Parameterised Stokes flow using the proper generalised decomposition in OpenFOAM

Vasileios Tsiolakis[‡,*,†], Ruben Sevilla[†], Matteo Giacomini[*], 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

ABSTRACT

Computational fluid dynamics has become an integral part of the automobile industry. In Volkswagen, OpenFOAM [1] is used for a variety of fluid dynamic disciplines, including external aerodynamics, aeroacoustics, cabin ventilation and engine flows. However, the necessity to test a large amount of configurations during the design stage introduces a major challenge.

In this work, a methodology to implement the proper generalised decomposition (PGD) [2] for parametric Stokes flow 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 build a reduced basis with no a priori knowledge of the solution. However, compared to other reduced order modelling techniques, the implementation of the PGD usually requires modification of the core routines of an existing solver. This work presents a non-intrusive implementation of the PGD for the numerical solution of Stokes flow problems using OpenFOAM. More precisely, the implementation of the PGD within the semi-implicit method for pressure linked equations (SIMPLE) [4] finite volume solver will be presented, with particular emphasis on the formulation proposed to enable a non-intrusive implementation.

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

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.

[4] Patankar, S.V. *Numerical heat transfer and fluid flow:* Hemisphere Publishing Corporation, 1981, Ch. 6.7.