Investigation on the capabilities of stabilized Galerkin approaches for the DNS and LES simulations of incompressible flow problems

A. Collard-Daigneault†, Laura Prieto Saavedra†∗, B. Blais†∗

† Research Unit for Industrial Flows Processes (URPEI)
Department of Chemical Engineering
École Polytechnique de Montréal, PO Box 6079, Stn Centre-Ville
QC, Canada, H3C 3A7 e-mail: bruno.blais@polymtl.ca

Key Words: CFD, high-order methods, continuous Galerkin, Galerkin least-square, LES

ABSTRACT

The finite element method (FEM) is seldom used in chemical engineering communities, where commercial (e.g. Ansys Fluent) or open-source (e.g. OpenFOAM) software based on the finite volume approaches are commonplace. The ease of extending FEM to high-order, methods which the order of convergence in the $L^2$ norm is greater than two [1], and its strong mathematical foundations make it an attractive candidate for Large-Eddy Simulation (LES) and/or Direct Numerical simulation (DNS). However, there is a significant absence of available open source software that leverages the capacity of high-order FEM for the incompressible flows commonly encountered in other engineering applications. In this work, we introduce a new open-source high-order CFD software for incompressible flow problems: Lethe [2]. It is built upon the well-established deal.II library [3], and leverages stabilized Galerkin Least-Squares formulation to solve turbulent problem through DNS or implicit LES. Through its deal.II heritage, Lethe supports dynamic mesh adaptation and is based on a solid C++ architecture. First, we present the underlying GLS formulation used within Lethe. We discuss the non-linear solution strategy used to formulate a fully implicit time-stepping scheme using BDF or SDIRK time integrators. Additionally, we present the strategies used to precondition and solve the linear system of equation using a monolithic approach or a Schur complement. We discuss both the cases of steady-state and transient problems, as well as the strategies used to reduce the computational time and increase robustness of the solver.

The resulting solver is used to study canonical benchmark problem turbulent flows. First, we investigate the turbulent cascade in Taylor-Green vortices at Re=1600 with meshes ranging from 10M to 100M degrees of freedom. It is shown, that the GLS scheme is capable reproducing the energy dissipation profile when high-order schemes are used. Then, we investigate the flow over periodic hills at Re=5600 where a very good agreement is obtained for both the averaged velocity profiles, but also the Reynolds stresses. We conclude by discussing the possible extensions to this work, notably in the field of solid-fluid flows.

REFERENCES

