

Gas-particle Model for Objects Rigid Motion in Fluids

L. Remaki¹, I.G. de Beristain¹ and J.M. Blanco²

¹ BCAM (Basque Center for Applied Mathematics) Alameda Mazarredo, 14. 48009 Bilbao
lremaki@bcamath.org; <http://www.bcamath.org>

² University of the Basque Country; School of Engineering Alameda Urquijo, s/n. 48013 Bilbao
jesusmaria.blanco@ehu.es; <http://www.ingeniaritza-bilbao.ehu.es/p224-home/es>

Object rigid motion in fluid simulation is of a great importance in many industrial applications. Typical case is rotors in turbomachinery. Many techniques to simulate rotating effects have been developed in the literature. The multiple rotating frame [1] and sliding surface methods are the mostly used. The first one is valid for steady states only and the second one that captures unsteadiness requires a physical moving mesh increasing significantly the computational cost and implementation complexity.

In this paper, we propose a CAD free model where the geometry is simulated by a characteristic function as in VOF [2] (volume of fluid) and a gas-particle model to simulate the motion of the moving part. A convection equation is coupled with Navier-stokes equations in a suitable manner to be developed in details in the final paper. The advantage of such a method is the simulation of the steady state with a single mesh, very easy to implement and its cost effectiveness, moreover it could be extended beyond the flow simulation to design optimization for instance. Tests using the proposed method and the classical CAD based approach will be achieved demonstrating the efficiency of the method.

REFERENCES

- [1] J.Y. Luo, A.D Gosman and I.R Issa, Prediction of impeller-induced flows in mixing vessels using multiple frames of reference. *Inst. Chem. Eng. Sympo. Ser., Vol.136(1994), 549-556*
- [2] C.W. Hirt, B.D. Nichols, Volume of fluid (VOF) method for the dynamics of free boundaries, *Journal of Computational Physics* **39** (1): 201–225. 1981