

A Novel Black-Box Massively Parallel Partitioned Approach to Fluid-Structure Interaction Problems

*Sam Hewitt¹, Alistair Revell¹ and Lee Margetts¹

¹ School of Mechanical, Aerospace and Civil Engineering, University of Manchester, Manchester, UK, M13 9PL

* Sam.Hewitt@postgrad.manchester.ac.uk

ABSTRACT

This paper describes new software based on a partitioned, black box approach for solving fluid structure interaction (FSI) problems. It couples the open source finite volume solver OpenFOAM, to the open source finite element solver ParaFEM [1]. This coupling is done using a conventional serially staggered scheme, that is first order accurate in time. The software has been tested using benchmark FSI problems, with long term goals to investigate the broad range of large scale FSI cases relevant to the wind energy industry. The aim is to demonstrate the potential benefits of such a tool, by delivering a unique capability to investigate novel concepts, to levels of physical realism not yet achieved.

Key Words: Fluid-Structure Interaction; High Performance Computing; Computational Fluid Dynamics; Finite Element Analysis; Partitioned Coupling

1. Introduction

Over the past thirty years the average size of offshore wind turbines has increased, with rotor diameters growing from 10m to 160m. It therefore becomes increasingly important to simulate the complex dynamics involved with unsteady and turbulent flow through rotating blades, and the large structural deformations of blades, in order to optimise their design.

The motivation is to produce software capable of solving coupled FSI problems to investigate the dynamics of wind turbines; both individually and ultimately in arrays relevant to farm configurations. The project will focus on coupling OpenFOAM to ParaFEM [1] in such a way as to minimise communication bottleneck and to maximise performance of the software.

The developed software uses a partitioned approach, with OpenFOAM and ParaFEM acting as blackbox fluid and structural solvers respectively. Using this approach makes use of already existing models and solution algorithms that have been previously validated, and the user has the versatility to select the type of CFD and FEM simulations independently from the coupling strategy. The coupling is achieved using a conventional serially staggered (CSS) scheme to solve the governing equations. The software will be highly parallel allowing large complex problems to be run with sufficient accuracy in a reasonable time on High Performance Computers.

FSI modelling offers the potential to accurately predict the deforming blade shape across a range of load conditions and thus improve the prediction of blade efficiency and noise production. FSI with ParaFEM not only provides fast stress analysis through parallel processing, but also the capability to incorporate more heavy weight structural integrity assessment through stochastic simulations [2], thermo-mechanical analysis [3] and multi-scale modelling of fracture [4].

2. Method

This section will give a brief description of the governing equations associated with the fluid and solid, and the interface conditions. Finally the system architecture and coupling algorithm between OpenFOAM and ParaFEM is described.

2.1. Fluid/OpenFOAM

The flow variables are described by the Navier-Stokes equations (1,2). The subscripts s and f represent the solid and fluid respectively.

$$\frac{\partial}{\partial t} + \nabla \cdot (\rho U_f) = 0 \quad (1)$$

$$\rho \frac{\partial U_f}{\partial t} + \rho U_f \cdot \nabla U_f - \nabla \cdot \sigma_f = f \quad (2)$$

Where U represents the velocity vector $[u,v,w]$, ρ the density, t the time and σ the stress tensor. The governing equations are discretised using the finite volume method (FVM).

2.2. Solid/ParaFEM

The solid is solved using an element by element variant of the finite element method (FEM), through the linear equation:

$$\sum_1^{nels} \{f_e\} = [K_e] \{x_e\} \quad (3)$$

Where f_e and x_e are the elements nodal forces and displacements respectively, and K_e represents the element stiffness matrix which is computed from the material properties and shape functions. $nels$ is the total number of elements in the mesh.

2.3. Interface

The interface between the fluid and solid is described by kinematic (4) and dynamic (5) equilibrium. Where n represents the unit normal.

$$U_f = \frac{dU_s}{dt} \quad (4)$$

$$\sigma_f \cdot n_f = -\sigma_s \cdot n_s \quad (5)$$

The kinematic condition ensures the spatial variables, velocity and displacement, at the interface between the fluid and structure are in equilibrium. The dynamic condition states that the force/stress exerted by the fluid and solid at the interface are equal and opposite.

2.4. System Architecture

The software considers OpenFOAM and ParaFEM as two black box solvers. This partitioned approach means the numerical schemes and discretisation methods, for the structure and flow, can be chosen independently. The modular approach makes the software robust and flexible for different types of problems. However if a dynamic structural model is used, care must be taken with the choice of time step so as to avoid spurious oscillations in the acceleration and traction at the interface [5].

The algorithm used in solving the FSI problem is shown in Figure 1a. The algorithm currently used is the most basic coupling technique, the conventional serially staggered scheme, developed in [6]. The coupling between the two programs is in its most simplistic form, a transfer of data. The difficulty comes from two sources. Moving data from an object orientated language (C++) to a procedural language (Fortran) and from a FVM to FEM. The quasi standard coupling interface MpCCI has been used for a similar problem in [7]. OpenFOAM contains good interpolation libraries, making the use of external packages unnecessary. OpenFOAM acts as the master program with ParaFEM being called as a subroutine at runtime. Figure 1b shows the data coupling, interpolation and data transfer between the two programs. This is described more comprehensively below.

1. Geometry : The initial geometry data (nodal coordinates and connectivity matrix) of the solid mesh are passed to ParaFEM, before the time loop begins.

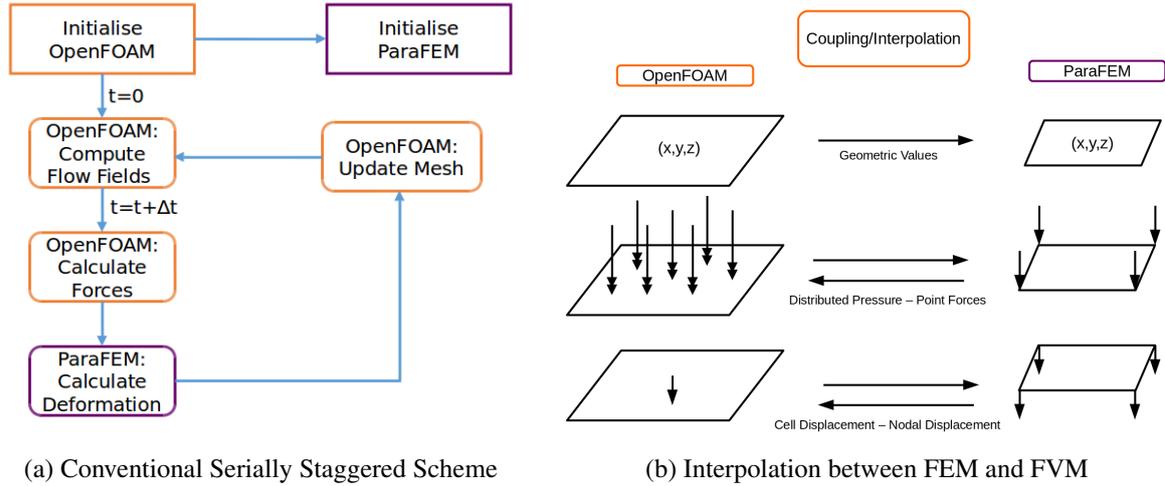


Figure 1: High level diagrams of the 1a Solution Algorithm & 1b Data Transfer

2. Dynamics : The pressure on each face, at the interface, is interpolated to a point force and a direction vector. This force is then distributed over the nodes of the FE element according to [1].
3. Kinematics : The nodal displacements are passed back to OpenFOAM that interpolates the nodal values into patch/face displacements.

3. Test Problems and Results

The software has been checked using the FSI benchmark problem proposed by Stefan Turek and Jaroslav Hron in [8]. This test involves the 2D laminar flow of an incompressible, Newtonian fluid, through a channel and around an elastic object (Figure 2). The values for the geometric parameters are shown in Table 1. The test involves fixing the elastic solid until the flow is fully developed before allowing the solid to move under the force of the fluid. The displacement at position A, is compared to the benchmark results over one full cycle.

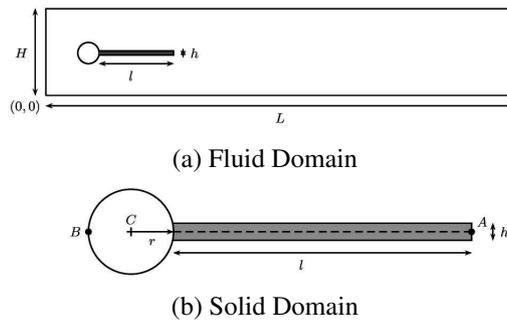


Figure 2: Schematic of the Hron and Turek benchmark case, taken from [8]

During the conference, the implementation of the software will be described before presenting the results and performance of the code for the FSI benchmark problem described earlier. Figure 3 shows an example of the problem run using the current FSI solver in OpenFOAM. This uses a finite volume solver for both the fluid and the solid.

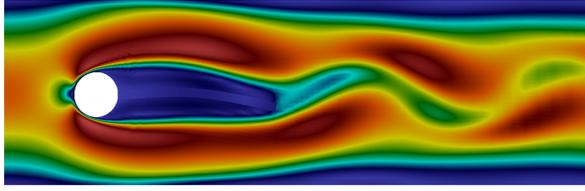


Figure 3: An example of Hron Turek Test case

Parameter	Value(m)
L	2.5
H	0.41
C	(0.2,0.2)
r	0.05
l	0.35
h	0.02
A	(0.6,0.2)
B	(0.2,0.2)

Table 1: Geometric Parameters from [8]

Scaling tests for both OpenFOAM and ParaFEM have been performed in the literature. Smith et al [1] has shown that ParaFEM scales well on up to 32,000 cores. while the HPC Advisory Council [9] have shown that OpenFOAM can scale well to 1024 cores.

4. Conclusions

This paper has described new software that couples the finite volume CFD solver, OpenFOAM with the finite element structural solver, ParaFEM. OpenFOAM has acted as the master, calling ParaFEM as a subroutine each time step. The modular set up of the software makes it flexible and robust. The code will be able to tackle larger and more complex problems, with good fidelity in the results, specifically to investigate physical phenomena that occur in wind turbines and ultimately arrays/farms.

Acknowledgements

The first author is supported by an Alstom/EPSCRC PhD Studentship. The authors would like to express their gratitude to Prof. Hrvoje Jasak and Dr Stefano Rolfo for their help at the 3rd UK and Ireland OpenFOAM User Meeting at the Hartree Center.

References

- [1] I. M. Smith, D. V. Griffiths, and L. Margetts. *Programming the Finite Element Method*. John Wiley & Sons, Ltd, 5th edition, 2014.
- [2] Jose David Arregui-Mena, Lee Margetts, D. V. Griffiths, Louise Lever, Graham Hall, and Paul M. Mummery. Spatial variability in the coefficient of thermal expansion induces pre-service stresses in computer models of virgin Gilsocarbon bricks. *Journal of Nuclear Materials*, 465:793–804, 2015.
- [3] Llion Evans, Lee Margetts, Valentina Casalegno, Louise Lever, Joe Bushell, Tristan Lowe, Andrew Wallwork, Philippe Young, Andre Lindemann, Marc Schmidt, and Paul Mummery. Transient thermal finite element analysis of CFC-Cu ITER monoblock using X-ray tomography data. *Fusion Engineering and Design*, 100:100–111, 2015.
- [4] A Shterenlikht and L Margetts. Three-dimensional cellular automata modelling of cleavage propagation across crystal boundaries in polycrystalline microstructures. *Proceedings of the Royal Society of London A: Mathematical, Physical and Engineering Sciences*, 471(2177), 2015.
- [5] Joris Degroote. Partitioned Simulation of Fluid- Structure Interaction. *State of the Art Reviews*, 20(3):185–238, 2013.
- [6] M. Lesoinne and C. Farhat. Higher-Order Subiteration-Free Staggered Algorithm for Nonlinear Transient Aeroelastic Problems. *AIAA Journal*, 36(9):1754–1757, sep 1998.
- [7] Michael Schäfer, Marcus Heck, and Saim Yigit. An implicit partitioned method for the numerical simulation of fluid-structure interaction. *Fluid-structure interaction*, pages 171–194, 2006.
- [8] Stefan Turek and Jaroslav Hron. Proposal for numerical benchmarking of fluid- structure interaction between an elastic object and laminar incompressible flow. *Lecture Notes in Computational Science and Engineering*, 53:371–385, 2006.
- [9] Scale Working Group. Computational Fluids Dynamics (CFD) Simulations at Scale OpenFOAM open source applications. Technical report, 2010. Available: http://www.hpcadvisorycouncil.com/best_practices.php.