International Conference on Control, Engineering & Information Technology (CEIT'14)
Proceedings - Copyright IPCO-2014
ISSN 2356-5608

Instability and interaction of the swirling macrostructure with the boundary layer in a cavity differentially heated

TALOUB Djedid[#1], BEGHIDJA Abdelhadi[*2]

[#]*Faculty of Science M'sila University y M'sila 28000, Algeria*

[1]djtaloub@yahoo.fr

[*1,2]*Laboratory of Renewable Energy and Sustainable Development (LRESD)*

Constantine1University, Constantine 25000, Algeria

[2]abeghidja1@yahoo.fr

***Abstract*— The study of the interaction of boundary layer with the flow and the influence of heat exchange in cavity for strong numbers of Rayleigh Ra=2.5x10^{10} is the subject of the various numerical simulations. The resulting field velocity can be dividing up into two fields: The flow of the average field in the cavity, and the field velocity fluctuating of this flow in the cavity. The cavity is the seat of instability generating or modulating the swirling structure. This work is in the continuity of in experiments undertaken former studies. It also fits in the more general prospect for the Dynamics of the Fluids, Turbulence and the Heat transfers. It contributes to the development of the methods allowing the comparison between the digital simulations and the experiment in the case of the turbulent flows comprising of the non stationary macrostructures. From this point of view the method of Large Eddy Simulation (known as LES) east considers beside that of RANS. Our objective consists in confronting our resulting with those from experimental on the one hand and those establish by other numerical studies. It is noted that a purely thermal approach on the fluctuations is to be developed and to treat. Finely This work presents a numerical validation of LES-WALE model using the results of the K-epsilon model, this study is based on 3D numerical simulation using FLUENT code calculates to determine the longitudinal velocity, the thermal fields for the configuration the and velocity vectors (U, V, W) for configuration in the plane of the recirculation zones for the case. Therefore, the results have good agreement with those of k-epsilon model, as they show the difference between the cases of flows.**

***Keywords*— convection, finite volume, parallelepiped, k-epsilon-LES.**

I. INTRODUCTION

To define the state of a fluid moving, four unknown functions must be given: three components of the vector velocity and pressure. These functions must be given in each point of the space field and at any moment. For that one has recourse to the Navier-Stokes equations which connect these parameters and which are deduced starting from the physical laws from conservation and the Newtonian laws from the movement. So that these equations are applicable to the flows of fluid, it should be supposed that the fluid in question is a continuous medium.

The large swirling structures are very anisotropic and are conditioned inter alia by the specific geometrical configuration of the flow considered. Recently and since the means of calculation allow it, a novel method is essential more and more even in the field of the industrial applications. On the basis of noted proceeding, the idea is to separate the scales from turbulence to solve numerically only the large swirling structure and to replace the effect of smallest by models under mesh. The motivation comes owing to the fact that the large structures are anisotropic and are responsible for all those mechanisms of exchange. The calculation direct as of these swirls gives to the results more credibility without making the cost very prohibitory. In addition more the small scales which are isotropic have as a main role the dissipation of energy. This characteristic makes easier the construction of a model universal to model their interaction with the calculated structures. By adopting this technique, one can only be gaining since in addition to their isotropic natures, these swirls are all the more small as the Reynolds number is high. While introducing, the model one avoids effort of calculation enormously compared to calculation DNS while keeping the deterministic aspect of the solution. This technique called Large Eddy Simulation is recognized currently more and more like a technique complementary to experimental measurements for comprehension, the prediction and it control turbulent flows. It makes it possible in particular to visualize the dynamics of the swirling structures with large scales.

This work is based on many previous studies both experimental and numerical, including Mergui [1, 2], Salat [3, 4] Lankhorst [5], Tian [6] Ampofo [7] examined numerically flows turbulent natural convection in a parallelepiped cavity. Many numerical investigations have also been conducted with cavities of modest size or Ra < 10^9 [18, 19] Beyond (Ra > 10^{10}).

International Conference on Control, Engineering & Information Technology (CEIT'14)
Proceedings - Copyright IPCO-2014
ISSN 2356-5608

We consider here a configuration of the normal convection in a fluid of cavity of parallelepiped, where a gradient in temperature is imposed between the walls.

There are examples off such applications has configuration in the solar systems, double glazed Windows, gold the description off the air flow within has room.

The aim of this work is to compare the results of fluent code to those proposed references.

A. *Description of the problem*

The physical model considered is schematized on the fig. 1. It is about a three-dimensional parallelepiped cavity of great dimension (H=2.46 m height, L= 0.385 m of width, D=0.72 m of depth), filled of air, whose walls are differential heated at a constant temperature according to table 1. We study eight different configurations. The flow in this cavity is turbulent with a number Ra =2.5 10^{10}.

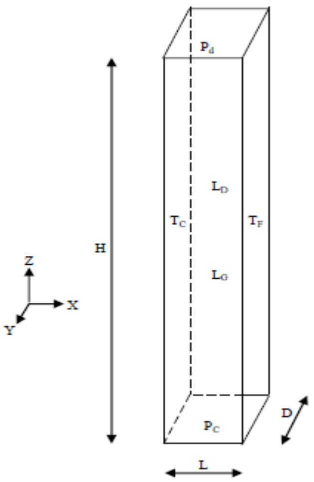

Fig. 1 Geometric configuration of the 3D model

TABLE I

VARIOUS STUDIED CONFIGURATIONS AND THEIR MODE OF REPRESENTATION

Configurations	C1	C2	C3	C4	C5	C6	C7	C8
	F	F	C	F	F	C	C	F
modes	FCCC	FFCF	FCCC	FMCM	FFCF	FCCC	FMCM	F FC F
	C	C	C	F	F	F	TF	TF

C :hot
F :cold
M :middle
TF:very cold
Tc: hot wall
Tf: cold wall
LD: right side wall
LG: left side wall
Pd: wall ceiling
Pc: wall Floor

		Pd	
TF	LG	Tc	LD
		Pc	

B. *Simplifying assumptions*

In order to obtain a simple mathematical model, one adopts the following assumptions:

- The flow is three-dimensional
- The fluid is Newtonian and incompressible
- The generated flow is turbulent
- The transfer of heat per radiation is negligible
- The physical properties of the fluid are constant except the density which obeys the approximation of Boussinesq in the term of pushed of Archimedes.

II. NUMERICAL SIMULATION OF TURBULENCE

Numerical simulation of turbulent and concepts related to the turbulence and that can provide adapted and effective modelling flows. We introduced the same equations governing our flows through this article and after posing problems of closing LES.

A. *Basic Equations*

The equations of evolution are used to describe the flow of an incompressible fluid in its movement. firstly reflects the conservation of mass locally, the other is conservation of movement quantity, and the third equation takes to reflect the heat transfer in the case of isothermal flow: energy equation.

The forms of these equations are different depending on the assumptions about the type of flow and fluid considered. In this study, we assume an incompressible fluid with constant thermodynamic properties (μ = const, ρ = constant, λ = constant, Cp = constant).

By applying a low-pass filter to results equations and the above assumptions, the system of equations to be solved in a filtered LES approach [8] we obtain:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \qquad (1)$$

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial}{\partial x_j}\left(\overline{u_j}\,\overline{u_i}\right) = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(2\nu \overline{S_{ij}} + \overline{\tau_{ij}}\right) \qquad (2)$$

$$\frac{\partial \overline{T}}{\partial t} + \frac{\partial}{\partial x_j}\left(\overline{u_i}\,\overline{T}\right) = \frac{\partial}{\partial x_j}\left(\alpha_f \frac{\partial \overline{T}}{\partial x_j} + \overline{\Theta_i}\right) \qquad (3)$$

$\overline{S_{ij}}$: Corresponds to the rate tensor of resolved deformation given by:

$$\overline{S_{ij}} = \frac{1}{2}\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) \qquad (4)$$

The new terms $\overline{\tau_{ij}}$ and $\overline{\Theta_i}$ from the filter respectively represent the tensor sub-mesh stress (or Reynolds stress) and heat flux sub-mesh and are defined by:

$$\overline{\tau_{ij}} = \overline{u_i}\,\overline{u_j} - \overline{u_i u_j} \qquad (5)$$

$$\overline{\Theta_i} = \overline{T}\,\overline{u_i} - \overline{T u_i} \qquad (6)$$

Modelling constraints sub-mesh $\overline{\tau_{ij}}$ is based on an assumption of sub-mesh viscosity (Boussinesq hypothesis) linking constraints in the mesh deformation rate tensor resolved $\overline{S_{ij}}$ [8]:

$$\overline{\tau_{ij}} - \frac{1}{3}\delta_{ij}\overline{\tau_{kk}} = 2\nu_t \overline{S_{ij}} \qquad (7)$$

"Eq. (2)" can be written as:

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial}{\partial x_j}\left(\overline{u_j u_i}\right) = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(2(\nu + \nu_t)\overline{S_{ij}}\right) \qquad (8)$$

The Smagorinsky model is based on an assumption of mixing length in which it is considered that the viscosity is proportional to the mesh length scale (here associated with the filtering equations, namely the characteristic mesh size) denoted Δ and a velocity scale determined by the product $\Delta\|\overline{S}\|$, where $\|\overline{S}\|$ is the norm of the strain rate defined tensor and resolved by [8]:

$$\left\|\overline{S}\right\| = \sqrt{2\overline{S_{ij}}\,\overline{S_{ij}}} \qquad (9)$$

Finally, writing the Smagorinsky model is as follows:

$$\nu_t = (C_s)^2\left\|\overline{S}\right\| \qquad (10)$$

The constant *Cs* is determined from the assumption of local equilibrium between production and dissipation of turbulent kinetic energy.

B. Numerical Schemes

We use the following calculation:

- The power-law schema: this one is best placed to capture the physical phenomena of heat transfer
- The discretization scheme of pressure velocity coupling is SIMPLE;
- The convergence criterion used is the under-relaxation.

C. Mesh

The influence of the size and the number of the nodes on the solution expressed by the heat transfer to the "heated" active part is illustrated by the temperature, for the configuration 1in the plan medium by Fig. 2.

An irregular distribution "geometrical continuation" of the nods is used to solve more precisely the physical phenomena present in particular in mode of boundary layer characterized by the existence of strong gradients in the parietal zones. Profiles vertical velocity becomes insensitive with the number of nods starting from the grid 150×75×75.

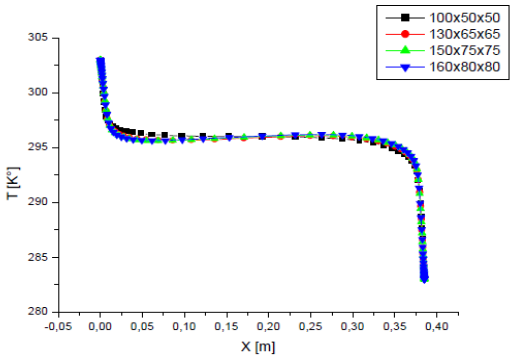

Fig. 2 Convergence of the profile of temperature for C1 in the y=D/2=0.36m plan and z=H/2=1.23m.

D. Validation

In order to verify the accuracy of the numerical results obtained in the present work, a validation of the numerical code was made taking into account some numerical studies available in the literature. Ampofo results [7] obtained in the case of a square cavity containing air, were used to test our simulation by Fluent. The comparison was made by considering the Rayleigh number 1.58 × 10^9. Comparison of velocity profiles along the plane medium shows excellent agreement.

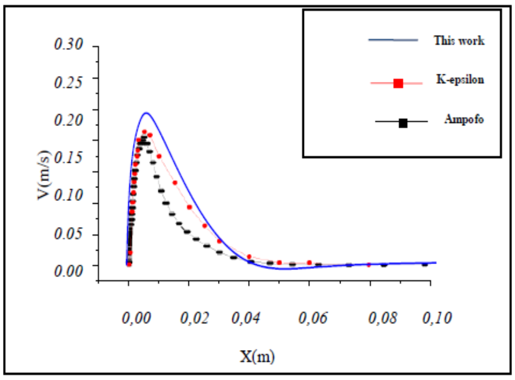

Fig. 3 Comparison of velocity profile V along the y = 0.36m for the configuration C1.

III. Results and Discussion

The results are represented in this work for a number of Rayleigh fixed at 2,5x10^{10} and the variation in temperature ΔT = 20°C, we go fixed the number of Prandtl Pr = 0, 71. The thermal fields are represented in fig. 4 for various configurations as represented in Table. 1

The recovered heat of the hot wall is transported by convection towards the cold wall and upwards, it is what explains the relatively high temperature near to the ceiling. One can notice that the temperature of the air raised in the case of the two hot side walls (configuration C1, C3 and C6), whereas in the other configurations the temperature of the air is less hot, and one notices the presence of the effect of the gradient in vertical temperature expressed by the push of Archimedes in the phenomenon of convection. Indeed the

ISSN 2356-5608

effect of the configuration C1 and C2 tends to amplify the transfer on revenge in the configurations C6, C7 and C8 it is less present.

Fig. 9 represents the profiles of temperature in the plan medium and for a height equalizes 1,23m, we notice that each configuration at a temperature means of the air in the heart of the cavity.

One can notice the thermal boundary layer near to the hot and cold wall with a thermal thickness of boundary layer approximately 0,02m.

The velocity vectors are represented in fig. 5. The flow is mono cellular to priority with the ascending fluid along the hot wall and goes down along the cold wall. However one notes a disturbance as well as the appearance of several zones of recirculation indicating a greater complexity and a higher degree of turbulence.

In fig. 6 we represent the field vertical velocity we clearly observe the rise of the fluid near to the hot wall and descend it near to the cold wall.

Fig. 7 presents the horizontal field velocity we observe near to the floor, the fluid flow of the cold wall towards the hot wall on the other hand near to the floor it is the reverse. The flow is mono cellular, as it appears in fig. 5 in addition it comprises first dominant swirling structure, located at a height varying from one configuration to another.

For fig. 8 we represent the profile vertical velocity. In this graph one can see the dynamic boundary layer near to the wall hot and cold, the thickness of this layer is equal approximately 0,1m.

In this study one saw the influence of the walls (the temperature imposed on each walls table. 1) on the flows for various configurations with high number of Rayleigh, thereafter. We established initially, the phenomenon of interaction of the swirling structures of the flow with the boundary layer.

We notice the effects on the profiles of temperature given in fig. 9 and those velocities fig. 8.

It appears clearly for certain configurations C1, C3 and C6, a strong imbalance of the stratification (not perfectly horizontal).

With the same height Z = H/2 = 1,23m the side faces has a direct influence on the velocity vector respectively with dimensions heat C2 and with dimensions cold C1 and intensity velocity 0,2m/s and 0,3m/s (see fig. 6 C1 and C2).

In fig. 5, we note the appearance of the nodes of recirculation, which one could allot to zones and places of interaction swirling structure/boundary layer.

Indeed, it is possible that it is the place also of release of turbulence.

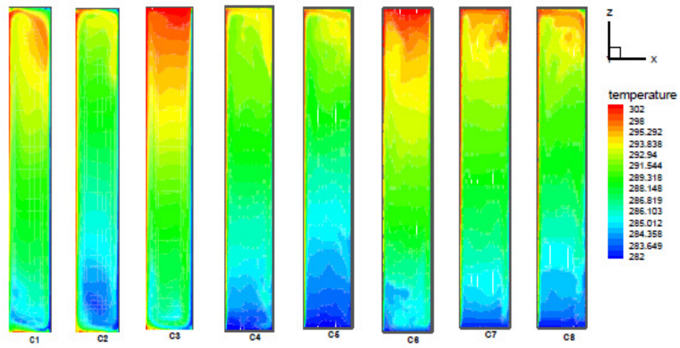

Fig. 4 Thermal fields for the eight configurations in the y=D/2=0.36m plan

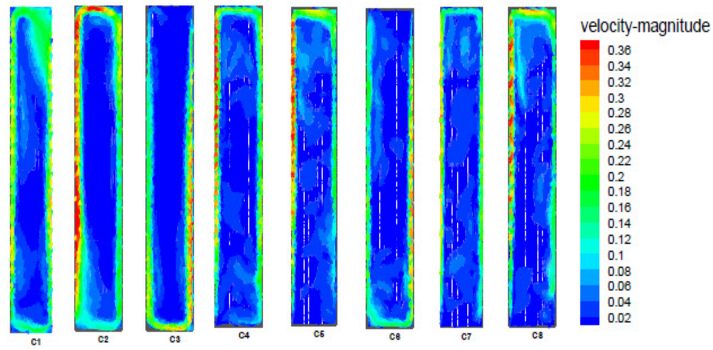

Fig. 5 Velocity vectors (U-W) for the eight configurations in the y=D/2=0.36m plan

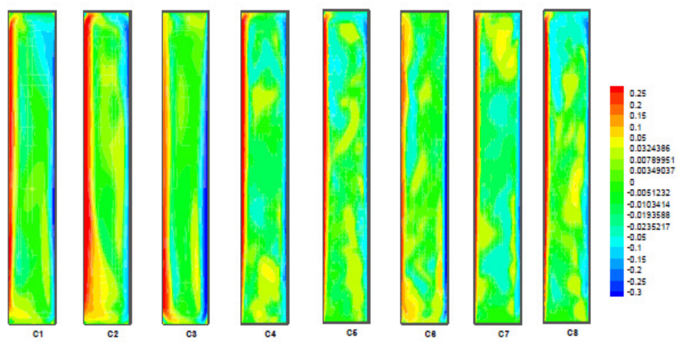

Fig. 6 Fields velocity vertical for the eight configurations in the y=D/2=0.36m plan.

ISSN 2356-5608

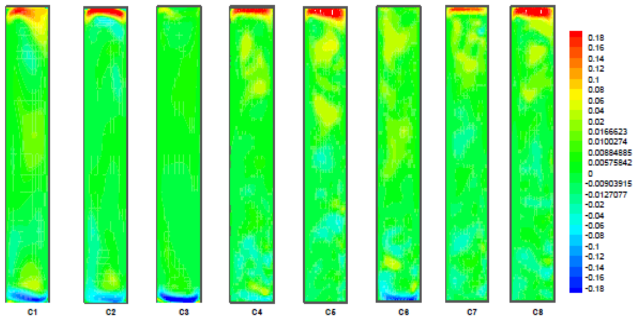

Fig. 7 Fields horizontal velocity for the eight configurations in the plan y=D/2= 0.36m

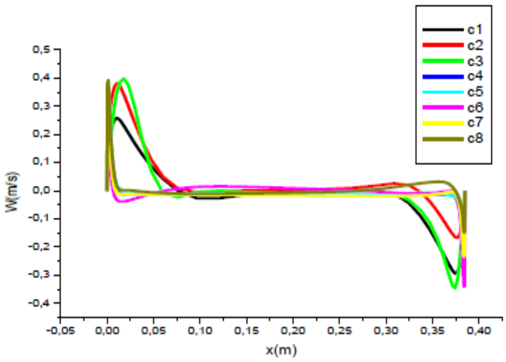

Fig. 8 Profile vertical velocity for the eight configurations in the y=D/2=0.36m plan and z=H/2=1.23m.

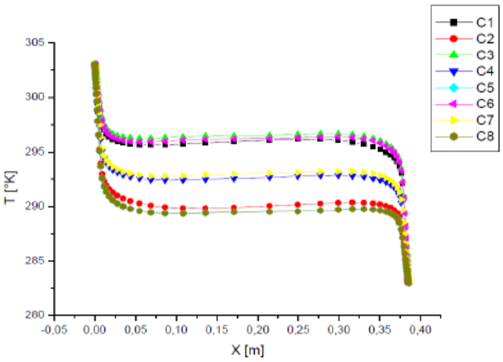

Fig. 9 Profile of temperature for the eight configurations in the y=D/2=0.36m plan and z=H/2=1.23m.

IV. Conclusion

In conclusion, we can confirm through this study the direct influence of side walls on the nature of the air flow in cavity. This influence results in the stratification of the mass of air which changes according to the configurations i.e. boundary conditions. Of other by, we highlighted the influence of number of Rayleigh thus of the ΔT (since the characteristics of the fluid remain unchanged), on the phenomena of convection and consequently on the zone of release of turbulent instability.

Taking into account the coherence of our results with those established numerically and in experiments, by others we succeeded in validating code Fluent the computer for the configurations studied with LES, and finished volumes.

References

[1] S. Mergui, F. Penot, Analyzes velocity and temperature of the air in natural convection in a cavity differentially heated with Ra=1.69 10^9, Int. J. Heat Mass Transfer 40 (14) 3427-3441.1991.

[2] S. Mergui, Experimental characterization of the flows of air of natural and mixed convection in a closed cavity, Thesis Doctorate, University of Poitiers.1993.

[3] J. Salat, F. Penot, Experimental approach of the natural convection of turbulent transition in a cubic cavity differentially heated VI Inter-University conference French-Quebec Thermals of the Systems, (26-28 mai 2003, Quebec).

[4] J. Salat, Contribution to the study of the three-dimensional natural convection in cavity differentially heated. Thesis of Doctorate of the Poitiers University, 2004.

[5] A.M. Lankhorst, D. Angirasa, C.J. Hoogendoorn, LDV measurements of buoyancy induced flows in an enclosure at high Rayleigh numbers, Exp. Thermal and Fluid Science, pp, 74-79.1993.

[6] Y.S. Tian, T.G. Karayiannis, Low turbulence natural convection in an air filled square cavity: Part 1. The Thermal and Fluid Flow Fields, IJHMT 43. Pp, 849-866.2000.

[7] F. Ampofo, T.G. Karayiannis, Experimental benchmark data for turbulent natural convection in an air filled square cavity, IJHMT 46 (19), pp, 3551-3572.2003.

[8] Chatelan, Larg eddy Simulation of the turbulent flows with heat transfers, Thesis Doctorate prepared at the Laboratory of Modeling and Development of Software (CEA Grenoble)

[9] F.X. Trias, A. Gorobets, M. Soria, A. OlivaD. Direct numerical simulation of a differentially heated cavity of aspect ratio 4 with Rayleigh numbers up to 10^{11}: Part I. Numerical methods and time-averaged flow, Int. J. Heat and Mass Transfer 53, pp, 665-673.2010.

[10] P.L. Betts, A.A. Dafa'Alla, Turbulent buoyant air flow in a tall rectangular cavity, UMIST?, Manchester, England.

[11] A. Dafa'Alla, P. Betts, Experimental stady of turbulent natural convection in a tall air cavity, Exp. Heat Transfer 9. Pp, 165-194. 1996.

[12] S.B. Pope, Turbulent Flows Chapter 13: Large-eddy simulation, Cambridge University Press, 2000.

[13] M.M. Lohaasz, P. Rambaud, C. Benocci, Flow features in a fully developed ribbed duct flow as a result of LES, in: ERCOFTAC International Symposium on Engineering Turbulence Modeling and Measurement, ETMM6, Sardinia, Italy, 2005,.

[14] A. Beghidja, Dj. Taloub, Effect of Rayleigh number on the turbulent structures with boundary layer in a differentially heated, In. J. Mechanics and Energy (IJME) Vol. 1, Issue 2, pp, ISSN: 2286-5845.2013.

[15] A. Beghidja, M. Bouafia, J.J. Vullierme, F. Yguel, A. Zammit, Modeling of the exchanges convective on the front verticals of cavities to great numbers of Rayleigh, General review of thermics Tom 34 (405). Pp, 515-526.1995.

[16] F. Ampofo, Turbulent natural convection of air in a non-partitioned or partitioned cavity with differentially heated vertical and conducting horizontal walls, Exp. Thermal and Fluid Science 29. pp,137-157. 2005.

[17] A. Beghidja, H. Gouidmi, R. Benderradji, E. Raouache, Natural convection and turbulent instability in cavity, Wseas Transactions on Heat and Mass Transfer, Issue 2, Vol 1. Pp, 515-526.2006.

[18] G. De Vahl Davis, Laminar natural convection in an enclosed rectangular cavity, In. J. Heat and Mass Transfer 11. pp, 675-693.

[19] S. Xin, P. Le Quéré, Direct numerical simulation of two dimensional chaotic natural convection in a differentially heated cavity of aspect ratio 4, J. Fluid Mech. 304 .1995.

[20] S.B. Pope, Turbulent Flows, Chapter 13: Large-eddy simulation, Cambridge University Press, 2000.

[21] J.H. Ferziger, M. Peric, Computational methods for fluid dynamics, Springer Verlag, 1996.