NUMERICAL PREDICTION OF MIXED TURBULENT AIRFLOW WITHIN A CAVITY USING VARIOUS TURBULENCE MODELS.

S. LEPERS [1], P. AUDE [2], P. DEPECKER [2], G. GUARRACINO [1]

[1] Laboratoire des Sciences de l'Habitat, DGCB (URA CNRS 1652), Ecole Nationale des Travaux Publics de l'Etat, Rue Maurice Audin, 69 518 Vaulx-en-Velin Cedex FRANCE
stephane.lepers@entpe.fr
[2] Centre de thermique de Lyon, Equipe Thermique du Bâtiment CETHIL-TB (UPRES.A CNRS 5008), Bât. 307, Institut National des Sciences Appliquées de Lyon, 20 Ave A Einstein, 69621 Villeurbanne, France

ABSTRACT

The overall objective of the present work is to evaluate the performance of three turbulence models, with a view to predicting characteristics of airflow within an anisothermic cavity. The standard K-ε model, the Renormalisation Group model (RNG) and the Reynolds stress model (RSM) are used in conjunction with the Fluent code. The accuracy and the relative performance of the three models are evaluated by comparing their numerical results with experimentally obtained data. This comparison is made for the constant flow rate of a two-dimensional turbulent mixed convective airflow. The results obtained are examined by comparing the parameters of velocity, turbulence energy and temperature. The general agreement between the models and the experimental results is quite good although significant differences were observed in some flow zones.

KEYWORDS

Anisothermal, Cavity, K-ε model, Mixed Convection, RNG, RSM, Simulation, Turbulence.

INTRODUCTION

Our study is set in the general context of air quality in areas of great size. Indeed, air quality and the transport of pollutants in large enclosed spaces have become of major concern, particularly in relation to sensitive industrial sites, areas relating to public health (hospitals, operating theatres), as well as in more commonly frequented locations (schools, homes) and buildings in general. Computer tools adapted to solving different building-related problems are widely used at present. The problems which we have considered require the prediction of flow characteristics within enclosed spaces (cavities) by taking into account particular physical phenomena related to the design of buildings - namely anisothermic cavities of great volume and low air velocities. Today, it is commonplace for such problems to be solved by the use of simulation techniques that have been derived from research into fluid mechanics. The objective of this work is to carry out a validation of CFD (Computational Fluid Dynamics) codes that were not originally designed for the treatment of low Reynolds number flows. For this purpose, we were interested in three different turbulence models: the standard K-ε model (K-ε), the Renormalisation Group K-ε model (RNG) and the Reynolds stress model (RSM).

EXPERIMENTAL CELL

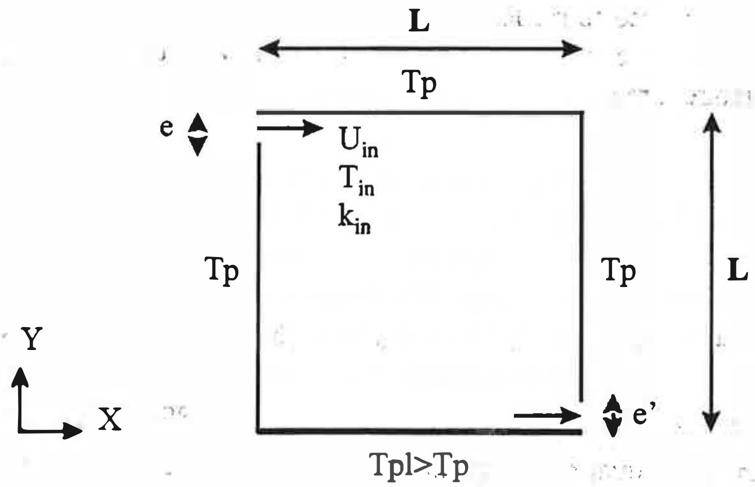

Fig. 1. Diagram of the cell under investigation.

The modelled experimental cell is managed by Blay and al. (1992, 1996) in the laboratory LET (Laboratoire d'Etudes Thermiques in Poitiers, France). The type of flow is mixed convective, turbulent and two-dimensional at a constant rate. It is generated by a horizontal jet opening into an enclosed square whose lower surface is maintained at a temperature higher than that of the other surfaces (see Fig. 1). All of these latter surfaces are isothermal, with the jet being at the same temperature as that of the "cold" surfaces. The geometrical characteristics and boundary conditions used are:

- geometry : e = 18 mm, e' = 24 mm, L = 1.04 m
- inlet slot : U_{in} = 0.57 m/s, k_{in} = 1.25.10^{-3} m²/s², T_{in} = 15.0 °C
- surfaces : Tp = 15° C, Tpl = 35.5 °C

Experimental results were obtained from studies carried out at the LET by Blay and al. (1992, 1996). The experimental device was designed in such a manner as to give rise to a two-dimensional flow. It consists of a square cavity of side length L, equipped with an air entrance slot of height e and an exit slot of height e'. The cavity itself is divided into three smaller cavities. Experimental measurements were carried out in the central cavity, with the design of the device permitting a quasi two-dimensional flow to be achieved within this central cavity (Blay and al., 1996).

TURBULENCE MODELS AND NUMERICAL SIMULATIONS

In defining a situation of steady flow, we have used three different turbulence models available in the FLUENT code (1996).

- the standard *K*-ε model (Launder and Spalding, 1974) calculates the turbulent viscosity according to the expression, $\mu_t = \rho\, C_\mu\, (k^2/\varepsilon)$, where C_μ takes its usual value equal to 0.09. The kinetic energy k and its rate of dissipation ε are obtained by the solution of their respective transport equations. Furthermore, we have used the standard surface logarithmic function (Launder and Spalding, 1974) for the modelling of the parietal zone.
- the ReNormalisation Group (RNG) model is also considered as one of the turbulent viscosity models. It has, however, been obtained from exact equations of the turbulence, by using renormalisation group mathematical techniques (Yakhot and al., 1986, 1992).

The RNG model is similar to the K-ε model, but by contrast takes account the variation of the turbulent Reynolds number in the expression of the turbulent viscosity. It includes a supplementary source term, R, in the equation of ε whose expression can be written as Eqn. 1. The parietal zone has been modelled here without the surface function by using a hyperbolic tangent mesh-grid.

$$R = \frac{C_\mu \rho \eta^3 (1 - \eta/\eta_0)}{1 + \beta \eta^3} \frac{\varepsilon^2}{k} \tag{1}$$

where $\eta_0 \approx 4.38$, $\beta = 0.012$, and $\eta \equiv Sk/\varepsilon$ with $S = \sqrt{2S_{ij}S_{ij}}$ and $S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$.

- the Reynolds stress model (Launder and al., 1975, 1989) calculates each stress, $\overline{u_i'u_j'}$, by solving transport equations simultaneously. In our case, the transport equation can be written as Eqn. 2. For the numerical solution of this model, results of the calculation obtained with the K-ε model were used to initialise our calculation.

$$u_k \frac{\partial \overline{u_i'u_j'}}{\partial x_k} = P_{ij} + d_{ij} + \Phi_{ij} - \varepsilon_{ij} \tag{2}$$

RESULTS OBTAINED

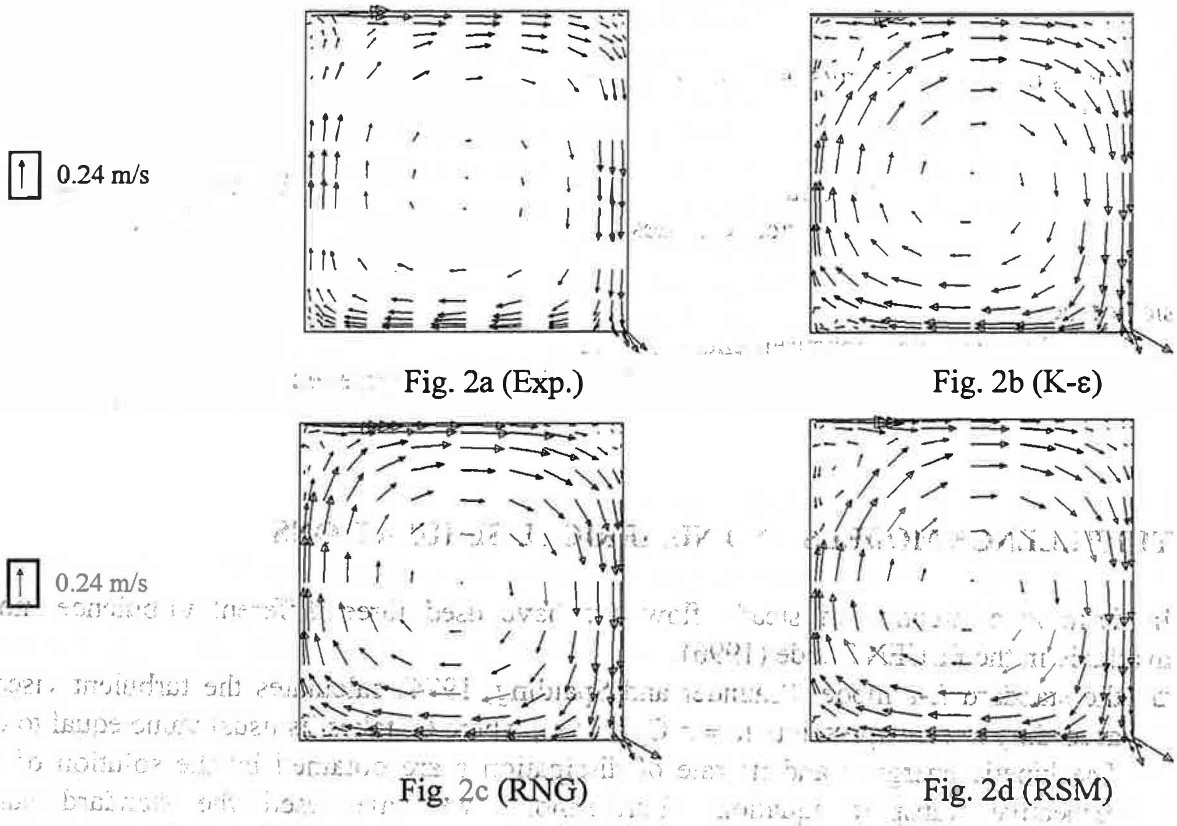

Figure 2. Velocity vector traces.
2a: Experimental. 2b: K-ε model. 2c: RNG model. 2d: RSM.

The general appearance of flow patterns is given by figures 2a to 2d. They show that the three models correctly calculated the flow characteristics in terms of mean velocity vectors. The presence of low velocities towards the middle of the cavity and nearly zero velocities at the centre can be observed in all cases. Similarly, it can be seen that the three recirculation zones observed in the experimentally obtained results are correctly reproduced by each of the models considered. Nevertheless, some differences can be noted:

- the recirculation zone situated directly under the jet is best described by the RNG model. More to the point, the *K*-ε and RSM models overestimate the extent of this zone.
- the recirculation zones situated in the left lower and right upper corners are similarly predicted in the three cases and correctly reproduce the experimental data.

However, in the various cases, one can observe a tendency for overestimation of velocities over the entire area and more particularly along the right surface to the level of the air exit slot.

Profiles Of Mean Velocity Fields And Of Turbulence Energy Within The Cavity:

Figures 3a to 3d represent profiles of mean velocity field and turbulence kinetic energy along the vertical axis at X = 520 mm (Fig. 3a and 3b) and the horizontal line at Y = 400 mm (Fig. 3c and 3d). By comparing the experimental results with the model used, one can see several important differences. First of all, in relation to the mean velocity fields (Fig. 3a and 3c), orders of magnitude are correctly predicted at the centre of the cavity, with differences of no more than 0.05 m/s. The differences increase as surfaces are approached and reach more than 0.1 m/s irrespective of the turbulence model used. For estimation of jet velocities, only the RNG model overestimates mean velocities by more than 50%. The two other models provide results that are more in agreement with the experimental data. Generally speaking, a common tendency for the three models to overestimate values of velocities was observed. Concerning the turbulence kinetic energy profiles close to the centre of the cavity, the models predicted an energy level that was too high. Observed differences varied from $0.5.10^{-3}$ to more than 1.10^{-3} m^2/s^2. However, the *K*-ε model provided results that were appreciably inferior to the two other models. Observed differences between experimental and modelled results became more significant for zones located in close proximity to the cavity's surfaces. Indeed, in these flow zones, the RNG model provided the best results. For experimental values at the points situated adjacent to the surfaces (at less than 5 cm), observed differences were in the order of 20% for the RNG model compared to more than 50% for the other two models. The *K*-ε model provided the worst results, with an overestimate of more than 100% for the first grid points

Profiles of temperature within the cavity:

Figures 4a and 4b show temperature profiles along the vertical line at X = 520 mm (Fig. 4a) and the horizontal line at Y = 520 mm (Fig. 4b). The three turbulence models provide correct estimations for the prediction of cavity centre temperatures, with observed differences remaining at less than 1 °C irrespective of the model. The RSM model predicts the temperature at the centre most accurately with a difference of 0.25 °C compared to the experimentally obtained value. More significant differences between modelled and experimental results can be observed when points in close proximity to the surfaces are considered. Indeed, irrespective of the surface studied, the RNG model provides the best estimates of temperature gradients near to that surface. The maximum difference observed was 1.5 °C for the point situated 1 cm above the heated surface, with the two other models overestimating this temperature by more than 5°C.

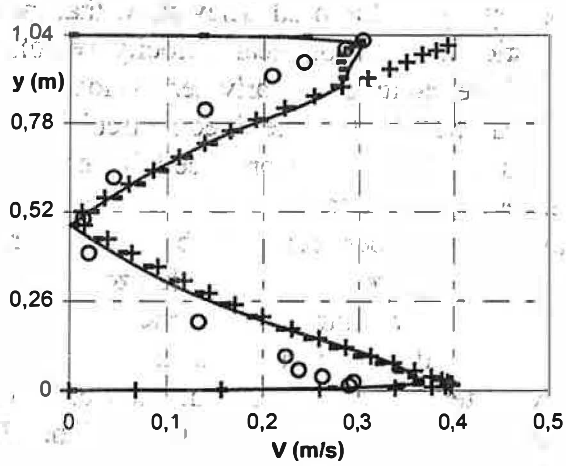

Fig. 3a.

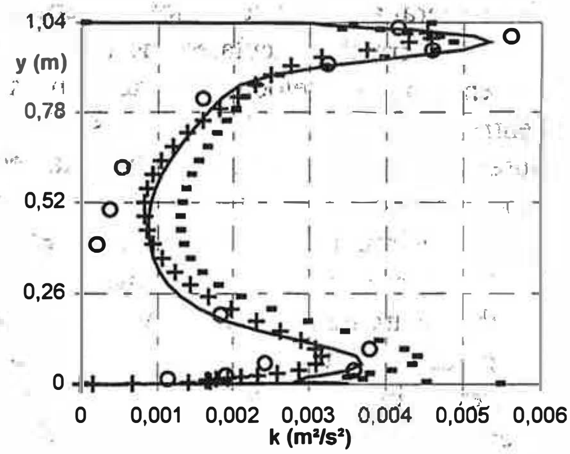

Fig. 3b.

o: exp. points, - - : k-ε, + : RNG, – : RSM

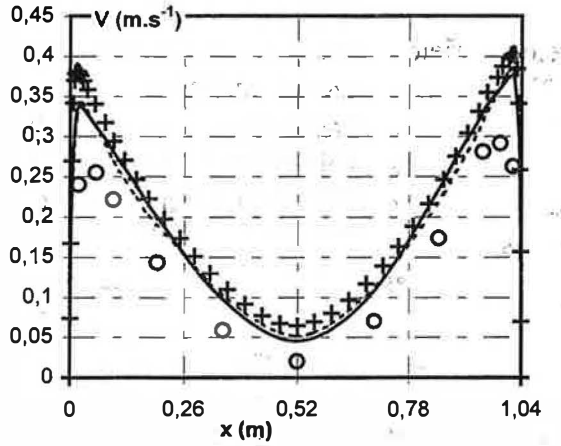

Fig. 3c.

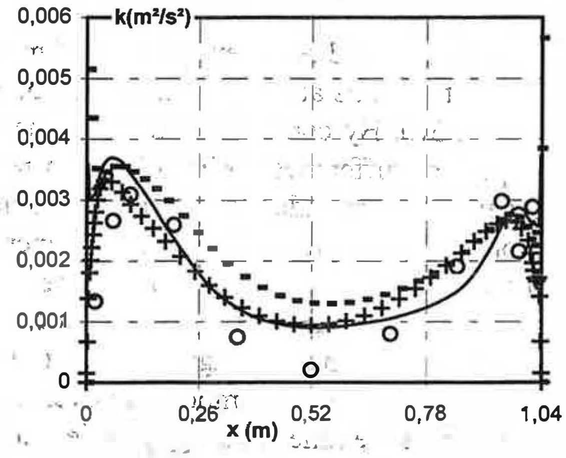

Fig. 3d.

Figure 3 a: Velocity profile on the vertical axis at X = 520 mm - b: Turbulence kinetic energy along X = 520 mm. - c: Velocity profile on the horizontal line at Y = 400 mm. - d: Turbulence kinetic energy along Y= 400 mm.

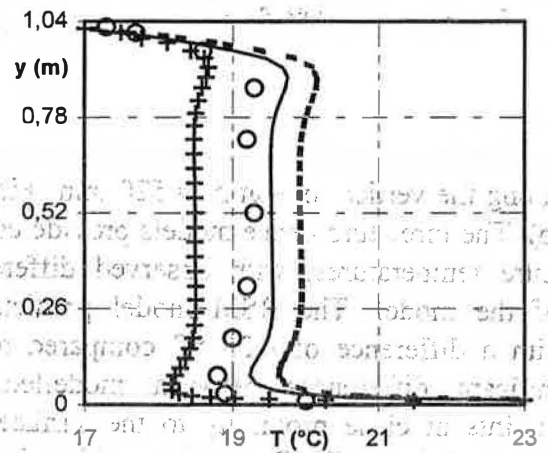

Fig. 4a.

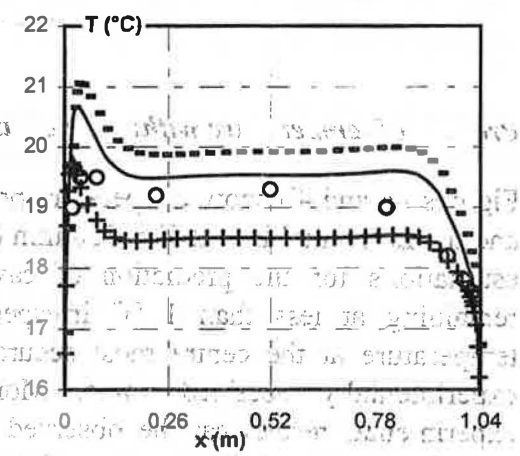

Fig. 4b.

Figure 4. a: Temperature values on the vertical line at X = 520 mm. b: Temperature values along the horizontal line at Y = 520 mm. (o: exp. points, - - : K-ε, + : RNG, —: RSM).

CONCLUSIONS

We have used three different turbulence models to predict a two-dimensional mixed convective flow within a cavity. The results obtained were in agreement with experimentally obtained data, even though non-negligible differences were observed in some flow zones. In the future pursuit of this work we hope to compare theoretical and experimental results by taking account of both experimental uncertainties and the uncertainties propagated by the different modelling techniques.

ACKNOWLEDGEMENTS

The authors are very grateful to Dr. D. Blay for the communication of experimental results.

LIST OF SYMBOLS USED

C_μ, β, η_o	constants of turbulence models	k	turbulence kinetic energy
d_{ij}	diffusion / spatial redistribution of energy	ε	rate of dissipation of k
e	height of air inlet slot	ε_{ij}	viscous dissipation
e'	height of air extraction slot	ρ	density of air
L	length of the cavity wall	μ_t	turbulent viscosity
R	source term in the equation of ε	Φ_{ij}	pressure-strain correlation
S_{ij}	mean strain	η	ratio of turbulent to mean strain time scale
S	2 times modulus of S_{ij}		
T	temperature		Subscripts
U, V, u_i, u_j	mean velocity	p	relative to the surfaces
P_{ij}	rate of production of turbulence energy	pl	relative to floor surface
$\overline{u_i' u_j'}$	Reynolds stress	in	relative to air flow

REFERENCES

Blay, D. (1996). Modélisation d'écoulements convectifs turbulents par la méthode des volumes finis. Proc. Modélisation numérique en thermique, pp. A7-1 à A7-30, Ile de Porquerolles, France.

Blay, D., Mergui, S., Niculae, C. (1992). Confined turbulent mixed convection in the presence of a horizontal buoyant wall jet. *ASME HTD, Fundamentals of mixed convection*, **213**, pp. 65-72.

Fluent Inc. (1996). Fluent User's Guide, Version 4.4.

Launder, B. E. (1989). Second-moment closure: present ... and future ? *International Journal of Heat and Fluid Flow*, **10** (**4**), pp. 282-300.

Launder, B. E., Reece, G. J., Rodi, W. (1975). Progress in the development of a Reynolds-stress turbulence closure. *Journal of Fluid Mechanics*, **68** (**3**), pp. 537-566.

Launder, B. E., Spalding, D. B. (1974). The Numerical Computation of Turbulent Flows. *Computer Methods in Applied Mechanics and Engineering*, **3**, pp. 269-289.

Yakhot, V., Orszag, A. (1986). Renormalization Group Analysis of Turbulence. I. Basic Theory. *Journal of Scientific Computing,* **1** (**1**), pp. 1-51.

Yakhot, V., Orszag, S. A., Thangam, S., Gatski, T. B., Speziale, C. G. (1992). Development of turbulence models for shear flows by a double expansion technique. *Phys. Fluids A*, **4**(**7**), pp. 1510-1520.