Advances in Computational & Experimental Engineering & Science

A Fully Coupled Resolution Method for Computing Laminar and Turbulent Flows

E. Didier[1] and A. R. J. Borges[1]

Summary

This work discusses the computation of the unsteady, laminar and turbulent, two dimensional incompressible viscous flow on unstructured cell-centered collocated grids. The computational approach, the so-called fully coupled resolution method, is presented as an alternative to classical decoupled approaches. This strategy is evaluate on the simulation of the flow around single and multiple circular cylinders, for which experimental data are available. Both robustness and computational efficiency of this approach are shown.

Introduction

The simulation of non-stationary flows in Computational Fluid Dynamics (CFD) requires proper modeling of the non-linear terms of Navier-Stokes equations. Due to the linearisation of these equations, it is necessary to perform one or more non-linear iterations during simulation process, to assure non-linearities residues convergence. However, traditional solution methods, like segregated method where momentum and pressure equations are solved successively, appear insufficiently accurate to deal with non-stationary flows or present slow non-linear convergence.

A bidimensional fully coupled (FC) resolution method associated to a finite volume approach formulation is developed to solve the Unsteady Reynolds Averaged Navier Stokes (URANS) equations [1]. This approach presents an attractive and emergent trend in CFD [2,3], and an alternative to standard decoupled (DC) method, like SIMPLE [4]. The main advantage of this FC method consists in an increased robustness due to an implicit and global treatment of the pressure-velocity coupling. Furthermore it allows faster convergence of non-linear residual than DC method, and therefore a particularly attractive procedure when dealing with unsteady flow simulations.

After the presentation of the numerical FC resolution method, simulations results for laminar and turbulent flows around circular cylinders are presented and discussed.

URANS Model

The URANS equations are written in a dimensionless form for an incompressible Newtonian viscous fluid and a turbulent flow:

[1] Departamento de Engenharia Mecânica e Industrial, Universidade Nova de Lisboa, Caparica, Portugal

Advances in Computational & Experimental Engineering & Science

$$u_{i,i} = 0 \tag{1}$$

$$u_{i,t} + \left(u_i u_j\right)_{,j} = \frac{1}{Re} u_{i,jj} - p_{,i} - \left(\overline{u'_i u'_j}\right)_{,j} \tag{2}$$

assuming a linear eddy-viscosity hypothesis for the turbulent Reynolds stresses,

$$-\left(\overline{u'_i u'_j}\right)_{,j} = \nu_T \left(u_{i,j} + u_{j,i}\right) - \frac{2}{3} k \delta_{ij} \tag{3}$$

Equations (1) to (3) involve the mean Cartesian velocity components u_i, the mean pressure p, the Reynolds number Re, the Kronecker symbol δ_{ij}, the turbulent kinetic energy k and the turbulent eddy viscosity ν_T which defines the eddy-viscosity model to be used. To close the set of equations, a two transport equations turbulence model, to solve the transport equations up to the wall, was choose: the k-ε model of Jones and Launder [5] and the k-ω model of Wilcox [6] adopted for both ability to simulate adverse pressure gradient and numerical robustness.

This formulation needs boundary conditions for velocity and turbulent model. Let us denote by Γ_C the boundary defined by the bodies geometries and by Γ_E the external boundary. For the velocity field, boundary conditions are given by:

$$\begin{aligned} &\text{on} \quad \Gamma_C : u_1 = u_2 = 0 \\ &\text{on} \quad \Gamma_E : u_1 = U_a \,;\; u_2 = 0 \end{aligned} \tag{4}$$

No pressure boundary condition is necessary at the external boundary in according with the adopted formulation.

The boundary condition for k-ε and k-ω turbulent models are the usual turbulent boundary conditions proposed respectively by Jones and Launder [5] and Wilcox [6].

Fully Coupled Resolution Method

This solver was constructed using a FC approach [2,3], adapted to unstructured cell centered collocated grids [1]. This FC approach is an attractive and emergent trend in CFD, and an alternative to standard DC approaches, like SIMPLE method [4].

In summary, the main advantage of this FC method consists in an increased robustness due to an implicit and global treatment of the pressure-velocity coupling. This technique allows fast convergence of non-linear residual than DC method (Figure 1), and appears as an attractive procedure for unsteady flow simulations.

The linearised system of bidimensional URANS equations and the reconstructed pressure equation [7] are discretised by the finite volume method for unstructured grids

Advances in Computational & Experimental Engineering & Science

and cell-centered collocated arrangement of unknowns. This implicit second order fully coupled resolution method uses second order scheme in time and space: three points Euler implicit scheme, Central Differencing Scheme, and Deferred Correction approach [8]. The linear system in velocity-pressure, which present a very ill-conditioned sparse and non-symmetric matrix, is solved using iterative algorithm resolution, like BiCGSTAB-ω with LU preconditioning.

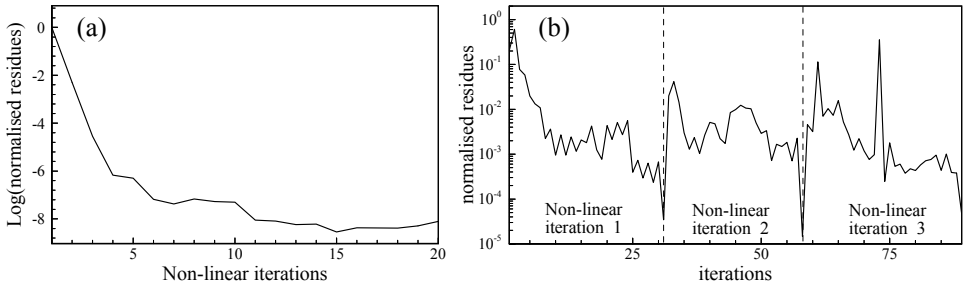

Figure 1: Flow around a cylinder, Re=3000. (a) Typical non-linear residual convergence; (b) Residual converge of linear system for three non-linear iterations.

Turbulence equations are discretised using the same schemes that the transport equations. The linear system for turbulence kinetic energy and turbulence dissipation issued from k-ε or k-ω model is first solved. The system is well conditioned and easily invertible by a classical CGSTAB algorithm.

CFD results and discussion

In order to validate and demonstrate the efficiency of FC resolution method on unstructured grid, computations are first performed for a single circular cylinder for Re=9500. A mixed unstructured grid is used with: 36000 cells, 400 nodes along the cylinder, 15 quadrilateral cells near the wall in the radial direction, a mesh refinement in the near wake. The first point in the boundary layer is located at e/D=0.0002 from the wall and the non-dimensional time step is Δt=0.005. The simulation of a 2s development of the flow takes 2.5 hours CPU time on a workstation Compaq, processor EV6, 500 MHz.

The number of non-linear iterations require to obtain a relative residue inferior to 5.10-3 is equal to 2 to 3. This accuracy is turned up to be sufficient to have agreement between numerical and experiments results as shown in Figure 2 and Figure 3.

The Fig. 2 gives the evolution of the circulation length. It is predicted with good accuracy, if compared with experimental data of Bouard and Coutanceau [9] and numerical results using a vorticity-stream function method [10].

Advances in Computational & Experimental Engineering & Science

Experimental data allows to compare profiles of the velocity component u_1 on the axis of the near wake of the cylinder for five stages of the development. As can be seen on Fig. 4, all five profiles of velocity are predicted with very good accuracy.

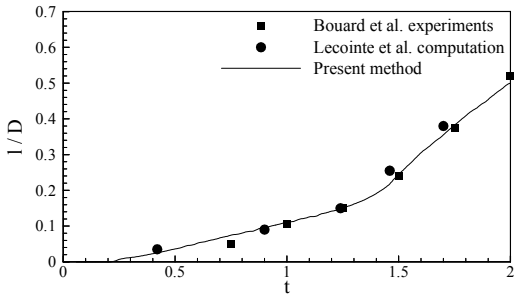

Figure 2. Circulation length during the flow development. *Re*=9500.

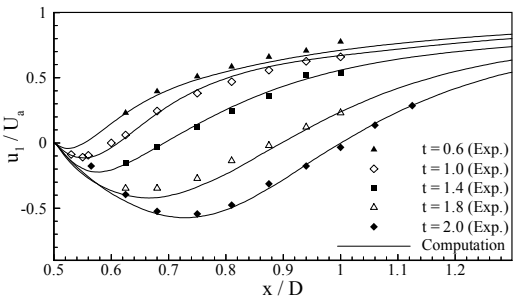

Figure 3. Profiles of u_1 on the near wake of a cylinder. *Re*=9500.

The second simulation for multiple circular cylinder configurations deals with two cylinders placed side by side, with a distance center to center *T/D*=1.5, normal to the flow, at *Re*=2200. Experiments reported by Summer et al. [11] allows to compare position of vortex center defined by *a* and *b*, respectively, the longitudinal distance between the vortex center and the rear of cylinder, the transversal length between the vortices center. Two types of vortex pairs appear: inside vortex *A* and outside vortex *B*.

A mixed unstructured grid is used, composed with 46000 cells, 400 nodes along each cylinder, 18 quadrilateral cells near the wall in the radial direction, and a mesh refinement in the near wake. The external boundary is located at twenty diameters of the cylinders. The first point in the boundary layer is located at *e/D*=0.0003 from the wall and the non-dimensional time step is Δt=0.01. The simulation of a 6s development of the flow takes 3.5 hours CPU time on a workstation Compaq, processor EV6, 500 MHz.

For 1.5< *T/D* <3.0 there are two interacting but distinct vortex streets. Obviously, for *T/D* large, cylinders generate two normal wakes, behaving quite independently of one another. At the Reynolds number *Re*=2200 and *T/D*=1.5, even though the wake is turbulent, the flow is essentially laminar. The doubled vortex street, for inside and outside vortices, is formed in which the vortices shed are in phase initially.

Figures 5 presents the vortices position. The numerical results compared with experimental ones show a good agreement. As can be seen on Fig. 5 right, the inside vortex is the first to be detached from the cylinder. The velocity of inside vortex center displacement is larger than the outside vortex center. The velocity of the first vortex center displacement and its position are in good agreement with experiments. In Fig. 5 left, inside and outside vortex center to center distances are respectively slightly overestimated and underestimated, due certainly to the inadequacy of grid refinement.

Advances in Computational & Experimental Engineering & Science

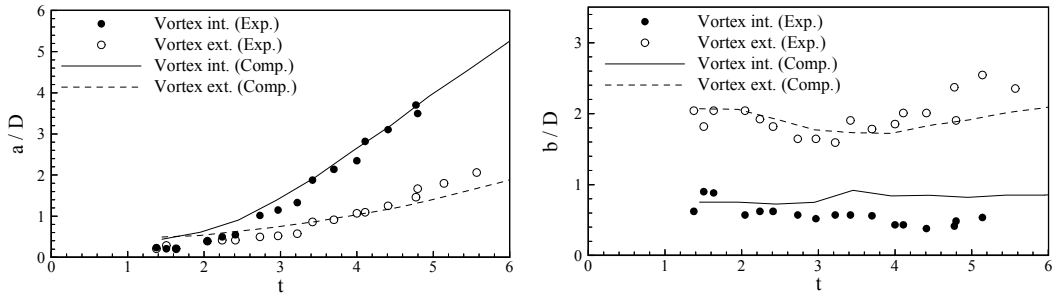

Figure 5. Vortex center position *a* and *b* for two cylinders. *T/D*=1.5, *Re*=2200.

The ultimate case deals with the uniform flow around a circular cylinder at *Re*=140000 studied experimentally by Cantwell and Coles [12]. This flow is a challenging one for conventional simulation. Indeed, the flow is laminar until a transition takes place after the first separation. Simulation is performed using a grid composed by 31500 cells, with a first point in the boundary layer located at *e/D*=0.00005.

We choose to compare the mean radial velocity profile at the rear of the cylinder. The Wilcox k-ω model gives reasonable results regarding to the k-ε model and comparing with experimental profile (Figure 6). The mean value of the drag coefficient is equal to 0.9 when using the standard k-ω model. The experimental value is 1.1. The Strouhal number is overestimated at 0.25 when, from experiments, it is equal to 0.23.

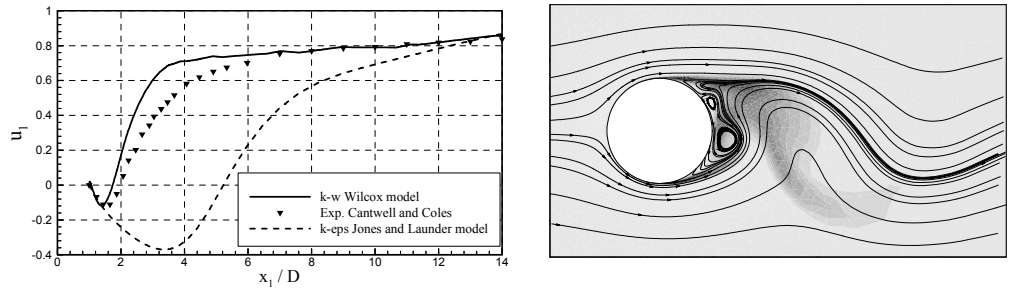

Figure 6. Right- Mean velocity component on the centerline at the rear of the cylinder. Left- Kinetic energy and stream line. *Re*=140000.

Concluding remarks

An original numerical method has been presented in this paper. The FC resolution method, which solves only one single linear system, leads to a strong velocity-pressure coupling which is essential for good convergence on non-linearities of linearised Navier-Stokes equations. The non-linear residue convergence is fast and allows a reduction of eight orders in a few iterations, much better than we get using DC methods like SIMPLE.

Advances in Computational & Experimental Engineering & Science

Simulation results demonstrate the robustness of the FC method for unstructured grids, and the ability of the FC solver to predict accurately laminar and turbulent flows around single and multiple bluff body geometries.

Reference

1 Didier, E. & Borges A.R.J. (2003): "Unsteady Navier-Stokes equations: a fully coupled method for unstructured mesh", *Proc. of the 12th Int. Conf. on Modelling Fluid Flow*, eds. T. Lajos & J. Vad: Budapest, pp. 814-821.

2 Deng, G.B., Ferry, M., Piquet, J. & Visonneau, M. (1991): "New fully coupled solutions of the Navier-Stokes equations", *Notes Num. Fluid Mech.*, Vol. 35, pp. 191-200.

3 Deng, G.B., Piquet, J., Vasseur, X. & Visonneau, M. (2001): "A new fully coupled method for computing turbulent flows", *Computers & Fluids*, Vol. 30, pp. 445-472.

4 Patankar, S. & Spalding, D. (1972): "A calculation procedure for heat, mas and momentum transfer in three-dimensional parabolic flows", *Int. J. Heat Mass Transfer*, Vol. 15, pp. 1787-1806.

5 Jones, W.P. & Launder, B.E. (1972): "The prediction of low Reynolds number phenomena with a two equation model of turbulence", *Int. J. Heat Mass Transfer*, Vol. 16.

6 Wilcox, D.C. (1993): *Turbulence Modeling for CFD*, DCW Industries: California.

7 Rhie, C.M. & Chow, W.L. (1983): "Numerical study of the turbulent flow past an airfoil with trailing edge separation", *AIAA Journal*, Vol. 21, pp. 1525-1532.

8 Kosla, P.K. & Rubin, S.G. (1974): "A diagonally dominant second-order accurate implicit scheme", *Computers fluids*, Vol. 2, pp. 207-209.

9 Bouard, R. & Coutanceau, M. (1980): "The early stage of development of the wake behind an impulsively started cylinder for $40<Re<10^4$", *J. Fluid. Mech*., Vol. 101, part 3, pp. 583-607.

10 Lecointe, Y. & Piquet, J. (1985): "Compact finite-difference methods for solving incompressible Navier-Stokes equations around oscillating bodies", *Chapter 1985-04, Von Karman Institute, Lecture Series*.

11 Summer, D., Price, S.J., & Paidoussis, M.P. (1997): "Investigation of impulsively started flow around side-by-side circular cylinders: application of particle image velocimetry", *J. of Fluids and structures*, Vol. 11, pp. 597-615.

12 Cantwell, & B. Coles, D. (1983): "An experimental study of entrainment and transport in the turbulent near wake of a circular cylinder", *J. Fluid. Mech*., Vol. 136, pp. 321-374.