Journal of Computational Physics 228 (2009) 2239–2255

Contents lists available at ScienceDirect

Journal of Computational Physics

journal homepage: www.elsevier.com/locate/jcp

Numerical model reduction of 2D steady incompressible laminar flows: Application on the flow over a backward-facing step

Y. Rouizi [a], Y. Favennec [b,*,1], J. Ventura [a], D. Petit [a]

[a] LET UMR CNRS 6608, ENSMA – Université de Potiers - ESIP, Téléport 2, 1 Avenue Clément Ader, BP 40109, 86961 Futuroscope Cédex, France
[b] LET UMR CNRS 6608, ENSMA – Université de Potiers - ESIP, 40 Avenue du Recteur Pineau, Bâtiment de Mécanique, 86022 POITIERS Cédex, France

ARTICLE INFO

Article history:
Received 4 April 2008
Received in revised form 28 November 2008
Accepted 2 December 2008
Available online 14 December 2008

Keywords:
Model reduction
Identification method
Inverse problem of Parameter estimation
Optimization
Adjoint method

ABSTRACT

The numerical solution of most fluid mechanics problems usually needs such a fine mesh that the associated computational times become non-negligible parts in any design process. In order to couple numerical modelling schemes with inversion or control algorithms, the size of such models needs to be highly reduced. The identification method is a way to build low-order models that fit with the original ones. The laminar flow over a backward-facing step is used as a test case. Presented solutions are found to be in good agreement with experimental and numerical results found in the literature.

1. Introduction

Many industrial problems involve separating and reattaching flows in channels, usually combined with recirculation bubbles. Heat exchanger flows, for instance, often bear such kind of behaviors. But despite the complexity of the flow topology, the entire behavior of most fluid flows is described by the so-called Navier–Stokes equations.

Since in most cases, these equations do not provide the known analytical solutions, many numerical methods have been developed over the years to solve them. The space discretization can be based on, among others, the finite element formulation, or, more usually, the finite volume method.

Among the types of flows which ensure separation and recirculation bubbles, the one around a backward-facing step can be regarded as having a very simple geometry while retaining rich flow features like the ones mentioned above. The understanding of its structure may thus lead to a finer analysis of what may spring with more complex geometries. The backward-facing step flow has often been used as a test case to assess the accuracy and efficiency of the codes developed from the methods mentioned above. Indeed, its geometry does not prove to be challenging for meshing, and the experimental data are available in plenty.

The literature offers many numerical and experimental studies on 2D steady incompressible flows over the backward-facing step. Its topology is known to depend on geometrical parameters, but is still determined mostly by the Reynolds number. It is currently accepted that the flow features are stable and steady up to $Re = 800$, when this number is calculated on the upstream mean velocity and hydraulic diameter [1–4]. It can be noted that, still depending on the Reynolds number, the flow may exhibit one or two recirculation regions of varying lengths.

* Corresponding author. Tel.: +33 5 49 45 36 85.
E-mail address: Yann.Favennec@univ-nantes.fr (Y. Favennec).
1 From September 2008, this author has been staying at Polytech' Nantes, Université de Nantes, La chantrerie - BP 50609, 44306 Nantes Cédex 3, France.

0021-9991/$ - see front matter © 2008 Elsevier Inc. All rights reserved.
doi:10.1016/j.jcp.2008.12.001

Any discretization method leads to the resolution of a matrix system of algebraic equations (AEs), instead of the integration of continuous partial differential equations (PDEs). For the solution of the matricial system to be close enough to the solution of the partial differential equations, the time-space discretization must be fine enough. This constraint usually leads to large matricial systems. Thus, depending on the mesh size and on the physics to be approximated, the computational price to pay to obtain a suitable solution can be high in terms of memory and CPU time.

When the solution of the model is to be found several times for particular applications, e.g. for inverse or optimization problems, then one has to re-assess the compromise made between accuracy and time consumption. One way to avoid the loss of accuracy is to consider reduction modeling techniques. These methods aim to solve a restricted number n of ordinary differential equations (ODEs) instead of the $N \gg n$ equations given by the "classical" discretization of the partial differential equations.

When considering reduction techniques, one may cite the methods using a basis change [5] reduction in the physical space of variables. For thermal linear problems, several methods coming from automatics have been successfully applied [6]. Though many reduction methods for application on linear systems exist, a few of them are viable for applications on nonlinear problems. On the one hand, the Proper Orthogonal Method coupled with the Galerkin projection (POD-G) has proved to be very efficient on fluid-type nonlinear problems where turbulence plays a non-negligible role [7–9]. On the other hand, the Modal Identification Method (MIM) has proved to be very efficient on diffusion-type nonlinear problems [10–12]. A comparison on a particular nonlinear diffusive problem between both the POD-Galerkin method and the Modal Identification Method recently proved that both methods are accurate and robust and that both can be formulated equivalently although the general ideas behind those two are completely different [13].

In this paper, a method derived from the Modal Identification Method is used to identify some reduced models related to some fluid mechanics problems. The reduction process leans on the solution of an inverse problem of parameter estimation: one defines the structure of the reduced model formulation before estimating the related vectors and matrices through the solution of an optimization problem. Let us insist here on an important point: the identification method aims at reproducing data that are supposed to be well described by the temporal equations of the detailed model, which is then a reference. Thus, when deriving the detailed model directly out of the Navier–Stokes equations, we ensure that we can use fields coming from any numerical codes, as long as those fields are a good approximation of a Navier–Stokes solution. It implies that the formulation of the reduced model is independent of the numerical code which provides the data to be used. It does not even depend on the class of numerical schemes used (e.g. finite differences and finite volume). The reduction method can even be used based on the experimental data. Consequently, the actual purpose of this paper is to evaluate the ability of the reduced model to reproduce and to predict the results obtained by a numerical code solving the Navier–Stokes equations. For reasons of simplicity and also because it is a very well-known code among fluid mechanics engineers, we chose to use the finite-volume code Fluent 6.3.26 to provide us with the flow data.

The developed identification method leads to consider some low-order models that are related to some high-order models. In this sense, one can speak of model reduction. Also, since the approach leans upon an optimization algorithm, one can also speak of compact modeling coupled with data fitting, sometimes referred as behavioural modeling. Actually, due to the fact that the matrices of the reduced models are not computed in direct way but rather identified in the modal form, one uses the terminology "modal identification method for model reduction".

This paper is organized as follows: In Section 2, the formulation of the detailed model is derived. The governing Navier–Stokes equations together with the boundary conditions are given. The variational problem is considered along with the special treatment of the pressure variable and the boundary conditions. This section, based upon the classical literature, e.g. [14,15] and also [16,17], eventually gives a formulation that is suitable for model reduction through the identification method. Note again that this formulation is not necessarily the one that is used to obtain the data before the estimation of the reduced model. Next, Section 3 gives the main keys for model reduction through the identification method. We detail there the reduced model formulation and its identification, which leans on the use of optimization algorithms. More precisely, one uses a gradient-type method where the gradient of the cost function is computed through the adjoint problem. In Section 4, we present some numerical results of model reduction for the backward-facing step problem. We find that the developed identification method seems to be well suited for model reduction in this particular but representative case. Eventually, Section 5 is dedicated to some conclusions and especially to future prospective works.

2. Formulation of the structure of the detailed model designed for model reduction

2.1. The governing equations

Let Ω be an open-bounded domain in $\mathbb{R}^d$, $(d = 2, 3)$, with a boundary $\partial\Omega$ and an outward pointing normal $\boldsymbol{n}$. For $T > 0$, we consider the problem of solving, for $\boldsymbol{u} : \Omega \times (0, T) \to \mathbb{R}^d$ and $p : \Omega \times (0, T) \to \mathbb{R}$, the time-dependent Navier–Stokes equations:

$$\begin{cases} \frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} - \nu\Delta\boldsymbol{u} + \frac{1}{\rho}\nabla p = 0 & \text{in } \Omega \times (0, T), \\ \nabla \cdot \boldsymbol{u} = 0 & \text{in } \Omega \times (0, T), \end{cases} \tag{1}$$

where $\boldsymbol{u}(\boldsymbol{x},\mathrm{t})$ is the flow velocity, $p(\boldsymbol{x}, t)$ is the pressure, $\nu > 0$ is the kinematic viscosity of the fluid, $\rho > 0$ is the fluid density and $\boldsymbol{x}$ is the collection of $(x_i), i = 1, \ldots, d$.