A RSM MODEL FOR THE PREDICTION OF HEAT AND MASS TRANSFER IN A VENTILATED ROOM

Frédéric Kuznik[1], Jean Brau [1], Gilles Rusaouen[1]

[1]Thermal Sciences Center of Lyon, UMR 5008, CNRS, INSA-Lyon, Université Lyon 1, Bât. Freyssinet, 20 rue de la Physique, 69100 Villeurbanne, France

ABSTRACT

A full-scale test room is used to investigate experimentally and numerically the velocity and temperature fields in the case of a mechanical ventilation. Detailed fields are measured for three cases of ventilation air temperature: an isothermal case, a hot case and a cold case. The experimental data are used to test two turbulence models: a first order k-ε turbulence model and a second order RSM turbulence model. The RSM model predicts the temperature and velocity fields better than the k-ε turbulence model. In particular, global values of velocity and temperature coming from experiments are in good agreement with the RSM turbulence model. This model is therefore recommended for simulation of ventilated enclosures with thermal effects.

KEYWORDS

Full-scale experiment, CFD, ventilation, k-ε turbulence model, RSM turbulence model

INTRODUCTION

Nowadays, the computational fluid dynamics codes (CFD codes) are industrially used for a large range of applications : automotive industry, machinery, electronics,... Naturally, it is then very tempting to use CFD codes in order to predict the indoor environment of building rooms. Such simulations can replace difficult and expensive experimentations to assess the human comfort for various configurations. The main problem lies in the way to model the airflow, and mainly in the choice of the turbulence model. In fact, many turbulence models exist and the majority of them were developed for highly turbulent flows which is not the case for building flows: then it is necessary to evaluate these various turbulence models.

In theory, the Navier-Stokes equations are only required to solve fluid dynamics problems. However, the direct numerical simulation (DNS) of these equations necessitates a too large computational cost: for a room like Minibat (which is presented part two of this article) mechanically ventilated at Re=10000, the numerical solution mesh requires 360 million of finite volumes (Schielstel 1993) !

The Large Eddy Simulation consists in filtering the small turbulence scales in order to solve only the large ones. This technique requires to model the filtered scales of turbulence which is called the subgrid-scale model. The simplest subgrid-scale model, the Smagorinsky one, is not suitable for the ventilated room prediction of flows (Davidson,1996, Kuznik and al. 2006). More complex subgrid-scale models exist and they are based on a dynamic filtering procedure. Their use is mainly reduced to two dimensional simulations for buiding rooms because of their computational cost (Zhang and al. 1999, Davidson 1997, Peng and Davidson 2000), but they predict correctly the flows in such cases.

Most of the simulations concerning the building rooms use based Reynolds Averaged Navier-Stokes (RANS) turbulence models. It consists in delinking the mean part (time average) and fluctuating part of the variables and solving only the equations for the mean part of each variables. Of course, some hypotheses are necessary to model the fluctuating part of the variables. The first order turbulence models link the Reynolds stress tensor to the mean velocity tensor via the Boussinesq hypothesis and the concept of turbulent viscosity. The second order turbulence models solve transport equations for the Reynolds stresses.

In order to calculate the turbulent viscosity, additional equations are needed for the first order turbulence models. The most commonly used additional equations concern the transport of k, the turbulent kinetic energy, and ε, the dissipation rate of k: these models are known as k-ε turbulence models. Various formulations of the transport equation exist: standard k-ε model, Renormalisation Group (RNG) k-ε model, Low Reynolds Number (LRN) k-ε model, k-ε "realizable" model,…These models are commonly chose for the simulation of turbulent flows in rooms (Nielsen 1998, Xu and Chen 2000, Weather and Spitler 1993, Teodosiu 2001, Awbi 1989, Posner and al 2003, Luo and al. 2004,...).

For the second order turbulence models, or RSM models (Reynolds Stress model), the components of the Reynolds stress tensor are unknowns of the problem. There are then computed using transport equations for these variables. 7 equations are added to the Navier-stokes ones. Few studies deal with the use of RSM models concerning mechanically

ventilated rooms (Chen 1990, Scalin and Nielsen 2004) even if the results with such models seem in good agreement with experimental data.

The object of this study is to compare a RSM model and a k-ε model for the prediction of airflow and temperature fields of a mechanically ventilated room. The k-ε "realizable" model is chose among the available k-ε models because it is the most appropriate one (Kuznik and al. 2007). Velocity and temperature fields are obtained by experimentations in a full-scale test room Minibat. Numerical approach was performed using CFD codes STAR-CCM+ and FLUENT (version 6.1.18). Comparisons between experimental data and numerical simulations allow to conclude concerning the ability of the models tested to predict the flows.

EXPERIMENTATIONS

Description of the experimental set-up

The experimental full-scale test room Minibat (CETHIL-INSA de Lyon, Allard and al. 1982) is shown on figure 1. The installation consists of an enclosure whose dimensions are 3.10m, 3.10m, 2.50m according to the coordinates directions (x,y,z). A thermal guard allows us to maintain the exterior faces walls at a uniform and constant value of around 20°C.

Our work only deals with tests carried with an axisymmetric jet coming from an air supply; the exact configuration can be found on figure 1. The jet is maintained at a fixed temperature by an air-treatment system. The ventilation system allows us to impose inlet and outlet flow rates which are measured with two flowmeters.

The test room has been equipped with thermocouples in order to measure the wall internal surfaces temperatures with a resolution of ±0.4°C, each face being equipped with nine thermocouples. The air temperature is measured with three Pt100 sensors with a resolution of ±0.2°C.
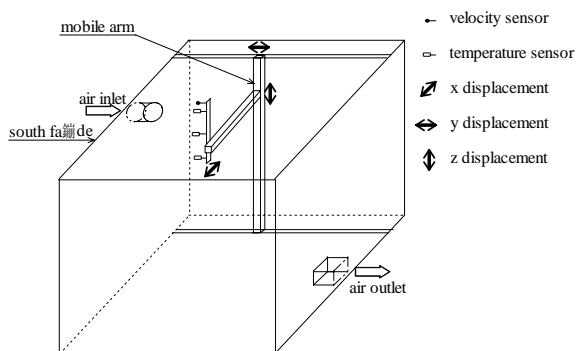

Figure 1 Experimental cell Minibat

The three components of the instantaneous velocity were measured via a three hot-wire probe. This probe has been calibrated in-situ along the three directions of the flow, taking into account the temperature of the fluid using a correlation developed in Kuznik 2005. The final calibration of the velocity probe was given with an uncertainty on the mean velocity measurement of ±0.05m/s. Only velocities with magnitude higher than 0.1m/s were measurable by our means. This type of probe has been chosen in order to get the mean velocity vectors and informations concerning the turbulent quantities.

A mobile arm allows us to move the temperature and air velocity sensor in the room in order to get complete fields of mean temperature and mean velocity magnitude. For a given position of the mobile arm, the quantities measured were the mean temperatures and 150000 samples of the three components of the velocity at a frequency of 5000Hz.

The experimental study was realized under steady state conditions and the characteristics of each case are given in table 1. This table presents the Reynolds and Archimede numbers based on the ventilation inlet diameter d and defined as follow:

$$R_{e,d} = \frac{U_{in}d}{\nu}$$

$$A_{r,d} = \frac{g\beta(T_{in}-T_m)d}{U_{in}^2}$$

T_m is the value of the mean temperature in the non-moving air zone (zone where the velocity is less than 0.05m/s), T_{in} the ventilation inlet temperature and U_{in} the ventilation inlet velocity. The table 3 shows the mean internal faces temperature of the walls during the experimentations (one face temperature is the mean value of the nine temperatures measured by the nine thermocouples of each face). These temperatures are used as boundary conditions for the numerical simulation.

Table 1
Global conditions during experimentations

	$R_{e,d}$	$A_{r,d}$	T_{in} (°C)	D (m³/h)
Isothermal case	13360	0	21.8	68
Hot case	12800	0.0097	30.6	65
Cold case	11760	-0.014	12.7	60

In conclusion, the experimental methodology permits us to obtain a complete boundary conditions description and the detailed dynamic and thermal

fields to compare with numerical data based on our models.

Results

In order to have a complete description of the different fields, 4 planes are investigated : the median plane located at x=1.55m and three vertical planes located at y=0.60m (at 3cm from the diffuser inlet), at y=1.10m (at 53cm from the diffuser inlet) and at y=1.60m (at 103cm from the diffuser inlet). The vertical planes allow to verify the symmetry of the flow according to the median plane. The median plane is scanned with 1760 positions of the mobile arm; each vertical plane is scanned with 792 positions. The spaces between each position are (2cm, 5cm, 2cm) according to (x, y, z). The figure 2 presents the measurement planes in the experimental cell.

Concerning the velocity, only the values superior to 0.1m/s were retained, according to the anemometer calibration. Then, the velocity measurements mainly concern the jet zone. The figure 3 presents the mean velocity isovalues obtained with our experimental means in the median plan. The hot case is reaching the ceiling faster than the isothermal case, due to the Archimede effect. The jet in the cold case doesn't adhere to the ceiling.
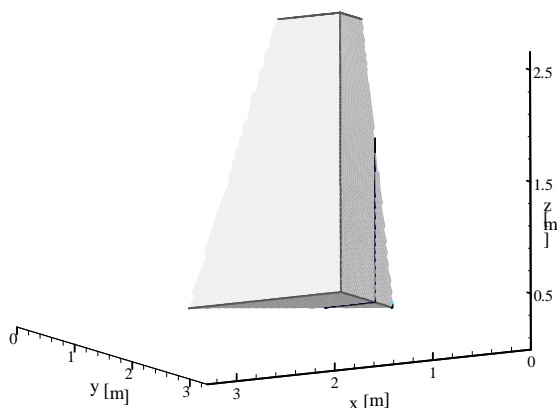

Figure 2 Test cell: measurement positions and numerical boundary conditions

Mean velocity results

The inlet diffuser has been designed to create an axisymmetric air jet in the room. For the three cases

tested, the measurements show flat velocity profiles at the inlet.

AIRFLOW MODELLING

This part is devoted to a description of the combined heat and fluid flow modelling. The mean air flow modelling principles are as follow: turbulence modelling, boundary conditions, computational domain discretization and numerical solution. Basically, our numerical model is based on commercial CFD codes, STAR-CCM+ (version 1.0) and FLUENT (version 6.1.18). Two different CFD codes are used because of the two turbulence models tested which are available in each software. The general purpose of the two codes is a finite volume, Navier Stokes solver.

Turbulence modelling

The two turbulence models are based on the Reynolds averaging of the Navier-Stokes and energy equations. The fluid is considered like an incompressible one with density computed via the ideal gas law considered as varying only with temperature.

The k-ε realizable model

This model uses the transport equations of k and ε to compute the turbulent viscosity. We employ in this study a revised k-ε turbulence model called the realizable k-ε model of Shih and al. 1995. Compared with the other k-ε models, the realizable one satisfies certain mathematical constraints on the Reynolds stress tensor, consistent with the physics of turbulent flows (for example the normal Reynolds stress terms must always be positive). Moreover, a new model for the dissipation rate is taken into account to predict the spread of both round and plane jets. This model is included in FLUENT code.

In FLUENT, the near wall treatment combines a two-layer model with enhanced wall functions. On the one hand, the first cell values of temperature and velocity are given by enhanced wall functions applicable to the entire near-wall region, according to the method of Kader 1993. On the other hand, the viscosity affected region is solved by the two-layer model of Wolfshtein, 1969: the demarcation of this region and the fully turbulent region (where the k-ε realizable equations are used) is determined by a wall distance based on the turbulent Reynolds number.

The RSM model

For the RSM model, the transport equations of the Reynolds stress tensor components are solved. The main advantage of this model is that it is better taking into account the turbulence anisotropy. Further to a bibliographical study, the quadratic RSM model of

Speziale and al. 1991 is used for our numerical model. This model is available in STAR-CCM+.

The near wall treatment uses the classical logarithmic wall functions that can be found in Kader 1993.
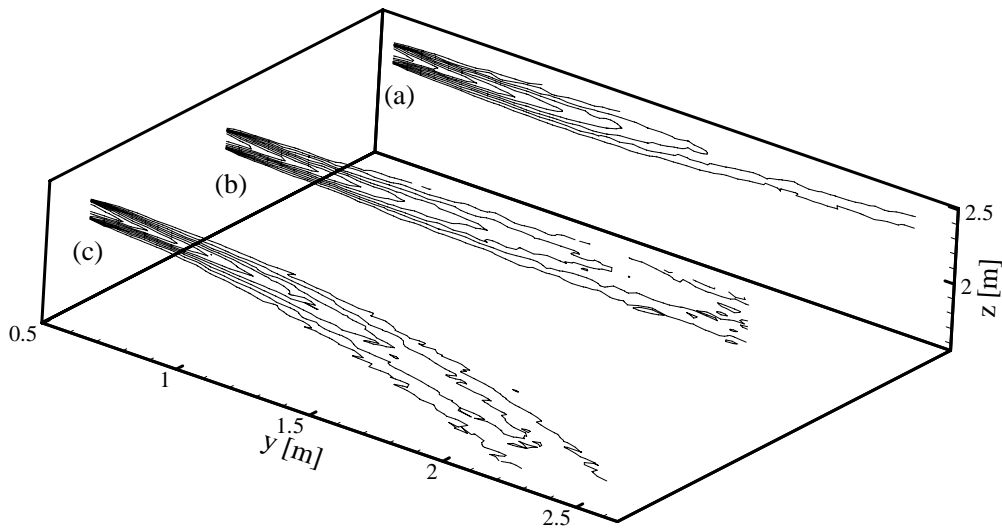

Figure 3 Mean velocity isovalues in the median plane and for the hot case (a),the isothermal case (b) and cold case (c)

Boundary conditions

The numerical solution precision deeply depends on the boundary conditions accuracy and the way that these conditions are integrated within the numerical model. In our case, there are three kinds of boundary conditions: air inlet conditions, air outlet conditions and wall boundary conditions.

In order to avoid errors due to the lack of knowledge about the exact physical parameters fields like temperature, velocity and pressure, we chose to model the air supply at the inlet (Kuznik and al. 2005). The inlet conditions are imposed far from the inlet diffuser, at a fully developed flow section (see figure 2). The velocity and temperature values are given as known values using the experimental data. Concerning the turbulence quantities, they are imposed assuming a fully developed duct flow upstream.

In the same way as the inlet boundary conditions, the outlet boundary conditions are imposed at a fully developed flow section. The outlet velocity is computed from mass balance, the gradients normal to flow direction of other variables are also set to zero at the exit section.

Finally, we need to provide boundary conditions of wall surfaces. Therefore, the classical no-slip boundary conditions are assured to the walls. We imposed either fixed values of temperatures using measured values at internal surfaces.

Discretization

The meshes are composed of finite volumes. Regular mesh cannot be used in our geometry and is not suitable for jet prediction. The available elements are then tetrahedral or polyhedral.

The k-ε realizable model

The mesh is designed using the pre-processor GAMBIT. The discretization of computational domain is achieved by means of an unstructured mesh. The grid contains tetrahedral elements obtained from a mesh generation algorithm based on the Delaunay criterion. Our final mesh is composed by 1599760 finite volumes

The RSM model

The mesh is designed using the pre-processor STAR-DESIGN. The discretization of the computational domain is achieved by means of an unstructured

mesh. The grid contains polyhedral elements, the final mesh being composed of 511963 finite volumes.

Numerical scheme

The solution method is based on the following main hypothesis: the diffusion terms are second-order central-differenced and the second-order upwind scheme for convective terms is used to reduce the numerical diffusion. The velocity-pressure coupling method is the SIMPLE algorithm. The multigrid scheme allows to accelerate the convergence as our model contains a very large number of control volumes.

The calculation time for a simulation with the RSM model was 30 hours with a Pentium IV 2.5Ghz, which is 14% more than for a calculation done with a k-ε realizable model.

RESULTS AND DISCUSSION

This section is devoted to an extensive comparison between the experimental data and numerical results concerning the experimental test cell Minibat in the three cases described previously. First, the maximum velocity and temperature decay curves are examined. Then, the airflow and temperature fields are more precisely compared by the means of various profiles and differences concerning global values. The jet trajectory in the median plan is a very important parameter of the airflow: that is why we compare the jet along z direction for experimental and numerical data. Only two profiles are shown for each case because they are caracteristic of the numerical models behaviors.

Maximum velocity and temperature decay curves

We first estimate the reliability of our models to predict the dynamics of the jet with the help of the maximum velocity and temperature curves.
The figure 4 presents the maximum velocity decay for the experimental data and the numerical models. The values of the maximum velocity U_m is normalized using the initial outlet velocity U_0.

For the three cases, the RSM model predicts better the maximum of the velocity., even if the numerical results don't match exactly the experimental data.

The figure 5 shows the maximum and minimumum temperature curves. The value of the temperature is normalized using $\Delta T_0 = T_0 - T_M$ with T_M the mean room temperature, T_0 the outlet temperature, and $\Delta T_m = T - T_M$.

The RSM model predcits reasonably the temperature decays, compared to the k-ε realizable model. As heat transfer deeply depends on turbulent values, it means that the RSM model predicts better the turbulent values occuring in the flow field.
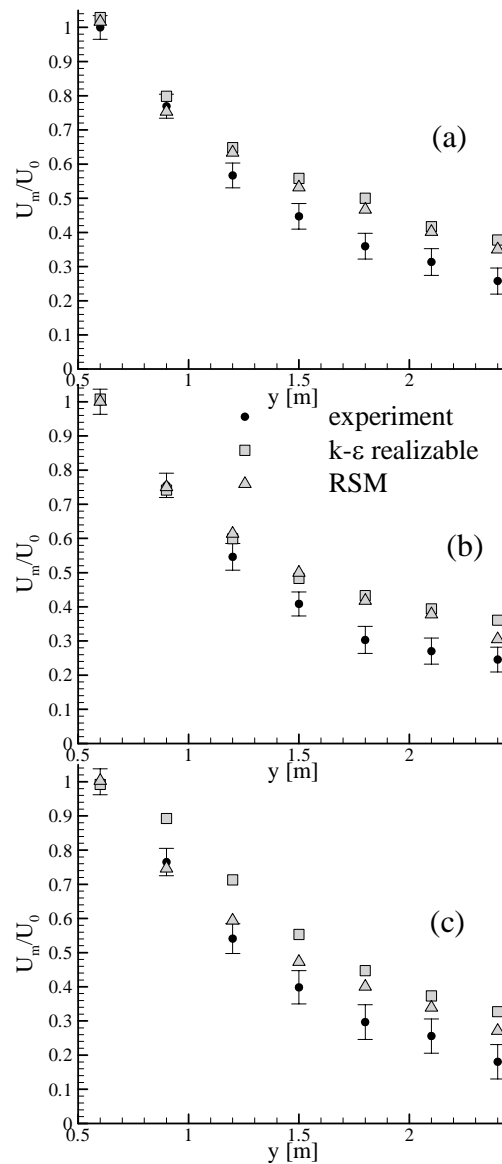

Figure 4 Maximum velocity decay curve for the isothermal case (a), hot case (b) and cold case (c)

Flow profiles analysis

All the profiles along z presented in this part belong to the median plan x=1.55m.

Isothermal case

The figure 6 shows the results obtained for the isothermal case. The RSM model predicts better the airflow than the k-ε "realizable" model, in particular the value of the velocity maximum and its position. For the RSM model, the difference between experimental data and numerical results reaches 32% for the velocity maximum while this difference is equal to 44% for the k-ε "realizable" model. For the jet trajectory in the median plan, the difference between experiment and CFD reaches 0.9cm for the RSM model and 5cm for k-ε "realizable" model.
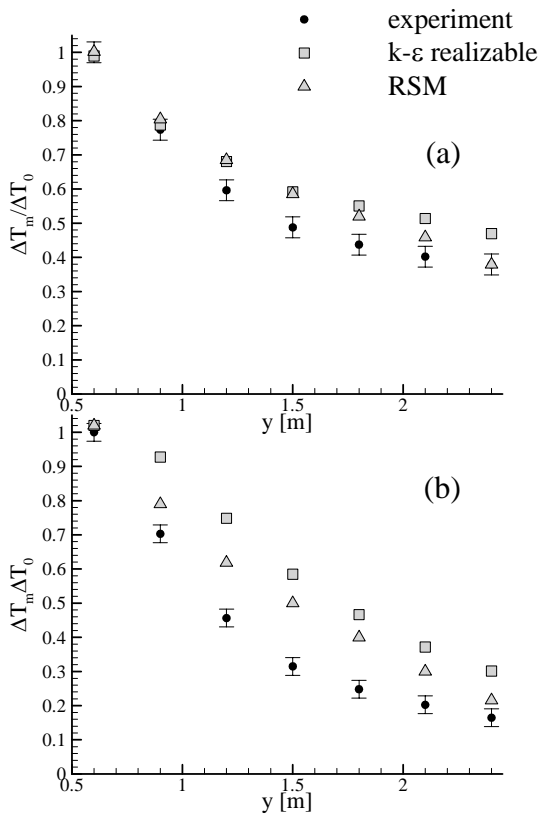
Figure 5 Maximum and minimum temperature decay curve respectively for the hot case (a) and the cold case (b)

Hot case

Velocity and temperature profiles obtained for the hot case are shown figure 7. The RSM model predicts better the airflow than the k-ε "realizable", just as well for the temperature field as for the velocity field.

For the RSM model, the difference between exprimental data and numerical results reaches 24% for the velocity maximum while this difference is equal to 45% for the k-ε "realizable" model. Concerning the temperature maximum, the difference between experiment and CFD reaches 0.1°C for the RSM model and 1°C for k-ε "realizable" model.

Cold case

The cold case, which is presented on figure 8, is the most representative of the modelling problems encountered. The dynamic of the flow is much more predicted with the second order model: 3cm of difference concerning the jet trajectory in the median plan for the RSM model against 30cm for the k-ε "realizable" model. The velocity and temperature fields are also better modeled using the second order model. For the RSM model, the difference between experimental data and numerical results reaches 28%

for the velocity maximum while this difference is equal to 46% for the k-ε "realizable". Concerning the temperature maximum, the difference between experiment and CFD reaches 1°C for the RSM model and 1.8°C for k-ε "realizable" model. The last result concerns the temperature of the occupied zone (zone under the jet where the velocity is low): the RSM model predicts this value with a difference of 0.1°C while the difference is equal to 1.4°C for the k-ε "realizable"! This last result shows the importance of the turbulence model on the heat transfer and the calculation of the thermal comfort due to the ventilation of the room.

Discussion

The comparison of experimental data and numerical results shows that the second order turbulence model predicts correctly the temperature and velocity fields in the three cases tested.

The turbulence analysis of the flows tested shows that the flow is highly anistropic (Kuznik 2005). The fisrt order turbulence models are not able to predict such anisotropy because of the Boussinesq hypothesis used. The second order turbulence models use the transport equations of the Reynolds stress components and then are able to model such anisotropy. In order to choose correctly a turbulence model, it is necessary to have an idea of the turbulent structures occuring in the flow.

CONCLUSIONS

The originality of this article lies in a detailed experiment combined with CFD numerical simulations. This allows to have a good knowledge of experimental conditions and to couple together experiment and modelling, necessary for a good analysis of the problem.

From an expert point of vue, the RSM model is recommended for the numerical simulation of a ventilated room with round or near 1 form factor ventilation air inlet. Even if the RSM model tested was quite correct, an improvement of this model can be done for the low velocity zone and has been proposed in Kuznik 2005.

Numerical simulation of flow occuring in building room is an important issue, as show the amount of publications. CFD can help to predict comfort of new ventilation systems (low energy) and problems linked to polluting agents (CO, humidity,…). Expertise concerning turbulence modelling is therefore really necessary.
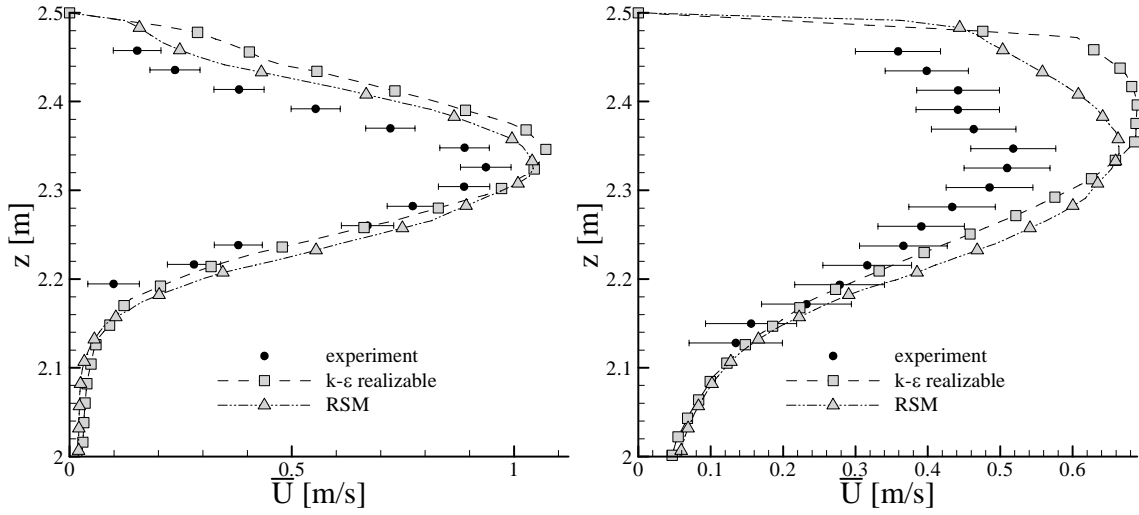
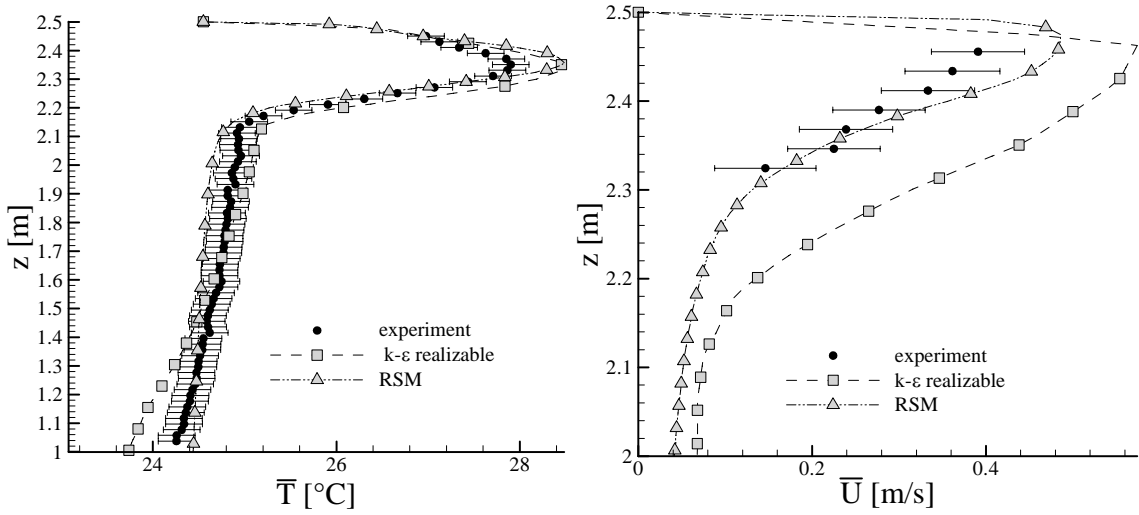*Figure 6 Mean velocity profiles at y=1.20m (left) and y=2.10m (right) – isothermal case*

Figure 7 Mean temperature profile at y=1.20m (left) and mean velocity profiles at y=2.40m (right) – hot case
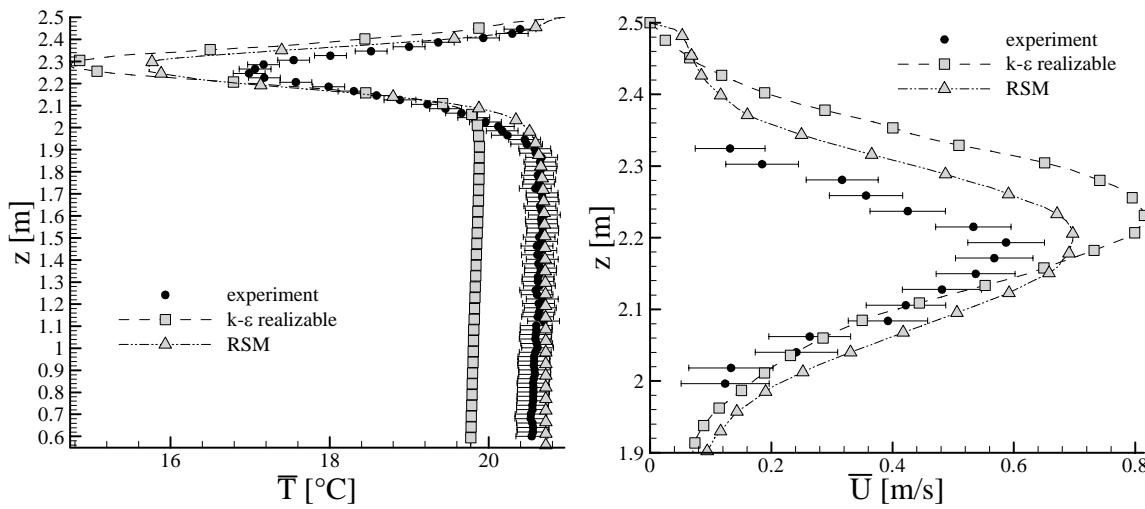

Figure 8 Mean temperature profile at y=1.20m (left) and mean velocity profiles at y=1.50m (right) – cold case

REFERENCES

Allard, F., Brau, J., Pallier, J.M. "Étude de l'incidence des apports solaires sur une cellule test en ambiance climatique simulée", Report n° 811106933328, INSA Lyon. 1982, 51p.

Awbi H.B. "Application of computational fluid dynamics in room ventilation", Building and Environment. 1989, 24(1),pp. 73-84.

Chen Q. "Prediction of room air motion by Reynolds-stress models", Building and Environment. 1996, 31(3),pp. 233-244.

Davidson L. "Implementation of a large eddy simulation method applied to recirculating flow in a ventilated room", Technical Report n° R9611, Aalborg University of Technology. 1996, 29 p.

Davidson L. "LES of recirculating flow without any homogenous direction: a dynamic one-equation subgrid model", Proceeding of the 7th International Symposium on Turbulence, Heat and Mass Transfer, Delft, Netherlands. 1997, pp 481-490.

Kader B. "Temperature and Concentration Profiles in Fully Turbulent Boundary Layers", International Journal of Heat and Mass Transfer. 1993, 24(9), pp 1541-1544.

Kuznik F. and Brau J. "Numerical and experimental investigation of a mechanically ventilated room: the impact of the inlet boundary conditions on CFD modeling of the ventilation system", International Journal of Ventilation. 2005, 4(2), pp.113-122.

Kuznik F. "Etude expérimentale des jets axisymétriques anisothermes horizontaux se développant près d'une paroi. Application à la modélisation numérique des cavités ventilées", PhD thesis, Nationale Institute of Applied Sciences, Lyon. 2005, 190p.

Kuznik F., Rusaouen G., Hohota R. "Experimetal and numerical study of a mechanically ventilated enclosure with thermal effects", Energy and Buildings. 2006 38(8) pp. 939-948.

Kuznik F., Rusaouen G., Brau J. "Experimental and numerical study of a full scale ventilated enclosure: Comparison of four two equations closure turbulence models", Building and Environment. 2007 42(3) pp. 1043-1053.

Luo S., Heikkinen J., Roux B. "Simulation of air flow in the IEA annex 20 test room – validation of a simplified model for the nozzle diffuser in isothermal case", Building and Environment. 2004, 39, pp. 1403-1415.

Nielsen P.V. "The selection of turbulence models for prediction of room airflow", ASHRAE Transactions. 1998, 104(1B), pp. 1119-1127.

Peng S.H., Davidson L. "The potential of large eddy simulation techniques for modelling indoor airflow", Proceeding of the 7th International Conference on Air Distribution in Rooms, Oxford, England. 2000, pp. 295-300.

Posner J.D., Buchanan C.R., Dunne-Rankin D. "Measurement and prediction of indoor air flow in a model room", Energy and Buildings. 2003, 35, pp. 515-526.

Scalin A., Nielsen P.V. "Impact of turnulent anisotropy near walls in room air flow", International Journal of Indoor Environment and Health. 2004, 14(3), pp. 159-168.

Schiestel R. "Modélisation et simulation des écoulements turbulents", Paris : Hermès, 1993, 442 p.

Shih T., Liou W.W., Shabbir A., Yang Z., Zhu J. "A new k-ε eddy viscosity model for high Reynolds number turbulent flows", Journal of Computer Fluids. 1995, 24(3), pp. 227-238.

Speziale C.G., Sarkar S., Gatski T. "Modeling the pressure-strain correlation of turbulence: an invariant dynamical systems approach", Journal of Fluid Mechanics. 1991, 227, pp. 245-272.

Teodosiu C. "Modélisation de systèmes techniques dans le domaine des équipements des bâtiments à l'aide des codes de type CFD", PhD thesis, National Institute of Applied Sciences, Lyon. 2001, 355p.

Xu W, Chen Q. "Simulation of mixed convection flow with a two layer turbulence model". Indoor Air. 2000, 10(4), pp. 306-314.

Weather J.W., Spitler J.D. "A comparative study of room airflow: numerical prediction using computational fluid dynamics and full-scale experimental", ASHRAE Transactions. 1993, 99(2), pp. 144-157.

Wolfshtein M. "The Velocity and Temperature Distribution of One Dimensional Flow with Turbulence Augmentation and Pressure Gradient", International Journal of Heat and Mass Transfer. 1969, 12, pp. 301-318.

Zhang W. and Chen Q. "A new filetered dynamic subgrid-scale model for large eddy simulation of indoor airflow", Proceeding of the 6th International IBPSA Conference, Kyoto, Japan. 1999, pp. D-03.