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

UNIFIED CONTINUUM FINITE ELEMENT SIMULATION OF TURBULENT FLOW FLUID-STRUCTURE INTERACTION

Johan Hoffman*[1], Johan Jansson[2,3], Niyazi C. Degirmenci[4], Jeannette Hiromi Spühler[5] and Bärbel Janssen[6]

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

Key words: *Unified Continuum Fluid-Structure Interaction, Finite Element Method, Turbulent Flow, Contact Model.*

Simulation of fluid-structure interaction poses several challenges with respect to stability and accuracy, in combination with turbulent flow the problem is even more challenging. In this paper we present our work based on a unified continuum finite element method, a monolithic method where the combined fluid-structure continuum is discretised by the same finite element method on one single mesh. In the original version of the method [1] we let the mesh follow the structure deformation and we use mesh smoothing to maximise mesh quality in the fluid part of the domain. This corresponds to a Lagrangian description of the structure and an Arbitrary Lagrangian-Eulerian (ALE) method for the fluid. This method is implemented in the FEniCS project as the software components DOLFIN-HPC and Unicorn, and is shown to scale near optimally up to tens of thousands of cores on massively parallel hardware architectures [2].

The unified continuum formulation effectively transforms the fluid-structure interaction problem into a multiphase flow problem, which is suitable for analysis. We derive a posteriori error estimates for the method, which we employ to construct adaptive algorithms that produce optimized meshes with respect to the error in some chosen goal functional. We present different strategies for mesh smoothing and adaptive mesh optimisation, and for large structure deformation we also investigate an alternative version of the method where the mesh is released from tracking the fluid-structure interface, which is then allowed to cut the mesh cells.

Furthermore, we describe a contact model suitable for the unified continuum framework. Turbulent fluid-structure interaction including contact is central in several of our applied modelling projects, e.g. simulation of the blood flow in the human heart including heart valves, and simulation of the human voice in the FP7 project Eunison [3].

REFERENCES

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

[2] 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.

[3] http://eunison.eu, 2013.