A 3D Multi-Block Mesh Interface Treatment For Finite Volume Fluid Flows Computations

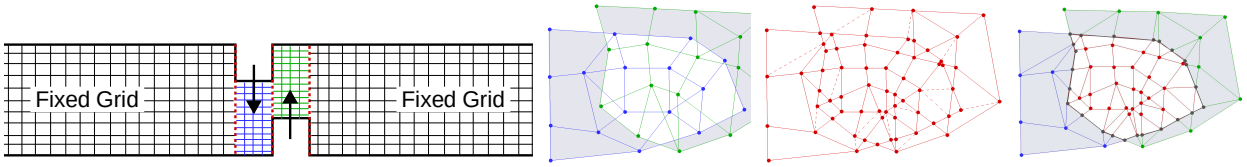
D. Vigneron, M. Drosson and J.-A. Essers

A&M Department, University of Liège

Chemin des chevreuils, 1, Bât B52/3,B-4000 Liège, Belgium

e-mails: *Didier.Vigneron@ulg.ac.be*, *M.Drosson@ulg.ac.be*, *JA.Essers@ulg.ac.be*

Computations of unsteady fluid flows with moving boundaries can be made on moving grids. Mesh deformation techniques have been designed to adapt the mesh to a new computation domain geometry at each time step. Nevertheless, for some types of boundary movements, these techniques will be unable to maintain a sufficiently good mesh quality. For example, figure 1a illustrates a duct flow stopped by two walls moving up and down. It is easy to understand that it is impossible to maintain the mesh quality if only one mesh is used for the whole domain. To avoid a necessary remeshing step, one solution is to use a multi-block mesh, composed of a number of meshes that can be deformed independently. The problem is to deal with the interfaces (represented in dashed red lines) between non conformal meshes and to detect fluid/fluid or fluid/wall interfaces. This paper presents an algorithm for the treatment of those interfaces for 3D unstructured grids, sliding on each other, in the framework of a cell-centered finite volume method (for which the fluxes computation for the discretization on non conformal meshes is straightforward), maintaining conservativity. The interface between two 3D mesh blocks is a (possibly non planar) surface mesh composed of triangles and/or quadrangles (Figure 1b). The algorithm must create, for the interface, a new mesh composed of faces having only one left and only one right neighbouring nodes except for those that are located on the walls. Therefore, it 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 neighbour and one right neighbour (Figure 1c). After this step, macro-faces are divided into triangles and quadrangles. Fluid/fluid and fluid/wall faces are detected to compute advective, viscous and boundary conditions fluxes (Figure 1d). Test-cases are presented to illustrate the convenience of using multi-block meshes, to verify the conservativity and to measure the CPU time needed by the algorithm.



(a) Simple 2D illustration (b) Initial interface (c) Intersection (d) New interface mesh

Figure 1: Interface treatment between two non conformal meshes. (a) 2D illustration. (b) Interface between two 3D non conformal meshes. Walls are illustrated in gray color. (c) Intersection computation. (d) Creation of one single interface mesh. Fluid/fluid and fluid/wall faces detection.

References

[1] J.D. Ramshaw. Conservative rezoning algorithm for generalized two-dimensional meshes. *Journal of Computational Physics*, 59:193–199, 1985.

[2] I. Lepot, D. Vigneron, J.-A. Essers, and O. Léonard. Implicit high-order geometrically conservative scheme for the solution of flow problems on moving unstructured grids. In Elsevier Science Ltd, editor, *Second MIT Conference on Computational Fluid and Solid Mechanics*, pages 992–996, MA,USA, June 2003.