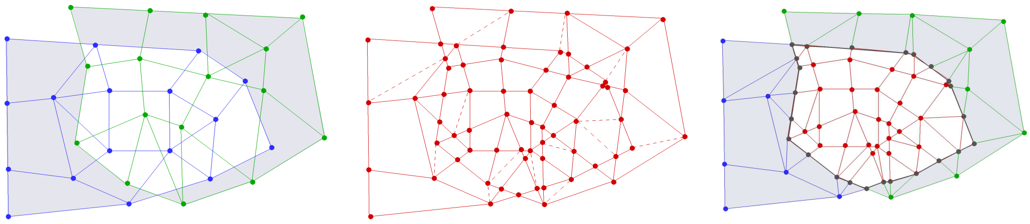


Figure 1: 2D illustration of a moving boundary problem involving the use of a multi-block mesh

This fact is illustrated with an example on Figure 1. Consider a flow in a rectangular duct. This flow is stopped by a mechanism consisting of two walls moving up and down respectively. To simulate such a flow, the grid must be deformed at each time step. It is easy to understand that it is impossible to maintain mesh quality if only one mesh is used for the whole of the domain. Therefore, one solution is to use four meshes that can be independently deformed. Two of them are fixed while the two others are easily deformed. In this multi-block mesh configuration, the problem is to deal with the interfaces (represented in dashed red lines) between the meshes and to detect fluid/fluid or fluid/wall interfaces. In this simple 2D case, the interface is a line. For 3D cases, the interface is a surface. This paper presents an algorithm for the treatment of those multi-block interfaces for 3D unstructured moving grids in the framework of a finite volumes method, maintaining the conservativity of the scheme.

The unsteady Navier-Stokes equations are integrated in space with a cell-centered finite volumes method on 3D moving grids. Therefore, they are written in conservative arbitrary Lagrangian-Eulerian (ALE) form and the discrete geometric conservation law (DGCL), assuring stability, is applied to the Crank-Nicolson scheme. The grid cells can be any kinds of polyhedra formed by any numbers of triangular and quadrangular faces. The unknowns are stored at the gravity center of each cell. These gravity centers are called the nodes. Two neighboring cells are separated by one or a number of faces that must be triangles or quadrangles. Therefore, one face has two neighboring nodes called the right and left nodes. The advective and viscous fluxes are integrated on each faces using the left and right values. If a face is situated on a boundary, it has only one left node and boundary conditions are imposed.



(a) (b) (c)

Figure 2: Interface treatment between two 3D non conformal meshes with walls. (a) Initial configuration. (b) Intersection of the two meshes is computed. (c) Fluid/fluid and fluid/wall interfaces are detected.

For such a 3D unstructured multi-block mesh, the interface between two mesh blocks is a (possibly non planar) surface. This interface is composed of the boundary meshes of the two 3D mesh blocks. This situation is illustrated on Figure 2. The two boundary meshes, which are 2D triangles and/or quadrangles meshes, are represented in blue and green colors. The gray zone represents the walls on which boundary conditions must be imposed (fluid/wall interface). The white zone is the interface of the two meshes (fluid/fluid interface). This paper presents an algorithm that creates a new mesh for the interface. Each face of this new mesh must have only one left and only one right neighbors expect for those that are situated on walls. The algorithm finds all edges intersections between the two meshes, creates new edges and new vertices and cuts all faces into a number of macro-faces that have only one left and one right neighbors. After this step, macro-faces are divided into triangles and quadrangles. Fluid/fluid and fluid/wall interfaces are detected to compute advective, viscous and boundary conditions fluxes.

Some test-cases will be presented to illustrate the convenience of using such multi-block meshes, to verify the conservativity and to measure the CPU time needed by the algorithm.

REFERENCES

[1] J.D. Ramshaw. “Conservative Rezoning Algorithm for Generalized Two-Dimensional Meshes”. *Journal of Computational Physiscs.*, Vol. **59**, 193–199, 1985.

[2] I.Lepot, D. Vigneron, J.A. Essers, O. Leonard ”Implicit High-Order Geometrically Conservative Scheme for the Solution of Flow Problems on Moving Unstructured Grids”, in *Second MIT Conference on Computational Fluid and Solid Mechanics*, June 23-27, 2003 / MA, USA.

[3] D. Vigneron, J.-M. Vaassen, J.A. Essers ”A Finite Volume Method with Conservative and Consistent Viscous Terms Discretization for Three-Dimensional Flows”, in *the 35th AIAA Fluid Dynamics Conference and Exhibit*, June 6-9, 2005 / Toronto, Canada.