Comparison of linear solvers for the incompressible Navier-Stokes equations in PFEM simulations

Martin Lacroix*, Février Simon, Eduardo Fernández-Sánchez,
Romain Boman, Jean-Philippe Ponthot*

LTAS-MN2L, Aerospace and Mechanical Engineering Department, University of Liege, Belgium.

* E-mail: martin.lacroix@uliege.be, JP.Ponthot@uliege.be

1 Introduction

The Finite Element Method (FEM) is a powerful numerical tool that has the advantage of being extremely versatile. In particular, the so-called Particle Finite Element Method (PFEM) aims at natively simulating free-surface flows with large deformations by the use of a remeshing procedure to capture the evolution of the domain (Cerquaglia, 2019). While performing large-scale and 3D simulations, the computational cost of numerical tools becomes an essential parameter to take into account (see Figures 1 and 2 for an example). In the PFEM, this computation time is mainly dedicated to the remeshing operation, but also to the linear solver that is called at each iteration of each time step for solving the finite element system. The objective of this master thesis is to compare the performances of different linear solvers available on the web, in the context of the PFEM.

2 Objective

The LTAS-MN2L group is developing a code based on the PFEM (Février, 2020) to solve the incompressible Navier-Stokes equations using a Lagrangian framework. The student is first asked to familiarize with the in-house software PFEM3D and understand the basics of its implementation. The second step consists of a literature review of linear algebra libraries which could turn out to be good candidates for the PFEM. For instance, PaStiX and PARDISO are likely to provide a significant advantage over the currently supported Eigen direct LU decomposition. The last step of the work is a comparison of the solvers on different test cases once incorporated into PFEM3D.

Background knowledge advised for the project : Basics in C++ programming, Introduction to numerical analysis (MATH0006-3), Finite Element Method (MECA0036-2).

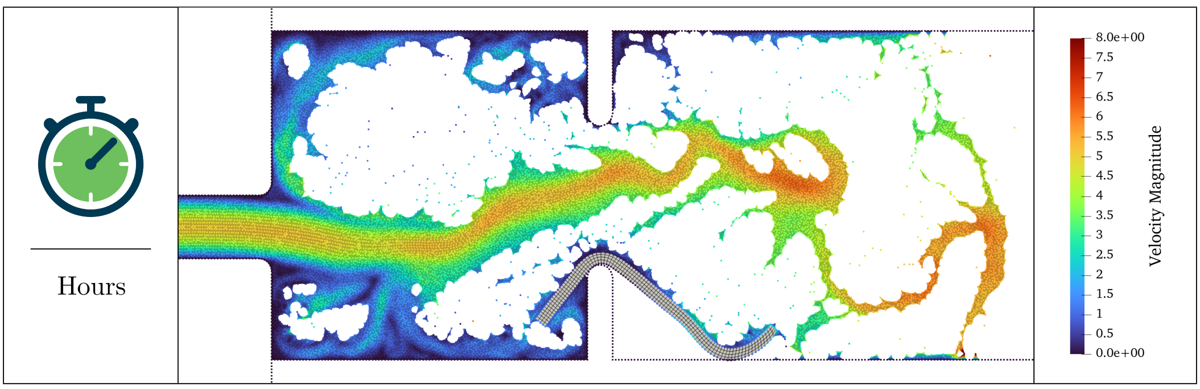

Figure 1: Fluid-structure simulation of a water jet on a solid wall. The PFEM is used for simulating the flow while the FEM is used for the induced deformation of the solid structure (in grey). More than 90% of the total computation time is dedicated to the linear solver.

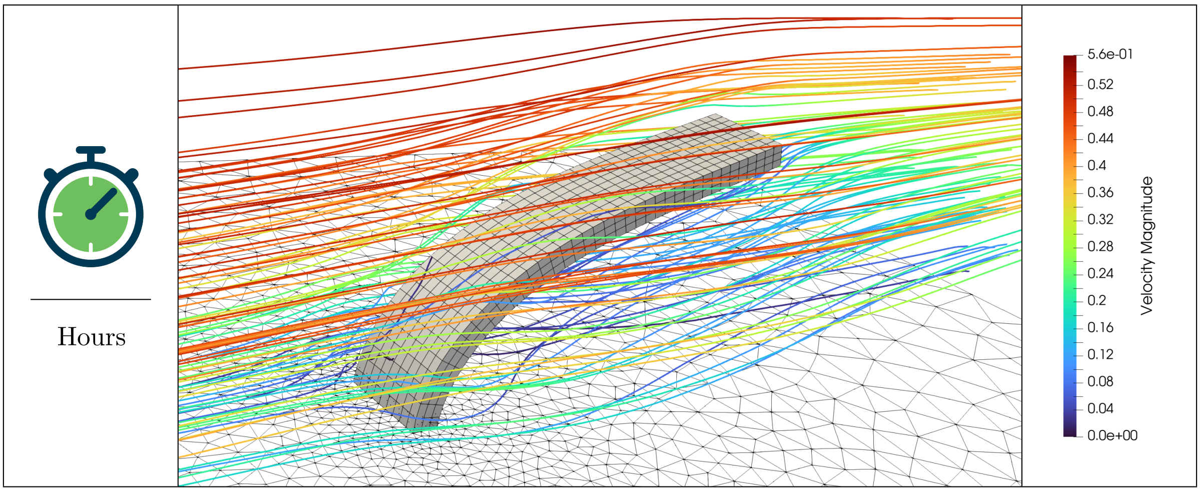

Figure 2: PFEM/FEM fluid-structure simulation of a 3D elastic flap in a cross-flow. More than 30% of the total computation time is dedicated to the linear solver (Lacroix et al., 2024).

References

Cerquaglia, Marco-Lucio. 2019. *Development of a fully-partitioned PFEM-FEM approach for fluid-structure interaction problems characterized by free surfaces, large solid deformations, and strong added-mass effects*. Vol. PhD thesis, University of Liege. `https://hdl.handle.net/2268/233166`.

Février, Simon. 2020. "Development of a 3D Compressible Flow Solver for PFEM Fluid Simulation." Master's thesis, University of Liege. `http://hdl.handle.net/2268.2/9010`.

Lacroix, Martin, Simon Février, Eduardo Fernández, Luc Papeleux, Romain Boman, and Jean-Philippe Ponthot. 2024. "A Comparative Study of Interpolation Algorithms on Non-Matching Meshes for Partitioned Fluid-Structure Interactions." *Computers & Mathematics with Applications* 155: 51–65.