Revue des Energies Renouvelables Vol. 14 N°1 (2011) 85 – 94
DOI: https://doi.org/10.54966/jreen.v14i1.243

Effect of urban morphology on wind flow distribution in dense urban areas

Saddok Houda[1*], Noureddine Zemmouri[2†], Rachid Athmani[3‡] and Rafik Belarbi[4§]

[1] University Mohamed Khider, Department of Civil Engineering, B.P. 145, Biskra, Algeria
[2] University Mohamed Khider, Department of architecture, B.P. 112 El Allia Nour, Biskra, Algeria
[3] University Mohamed Khider, Department of Mechanical Engineering, B.P. 145, Biskra, Algeria
[4] University of La Rochelle, Department of Civil Engineering, 23 Av. Albert Einstein, 17071 la Rochelle cedex 9, France

(reçu le 10 Janvier 2011 – accepté le 28 Mars 2011)

Abstract - *Environmental flow modelling can be achieved either by using wind-tunnel studies or via the utilization of Computational Fluid Dynamics (CFD) techniques. Conducting wind-tunnel studies may be expensive and time consuming, particularly in the event of additional test requirements with modified building and/or environmental configurations. On the other hand, CFD can provide significant cost benefits for assessing and optimizing engineering design solutions related to environmental concerns and appear attractive as a potential alternative tool. By using CFD in this manner it is anticipated that these types of advanced performance-based studies will be a useful tool and essential aid for urban designers and environmental planners. This paper discusses ongoing development and application of CFD simulations through a case study using CFD software for simulating air flow around a specific vernacular urban fabric of Ghardaïa. The potential of CFD for prediction of wind speeds around a complicated urban environment and a complex fabric structures is investigated. As the 3D solid model of the geometry was not available a captured Google Earth image of the fabric structure was used as a backdrop in the AC3D software, supplied as a utility to Phoenics. The outlines of the buildings were then traced to create polygons, which were then extruded to produce the individual buildings. Finally, the entire scene was exported to the VR-Editor as a single Phoenics object.*

Résumé - *La modélisation des écoulements dans l'environnement extérieur autour des bâtiments soit par la voie expérimentale en soufflerie ou par l'utilisation de la mécanique des fluides numériques (Computational Fluid Dynamics, 'CFD'). Les expérimentations en soufflerie peuvent êtres longues et coûteuses, en particulier dans le cas des configurations urbaines complexes où il est souvent nécessaire de faire de nombreux essais de modifications de paramètres et configuration de l'environnement. D'autre part, la CFD est rentable du point de vue financier quant à l'évaluation et l'optimisation des systèmes liés au développement durable à l'échelle urbaine et ou architecturale. La CFD apparaît donc comme un outil efficace d'aide à la décision pour l'architecte et l'urbaniste. Cet article décrit une procédure de modélisation, de simulation et d'évaluation des écoulements pour des configurations urbaines particulières en termes de formes, d'aspect*

[*] saddok@gmx.fr
[†] zemmouri@fullbrightmail.org
[‡] rachatman@yahoo.fr
[§] rafik.belarbi@univ-lr.fr

et de morphologie. Le choix de la ville de Ghardaïa, comme exemple d'étude, est pour ce cas précis, amplement significatif et démonstratif.

Keywords: CFD - Complex urban fabrics - Buildings - CAD - Flow - Model.

1. INTRODUCTION

There is a need to properly develop the application of Computational Fluid Dynamics 'CFD' methods in support of air quality studies. CFD models are emerging as a promising technology for such assessments, in part due to the advancing power of computational hardware and software [Neophytou, 2005]. CFD, simulations have the potential to yield more accurate solutions than other methodologies because it is a solution of the fundamental physics equations and includes the effects of detailed three-dimensional geometry and local environmental conditions.

The results of CFD simulations can both be directly used to better understand specific case studies as well as be used to support the development of better-simplified algorithms that may be generally applied [Jal, 2003].

The city of Ghardaïa, in the M'Zab valley, was taken as the case study with its dense fabric which demonstrates the ideas of building cool, shaded, airy structures in a hot dry climate.

This specific built environment with its complex and organic geometry will help to demonstrate the ability of CFD techniques to tackle the most complicated situations.

2. MATHEMATICAL MODEL

2.1 Physical model

The Phoenics 3.5 CFD program was used in this research [Cham Phoenics, 2005]. The turbulence model was the modified $k-\varepsilon$ model by MMK. The standard $k-\varepsilon$ model is known to yield unsatisfactory predictions when applied to flow around bluff bodies as encountered for example in wind-engineering applications. The model fails to reproduce well the surface pressure distribution around a body because of the tendency to overestimate turbulent production in the impingement region on the frontal area of the body [Tsuchiya *et al*., 1997].

All the objects that will be included in the simulation must be prepared in a step-by-step procedure starting with the capture of the area of investigation using Google Earth 4.2 Professional [Google Earth, 2007]. The image is then imported in the AC3D software where the outlines of the buildings were traced and then extruded to produce the individual buildings [AC3D, 2009]. The dimensions and the geometry of the objects are scaled to fit the real size of the configuration.

It is well known that the CAD files created or generated by CAD packages do not necessarily guarantee that the facets are consistent with each other in respect of inward- and outward-looking direction or define closed volumes.

Phoenics requires that facets should have a direction sense in order that it can know on which side is the fluid and on which the solid; and of course facets which share an edge should be in agreement on the matter. In most cases there a need to fix and repair the overlapping surfaces in order to simulate complicated configurations.

In the present case, Magics 12.0 software [Materialise Magics, 2007] has been used to correct the file containing the urban geometry of the case study as shown in figures 3 and 4.

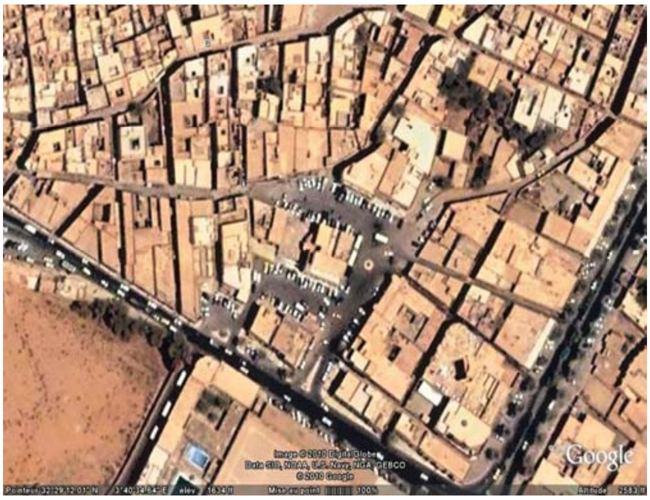

Fig. 1: Ghardaia complexe urban morphology

Fig. 2: Tracing the complex urban morphology using AC3D

2.2 Numerical simulation

The governing equations of incompressible turbulent wind flow around buildings are continuity and the Reynolds-averaged Navier - Stokes equations, expressed as follows:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial U_i U_j}{\partial x_i} = -\frac{\partial P}{\partial x_i} + B_i + \rho \sum_j \left[\nu_L \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_j} \right) - \overline{u_i u_j} \right] \tag{2}$$

Fig. 3: Defective 3D urban configuration tracing result using AC3D

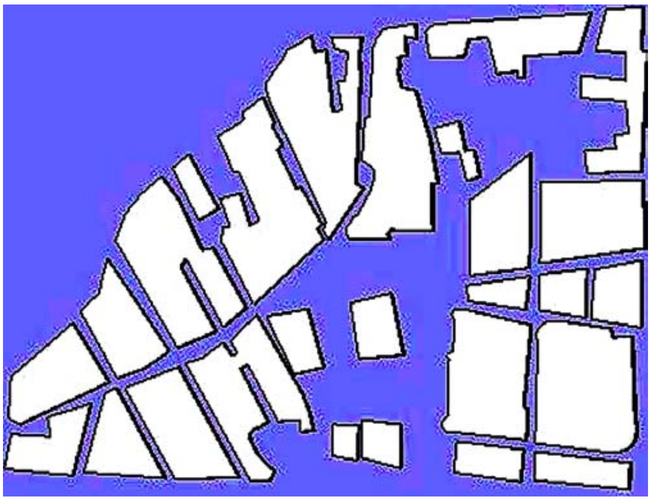

Fig. 4: Repaired 3D urban configuration tracing result using Magics

2.2.1 Turbulence model

The turbulent stresses appeared in equation (2), use the eddy-viscosity concept to determine the Reynolds stresses from:

$$-\rho\times\overline{u_i\,u_j} = \mu_\tau\times\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_j}\right)-\rho\times K\frac{2d_{ij}}{3} \tag{3}$$

where

$$\mu_r = C\times\rho\times V_s\times L_s \tag{4}$$

and where C is an empirical constant, and V_s and L_s are turbulence velocity and length scales which characterize the Large-Scale turbulent motion.

A reasonable value of the kinematic eddy viscosity ν_t can be estimated by taking $C = 0.01$ and V_s as a typical mean-flow velocity and the length scale L_s as ~10 % of the flow width:

$$\nu_t = \frac{\mu_\tau}{\rho} \tag{5}$$

The standard high-Re form of the $k - \varepsilon$ model employs the following turbulence transport equations:

$$\rho \frac{\partial}{\partial x_i}\left[U_i K - \frac{\nu_t}{Pr_K}\frac{\partial K}{\partial x_i} \right] = \rho\left(P_K + \Gamma_b - \varepsilon\right) \tag{6}$$

$$\rho \frac{\partial}{\partial x_i}\left[U_i \varepsilon - \frac{\nu_t}{Pr_\varepsilon}\frac{\partial \varepsilon}{\partial x_i} \right] = \rho \times \frac{\varepsilon}{K}\left(C_{1e} P_K + C_{3e}\Gamma_b - C_{2e}\varepsilon\right) \tag{7}$$

The kinematic turbulent (or eddy) viscosity is given by:

$$\nu_t = C_\mu \times C_d \times \frac{K^2}{\varepsilon} \tag{8}$$

The model constants are: $C_\mu = 0.5478$, $C_d = 0.1643$, $Pr_K = 1.0$, $Pr_{\varepsilon=K} = 1.314$, $C_{1e} = 1.44$, $C_{2e} = 1.92$ and $C_{3e} = 1.0$.

The buoyancy production Γ_b is < 0 for stably-stratified layers, so that K is reduced and turbulence damped. For unstably-stratified layers, Γ_b is positive and turbulence is augmented.

The constant C_{3e} has been found to depend on the flow situation. It should be close to zero for stably-stratified flow, and close to 1.0 for unstably-stratified flows. The default value of C_{3e} is 1.0.

As it has been stated before the MMK $k - \varepsilon$ model adopted here for computational efficiency and accuracy. In fact the MMK $k - \varepsilon$ model is derived from KL $k - \varepsilon$ model which a modified $k - \varepsilon$ standard model.

The KL $k - \varepsilon$ model differs from the standard model only in that the volumetric production rate of turbulence energy is calculated from the ad-hoc replacement:

$$P_K = \nu_t \times S \times \Omega \tag{9}$$

Rather than

$$P_K = \nu_t \times S^2 \tag{10}$$

Where S and Ω are, respectively, strain and vorticity parameters defined by:

$$S^2 = \frac{U_{i,j} + U_{j,i}}{2} \tag{11}$$

$$\Omega^2 = \frac{U_{i,j} - U_{j,i}}{2} \tag{12}$$

The excessive k levels in impingement regions are produced by S. Now, in simple shear flows the modification has little effect because Ω and S are essentially equal, whereas in stagnation regions W is nearly zero so that turbulence production is much reduced by the modification

The MMK $k-\varepsilon$ model differs from the standard model in that the eddy viscosity ν_t is computed from:

$$\nu_t = F_\Omega \times C_\mu \times C_d \times \frac{K^2}{\varepsilon} \tag{13}$$

where the multiplier F_Ω is calculated from:

$$F_\Omega = \min\left(1.0, \frac{\Omega}{S}\right) \tag{14}$$

So that the standard model is recovered whenever $\frac{\Omega}{S} > 1$.

3. GEOMETRY AND SOLUTION DOMAIN

A Cartesian coordinate system has been employed, and the urban geometry and solution mesh have been created using Phoenics-VR. The size of the domain has been taken as a multiple of the characteristic height of the tallest building which is in this case 18 meters, as recommended by Hall [Hall, 1996] who suggests that the distance between any edge of the domain and the buildings must be at least five times of the characteristic height of the building.

The domain covers the entire area of 350 m × 250 m, including all the buildings and surrounding areas. The height of 50 m from the ground in the vertical direction of the calculation domain provides about 30 m of open space above the tallest building. The mesh contained 100 subdivisions in the x direction, 80 in y direction and 10 in z direction, resulting in 80.000 cells in a Cartesian coordinate system.

4. BOUNDARY CONDITIONS

An inflow condition was applied at the leftmost (upwind) side (y-z plane) of the domain as given in Figure 5. An inlet wind speed of 25 m/s has been chosen as representing the average wind conditions in the region. In addition the orientation of the urban configuration has been selected according to the prevailing wind direction.

A pressure outlet condition was applied at the rightmost (downwind) side (y-z plane) of the domain.

5. SOLUTION OF THE EQUATIONS

In Phoenics the pressure-velocity coupling in the incompressible flow is resolved through the Simplest algorithm that is a variant of the well known Simple algorithm of Patankar and Spalding [Patankar, 1972].

The convective and diffusive fluxes are combined by the Hybric scheme, employing a staggered grid. The volume integrated algebraic equations of velocity components are solved by Jacobi-Point by Point method, whereas those of other variables are solved by the whole-field solution procedure.

The whole-field solution procedure is an extension of Thomas algorithm to 3-D equations.

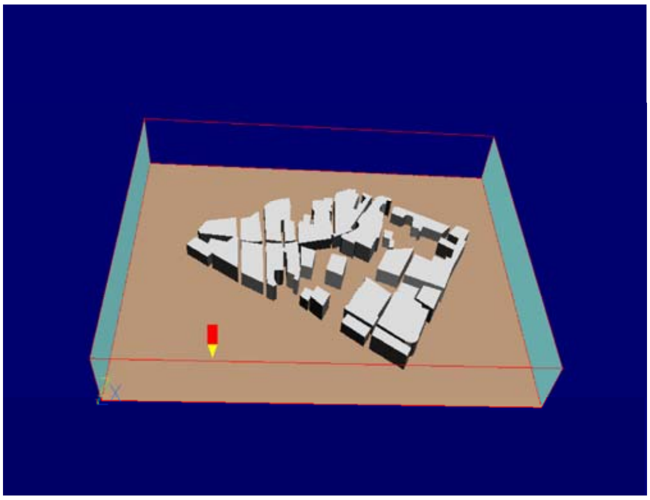

Fig. 5: Setup of the 3D urban configuration for simulation under Phoenics

6. RESULTS AND DISCUSSION

Images were generated to show various views of the velocity fields. The horizontal distribution of airflow at the height of 8 m in the sample area, whose size is 87500 meters square, is shown in Figure 5.

At this scale, the shape and arrangement of individual buildings affect the formation of the field of wind.

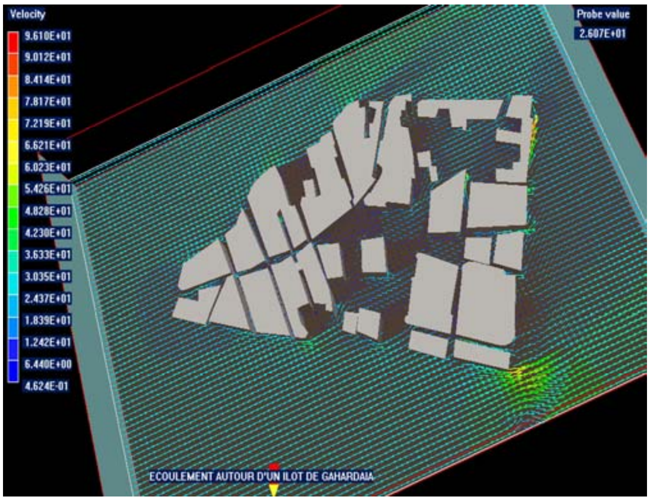

Fig. 6: The horizontal distribution of airflow at the height of 8 m in the sample area

It is seen from Figures 6, 7, 8 and 9 that the velocity of air flux varies drastically with urban density. In the present case the urban configuration acts as a wind shield reducing the air velocity inside the urban fabric, whereas the maximum velocities are observed at the boundaries of the lot.

In large open spaces among buildings, although the wind direction is complicated, the wind speed is relatively low. In addition it is clear that dense and organic forms appear to create a better sheltered outdoor environment.

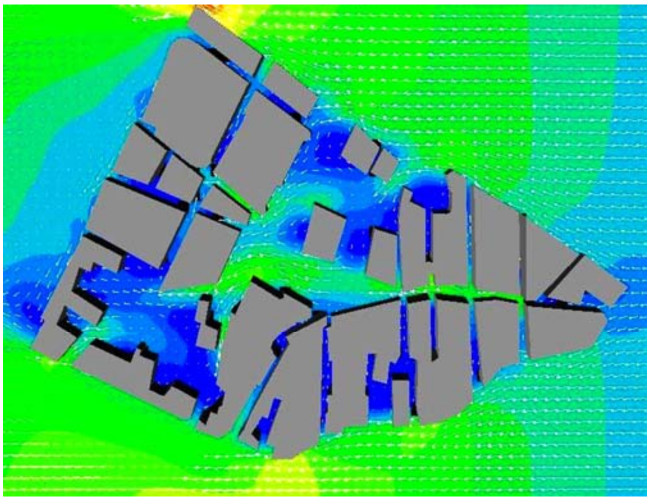

Fig. 7: The horizontal distribution of airflow at the height of 8 m in the sample area using contours

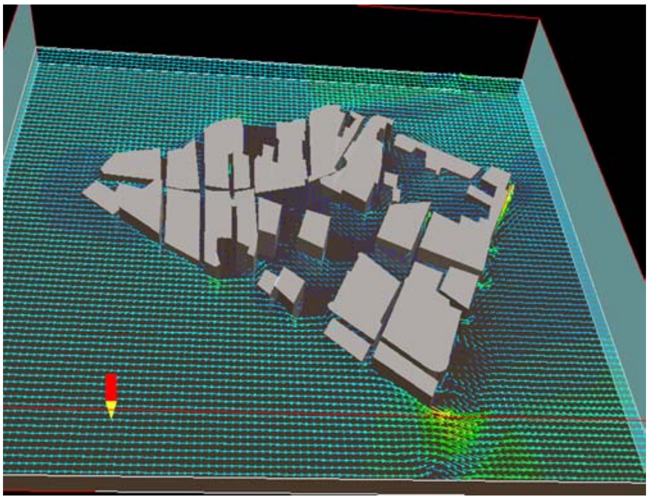

Fig. 8: 3D view of the horizontal distribution of airflow at the height of 8 m in the sample area

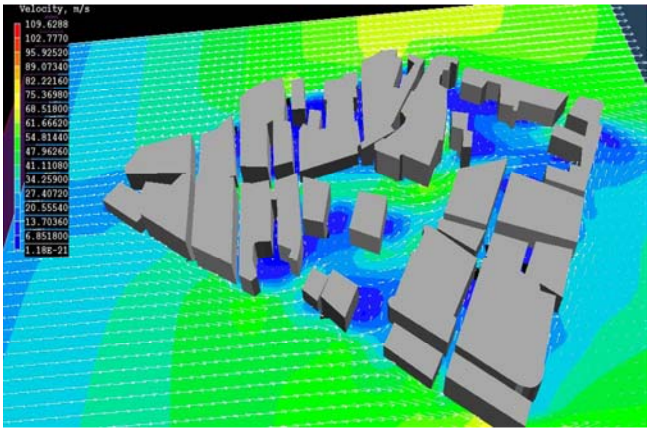

Fig. 9: Scaled 3D view of the horizontal distribution of airflow at the height of 8 m in the sample area using contours

7. CONCLUSION

A numerical study of velocity field in a densely vernacular urban fabric has been undertaken and discussed here. It was found that the velocity distribution can considerably change with the building shape and urban configuration.

By using computational fluid dynamics (CFD) useful information can be drawn and used by urban developers, architectural designers and environmental planners to promote natural ventilation, a good measure for reducing energy use in buildings and providing better outdoor air quality.

It has been also demonstrated the ability of combining different tools from picture capture, image correction to CFD simulation to assess a natural phenomena.

REFERENCES

[1] M. Neophytou and R. Britter, '*Modelling the Wind Flow in Complex Urban Topographies: A Computational-Fluid-Dynamics Simulation of the Central London Area*', Proceedings of the 5th GRACM International Congress on Computational Mechanics, Limassol, Chypre, 2005.

[2] E.N. Jal, '*Applying CFD to Environmental Flows*', Report of Connell Wagner Pty Ltd, Australia, 2003.

[3] Software, '*Cham Phoenics, Version 3.5.1*', Cham Ltd., London, UK, 2005.

[4] M. Tsuchiya, S. Murakami, A. Mochida, K. Kondo and Y. Ishida, '*Development of a New* $k-\varepsilon$ *Model for Flow and Pressure Fields Around Bluff Body*', Journal of Wind Engineering and Industrial Aerodynamics, Vol. 67-68, pp. 169 - 182, 1997.

[5] Software, '*Google Earth Version 4.2*', 2007.

[6] 3D Design Software, '*AC3D 6.5.28*' copyright © 2009 Inivis ltd.

[7] Software, '*Materialise Magics Version 12.0.1.2*', 2007.

[8] R.C. Hall, '*Evaluation of Modelling Uncertainty – CFD Modelling of Near-Field Atmospheric Dispersion*', Project EMU, Final Report, WS Atkins Consultants Ltd, UK, 1996.

[9] S. Patankar and D. Spalding, '*A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows*', International Journal of Heat and Mass Transfer, Vol. 15, N°10, pp. 1787 – 1806, 1972.