11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

R-ADAPTIVITY FOR FLUID-STRUCTURE INTERACTION, DEFORMING DOMAINS AND INTERNAL BOUNDARIES

Müller K.[*1], **Spühler J.**[2], **Degirmenci N.C.**[3,4], **Jansson J.**[5] **and Hoffman J.** [6]

[1] Department of High Performance Computing and Visualization, KTH Royal Institute of Technology, SE-10044 Stockholm, Sweden, kasparm@kth.se, www.csc.kth.se/∼kasparm
[2] Department of High Performance Computing and Visualization, KTH Royal Institute of Technology, SE-10044 Stockholm, Sweden, spuhler@kth.se, www.csc.kth.se/∼spuhler
[3] Department of High Performance Computing and Visualization, KTH Royal Institute of Technology, SE-10044 Stockholm, Sweden, ncde@kth.se, www.csc.kth.se/∼ncde
[4] Department of High Performance Computing and Visualization, KTH Royal Institute of Technology, SE-10044 Stockholm, Sweden, jjan@kth.se, www.csc.kth.se/∼jjan
[5] Basque Center for Applied Mathematics BCAM, Spain, jjansson@bcamath.org, www.bcamath.org/es/people/jjansson
[6] Department of High Performance Computing and Visualization, KTH Royal Institute of Technology, SE-10044 Stockholm, Sweden, jhoffman@kth.se, www.csc.kth.se/∼jhoffman

Key words: *R-Adaptivity, FSI, Deforming Domain, Internal Boundaries, Moving Grid.*

In fluid-structure interaction, flow in deforming domains and multiphase flow with internal boundaries, the computational domain is changing with time. When using finite element methods where the computational mesh is matching the domain, the mesh is deformed. This repositioning of the vertices on the mesh boundary affects the mesh quality, the accuracy and the efficiency of the computation. We present a parallel computational framework based on an r-adaptive method which handles these challenges and we illustrate the method in computational tests.

The presented work is based on stabilized finite element methods, implemented as the software components DOLFIN-HPC and Unicorn, in the FEniCS project[1].The fluid is modeled by the incompressible Navier-Stokes equations and the fluid-structure interaction model is monolithic where the fluid and the structure are modelled as one single continuum discretized on the same computational mesh [2]. The deforming structure is described in Lagrangian coordinates and the fluid in an Arbitrary Lagrangian-Eulerian (ALE) formulation. In the case of internal boundaries we align the mesh points along the interface. Mesh smoothing is applied to improve the mesh quality and to concentrate the mesh resolution to optimise accuracy, based on a posteriori error estimation.

Besides presenting the framework, test cases are shown illustrating the key challenges: Simulation of the blood flow in the left ventricle of the heart by prescribing the movement

of the wall in time and space, simulation of multiphase stratified flow modelling the ocean-athmosphere system, and fluid-structure interaction of air and vocal folds in a model of human phonation.

REFERENCES

[1] J. Hoffman et.al.. Unicorn: parallel adaptive finite element simulation of turbulent flow and fluid-structure interaction for deforming domains and complex geometry, *Computer and Fluids*, Vol.**80**, pp.310-319, 2013.

[2] J. Hoffman, J. Jansson and M. Stöckli, Unified continuum modeling of fluid-structure interaction, Mathematical Models and Methods in Applied Sciences, Vol.21(3), pp.491-513, 2011.