Chapter 4

Fully Conservative Multiblock Method for the Resolution of Turbulent Incompressible Flows

Main contents of this chapter are published as
J. Cadafalch, C.D. Pérez-Segarra, M. Sòria and A. Oliva. Fully conservative multiblock method for the resolution of turbulent incompressible flow. *In Proceedings of the Fourth ECCOMAS Computational Fluid Dynamics Conference*, volume 1.2, pages 1234-1239, Athens, 1998.

Abstract. This paper presents a fully conservative finite-volume overlapping multiblock formulation for the analyses of turbulent natural and forced internal flows. In order to study in detail the well-posedness of the method, the work is restricted to simple scenarios (Cartesian meshes which overlap in arbitrary directions, and two-dimensional flows). The turbulence is modeled by means of two-equation low-Reynolds-number k-ε models. The mass and momentum equations are solved by a coupled algebraic multigrid method, while the scalar transport equations (energy, k and ε) are solved segregately by a multigrid solver. The free convection in a tall cavity is studied with decomposed and non-decomposed domains. The numerical errors are compared, and the generalized Richardson extrapolation is adopted to estimate the order of accuracy of the solution.

4.1 Introduction

Recently, many efforts have been made by the computational fluid dynamics community in the research of methods that provide a higher computational power. Coupled solvers for the resolution of the Navier-Stokes equations have been shown to be more efficient than segregated solvers in many situations [1]; the parallel computation reduces the execution time [2]; and the multiblock technique has been adopted by many scientists as a tool which, combined with body-fitted coordinates, can give structured grids the capability of fitting into complex geometries [3], as an alternative to the non-structured grids and the blocking off technique (approximation of the geometry in a stepwise manner).

In the multiblock method the global domain is divided up in several subdomains or blocks. The governing equations are solved in each subdomain, and information of the dependent variable values is transferred between each other in the sharing boundaries.

During the last decade, many studies have been carried out on the multiblock technique for the analyses of incompressible flows. They can be classified into two groups: Patched Grid, in which the domain is divided into subdomains without overlapping zones [4]; and Overlapping Grid which divides the domain into overlapped subdomains [5]. The most important difference between the works presented by different authors is the way of transferring information between subdomains. Interpolation schemes that preserve local fluxes of the physical quantities between the subdomains are known as conservative schemes; on the other hand, interpolation schemes such as Lagrangian interpolations are known as non-conservative. It is widely accepted that conservative schemes yield to better results. However, some so-called conservative schemes which only preserve mass fluxes can lead to wrong solutions [6].

A multiblock method should accomplish the well-posedness conditions; i.e. the adopted interpolation scheme should not affect the result of the differential equations. This aspect clearly depends on the nature of the PDEs. Henshaw and Chesshire [5] demonstrate that for one 2nd order PDE, non-conservative and conservative interpolations are well-posed. If this conclusion is extrapolated to the resolution of the Navier-Stokes equations (in two-dimensional problems two second-order PDEs coupled by a first-order PDE), Lagrangian interpolation could be thought to be a suitable interpolation scheme, which has been shown in previous works to be false [7].

The purpose of this paper is to demonstrate that the fully conservative formulation based on finite-volume techniques presented here preserves the well-posedness condition. Therefore, a simple two-dimensional problem is studied. The extension of this formulation to three-dimensional situations and body-fitted coordinates will be considered in a future work.

The turbulence is modeled by means of two equation low-Reynolds number k-ε models. The discretized governing equations are obtained using the finite-volume method on staggered structured Cartesian grids. The mass and momentum equations are solved by a coupled algebraic multigrid method [1], and the energy, the turbulent kinetic energy and its dissipation rate equations are solved segregately by means of a multigrid solver. A global explicit procedure for the information transfer between subdomains is adopted.

A post-processing tool based on the Richardson extrapolation [8] has been developed to study the order of accuracy of the numerical solutions. The numerical errors are also studied. This tool requires solving each problem adopting an h-refinement criterion (use of different meshes related by a mesh ratio r; in this work $r = 2$ and five levels of refinement are studied). The numerical solutions of the free convection in a tall cavity using a decomposed domain and a non-decomposed domain are compared. The well-posedness condition of the multiblock method requires the order of accuracy and the numerical errors of both solutions to be similar.

4.2 Formulation

4.2.1 Governing equations

The time-averaged governing equations of the fluid flow (continuity, momentum and energy) assuming: the Boussinesq approximation (density variations are relevant only in the buoyancy terms of the momentum equations), fluid Newtonian behavior, negligible heat friction and influence of pressure on temperature, non-participant radiating medium, may be written in tensor notation as:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{4.1}$$

$$\frac{d(\rho u_i)}{dt} = -\frac{\partial p_d}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} - \rho\beta(T - T_o)g_i \qquad (i = 1, 2) \tag{4.2}$$

$$\frac{d(\rho T)}{dt} = -\frac{1}{c_p}\frac{\partial \dot{q}_i}{\partial x_i} \tag{4.3}$$

where:

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \rho\overline{u_i' u_j'} \tag{4.4}$$

$$\dot{q}_i = -\lambda \frac{\partial T}{\partial x_i} + c_p \rho \overline{u_i' T'} \tag{4.5}$$

and, x_{i} is the Cartesian coordinate in the i-direction ($x_1{=}x$, $x_2{=}y$); t the time; u_{i} the mean velocity in the i-direction ($u_1{=}u$, $u_2{=}v$); p_{d} the mean dynamic pressure; T the mean temperature; g_i the i-component of the gravitational acceleration vector ($g_1 = 0$,$g_2 = g$); T_o is the reference temperature; and ρ, μ, β, λ, c_{p} are respectively: the density, the dynamic viscosity, the coefficient of thermal expansion, the thermal conductivity and the specific heat at constant pressure. The turbulent fluctuating velocity in the $\mathrm{x_i}$-direction and the fluctuating temperature are indicated by u_{i}' and T'.

The turbulence is modeled using a low-Reynolds k-ε turbulence models. The turbulent stresses and the turbulent heat fluxes are evaluated by analogy with the Stokes viscosity law and the Fourier heat conduction law. Thus, these terms are written in the form:

$$\rho \overline{u_i' u_j'} = -\mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) + \frac{2}{3} \rho k \delta_{ij} \tag{4.6}$$

$$\rho \overline{u_i' T'} = -\frac{\mu_t}{\sigma_t} \frac{\partial T}{\partial x_i} \tag{4.7}$$

where: μ_{t} and σ_{t} are the turbulent viscosity and the turbulent Prandtl number, and δ_{ij} is the Kronecker delta. The turbulent Prandtl number is usually taken as a constant. The turbulent viscosity is related to the turbulent kinetic energy (k) and the dissipation of turbulent kinetic energy (ε) by means of the empirical expression of Kolmogorov-Prandtl.

The turbulent kinetic energy and its dissipation are obtained from their transport equations. Although the exact form of these equations results from the Navier-Stokes equations, empirical approximations of some terms are necessary. The resulting k-ε equations, together with the Kolmogorov-Prandtl expression, can be written, after taking low-Reynolds-number effects into account, as:

$$\mu_t = c_\mu f_\mu \frac{\rho k^2}{\tilde{\varepsilon}} \tag{4.8}$$

$$\frac{d(\rho k)}{dt} = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + P_k + G_k - (\rho \tilde{\varepsilon} + D) \tag{4.9}$$

$$\frac{d(\rho\tilde{\varepsilon})}{dt} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\tilde{\varepsilon}}{\partial x_i}\right] + c_{\varepsilon 1}\left[f_1 P_k + c_{\varepsilon 3} G_k\right]\frac{\tilde{\varepsilon}}{k} + E - c_{\varepsilon 2} f_2 \frac{\rho\tilde{\varepsilon}^2}{k} \tag{4.10}$$

where the variable $\tilde{\varepsilon}$, defined as $\tilde{\varepsilon}=\varepsilon\text{-}D/\rho$ is introduced in some turbulence models for computational convenience in order to obtain a zero value of $\tilde{\varepsilon}$ at the wall. The shear production and buoyancy production/destruction of turbulent kinetic energy are respectively $P_k = -\rho\overline{u_i' u_j'}\partial u_i/\partial x_j$ and $G_k = -\beta\overline{u_i' T'} g_i$. In these terms the turbulent stresses and heat fluxes are evaluated using the standard eddy-diffusivity model.

The boundary conditions and the empirical values (c_μ, $c_{\varepsilon 1}$, $c_{\varepsilon 2}$, $c_{\varepsilon 3}$, σ_k, σ_ϵ, σ_t, f_μ, f_1, f_2, D and E) depend on the turbulence model adopted.

4.2.2 Multiblock interpolation scheme

The resolution of the above-presented governing equations by means of multiblock method requires special boundary conditions in the boundaries of the overlapping zones.

For the averaged Navier-Stokes equations, the entrance mean velocity is calculated via local mass balances, and the tangential mean velocity using local balances of the tangential-momentum fluxes. This procedure has been proved to be suitable in previous works on laminar simply connected incompressible flows [6][7].

The scalar fields (k, ε and T) are coupled with the Navier-Stokes equations in the coefficients of the momentum diffusion (turbulent viscosity) and in the body force term. However, in the multiblock method they behave as if they were non-coupled single 2nd order PDEs. A conservative scheme based on Lagrangian interpolations in one subdomain, and which preserves convective and diffusive fluxes in the other, has been proved to be well-posed for the temperature field [7]. In this work, this procedure is used for all the scalar fields.

4.2.3 Post-processing tool

In order to evaluate the quality of the numerical solutions, the numerical error and the order of accuracy (p) are estimated using a post-processing tool described in detail in [6]. An h-refinement treatment is adopted. The problem is solved on different meshes related by a mesh ratio r (in this work r=2 and five levels of refinement are studied). According to the Richardson extrapolation [8], with three solutions of a problem (ϕ_1, ϕ_2 and ϕ_3) obtained on the grids $h_1=h$ (fine grid), $h_2=rh$ (middle grid) and $h_3=r^2h$ (coarse grid), the order of accuracy of the numerical solution can be determinated by the following equation when the assumptions of smoothness and monotone error convergence in the mesh spacing apply:

$$p = \frac{\ln\left(\frac{\phi_2-\phi_3}{\phi_1-\phi_2}\right)}{\ln r} \tag{4.11}$$

A map of the estimated order of accuracy is calculated with three consecutive mesh levels.

A single grid independent solution is also carried out. It is considered to be the "exact" or reference solution, and maps of errors for each refinement level are obtained by direct comparison to this reference solution.

In both the error map and the numerical accuracy map, as the compared meshes do not have coinciding nodes, it has been necessary to interpolate information from the solutions. Bi-quadratic interpolations have always been used so as not to introduce uncertainties in the post-processing study.

The averaged p and the averaged error have been adopted as global estimates of the global order of accuracy and the global numerical error. As the calculation of p requires three consecutive mesh levels, and the h-refinement treatment has been carried out on five levels, only three p estimators are calculated. On the other hand, the numerical error estimation is evaluated for each level of refinement.

This post-processing tool has to be applied segregately on all the variables of the problem.

RESULTS

The fully conservative multiblock method has been tested on the resolution of the incompressible turbulent air flow (Pr=0.71) in a tall rectangular cavity (aspect ratio=30) with differentially heated vertical walls (Ra=$2.43{\cdot}10^{10}$) and adiabatic top and bottom walls.

The turbulent empirical values and functions proposed by To and Humphrey [9] and the boundary conditions k=0 and $\partial\varepsilon/\partial \mathrm{x_n}$=0 have been adopted.

Both the first-order accurate upwind difference scheme (UDS) and the high-order SMART scheme [10] for the convective terms have been adopted, together with second-order accurate central differences for the diffusive terms.

Rectangular structured meshes of $n_\mathrm{x} * n_\mathrm{y}$ control volumes, concentrated at the walls by means of a tanh-like function [11] with concentration factors κ_x and κ_y have been used (κ_x: concentration factor in the x-Cartesian direction, κ_y: concentration factor in the y-Cartesian direction), see Fig. 4.1.

An h-refinement study using a single-domain and two overlapped domains has been performed with five refinement levels of n=10, n=20, n=40, n=80 and n=160 (where n_x=n_y=n for the non-decomposed domain, and n_x=n and n_y=$n/2+1$ for the decomposed domain). In the overlapping zone of the decomposed domain, which

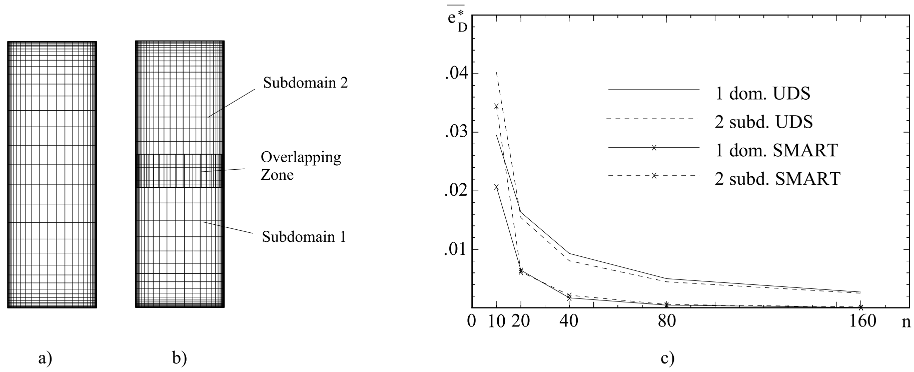

Figure 4.1: (a) Grid for the non-decomposed domain study (n_x=n_y=n , κ_x=4, κ_y=3). (b) Grid for decomposed domain study (n_x=n, n_y=$n/2+1$, κ_x=4, κ_y=3; subdomain 2: n_x=n, n_y=$n/2+1$,κ_x=3, κ_y=2). (c) Averaged absolute normalized error of the T-field vs. discretization level for different numerical schemes (UDS and SMART).

contains at least two nodes of the involved subdomains, no concentration factor in the y-Cartesian direction has been used in order to avoid convergence difficulties that could arise from relating meshes with highly different sizes. in order to point out the generality of the Interpolation scheme adopted, in the subdomains of the decomposed-domain study the meshes have been concentrated with different concentration factors in such a way that the control volumes faces of both do not coincide, see Fig. 4.1b.

The solution in a single block with $320 * 320$ control volumes refined at the walls (κ_x=4, κ_y=3) and using the SMART scheme has been taken as the reference solution for the error studies.

The numerical error and the order of accuracy of the study on the single domain and on the decomposed domain have been calculated and compared for the variables u, v, T, k and ε. As similar results have always been achieved, only the corresponding to the T-field will be presented.

Fig. 4.1c compares the evolution of the average normalized error of the temperature field (normalized by the temperature difference between the vertical walls) for the single domain and for the decomposed domain studies. The discretization errors are similar when the same levels of refinement are compared, and both vanish with the mesh. Only some discordance appear for the coarsest grid (n = 10), where the numerical solution is far from the grid independent solution and little differences in the mesh spacing can lead to important differences in the solution.

Table 4.1 compares the average p values of the T-field calculated for the study on

grid	$\bar{p}$			
	UDS		SMART	
$n_3/n_2/n_1$	1 dom	2 subd	1 dom	2 subd
10/20/40	1.33	1.15	1.79	2.37
20/40/80	1.18	1.35	2.18	1.77
40/80/160	1.15	1.23	1.64	1.75

Table 4.1: Average p of the T-field for different numerical schemes (UDS and SMART).

the non-decomposed domain and on the decomposed domain. For example, with the set of coarsest solutions (n_3=10, n_2=20 and n_1=40), the estimated order of accuracy for the solution on the single domain corresponding to the UDS and SMART are respectively 1.33 and 1.79, and the values for the study on the decomposed domain are 1.15 and 2.37. The comparison between levels of refinement involves the hypothesis of equality of the grid spacing. As different concentration factors have been adopted for the decomposed domain study, some differences in the values of the estimated order of accuracy could be expected a priori. However, good agreement is always achieved between the values calculated for both studies. All of them are also reasonable values: around 1 when the convective terms are modeled by means of the first order UDS scheme, and around 2 for the cases with the high order SMART scheme.

4.3 Conclusions

The multiblock method is an efficient tool used for improving the power of the computational fluid dynamics and heat transfer. Some multiblock approaches for incompressible flows have been developed recently by different authors. Some of these approaches have been shown in previous works not to be well-posed although in some problems, generally those with low gradients, they can lead to satisfactory results (e.g. conservative named schemes based only on the local conservation of mass). Therefore, in order to have confidence in a multiblock method, it is important to study it in detail before it is applied to the resolution of complex problems.

The present work demonstrates that the fully conservative multiblock formulation adopted for turbulent incompressible flows preserves the order of numerical accuracy and does not introduce additional uncertainty in the numerical solution, thus, it accomplishes the well-posedness condition.

4.4 Acknowledgments

This work has been financially supported by the Comisión Interministerial de Ciencia y Tecnología, Spain (project number TIC724-96) and by the Comissionat per Universitats i Recerca de la Generalitat de Catalunya.

Nomenclature

c_p	specific heat at constant pressure
$c_{\varepsilon 1}, c_{\varepsilon 2}, c_{\varepsilon 3}, c_\mu$	empirical constants in the turbulence models
D, E	extra terms in the k and ε equations
e_D	discretization error
f_1, f_2, f_μ	empirical functions in the turbulence models
g	gravitational acceleration
G_k	buoyancy production/destruction of k
k	turbulent kinetic energy
n	number of nodes
p	order of accuracy
p_d	dynamic pressure
P_k	shear production of k
Pr	Prandtl number
r	mesh ratio in the h-refinement
Ra	Rayleigh number
t	time
T	temperature
u, v	Cartesian velocity components
x, y	Cartesian coordinates

Greek symbols

β	thermal expansion coefficient
δ_{ij}	Kronecker delta
ε	dissipation rate of k
κ	concentration factor
λ	thermal conductivity
μ	dynamic viscosity
μ_t	turbulent viscosity
ρ	density
$\sigma_k, \sigma_\varepsilon, \sigma_t$	turb. Pr for k, ε and T
τ_{ij}	shear-stresses

Superscripts

$'$	fluctuating quantity
$*$	non-dimensional value
$-$	average value

Subscripts

i, j	Cartesian directions
o	reference value

References

[1] P.S. Sathyamurthy. *Development and evaluation of efficient solution procedures for fluid flow and heat transfer problems in complex geometries.* PhD thesis, University of Minnesota, 1991.

[2] D. Drikakis, E. Schreck, and F. Durst. Performance analysis of viscous flow computations on various parallel architectures. *Journal of Fluids Engineering*, 116:635–841, 1994.

[3] Y. Zang and R.L. Street. A composite multigrid method for calculating unsteady incompressible flows in geometrically complex domains. *International Journal for Numerical Methods in Fluids*, 20(5):341–361, 1995.

[4] D. Lee and J.J. Chiu. Computation of physiological bifurcation flows using a patched grid. *Computers Fluid*, 21(4):519–535, 1992.

[5] G. Chesshire and W.D. Henshaw. A scheme for conservative interpolation on overlapping grids. *SIAM Journal of Scientific Computing*, 15(4):819–845, 1994.

[6] J. Cadafalch, J. Salom, M. Costa, and A. Oliva. Domain decomposition as a method for the parallel computing of laminar incompressible flows. In *Proceedings of the Second European Congress on Computational Methods in Applied Sciences and Engineering (ECCOMAS)*, pages 845–851, 1996.

[7] J. Cadafalch, A. Oliva, C.D. Pérez-Segarra, M. Costa, and J. Salom. Comparative study of conservative and nonconservative interpolation schemes for the domain decomposition method on laminar incompressible flows. *Numerical Heat Transfer, Part B*, 35(1):65–84, 1999.

[8] I. Celik and Vei-Ming Zhang. Calculation of numerical uncertainty using richardson extrapolation: application to some simple turbulent flow calculations. *Journal of Fluids Engineering*, 117:439–445, 1995.

[9] W.M. To and J.A.C. Humphrey. Numerical simulation of buoyant, turbulent flow-1. free convection along a heated vertical flat plate. *International Journal of Heat and Mass Transfer*, 29:573–592, 1986.

[10] P.H. Gaskell and A.K.C. Lau. Curvature-compensed convective transport: SMART, a new boundedness-preserving transport algorithm. *International Journal for Numerical Methods in Fluids*, 8:617–641, 1988.

[11] C.D. Pérez-Segarra, A. Oliva, M. Costa, and F. Escanes. Numerical experiments in turbulent natural and mixed convection in internal flows. *International Journal for Numerical Methods for Heat and Fluid Flow*, 5(1):13–33, 1995.