

**Fida Zair**  
PhD student  
Moulay Ismail University  
Faculty of Sciences  
Morocco

**Mhamed Mouqallid**

Professor  
Moulay Ismail University  
Ensam Meknes  
Morocco

**El houssine Chatri**

Professor  
Moulay Ismail University  
Faculty of Sciences  
Morocco

**Rabii El Maani**

Assistant Professor  
Moulay Ismail University  
Ensam Meknes  
Morocco

# The Effect of the Presence of Obstacles on the Emission Dispersion Emited by a Bent Chimney

*It is proposed to study numerically the dispersion of polluting emissions in turbulent conditions around a two-dimensional obstacle. The effect of the presence of obstacles on the flow from a bent chimney is mainly addressed. A numerical simulation of the dispersion of pollutants emitted by the chimneys was performed using the simulation code. The numerical method used to solve the equations describing the flow is the finite volume method (FVM) and the mesh size adopted is non-uniform, very tight near the stack and obstacle. The results found essentially show that the presence of obstacles modifies the flow.*

**Keywords:** Dispersion; Pollutant; Bent chimney; Computational Fluid Dynamics; Finite volume.

## 1. INTRODUCTION

The study of pollutant dispersion is very complex and requires, first of all, a good understanding of the behaviour of a flow in the presence of obstacles [1].

In this context, Mirzai and Al [2] experimentally studied the structure of a flow around an obstacle. They showed that the deposition and transport of the pollutant depends on the shape and orientation of the building as well as the nature of the boundary layer.

Poitras and Al [3] studied a numerical simulation of fluid flows around a building model using a vortex method. For this study, the fluid is considered viscous and incompressible, the two-dimensional and unsteady flow and the Reynolds number used for the simulations is 400, for two geometries ( $b/h=1$  and  $b/h=2$  such that  $b$  is the length of the obstacle, and  $h$  is the height of the obstacle). They found that no significant difference was observed between these two flows except for the length of the recirculation zone downstream of the obstacle.

Hervé and Al [4] studied the experimental characterization of flow and dispersion around a two-dimensional obstacle. The objective of this thesis is a fine experimental characterization of flow and turbulent dispersion around an obstacle placed in a surface boundary layer, in order to evaluate the validity of RANS models for their application to the study of atmospheric dispersion.

Initially, they used hot wire anemometry, laser Doppler anemometry and particle image anemometry techniques to determine the velocity field in a rough surface boundary layer and around a two-dimensional obstacle of square cross-section.

Marvoidis and Al [5] experimentally studied the dispersion of the pollutant around an isolated obstacle. The purpose of this study is to vary the shape of the

building (cube, cylinder and a large obstacle) and its orientation in relation to the wind direction.

They showed that pollutant concentrations are affected by the shape of the obstacle and the lateral location of the source relative to it.

Yakhot and Al [6] studied with direct numerical simulation (DNS) the turbulent flow around a cube placed on the bottom of a channel. The results found confirm the unsteadiness of the flow caused by the unstable interaction of a horseshoe vortex upstream of the cube and on these lateral sides. They showed in this work that the negative production of turbulence is expected in the area upstream of the obstacle where the horseshoe vortex begins to form mainly.

Zhang and Al [7] carried out flow velocity measurements for a jet obliquely discharged into a crossflow in a duct. The flow measurements were made with optical fiber laser Doppler velocimetry. The obtained velocity data revealed that the inclined jet is effective in generating longitudinal vortices in the crossflow.

Hwang and Yang [8] conducted a numerical study of vortex structures around a cube in a channel. The objective of this work is to study the characteristics of the coherent vortex structures produced by the presence of the obstacle, including horseshoe vortex systems upstream of the obstacle, as well as lateral vortices in the vicinity of the two lateral faces of the cube. They found in this project that the approach of the flow towards the obstacle prevents the gradient of adverse pressure that produces a three-dimensional separation of the boundary layer, which leads to the formation of horseshoe vortices. They also found that with the increase in Reynolds' number, the horseshoe vortex structure becomes complex and the number of eddies increases in pairs.

Huptas and Elsner [9] worked on stationary and unsteady flow around two square obstacles and presented numerical results on flow around a square obstacle in the first case and around two square obstacles in the second case using the FLUENT. For the first resolution, they showed that the thickness of the

Received: December 2019, Accepted: July 2020

Correspondence to: Fida Zair, PhD student

Moulay Ismail University,

Faculty of Sciences, Morocco

E-mail: fidae.zair@gmail.com

doi:10.5937/fme2004882Z

© Faculty of Mechanical Engineering, Belgrade. All rights reserved

boundary layer can influence the shear layer near the wall. And for the second resolution, they found that the union of the two vortex zones downstream of the first cube and upstream of the second makes the flow more turbulent.

Guo and Al [10] also used the LES to simulate the behavior of the film cooling flows; they also examined the effect of the inclination angle but they added the impact of the blowing ratio on the cooling performance and then on the cooling efficiency. They concluded that bigger blowing ratios and injection angles worsen it.

Fillippini and Al [11] studied the flow around the cubes placed on a channel using LES (large Eddy simulation). The selective structure model was used for the determination of turbulent viscosity. The flow around these geometries has very complex phenomena such as horseshoe-swirl vortices and recirculation regions. The main objective of this study is to identify the flow around the cubes in a channel for a Reynolds number equal to 22000. The results obtained are in accordance with the experiment both qualitatively and quantitatively. Among the results found, they showed that with the increase in the ratio  $S/H$ , such that  $S$  is the distance between the cubes, and  $H$  is the cube side; the average drag coefficient increases during the second cube while it is about constant for the first cube.

Jiang and Al [12] conducted work on turbulent flow around a square cylinder placed near a solid wall. The study consists in comparing the simulation and experimentation results for the configurations:  $S/D=1$  (periodic case) and  $S/D=0.25$  (stationary case) such that  $S$  is the distance between the cylinder and the solid wall,  $D$  is the diameter of the cylinder. The turbulence intensity is about 1. The Reynolds number is in the range of 10000 to 100000 and they found that vortex formation is related to the separation of flow at the leading edge that gives rise to shear stresses on each of the lateral surfaces of the cylinder, and they showed that if the cylinder is placed near the wall; the vortex detachment can be completely eliminated.

Hallek and Al [13] studied a two-dimensional numerical simulation of a turbulent flow around two cavities. Their purpose is to study the interaction of a boundary layer with two cavities and to characterize the dynamic structure of the flow. In the case of a single large cavity, the structure ensures the existence of a large recirculation where there are two vortices born; one small at the foot of the first step and the other larger one that covers the entire cavity before escaping after reattaching. In the second case, this phenomenon was blocked by the existence of the obstacle and the creation of a new cavity. The study of the profiles speeds, in different sections of the area shows that the profile of entry is more laminar.

Gera and Al [14] have studied with CFD (Computational Fluid Dynamics) the unsteady 2D flow around a square obstacle. The simulation was carried out for a flow around a square cylinder in order to analyse the wake behaviour. The Reynolds number ( $Re$ ) considered in the range 50-250 so that flow is laminar. The main objectives of this were to capture the features of flows past a square cylinder in a domain with the use of CFD. Finite volume method has been used with

lagged grid arrangement. The incompressible SIMPLE algorithm was used for the velocity pressure coupling. The second order discretisation was used both for space and time. A high resolution grid has been used to avoid spurious oscillations and to keep the errors within limits. The lift coefficient and velocity component in the wake region were monitored for calculation of Strouhal number. The variation of Strouhal number with Reynolds number was found from the analysis.

Liu and Al [15] carried out a numerical study of a flow around a square cylinder. This study focuses on the admission surface of the domain. To overcome such a situation problem, he proposed that the condition of the intake surface should be variable over time. For In addition, the intensity of the turbulence will be considered upon admission of the domain as a boundary condition. It has noted as a result that the forces acting on the cylinder are directly related to the flow structure and vortex formation at the leading edge. These wormholes act on top of each other and remain attached with the obstacle.

Omidyeganeh and Al [16] conducted a Numerical simulation of the wind flow around cube in channel. A code has been developed in C++ to solve the Navier-Stokes and continuity equations using large eddy simulation for the turbulent flow. The finite volume formulation applied on the staggered grid arrangement for a structured mesh system. Smagorinsky-Lilly sub-grid scale (SGS) model was used which can be substituted with other models easily. The discretisation scheme for all terms except the convection terms, which adopts a TVD scheme, was the central difference scheme. The PISO algorithm had found to be faster than the SIMPLE algorithm for all cases and problems. The laminar flow approaching a cube in the atmosphere boundary layer and in channel was calculated and compared with experiments of turbulent flow. The laminar inflow boundary condition significantly changes the flow pattern around the obstacle. Also the mostly needed modifications had been proposed and estimated.

Ankur and Al [17] carried out work on wind flow around a square plate. The ADINA computer software using the finite element method was used for the simulation. The turbulence model  $k-\epsilon$  was used. In this project, he increased the Reynolds number to reduce the viscosity from 0.01 to 0.0001 N.s/m<sup>2</sup> in three different models. This helps to study the effect of non-linearity and the various measures that are required for the solution to converge, and he found that these techniques used by the ADINA software allow the downstream flow of the plate to be evaluated. This analysis can be applied during the design phase to improve the aerodynamic structure and reduce forces. Among the results obtained, he found that with the increase in the Reynolds number, the vortexes downstream of the plate are increased in parallel.

Damien and Al [18] worked on a laminar flow around a square cylinder. The flow in regime established around a square cylinder placed on a plate plane is examined by visualizations of the measurements by velocimetry by imaging of particles (PIV) and numerical simulations for a Reynolds number

of 1000. They have observed in this work that the existence of a vortex horseshoe, the presence of vortices vertical and horizontal axes show the complexity of this flow. In the region upstream, the fluid hits the profile and also tries to bypass it by passing through these sides or over it. On the region above the cube, the flow results from the interaction of two flows, the fluid that is diverted by the upstream side above the profile is accelerated and takes off over the width of the cube. The calculations show them a stationary system of four eddies upstream of the more confined obstacle. This leads to the formation of eddies vertical drop above the cube larger than by experience.

Cheng and Al [19] simulated a linear shear flow incompressible two-dimensional above a square tube. They showed the effect of the shear rate  $\tau$  on the frequency of vortex detachment from the cylinder. The results obtained show that the vortex behind the cylinder is highly dependent on the shear rate and Reynolds number. For a number  $Re = 50$ , the effect of a small number  $\tau$  causes an alternating vortex separation followed by an intensity uneven, whereas for a  $Re > 50$  and a high value of  $\tau$ , removes the disbond of the cylinder vortex. Differences in the strength and size of vortices on the sides the upper and lower parts of the cylinder become more pronounced as they progress that the number  $\tau$  increases.

Alan and Huber [20] studied in a wind tunnel the influence of the width and orientation of obstacles in relation to the wind. It examined the profiles of concentrations in the wake of buildings. The width-to-height ratios of the latter vary from 2 to 22 for orientation angles between -30 degrees and + 60 degrees, the height of the chimney is equal to 1.5 times the height of the building. The dispersion of smoke from a chimney placed near an obstacle is maximum for a width/height ratio equal to 10.

Bhourri and Al [21] experimentally studied the dispersion of pollutants from a stack. The two measurement techniques adopted are PIV, which allows the determination of velocity and vorticity fields, and laser tomography, which allows the visualization of the evolution of the feather along a long distance in a vertical or horizontal plane. The experimental parameters are the size, diameter of the stack and velocity ratio.

Bhourri and Al [22] present in their numerical study the flow structure from a stack bent around a parallelepiped. The particle image velocimetry (PIV) technique was applied to generate an experimental image. In this study, the influence of the obstacle's angle of attack on the flow structure is studied. The results show that this parameter affects the dynamics and mass characteristics.

When the obstacle is normal to the flow coming in the opposite direction, the recirculation area is indeed large and a higher deposition of pollutants is observed. In fact, the inclination of the building is effective in reducing the concentration of pollutants on the facades and on the ground.

Bhourri and Al [23] are studying the flow from a bent stack over a downstream obstacle, in which they have tried to evaluate the effect of several parameters on the

resulting flow characteristics such as velocity ratio, obstacle spacing, obstacle arrangement and geometry. They showed that wind speed affects pollutant concentration and the geometry of the building causes changes in plume behaviour.

In our case, it is a numerical simulation of the dispersion of pollutant emissions from a bent stack in the presence of obstacles. During this work, we propose to study the influence of the presence of obstacles on the dispersion phenomenon.

## 2. THEORETICAL FORMULATION OF THE PROBLEM

A bent chimney of height  $h$  and diameter  $d$  that emits a chemically inert mixture of air and smoke at the ejection rate  $U_0$  and temperature  $T_0$ , in the presence of a parallelepipedally shaped obstacle placed downstream, is considered. This plume is subjected to a wind speed of  $U_\infty$ , the ambient temperature is  $T_\infty$ . (Figure 1)

The resulting flow is assumed to be two-dimensional, turbulent and stationary on average. It thus responds to Navier Stokes' equations which, discretized with Favre's decomposition, are written as follows:

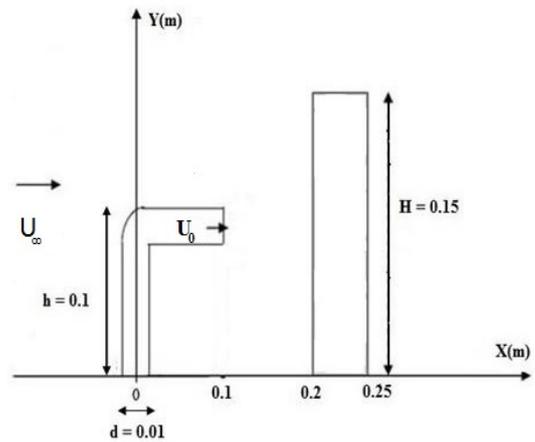


Figure 1. Geometry diagram

Continuity equation:

$$\frac{\partial \rho U_i}{\partial x_i} = 0. \quad (1)$$

Momentum equations:

$$\frac{\partial}{\partial t}(\rho U_i) + \frac{\partial(\rho U_j U_i)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i. \quad (2)$$

Energy equation:

$$\frac{\partial}{\partial t} \left[ \rho \left( h - \frac{P}{\rho} + \frac{1}{2} U_i U_i \right) \right] + \frac{\partial}{\partial x_i} \left[ \rho U_j \left( h + \frac{1}{2} U_i U_i \right) \right] = \frac{\partial U_i \tau_{ij}}{\partial x_j} - \frac{\partial q_j}{\partial x_j}. \quad (3)$$

where:

$e$ : Internal energy.

$h = e + \frac{P}{\rho}$ : The specific enthalpy and  $\tau_{ij}$  is the tensor

of viscous stresses.

$q_j$  : The heat flow calculated in general from Fourier's

$$\text{law } q_j = -\lambda \frac{\partial T}{\partial x_j}, \text{ where } \lambda \text{ is the thermal conductivity}$$

of the fluid and T its temperature.

Closing the equation system requires the use of a turbulence model. In our work we used first-order turbulence models: k-ε Standard, k-ε Realizable, k-ε RNG, k-ω SST and k-ω Standard, and second-order RSM.

### 3. NUMERICAL METHOD

In this study, the mesh size adopted (Figure 2) is non-uniform, very tight near the chimney and obstacle.

The field of study is composed of a bent chimney that injects a pollutant that interacts with a transversal flow composed of ambient air. The chimney is of diameter d, height h and admits an elbow of length l. A parallelepipedic obstacle of height H is placed downstream of the source.

Moreover, we propose to introduce a temperature gradient between the interacting flows  $\Delta T = 100 \text{ K}$  and the velocity ratio is equal to  $R = 1.6$ . The calculation range is wide enough so that the boundaries of the range do not disturb the flow.

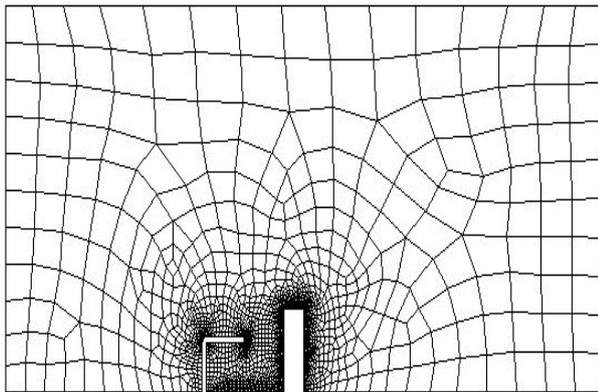


Figure 2: Mesh distribution for a bent chimney in the presence of an obstacle

### 4. RESULTS AND DISCUSSIONS

Figure 3 gives the field of velocity vectors which shows both the acceleration of the flow near the obstacle and the appearance of recirculation zones before and after the obstacle.

Figure 4, which presents the contour of the temperature, displays the evolution of pollutants around the obstacle. It is noted that the building has a great influence on the diffusion of contaminants ejected through the chimney.

We present here the results relating to the evolution of the longitudinal velocity and the vertical velocity in comparison with the experimental results obtained by Bhouri's PIV [23].

Figures 5-10 below show our simulation results using the five turbulence models (k - ε Standard, k - ε Realizable, k - ε RNG, k - ω Standard, k - ω SST and RSM) against the results of I. Baouab and Al [23].

The first position corresponds to the upstream of the obstacle  $X = 0.14 \text{ m}$ , the second is placed on the roof of

the building  $X = 0.21 \text{ m}$  and the last is located downstream of the obstacle  $X = 0.29 \text{ m}$ .

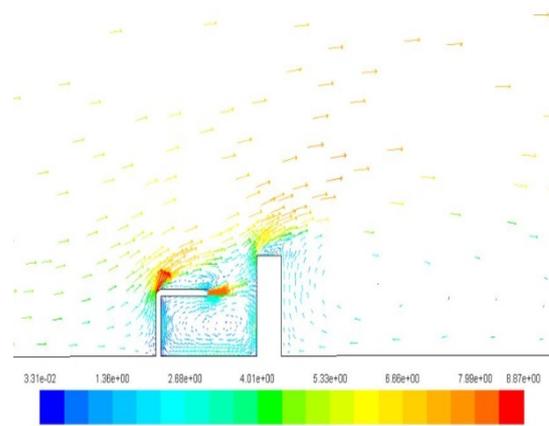


Figure 3: Field of velocity vectors

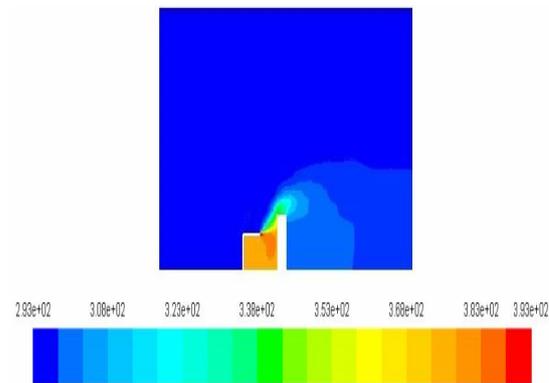


Figure 4: Contour of the temperature

#### 4.1 Evolution of the longitudinal Velocity and of the vertical velocity

Observes that in addition to the satisfaction of the other models (k-ε Standard, k-ε Realizable, k-ε RNG and RSM ), both models k-ω SST and k-ω Standard give a good agreement.

It can be seen that the variation in longitudinal and transverse velocity is non-regular and depends almost entirely on the presence of the obstacle, which has an effect on the flow. It can also be noted that the presence of an obstacle considerably modifies the evolution of the speed downstream and upstream of the obstacle.

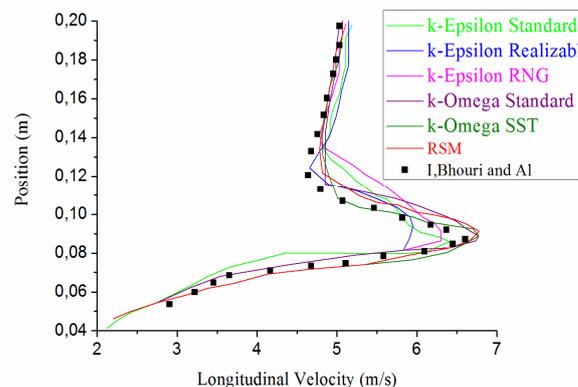


Figure 5: Evolution of the longitudinal velocity in the area of the domain  $X = 0.14 \text{ m}$

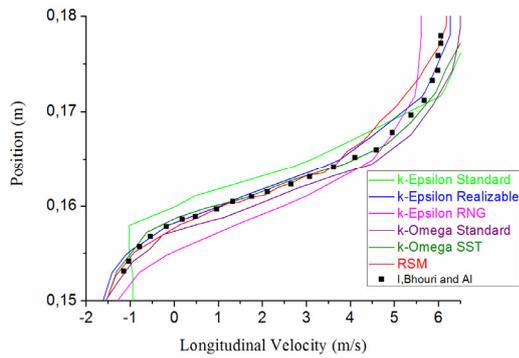


Figure 6: Evolution of the longitudinal velocity in the area of the domain  $X = 0.21$  m

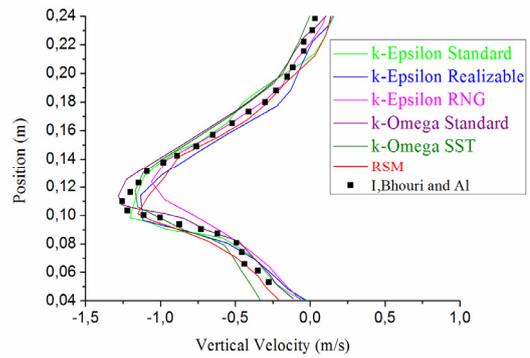


Figure 10: Evolution of the vertical velocity in the area of the domain  $X = 0.29$  m

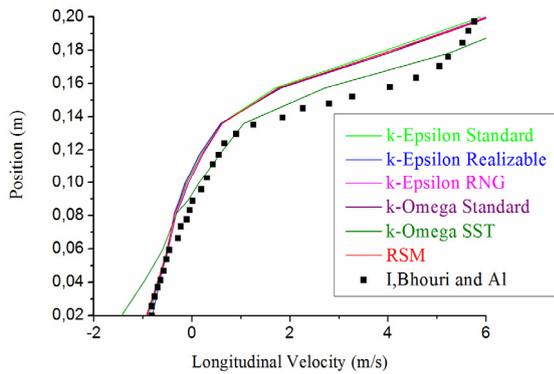


Figure 7: Evolution of the longitudinal velocity in the area of the domain  $X = 0.29$  m

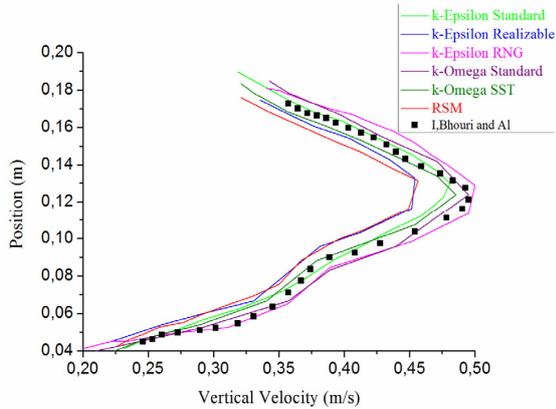


Figure 8: Evolution of the vertical velocity in the area of the domain  $X = 0.14$  m

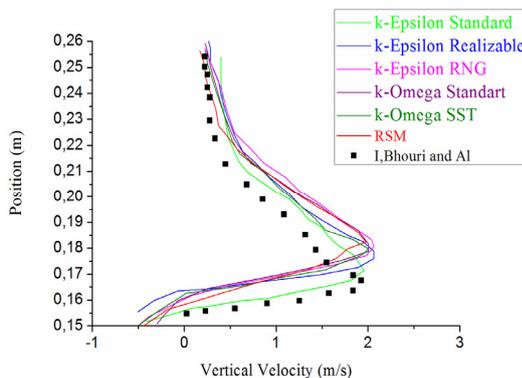


Figure 9: Evolution of the vertical velocity in the area of the domain  $X = 0.21$  m

#### 4.2 Evolution of Reynolds stress

Figures 11 and 12 represent the variation of Reynolds stress for the position  $X = 0.13$  m (between the chimney and the obstacle) using in this part the RSM model of the simulation. According to the results found, we have a good agreement between our results and those of I. Baouab and Al [23].

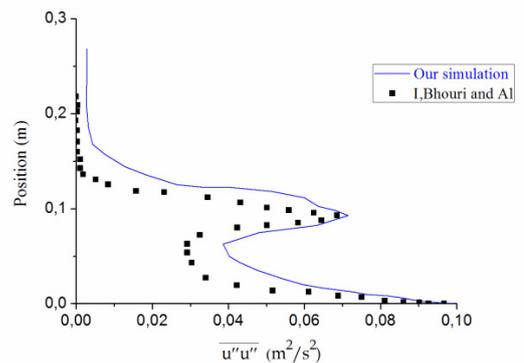


Figure 11: Evolution longitudinal of stress for position  $X = 0.13$  m

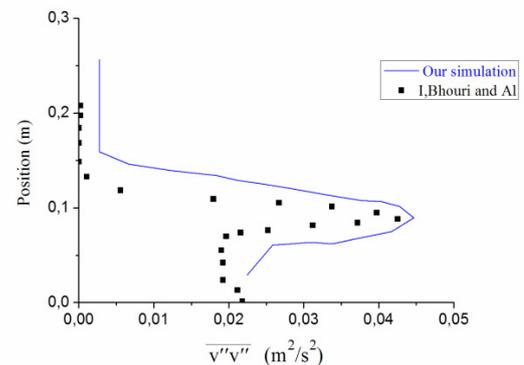


Figure 12: Evolution vertical of stress for position  $X = 0.13$  m

#### 4.3 Comparison between a bent chimney and a straight chimney

In this part we study the variation of the longitudinal velocity for different positions in both cases of configurations.

The figure 13 shows the configuration of straight chimney. And the figures 14-16 show that the

dispersion of the pollutant differs from one configuration to another, which means that the straight chimney geometry is the most suitable because it promotes further dispersion.

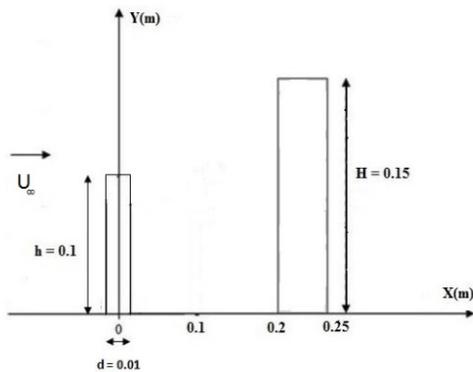


Figure 13: Geometry diagram

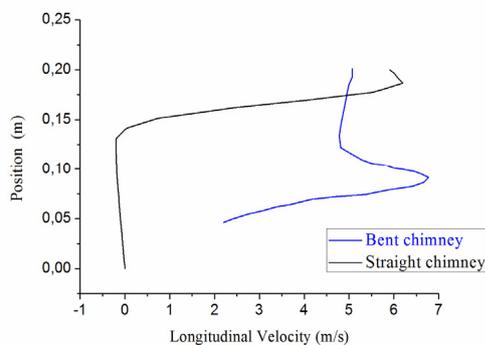


Figure 14: Evolution of the longitudinal velocity in the area of the domain  $X = 0.14$  m

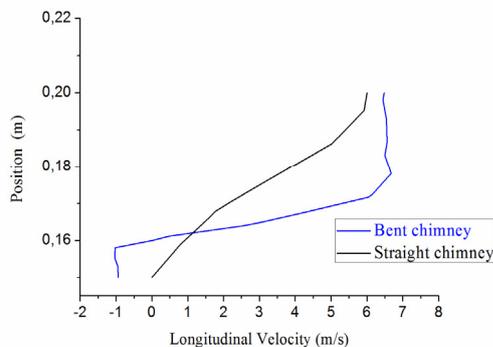


Figure 15: Evolution of the longitudinal velocity in the area of the domain  $X = 0.21$  m

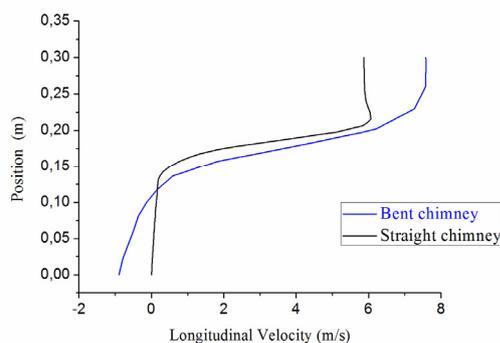


Figure 16: Evolution of the longitudinal velocity in the area of the domain  $X = 0.29$  m

## 5. CONCLUSION

In this work, we presented a numerical study of the dispersion of stack emissions from bent chimneys in turbulent conditions around a two-dimensional obstacle downstream under different zones within a constant temperature crossflow.

This type of study determines the influence of the presence of obstacles on the flow emanating from a stack. First, we started by validating the experimental speed data with our numerical simulation based on the finite volume and a good agreement was revealed.

Then, we worked on the two turbulence models  $k - \omega$  Standard and  $k - \omega$  SST in which we found that the evolution of the longitudinal and transverse velocity for the different positions given gives a better congruence, too.

The results found essentially show that the presence of an obstacle downstream modifies the dispersion of pollutants as well as the appearance of vortex zones downstream and upstream of the obstacle.

The comparison between the two geometries shows that the shape is a very influential factor on the dispersion ejected from the chimney: The straight configuration is the most preferable because it favours the dispersion of additional pollutants in the air, which justifies the choice of the straight chimney by the majority of industrial plants.

## REFERENCES

- [1] Bouterra, M. and Al.: Simulation Numérique Bidimensionnelle d'un Ecoulement Turbulent Stratifié autour d'un Obstacle, International Journal of Thermal Sciences, Vol. 41, pp. 281 – 293, 2002.
- [2] Mirzai, H. M. and Al.: Wind tunnel investigation of dispersion of pollutants due to wind flow around a small building, Atmospheric Environment, Vol. 28, pp.1819-1826, 1994.
- [3] Poitras, G. J. and Al.: Etude de l'écoulement de fluides autour de modèles de bâtiments, PhD thesis, université de Moncton, 1998.
- [4] Hervé, G. and Al.: Caractérisation expérimentale de l'écoulement et de la dispersion autour d'un obstacle bidimensionnel, PhD thesis, Université de Lyon, 2015.
- [5] Marvoidis, I. and Al.: Fields and wind tunnel investigation of plume dispersion around single surface obstacles, Atmospheric Environment, 37, pp. 2903-2918, 2003.
- [6] Yakhot, A., Liu, H. and Nikitin, N.: Turbulent flow around a wallmounted cube: A direct numerical simulation, International journal of fluid flow, 27, pp.994-1009, 2006.
- [7] Zhang, X.: Turbulence measurements of an inclined rectangular jet embedded in a turbulent boundary layer, International Journal of Heat and Fluid Flow, 21, pp. 291-296, 2000.
- [8] Hwang, J. And Yang, K.: Numerical study of vertical structures around a wall-mounted cubic obstacle in channel flo, Vol. 16, No. 7, 2010.

- [9] Huptas, M. And Elsner, W.: Steady and unsteady: simulation of flow structure of two surface-mounted square obstacles, Task quarterly 12 No. 3, pp.197-207.
- [10] Guo, X., Shröder, W. and Meinke, M.: Large eddy simulations of film cooling flows, Journal computers and fluids, 2005.
- [11] Filippini, G., Franck, G., Nigro, N.: Large Eddy Simulations of the flow around a square cylinder, Mecanica Computacional, Vol. 24 A.Larreteguy (Editor) Buenos Aires, Argentina, 2005.
- [12] Jiang, Y. and Al.: Using large eddy simulation to study airflows in and around buildings, ASHRAE Transactions, 2003.
- [13] Hallek, K., Al.: Simulation numérique bidimensionnelle d'un écoulement turbulent autour de deux cavités, Revue des Energies Renouvelables, Vol. 10, No. 4, 2007.
- [14] Gera, B., Pavan K.Sharma and Singh, R. K.: CFD analysis of 2D unsteady flow around asquare cylinder, Intrenanational Journal of Applied Engineering Research , DINDIGUL, Vol. 1, No. 3, 2010.
- [15] Liu, Z.: Square cylinder Large Eddy Simulation based on random inlet boundary condition, Journal of Applied Fluid Mechanics, Vol. 3, No. 1, pp. 35-45, 2010.
- [16] Omidganeh, M. and Abedi, J.: Numerical simulation of the wind flow around cube in channel, BBAA VI International Colloquium on Bluff Bodies Aerodynamics-Applications Milano, Italy,juillet 20-24-2008
- [17] Bajoria, A.: Analysing wind flow around the square plate using ADINA, Massachusetts Institute of Technology-May 2008.
- [18] Damien, C., Laurent, D., Sébastien, R. and Pierre, J. : Laminaire autour d'un cylindre carré .comparaison calcul experience, Nancy, 3-7 Septembre 2001.
- [19] Cheng, M., Whyte, D. S. and Lou, J.: Numerical simulation of flow around a square cylinder in uniform-shear flow, Journal of Fluids and Structures, Vol. 23, pp. 207– 226, 2007.
- [20] Huber, A.: The influence of building width and orientation on plume dispersion in the wake of building, Atmospheric Environment, Vol. 23, pp. 2109-2116, 1989.
- [21] Bournout, Ph.: Experimental study of the plume emitted by a smokestack, Proceeding of PSFVIP, 2003.
- [22] Baouab, I. B., Bournout, Ph., Mahjoub, S. N., Mihiri, H. and Le palec, G.: Dispersion of a bent chimney fume around a variably oriented building, Mechanical Engineering Science, Vol. 225, pp. 843-852, 2010.
- [23] Baouab, I. B., Mahjoub, S. N., Mihiri, H., Bournout, Ph. and Le palec, G.: Dynamic and mass transfer characteristics of the flow issued from a bent chimney around buildings, Heat Mass Transfer, Vol. 49, pp. 337-358, 2013.

## NOMENCLATURE

$x, y, z$	Cartesian coordinates
$P$	Pressure
$T$	Temperature
$U, V$	Velocity components along $x$ and $y$
$R$	Velocity ratio
$u_i$	Velocity components along the $i$
$\frac{u_i u_j}{\rho}$	Reynolds stress
$d$	Chimney diameter
$h$	Chimney height
$H$	Obstacle height
$e$	specific internal energy
$h$	specific enthalpy
$k$	kinetic energy of turbulence
$q_j$	heat flow calculated
$t$	time

## Symboles grecs

$\rho$	Density
$\varepsilon$	Dissipation rate of the turbulent kinetic energy
$\omega$	specific rate of dissipation.

## УТИЦАЈ ПРИСУСТВА ПРЕПРЕКА НА ДИСПЕРЗИЈУ ЕМИСИЈЕ ИЗ САВИЈЕНОГ ДИМЊАКА

**Ф. Заир, М. Мукалид, Ел Х. Шатри, Р. Ел Мани**

Предлаже се нумеричко истраживање дисперзије загађујућих емисија у турбулентним условима око дводимезионалне препреке. Рад се претежно бави утицајем присуства препрека на проток загађујуће емисије из савијеног димњака. Нумеричка симулација дисперзије полутаната коју емитују димњаци изведена је помоћу симулационог кода. Једначине којима се описује проток полутаната су решаване методом коначних запремина, док су окца мрежице била неједнаке величине, веома згуснута у близини нагомиланих честица и препреке. Резултати показују да се проток емисије модификује присуством препрека.