Implementation of a moving immersed boundary method on a dynamically refining mesh with automatic load balancing.

Kyle G. Mooney[1] and Jacques Papper[2]

[1]Icon Technology & Process Consulting LLC. 4555 Lake Forest Drive, Suite 650 Cincinnati, OH USA, k.mooney@iconcfd.com
[2]Icon Technology & Process Consulting Ltd., 7 Rue Auber 31000 Toulouse France, j.papper@iconcfd.com

March 25, 2015

Abstract

A vast majority of finite volume CFD simulations are performed on boundary fitted meshes. While this approach enables high fidelity representations of geometries of interest, many complications arise when moving or deforming boundaries are required. Some approaches to handling deforming domain simulations may be local or global remeshing, node smoothing, or non-conformal sliding interfaces. Another approach is the immersed boundary (IB) method which avoids the need for boundary fitted meshes entirely and merely requires a geometry surface to be 'immersed' within the surrounding base mesh.

In this work the immersed boundary method implementation by Jasak et al [1] is extended to allow free deformation and motion of the boundary within the base mesh. Here the tri-surface geometry can be moved and deformed freely while also imposing appropriate surface velocity boundary conditions on the flow field. When combined with automatic mesh refinement and de-refinement, greater local mesh resolution can be committed to the IB interface zone or areas of strong gradients with no a priori knowledge of their locations. Furthermore, a dynamic run-time load-balancer [2] is used to avoid processor bottlenecks as mesh elements are added and removed as the IB surface displaces through the domain.

1

[1]Hrvoje Jasak, Damir Rigler, and Zeljko Tukovic. Finite volume immersed boundary method for turbulent flow simulations. 9th OpenFOAM Workshop, Zagreb Croatia, June 2014.

[2]Kyle Mooney, Jacques Papper, and Tyler Voskuilen. Performance evaluation of existing and new VOF simulations techniques: solving, interface treatment, and dynamics meshes. 9th OpenFOAM Workshop, Zagreb Croatia, June 2014.