ParCFD'2020
32^{nd} International Conference on Parallel Computational Fluid Dynamics
May-11-13 2020, Nice FRANCE

COMPARISON OF SLIDING MESH AND IMMERSED BOUNDARY METHODS IN UNSTEADY SIMULATIONS

J. VERA*, J. MUELA*,† F. FAVRE‡ AND C. OLIET*

*Heat and Mass Transfer Technological Center (CTTC) Universitat Politècnica de Catalunya-BarcelonaTech (UPC) ESEIAAT, Colom 11, 08222, Terrassa (Barcelona), Spain
E-mail: cttc@cttc.upc.edu, Web page: http://www.cttc.upc.edu

†Termo Fluids S.L., Av. Jacquard 97-E, 08222, Terrassa (Barcelona), Spain.
Web page: http://www.termofluids.com/

‡Instituto de Ingeniería Mecánica y Producción Industrial (IIMPI), Universidad de la República, Uruguay

Key words: Sliding-Mesh, Dynamic Mesh, Immersed Boundary Method

Abstract. The feasibility and accuracy of two methods for simulating moving bodies inside a fluid are studied. Different cases are presented using the Immersed Boundary Method and the Sliding Mesh technique. The results obtained by both methods are compared and assessed. These results allow to study the accuracy of both methods as well as their computational feasibility, in terms of computation time, mesh requirements and parallel scalability.

1 INTRODUCTION

In some cases of interest aimed to be studied by means of Computational Fluid Dynamics (CFD) certain moving objects within the simulation domain have to be considered (e.g., wind turbines, turbo-machinery, etc.) In order to properly describe the effect that these objects have on the fluid movement different approaches can be used. The objective of this work is to study the efficiency, accuracy and limitations of two approaches used in this field, the Immersed Boundary Method and the Sliding Mesh technique.

The Immersed Boundary Method (IBM) was first described by Peskin [1] and several improvements since the first approach have been made [2–5]. The method consists in adding a source term to the Navier-Stokes equations to describe a solid immersed in a fluid. The interface of the solid is located with a Lagrangian set of points, and the fluid is solved in a static Eulerian mesh, modifying the equations at the vicinity of the solid. This method allows the versatility of describing a solid at various positions without needing it to be conformal to the mesh. Moreover, it also allows to describe moving objects. The Sliding Mesh method [6, 7] is one of the most common techniques of dynamically connecting meshes, along with the chimera (overset) method [8] It is mainly used in cases where the solid is rotating along a fixed axis, (e.g., wind turbines, turbo-machines, etc.). This approach consists on employing two different meshes: the first one uses an static

Eulerian framework, while the second mesh is rotating with the solid, in an Eulerian-Lagrangian framework. These two meshes are none-overlapping and share a common boundary, where the communication between them is performed.

Both methods can describe the movement of objects, so it would be of interest to determine which one is more suitable for each application.

Immersed Boundary Methods have the limitation that as the *Re* number increases, the meshing near the boundary must be refined. Not having a constant boundary requires a finer mesh at all the possible positions of the solid. Another disadvantage regarding the meshing is that prismatic layers cannot be used because the normal at the surface is also dynamic. This two drawbacks increase the computational cost for turbulent cases. In addition, as the mesh size increases, the localization of the solid body for each cell and the modification of equations can be very time consuming.

On the other hand, Sliding Mesh Methods allow a refinement near the body, even prismatic layers can be used, so more adequate and optimal meshes can be used. However, this method does not allow the versatility of the IBM in the movement of the solid objects. The performance of Sliding mesh technique is very dependent on how the communication between both domains through the shared boundary is performed. Therefore, how the information is transfered in the sliding-boundary is a key aspect both in the accuracy of the method and its performance.

2 METHODOLOGY

The first case used to compare the methodologies is based in the standard case of flow past a circular cylinder. This case is modified so that the cylinder is rotating around an eccentric axis, placed at the cylinders boundary, as shown in Figure 1.

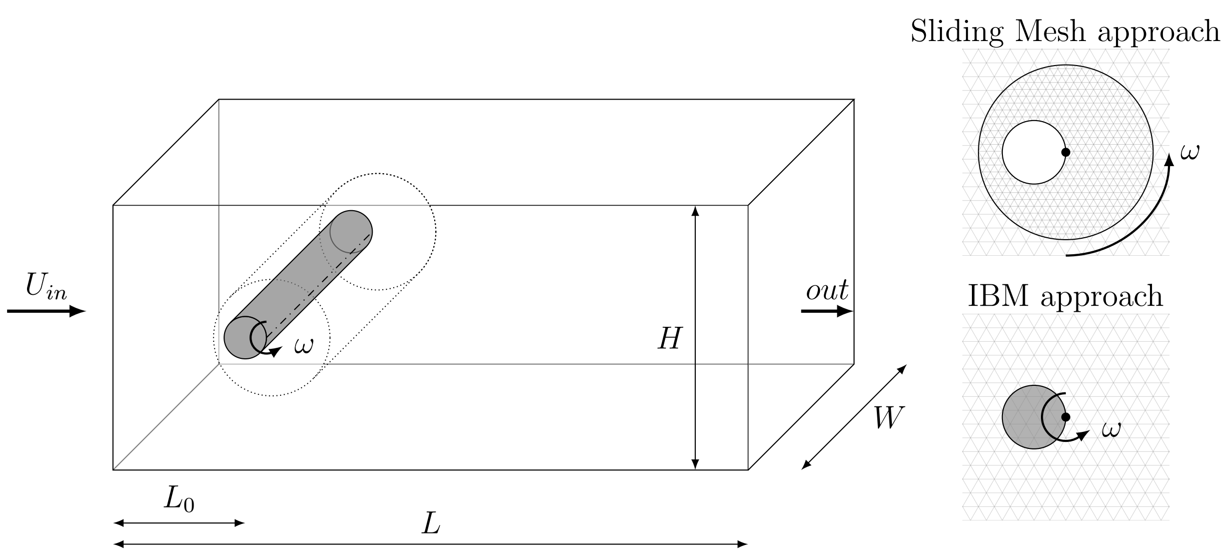

Figure 1: Geometry and approach of the case

The Reynolds number of the case is $Re = \rho U_{in} D/\mu$, where U_{in} is the inlet velocity and

D is the cylinder diameter. The rotating velocity of the cylinder is set to $\omega = U_{in}/D$

For the Immersed Boundary Method, an STL surface defines a cylinder, and a discrete forcing approach is used [2]. The discrete forcing approach consists in modifying the discretized Navier-Stokes equations adding a forcing term that imposes the velocity of the cell at the vicinity of the solid. A cut cell approach [9] is used in cases where more accuracy is required. The cut cell approach consists on modifying the Poisson equation coefficients accounting for the shape of the cell when intersected with the solid.

The mesh is refined as the Re number of the case increases, this refinement is performed specially in the region where the solid moves, and the results of the case are compared to the Sliding Mesh case results.

For the Sliding mesh approach, two different meshes are used, a fixed mesh and a moving one. In order to stitch the two independent meshes, an efficient method to locate the edges and faces to be intersected is employed. The algorithm consists on an improved version of the methodology developed in Muela et al. [10]. The new method employs auxiliary Lagrangian particles that move move jointly with the rotary domain, allowing tho optimize the projection-intersection step. The intersection is performed by means of the Sutherland-Hodgman algorithm [11]. The new faces resulting from the projection-intersection step are employed to reconstruct the topology of the mesh at each iteration. This new topology at the sliding boundary is employed to properly modify the convective and diffusive operators, as well as the Poisson equation.

3 PRELIMINARY RESULTS

Some simulations have already been performed, different time instants and a velocity plot for a simulation of a rotating cylinder at low Re number using the IBM is shown in Figure 2.

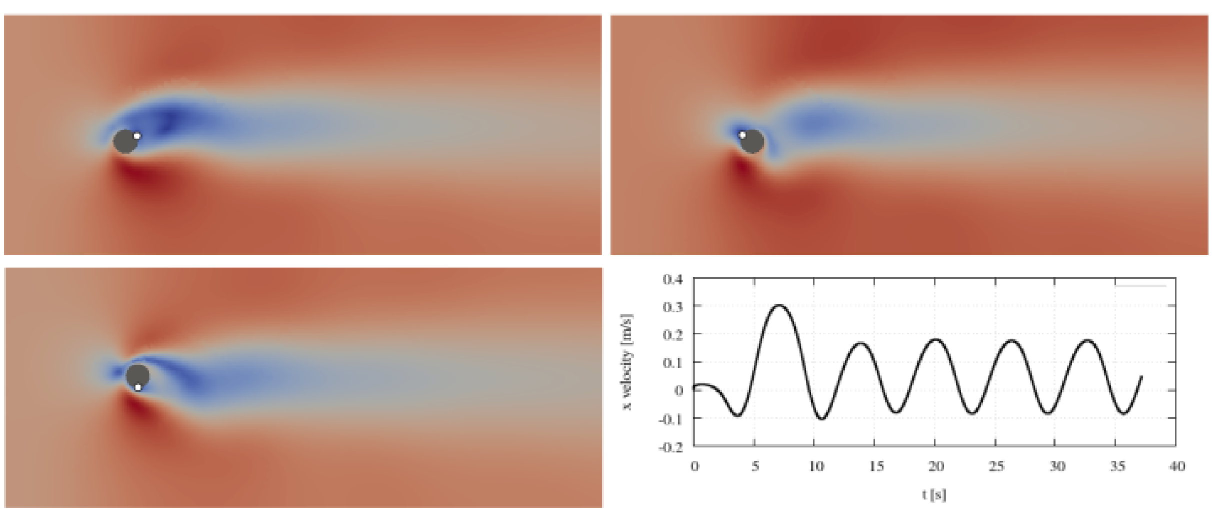

Figure 2: Different time instants an velocity plot at $x = 10$ m of the rotating cylinder case solved using the immersed boundary method

ACKNOWLEDGMENTS

This work has been financially supported by the Ministerio de Educación y Ciencia (MEC), Spain, within the project *"Algorimtos numéricos avanzados para la mejora de la eficiencia energética en los sectores eólico y solar-termico: Desarrollo/adaptación a nuevas arquitecturas computacionales"* (ref. ENE2017-88697-R).

References

[1] Charles S Peskin. Flow patterns around heart valves: a numerical method. *Journal of computational physics*, 10(2):252–271, 1972.

[2] J Mohd-Yusof. methods for complex geometries. *Annual Research Briefs, Center for Turbulence Research, Stanford University*, page 325, 1998.

[3] Elias Balaras. Modeling complex boundaries using an external force field on fixed cartesian grids in large-eddy simulations. *Computers & Fluids*, 33(3):375–404, 2004.

[4] Rajat Mittal and Gianluca Iaccarino. Immersed boundary methods. *Annual Review of Fluid Mechanics*, 37:239–261, 2005.

[5] EA Fadlun, R Verzicco, Paolo Orlandi, and J Mohd-Yusof. Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations. *Journal of computational physics*, 161(1):35–60, 2000.

[6] Man Mohan Rai. A conservative treatment of zonal boundaries for euler equation calculations. *Journal of Computational Physics*, 62(2):472–503, 1986.

[7] Eric L Blades and David L Marcum. A sliding interface method for unsteady unstructured flow simulations. *International Journal for Numerical Methods in Fluids*, 53(3):507–529, 2007.

[8] Joseph L Steger, F Carroll Dougherty, and John A Benek. A chimera grid scheme.[multiple overset body-conforming mesh system for finite difference adaptation to complex aircraft configurations]. *American Society of Mechanical Engineering*, 5:59–69, 1983.

[9] Donald Keith Clarke, HA Hassan, and MD Salas. Euler calculations for multielement airfoils using cartesian grids. *AIAA journal*, 24(3):353–358, 1986.

[10] J Muela, D Martínez, O Lehmkuhl, Carlos David Pérez-Segarra, and A Oliva. New parallel method for adjacent disconnected unstructured 3d meshes. *International Journal of Computational Fluid Dynamics*, 30(6):388–394, 2016.

[11] Ivan E Sutherland and Gary W Hodgman. Reentrant polygon clipping. *Communications of the ACM*, 17(1):32–42, 1974.