6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11-15 June 2018, Glasgow, UK

A non-intrusive approach to the proper generalised decomposition for Oseen flow in OpenFOAM

Vasileios Tsiolakis[‡,*,†], Matteo Giacomini[*], Ruben Sevilla[†], Carsten Othmer[‡] and Antonio Huerta[*]

[‡]Volkswagen AG
Brieffach 011/1777, D-38436, Wolfsburg, Germany
e-mail: vasileios.tsiolakis@volkswagen.de
[*]Laboratori de Càlcul Numèric (LaCàN)
Universitat Politècnica de Catalunya
Jordi Girona, 1-3, Campus Nord Edifici C2, 08034 Barcelona, Spain
[†]Zienkiewicz Centre for Computational Engineering
College of Engineering, Swansea University
Swansea SA1 8EN, Wales, UK

Computational fluid dynamics (CFD) is an integral part of the automobile industry. However, the aerodynamic optimisation process is known to introduce a major bottleneck in the design cycle, due to the necessity to test a large number of configurations. Using reduced order modelling (ROM) techniques, a methodology to introduce real-time aerodynamic design would lift this bottleneck by compacting, after an initial computational investment, all steps of the optimisation cycle into one.

OpenFOAM [1] has become an industry standard for CFD. As such, in this work, a methodology to implement the proper generalised decomposition (PGD) [2] for parametric Oseen flow problems in OpenFOAM is proposed, by considering parameters of the problem (e.g. the physical properties of the fluid) as extra dimensions of the generalised solution [3]. PGD has gained popularity due to its ability to compute a reduced basis with no a-priori knowledge of the problem. However, compared to other ROM techniques, its implementation usually requires modification of the core routines of an existing solver. This work, as part of a research aiming towards a daily industrial application, presents a non-intrusive approach for the implementation of the PGD for the numerical solution of Oseen flow problems using OpenFOAM. Particular emphasis will be given on the formulation proposed to enable the non-intrusive implementation.

Numerical examples will be used to demonstrate the potential of the proposed methodology for solving Oseen flow problems where the viscosity is considered an extra coordinate of the generalised solution.

This project is part of the Marie Skłodowska-Curie ITN-ETN AdMoRe funded by the European Union Horizon 2020 research and innovation program with grant number 675919.

REFERENCES

[1] *OpenFOAM The Open Source CFD Toolbox* Programmer's Guide, OpenCFD Limited, 2016.

[2] Ammar A., Mokdad B., Chinesta F., Keunings R. *A new family of solvers for some classes of multidimensional partial differential equations encountered in kinetic theory modelling of complex fluids*, J. Non-Newtonian Fluid Mech. 139, 2006, p. 153–176.

[3] Ammar, A., Huerta, A., Chinesta, F., Cueto, E., and Leygue, A., *Parametric solutions involving geometry: a step towards efficient shape optimization.* Computer Methods in Applied Mechanics and Engineering, 2014, p. 178–193.