Robustness and challenges in 3D FSI simulations with Chimera-based flow solvers: Application to flexible structures in environmental flows.

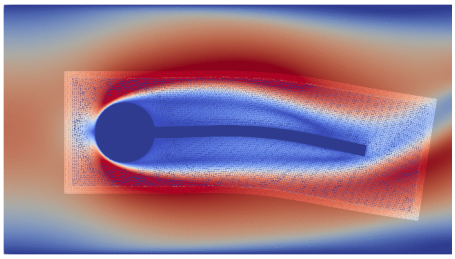
A. Ghantasala*, N. K. Narayanan†, R. E. Meethal†, R. Wüchner† and K.-U. Bletzinger†

* † Lehrstuhl für Statik,
Technische Universität Mnchen,
Arcisstr. 21, 80333 München, Germany
E-mail : aditya.ghantasala@tum.de* , web page : https://www.st.bgu.tum.de

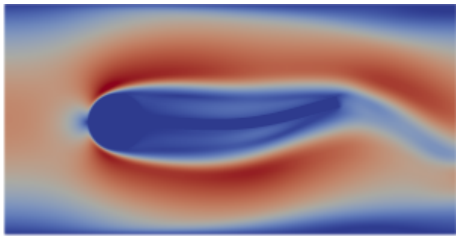
ABSTRACT

Problems involving fluid simulations have a wide spread applications. Some of these involve extreme modifications of fluid domain, for example configuration optimization and coupled problems like fluid-structure interaction simulations involving highly flexible structures. Many approaches like re-meshing, mesh deformation, embedded and chimera techniques have been used in the past to address the issue of changing fluid domain during these simulations. Although, some of these approaches are suitable for such problems, they have their limitations and targeted application range. Among these, Chimera technique offers a promising compromise.

This contribution discusses the implementation details of Chimera technique for fluid solver using finite element formulation and challenges involved in using this fluid solver in fluid-structure interaction simulations. The implementation of this is done in Finite-Element based open-source framework KRATOS[2]. In the Chimera technique, the fluid domain is divided into a number of overlapping sub-domains called patches. These patches are coupled to a background mesh during the solution process to obtain the solution. This coupling of patches and the background is done monolithically for the solution on entire fluid domain. A coupled simulation involving a chimera formulated fluid solver, especially when changes in fluid domain are involved, poses different challenges in terms of including the patches in the background domain. One such application involving fluid domain changes is fluid-structure interaction simulation. Different challenges involved in fluid-structure simulation, their illustration and the way of tackling them is outlined in detail in this contribution. Figure 1b shows the deformed patch mesh for an fluid-structure simulation and corresponding magnitude of velocity distribution which is continuous across the patch and background domains.



(a) Illustration of Background and patch meshes



(b) Velocity magnitude

REFERENCES

[1] H. Hadžić. Development and Application of a Finite Volume Method for the Computation of Flows Around Moving Bodies on Unstructured, Overlapping Grids. *PhD Thesis, Technische Universtt Hamburg, 2005.*

[2] P. Dadvand, R. Riccardo and E. Oñate. An object-oriented environment for developing finite element codes for multi-disciplinary applications. *Archives of computational methods in engineering* (2010) **17(3)**:253297.