Ninth International Conference on Computational Fluid Dynamics (ICCFD9), Istanbul, Turkey, July 11-15, 2016 **ICCFD9-xxxx**

A POD model for boundary conditions in CFD

Michel Bergmann[1,2], Andrea Ferrero[1,2], Angelo Iollo[1,2]

[1] Inria Bordeaux Sud Ouest, Team MEMPHIS, 200 avenue de la vielle tour, 33405 Talence, France
[2] Université de Bordeaux, IMB, 351 cours de la libération, 33405 Talence, France

Corresponding author: michel.bergmann@inria.fr

Abstract: A numerical approach to accelerate the numerical simulation of fluid flows is proposed. Solving the full detailed model, *i.e.* the Navier-Stokes Equations (NSE), can be very costly for wake flows around bodies where a very large computation domain is needed. Indeed, the far field can have a singnificant influence on the flow near the body and thus imposing the appropriate boundary conditions is necessary. We propose a Galerkin free Proper Orthogonal Decomposition (POD) reduced order model (ROM) to overcome this difficulty by coupling the resolution of the NSE in a small computational domain with a POD ROM.

Keywords: Proper Orthogonal Decomposition, Reduced Order Model, Computational Fluid Dynamics.

1 Introduction

One of the main problem for computational fluid dynamics is the imposition of appropriate boundary conditions that allows coherent structures to go out properly from the computation domain. To this end, absorbing and non-reflecting boundary conditions have been developed [1]. The use of these kind of boundary conditions requires however a quite large computational domain. One way to overcome this difficulty is to impose an actual flow as boundary conditions to reduce the computational domain. We propose to impose boundary conditions projecting the solution of the full detailed model inside a small computational domain on a robust POD basis computed from actual flows on a larger domain. The computation of a robust POD basis [2] still requires several computations of the detailed model on a large domain but it is computed offline once for all, as long as the POD basis represents correctly the input parameter space. This approach is briefly described in what follows.

2 Problem Statement

The method is developed on the classical two dimensional circular cylinder wake flow. We will apply this strategy to compute the three dimensional wake flow around a visco-elastic beam airfoil with two-way fluid structure interactions. The NSE are discretized in space on a cartesian mesh. The body geometry is implicitly represented using a level set function, and the body is taken into account in the NSE thanks to a second order penalization method. All the details can be found in [3].

We start to solve the NSE in a large computational domain Ω_{POD} on a coarse grid for several input parameters obtained by the sampling proposed in [2]. We use artificial boundary conditions on Σ as in [1]. We then build a $N-$dimensional POD basis for velocity $\{\mathbf{\Phi_i}\}_{i=1}^{N}$ and pressure $\{\Psi_i\}_{i=1}^{N}$ that are robust to represent the subspace spanned by the input parameter space. The input parameter space is the Reynolds number (2D cylinder wake) and the elastic beam parameters (3D airfoil wake). This step can be done offline.

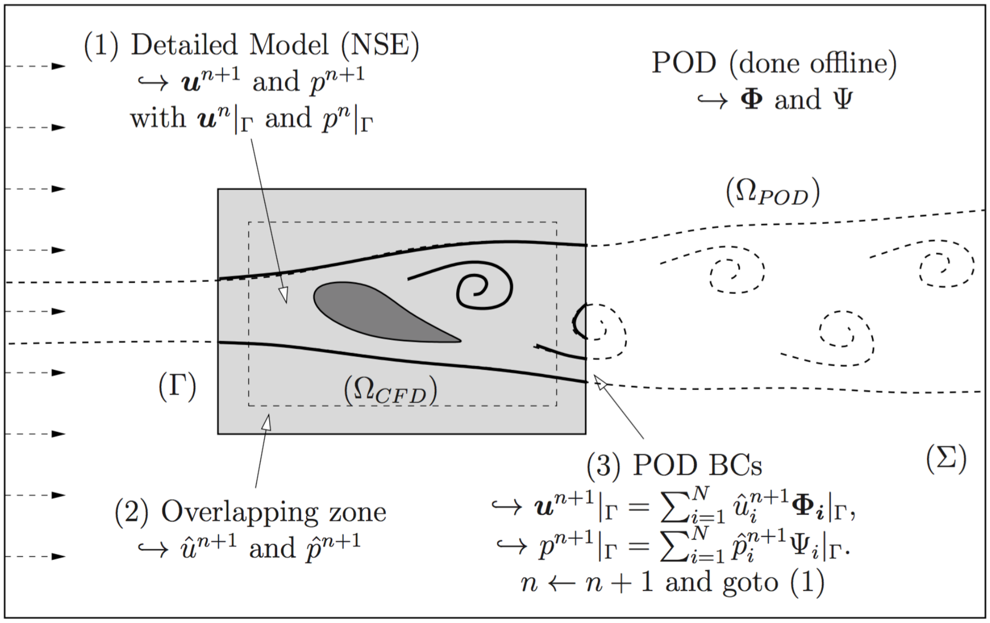

Figure 1: Sketch of the computational domain and methods.

We then solve the NSE on a finer mesh in a small domain Ω_{CFD} (numerical zoom) around the body. The boundary conditions for $\mathbf{x} \in \Gamma$ are then determined using the following POD representation

$$\mathbf{u}(t, \mathbf{x}) = \sum_{i=1}^{N} \hat{u}_i(t) \mathbf{\Phi_i}(\mathbf{x}) \quad \text{and} \quad p(t, \mathbf{x}) = \sum_{i=1}^{N} \hat{p}_i(t) \Psi_i(\mathbf{x}).$$

The sets of coefficients $\hat{u}(t)$ and $\hat{p}(t)$ are determined projecting the NSE solution on $span\{\mathbf{\Phi}\}$ and $span\{\Psi\}$ on a predefined overlapping region. Since we are using a fractional step method to solve the NSE in time, we may require $\hat{u}_i(t + \Delta t)$ and $\hat{p}_i(t + \Delta t)$ that can be obtained extrapolating previous values.

Figure 1 presents a sketch of the computational domain as well as the different methods used.

3 Conclusion and Future Work

The method proposed allows to significantly reduced the computational costs to simulate a wake flow. The next step is to improve the simulation using an overset approach. The detailed model will be solved on a body fitted Chimera mesh allowing precise computations of the boundary layers while the POD computation will still be done on a background cartesian mesh. This is part of the european project H2020 AEROGUST.

References

[1] G. Jin and M. Braza. A Nonreflecting Outlet Boundary Condition For Incompressible Navier-Stokes Calculations. *Journal of Computational Physics*, 107:239–253, 1993.

[2] M. Bergmann, T. Colin, A. Iollo, D. Lombardi, O. Saut, and H. Telib. Reduced Order Models at Work in Aeronautics and Medicine. *Modeling, Simulation and Applications*, 9, 2013.

[3] M. Bergmann, J. Hovnanian, and A. Iollo. An accurate cartesian method for incompressible flows with moving boundaries. *Communications In Computational Physics*, 15:1266–1290, 2014.