Finite volume computation of internal turbulent flows promoted by wind action and heat release employing a 3D nonorthogonal collocated grid system

J.F.A. Dias Delgado

Faculdade de Ciências e Tecnologia, Universidade Nova de Lisboa, 2825 Monte de Caparica, Portugal

ABSTRACT

A general numerical method for the solution of the complete 3D averaged Navier-Stokes equations is presented. This method uses nonorthogonal co-ordinates, Cartesian velocity components and a pressure-velocity-coupling algorithm suitable for nonstaggered grid systems. A k-ε two-equation turbulence model is used with this method to calculate the turbulent flow inside a heated space in communication with the outside environment through two small ventilation openings.

INTRODUCTION

Natural ventilation flows inside buildings are promoted by the combined action of thermal buoyancy and external wind. These natural flows present almost the cases complex geometries. For flow calculations with complex geometries body-fitted coordinate system must be used. In this work a strong conservation form of the governing equations is used together with a nonstaggered nonorthogonal grid system. Nonstaggered grid system are preferred because there is only one set of control volumes, which simplifies the code. In order to ensure pressure-velocity-coupling in the nonstaggered grid system, the pressure-weighted interpolation method proposed by Rhie and Chow [5] is used.

Natural flows to be simulated are particularly dependent on the outdoor conditions, namely on wind intensity and direction. For this reason the present rational approach of the problem combines a numerical simulation of the dynamic and thermal governing equations with experimental information on wind pressure distribution at the ventilation openings and pressure loss coefficients of the ventilation openings. This experimental information is used as a suitable boundary conditions to link the internal flow with the outside environmental conditions and to calculate the boundary values of velocity at the ventilation openings.
GOVERNING EQUATIONS

To simulate the turbulent flow the well-known k-ε two-equation turbulence model [3] is used. Using the Cartesian components of vectors and tensors, and expressing the divergence operator in the strong conservation form, the governing equations for the averaged flow can be written in a general nonorthogonal coordinate system \( \xi^j \) as follows:

\[
\frac{\partial}{\partial \xi^j} \left( \rho u_k \beta^j_k \right) = 0
\]  \( \text{(1)} \)

\[
\frac{\partial}{\partial \xi^j} \left( \rho u_k \beta^j_k u_i \right) = \frac{\partial}{\partial \xi^j} \left[ \frac{1}{J} (\mu + \mu_t) S_{ik} \beta^j_k \right] - \frac{\partial}{\partial \xi^j} \left[ (p + \frac{\rho}{2} \rho k) \beta^j_i \right] + J \Delta \rho g_i
\]  \( \text{(2)} \)

\[
\frac{\partial}{\partial \xi^j} \left( \rho u_k \beta^j_k T \right) = \frac{\partial}{\partial \xi^j} \left[ \frac{1}{J} \left( \frac{\mu_t}{\sigma} + \frac{\mu_i}{\sigma_T} \right) \frac{\partial T}{\partial \xi^m} \beta^m_k \beta^j_k \right] + JS_T
\]  \( \text{(3)} \)

\[
\frac{\partial}{\partial \xi^j} \left( \rho u_k \beta^j_k k \right) = \frac{\partial}{\partial \xi^j} \left[ \frac{1}{J} \left( \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial \xi^m} \beta^m_k \beta^j_k \right] + J (G - \rho \varepsilon)
\]  \( \text{(4)} \)

\[
\frac{\partial}{\partial \xi^j} \left( \rho u_k \beta^j_k \varepsilon \right) = \frac{\partial}{\partial \xi^j} \left[ \frac{1}{J} \left( \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial \xi^m} \beta^m_k \beta^j_k \right] + J \left( \frac{C_1}{k} G - C_2 \frac{\rho \varepsilon^2}{k} \right)
\]  \( \text{(5)} \)

\[
\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}
\]  \( \text{(6)} \)

\[
\rho = \frac{p}{RT}
\]  \( \text{(7)} \)

with

\[
S_{ik} = \frac{1}{J} \left[ \left( \frac{\partial u_i}{\partial \xi^m} \beta^m_k + \frac{\partial u_k}{\partial \xi^m} \beta^m_i \right) - \frac{2}{3} \delta_{ik} \frac{\partial u_p}{\partial \xi^m} \beta^m_p \right]
\]  \( \text{(8)} \)

\[
G = \frac{\mu_t}{J^2} \left( \frac{\partial u_i}{\partial \xi^m} \beta^m_k + \frac{\partial u_k}{\partial \xi^m} \beta^m_i \right) \frac{\partial u_k}{\partial \xi^n} \beta^n_i - \frac{2}{3} \delta_{ik} \left( \frac{\mu_t}{J^2} \frac{\partial u_p}{\partial \xi^m} \beta^m_p + \frac{1}{\rho k} \right) \frac{\partial u_k}{\partial \xi^n} \beta^n_i
\]  \( \text{(9)} \)

In these equations

\[
\beta^j_k = J \frac{\partial \xi^j}{\partial x_k}
\]  \( \text{(10)} \)

represents the cofactor of the kth row and jth column in the Jacobian matrix (J) of the coordinate transformation \( x_j = x_j(\xi^k) \), where \( x_j \) is the reference Cartesian co-ordinate system, and J is the Jacobian of the transformation. In the form presented, momentum equations were subtracted by the static pressure equation to show more clearly the nature of the buoyant term, and pressure
p = (P - P₀) and Δρ = (ρ - ρ₀) are referred to the outdoor conditions (P₀, ρ₀). For the empirical constants of the k-ε model (C₁, C₂, σ_k, σ_ε) were assigned the standard values proposed by Launder and Spalding [3]. For the Prandtl Schmidt number σ_T a value of 0.9 was used.

DISCRETIZATION PROCEDURE

The finite difference form of the governing equations is obtained using a finite volume method based on a nonstaggered grid system adapted to a general nonorthogonal mesh. These finite control volumes are bounded by six faces defined by the coordinates of their eight vertices and all variables are stored at their geometrical center (see Figure 1). It is assumed that in the computational domain Δξ₁ = Δξ₂ = Δξ₃ = 1, consequently in the physical domain the volume of these control elements is equal to δV = JΔξ₁Δξ₂Δξ₃ = J.

![Figure 1: Control volumes: (a) physical domain; (b) computational domain.](image)

For discretization purposes equations (1-5) are cast in the following general transport equation

\[
\frac{\partial}{\partial \xi_j} (ρu_k β^j_k φ) = \frac{\partial}{\partial \xi_j} \left[ J \frac{Γ_φ}{J} \frac{\partial φ}{\partial \xi_m} β^m_k β^j_k \right] + JS_φ
\]

Taking into account the Gauss divergence theorem, integration of this equation over the control volumes results in the following equation for the mean fluxes through the cell faces

\[
I_e - I_w + I_n - I_s + I_d - I_u = S
\]
where $I_i$ stand for the total flux of $\phi$ that passes through the cell face normal to the constant $\xi^i$ coordinate and $S$ is the linearized averaged source term. The total flux of $\phi$ consists of two contributions: the convection term $I_i^C$ and the diffusion term $I_i^D$, i.e., $I_i = I_i^C + I_i^D$.

The diffusion terms $I_i^D$ are evaluated using central differences discretization, considering the east control volume face

$$I_e^D = \left( \frac{\Gamma_e}{\delta V} \right) (\phi_E - \phi_P) (\beta_k^1 \beta_k^L) e + \left( \frac{\Gamma_e}{\delta V} \right) \left[ (\phi_n - \phi_s) e (\beta_k^2 \beta_k^L) e + (\phi_d - \phi_u) e (\beta_k^3 \beta_k^L) e \right] \quad (13)$$

The first term on the right hand side of this equation comes from the normal derivatives in the diffusion term and is computed implicitly. The second term comes from cross derivatives and results from nonorthogonality of the grid lines. It is computed explicitly by linear interpolation of the adjacent nodal values, i.e., $(\phi_n - \phi_s) e = f_1 (\phi_n - \phi_s)_E + (1 - f_1) (\phi_n - \phi_s)_P$. In this equation $f_1$ is a linear interpolation factor defined as $f_1 = \Delta x_{pe} / \left( \Delta x_{pe} + \Delta x_{ce} \right)$, where $\Delta x_{pe}$ and $\Delta x_{ce}$ are the length of the straight line connecting the face center $e$ to the P and E node points, respectively. The values of $\phi_{n,s}$ are also obtained by linear interpolation of the adjacent nodal values considering the equivalent interpolation factors $f_2$ and $f_3$ in the remainder directions.

The convective terms $I_i^C$ are evaluated at the control volume faces which lie between the point P and any surrounding point, considering the east control volume face

$$I_e^C = F_e \phi_e = \rho_e \left( u_1 \beta_1^L + u_2 \beta_2^L + u_3 \beta_3^L \right) e \phi_e \quad (14)$$

The cell face values of the convected variable $\phi_e$ are obtained implicitly using a QUICK based convection discretization scheme for velocity components and the hybrid central/upwind convection discretization scheme for the remainder variables (details of this procedure can be found in [1]). The cell face mass flux $F_e$ are evaluated explicitly using an interpolation method suitable for nonstaggered mesh that will be presented in the next section.

The convective and diffusive fluxes through the additional cell faces of the three dimensional grid used are computed similarly and the resulting discretized transport equation is written in the following linearized form

$$a_p \phi_p = \sum a_{nb} \phi_{nb} + S \quad (15)$$

where the summation extends over the neighbor grid points of P, the coefficients $a_{nb}$ are related to the convection and diffusion contributions and $S$ contains all other terms which are not computed implicitly through the nodal values of $\phi$, such as cross derivatives, pressure gradient and buoyant terms in momentum equations.
PRESSURE VELOCITY COUPLING

In nonstaggered grids the use of linear interpolation to estimate cell face velocities conduces to the decoupling between pressure and velocities, leading to spurious pressure oscillations. To avoid this problem, linear interpolation for pressure and momentum interpolation for velocities has successfully been proposed by Rhie and Chow [5]. To avoid dependence on the underrelaxation factors used in momentum equation a corrected version of Rhie and Chow method proposed by Majumdar [4] is used in the present work to interpolate the velocity components at the cell faces. Therefore, considering the east face, velocity components are interpolated as follows

\[ u_{e} = \alpha \left( \sum a_{n}^{u} u_{inb} + S' - (p_{n} - p_{s}) \beta_{i}^{2} - (p_{d} - p_{u}) \beta_{i}^{3} \right) \left( p_{E} - p_{P} \right) + (1 - \alpha) u_{e}^{old} \]

(16)

where \( \alpha \) is the underrelaxation factor for velocity components and \( S' \) is the source term of the discretized momentum equation which does not include the pressure terms that are written explicitly. The overbar denotes a linear interpolated value between grid nodes P and E. With this formulation, cell face velocity depends on the pressure at the two neighboring nodes as in the staggered practice and coupling between pressure and velocity is ensured.

To compute the pressure field (which does not have an explicit differential equation to be discretized) the SIMPLEx algorithm proposed by Kobayashi and Pereira [2] is used. This algorithm is a modified version of the well-known SIMPLE algorithm suitable to guarantee mass conservation in nonstaggered formulations. It consists of a predictor step and a corrector step for calculation of velocity and pressure fields. In the first step, with the velocity and pressure fields guess or calculated in the previous iteration, the coefficients of momentum equation are calculated to obtain a new velocity field. In the second step, are calculated the new values of mass fluxes through cell faces, using equation (16), and the coefficients of a pressure-correction equation that is solved to obtain the pressure-correction field. This pressure-correction field is then used to update velocity and pressure fields, and the fluxes through cell faces. This process is repeated until convergence is achieved.

BOUNDARY CONDITIONS

The internal flows to be simulated are in communication with the outside environment through small ventilation openings and are particularly dependent on wind intensity and direction. For economical reasons the calculation domain of the flow does not extend outside the internal space. Wind pressure outside the ventilation openings is given as a boundary condition, outside the boundary of the calculation domain, making

\[ p_{out} = C_{p} \frac{1}{2} \rho_{o} u_{o}^{2} \]

(17)
where \( u_o \) is the reference wind velocity and \( C_p \) is the local wind pressure coefficient. Velocity at the ventilation openings \( u_v \) is considered normal to the openings surface and is calculated at the boundary of the computational domain by the pressure loss expression

\[
(p_{\text{out}} - p_{\text{int}}) = \zeta \frac{1}{2} \rho_v u_v |u_v|
\]

(18)

where \( \zeta \) is the pressure loss coefficient of the opening. This expression links the outdoor wind pressure conditions \( p_{\text{out}} \) with the internal pressure conditions just inside the calculation domain \( p_{\text{int}} \).

At the walls the generalized log law, described in detail by Launder and Spalding [3], is employed as an artificial boundary condition that connect the wall shear stress to the variables just outside the viscous sub layer. For the temperature equation, the global heat transfer coefficient of the wall, \( U \), is used to estimate the heat flux through the wall. This heat flux is introduced as a source term for the control volumes adjacent to the wall

\[
S_T = -U(T - T_o)A_w / c_p
\]

(19)

where \( A_w \) is the area of the cell face in contact with the wall and \( c_p \) is the specific heat at constant pressure of the air. The heat power dissipated by the heating plate is introduced in temperature equation in a similar way

\[
S_T = \dot{q}A_w / c_p
\]

(20)

where \( \dot{q} \) is the value of the heat power dissipation per unity of area.

Pressure at the boundaries of the calculation domain is established by extrapolation from interior values with the condition of vanishing normal derivatives there, unless for the upper and lower horizontal walls, were due to thermal buoyancy dominance of the flow, it was assumed a linear variation of pressure. For the pressure-correction variable of the SIMPLES algorithm it was assumed that their normal derivatives vanishes on all boundaries of the computation domain.

Figure 2: Geometry of the flow and wind pressure distribution.
PROBLEM DEFINITION AND RESULTS PRESENTATION

The flow configuration adopted in the present work is presented schematically in Figure 2. It consists of an internal space with a square floor of 25x25 cm², two opposed rectangular walls with 6.5 cm height and a ridge-roof with a slope of 50%. This internal space communicates with the outside environment by two ventilation openings made on the symmetry plane of the space, one with 4.3x4.0 cm² on one of the rectangular walls and the other with 5.0x4.0 cm² on the roof. The pressure loss coefficient of the openings ζ was made equal to 2.5. Also in the symmetry plane, near the floor, a vertical heating plate with 3.6x6.0 cm² was placed. The walls of this internal space were considered made of Perspex plates with 1.0 cm thick with a global heat transfer coefficient U equal to 3.875 w/m²°C.

The wind pressure coefficients $C_p$ outside the ventilation openings made on the wall and on the roof were considered equal to 0.72 and -0.48, respectively. These values were taken from the wind pressure distribution around the symmetry plane of a model with this geometry presented in Figure 2. These results were obtained by the author with the model deeply immersed in a turbulent boundary layer with a power law $u / u_\infty = (x_2 / x_{2\infty})^{1/2.75}$ and are referred to the wind velocity at the height of the corner of the rectangular wall and roof.

An algebraic grid generation was used in order to allocate the grid spacing in the physical domain. The boundary points at the upper and lower walls were selected and connected by straight lines. The grid generated is repeated for each $x_3$ constant plane. A mesh with 22x22x16 grid nodes was used. Figure 3 shows the nonorthogonal grid used. Due to symmetry only one half of the flow field was calculated.

![Figure 3: Nonorthogonal mesh used (22x22x16 grid nodes)](image)

Numerical simulation was carried out for different reference wind velocities, maintaining the heat power dissipated by the heating plate constant and equal
Advanced Computational Methods in Heat Transfer

\[ u_0 = 0.0 \text{ m/s} \]
\( (U_v = 0.18 \text{ m/s}) \)

\[ u_0 = 1.0 \text{ m/s} \]
\( (U_v = 0.51 \text{ m/s}) \)

\[ u_0 = 3.0 \text{ m/s} \]
\( (U_v = 1.42 \text{ m/s}) \)

Figure 4: Results of computations, velocity fields.
Figure 5: Results of computations, temperature fields.
to 50 w. In Figure 4 are presented the velocity fields and in Figure 5 the temperature fields simulated for reference velocities $u_o$, respectively equal to 0.0, 1.0 and 3.0 m/s. These results are presented for three vertical parallel planes: the first one near the symmetry plane; the second one at half distance between the symmetry plane and the opposed wall; and the third one near the opposed wall. Temperature fields are presented in the nondimensional form $(T - T_o) / (T_v - T_o)$, where $T_v$ is the mean temperature at the outlet opening. In Figure 4 are also presented the mean values of velocity at the entrance opening $U_v$ and in Figure 5 the values of $(T_v - T_o)$ obtained for each simulation.

These results show clearly the influence of the increasing value of wind action on the flow patterns inside the heated space. The increase of the ventilation rate promoted by wind action progressively bends the thermal plume formed above the heating plate, until the situation for which the flow is no more deflected by buoyancy effects, and takes with it directly to the outside environment the major part of the heat released by the heating plate.

CONCLUSIONS

A computational procedure based on the finite volume method using nonstaggered, nonorthogonal grids was presented. This computational method was successfully used with a k-ε two-equation turbulence model to predict the flow promoted by thermal buoyancy and wind action inside a heated space with a complex geometry.

We acknowledge the financial support for the realization of this work provided by JNICT contract PIBC/C/CEG/1374/92.

REFERENCES