

2008

CFD Modeling and Validation of Temperature and Flow Distribution in Air-Conditioned Space

Jahar Sarkar
Institute of Technology

Soumen Mandal
Institute of Technology

Follow this and additional works at: <http://docs.lib.purdue.edu/iracc>

Sarkar, Jahar and Mandal, Soumen, "CFD Modeling and Validation of Temperature and Flow Distribution in Air-Conditioned Space" (2008). *International Refrigeration and Air Conditioning Conference*. Paper 972.
<http://docs.lib.purdue.edu/iracc/972>

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.

Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at <https://engineering.purdue.edu/Herrick/Events/orderlit.html>

CFD Modeling and Validation of Temperature and Flow Distribution in Air-Conditioned Space

Jahar SARKAR*, Soumen MANDAL

Department of Mechanical Engineering,
Institute of Technology, B.H.U,
Varanasi-221005, India

*Phone: + 91-9919787557, Fax: 91-542-2368157;
Email: jahar_s@hotmail.com

ABSTRACT

Creating suitable thermal conditions for satisfying human desires, the thermal comfort has been recognized to be an essential requirement of the indoor environment. Thermal comfort is related to temperature and airflow distributions in air conditioning space, which play an important role in optimum design of air-conditioning system or ventilation system. No such general CFD model can be applicable for airflow and temperature distribution in room due to the suitability of turbulent models are dependent on application, operating conditions and other factors. The main aim of the present study is to investigate the airflow and temperature distribution of air-conditioning space.

For this purpose, an experimental study on flow pattern and velocity distribution in an enclosed space with single inlet and outlet has been conducted. The flow pattern, velocity and temperature distributions in the above enclosed space have been investigated using three dimensional CFD simulation and the results obtained from the simulation are validated with experimental data. Results show that $\kappa - \varepsilon$ and $k - \omega$ models give more closer results of velocity and temperature distributions compared to Reynolds stress model, although all these three give very narrow results at inlet plane and Reynold stress model show more deviation with experimental data compared to $\kappa - \varepsilon$ and $k - \omega$ models for velocity distribution.

Keywords: Thermal comfort, air-conditioning, CFD, turbulent model, validation

1. INTRODUCTION

Since the distribution of air velocity and temperature in a ventilated space is not uniform, whole flow field airflow patterns and air velocity distributions are essential information to understand performance of ventilation systems, occupant comfort and well-being. Prototype studies are expansive and time consuming, largely due to limitations of the current available measurement technologies and instrumentation. Interests in simulating the airflow through large openings in buildings, such as windows or doors, allowing for bidirectional flow, are increasing. However, perfect modeling of air flow and temperature distribution is very tricky due to suitability of different turbulent models (Nielsen, 1998) used for CFD simulation of room air flow and hence validation is essential.

An extensive literature review on the application of CFD to building ventilation and IAQ (indoor air quality) problems was performed by Emmerich (1997), which includes room airflow case for various ventilation system, and strategies and room configurations; flow from diffusers; modeling effects of occupants; exhaust ventilation system performance; wind pressure distribution for flow around buildings; thermal and air flow performance in large enclosures; pollutant transport including partials and moisture; air curtains; pressure loss in ducts; and coupling of CFD programs with multi-zone air flow models and/or building energy simulation models, and it was determined that Large eddy simulation has unique capabilities compared to other CFD model. Emmerich and McGrattan (1998) described a three-dimensional, large eddy simulation (LES) model developed for studying the transport of smoke and hot gases during a fire in an enclosure. Williams et al. (1994) described three-part paper documents research completed on the topical aspects of computational fluid dynamics (CFD) associated with the room air motion problem. Eftekhari et al. (2002) investigated airflow distribution inside a test room, which is naturally ventilated

through adjustable louvers for summer conditioned. Abanto et al. (2004) described the study of the numerical simulation of airflow and the prediction of comfort properties in a visualization room of a research centre. Fan (1995) has used various models to solve the air and contaminant distribution in rooms. Chen and Xu (1998) have proposed a new zero-equation model to simulate three dimensional distributions of air velocity, temperature, and contaminant concentrations in rooms. Zhao et al. (2003) proposed new numerical method and claimed that it can correctly simulate air velocity and temperature distributions in the room except in a few positions by less computing time than using conventional CFD methods. Jouvray et al. (2007) used non linear RANS model for complex geometry, which showed impressive performance for validation.

Zhao et al. (2001) developed a PIV (Particle Image Velocimetry) measurement system for indoor airflow field studies. Quantative air velocity distribution can be obtained to validate numeric models and analyze ventilation strategies. The results showed that the PIV technique can be effective method to quantitatively measure the room air velocity, especially for those regions with very low velocity. Posner et al. (2003) compared result from relatively simple three dimensional numerical simulations (CFD) with laser Doppler anemometry (LDA) and particle image velocity meter (PIV) experimental measurement of indoor air flows in a one tenth scale model room. Stamau and Katsiris (2006) used the SST based $k - \omega$ model to calculate air-flow velocities and temperatures in a model office room. Calculations were compared with experiments and with the results of the standard $\kappa - \varepsilon$, RNG $\kappa - \varepsilon$ model and the laminar model and concluded that all the three tested turbulent models predict satisfactorily the main qualitative features of the flow and the layered type of temperature fields and computations with the SST $k - \omega$ based model showed the best agreement with measurements.

The main aim of the present study is to investigate the airflow and temperature distribution in a model room by both experimentally with single inlet and outlet and numerically using three dimensional CFD simulation and validation of simulation results with experimental data for different flow rates at inlet.

2. CFD SIMULATION

The physical model of the ventilated room for which the numerical and experimental studies were conducted is shown in Figure 1. A rectangular enclosure made of glass of dimensions of 1m length, 0.67m width and 0.67 m height, with single inlet and outlet has been taken as the physical model for this study. Both the supply and exhaust ducts are 0.34 m length and 0.0416m diameter. The inlet duct is situated at the height of 0.50 m and outlet duct is at the height of 0.16 m from the base.

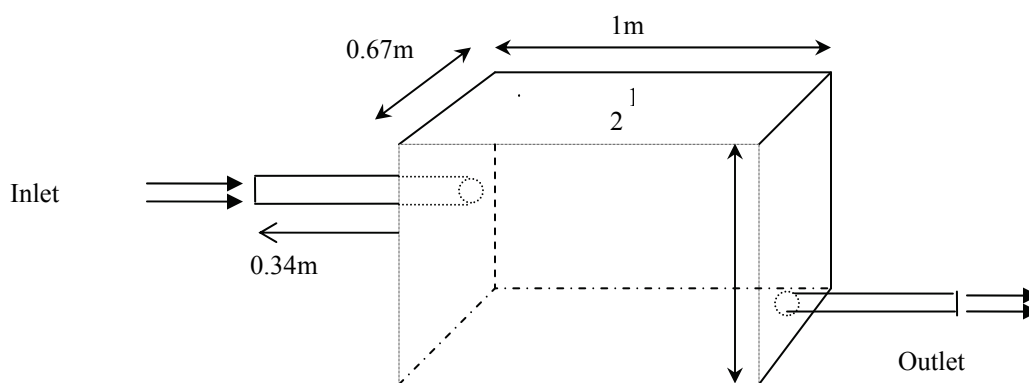


Figure 1: Ventilated room model

The following continuity, momentum and energy equations are applicable for constant properties and describing the three-dimensional fluid flow and heat transfer in the air-conditioned or ventilated space:

$$\frac{\partial(\rho\bar{u})}{\partial x} + \frac{\partial(\rho\bar{v})}{\partial y} + \frac{\partial(\rho\bar{w})}{\partial z} = 0 \quad (1)$$

$$\rho \left(\bar{u} \frac{\partial \bar{u}}{\partial x} + \bar{v} \frac{\partial \bar{u}}{\partial y} + \bar{w} \frac{\partial \bar{u}}{\partial z} \right) = -\frac{\partial \bar{p}}{\partial x} + \mu_{eff} \left(\frac{\partial^2 \bar{u}}{\partial x^2} + \frac{\partial^2 \bar{u}}{\partial y^2} + \frac{\partial^2 \bar{u}}{\partial z^2} \right) \quad (2)$$

$$\rho \left(\bar{u} \frac{\partial \bar{v}}{\partial x} + \bar{v} \frac{\partial \bar{v}}{\partial y} + \bar{w} \frac{\partial \bar{v}}{\partial z} \right) = -\frac{\partial \bar{p}}{\partial y} + \mu_{eff} \left(\frac{\partial^2 \bar{v}}{\partial x^2} + \frac{\partial^2 \bar{v}}{\partial y^2} + \frac{\partial^2 \bar{v}}{\partial z^2} \right) \quad (3)$$

$$\rho \left(\bar{u} \frac{\partial \bar{w}}{\partial x} + \bar{v} \frac{\partial \bar{w}}{\partial y} + \bar{w} \frac{\partial \bar{w}}{\partial z} \right) = -\frac{\partial \bar{p}}{\partial z} + \mu_{eff} \left(\frac{\partial^2 \bar{w}}{\partial x^2} + \frac{\partial^2 \bar{w}}{\partial y^2} + \frac{\partial^2 \bar{w}}{\partial z^2} \right) \quad (4)$$

$$\rho c_p \left(\frac{\partial \bar{T}}{\partial t} + \bar{u} \frac{\partial \bar{T}}{\partial x} + \bar{v} \frac{\partial \bar{T}}{\partial y} + \bar{w} \frac{\partial \bar{T}}{\partial z} \right) = k_{eff} \left(\frac{\partial^2 \bar{T}}{\partial x^2} + \frac{\partial^2 \bar{T}}{\partial y^2} + \frac{\partial^2 \bar{T}}{\partial z^2} \right) \quad (5)$$

Where, effective viscosity and thermal conductivity are given by,

$$\mu_{eff} = \mu + \mu_t, \quad k_{eff} = k + k_t \quad (6)$$

To numerical investigation of airflow and temperature distributions in ventilated room the commercial software Fluent 6.0 has been used. Fluent mesh generator 'Gambit' was used to build the computational mesh for model room. Extension of 0.34m long to the supply and exhaust ducts were used to facilities boundary condition specifications that did not compromised local flow field accuracy.

The governing equations are discretized using the control volume method, where a whole computational domain is divided into small control volumes. A scalar grid point is located at the center of each control volume, storing the values for scalar variables such as pressure, temperature and species. In order to ensure the stability of numerical calculation, velocity components are arranged on different grid points, staggered with respect to scalar grid points. Discretized equations for a variable are formulated by integrating the corresponding governing equation over the three-dimensional control volumes. The three turbulent models ($\kappa - \varepsilon$ Model, $k - \omega$ Model and Reynolds stress model) have been used to solve the governing equations in Fluent.

The following boundary conditions have been used in CFD simulation:

At inlet: $V_x = V_i, \quad V_y = 0, \quad V_z = 0; T = T_i$

At outlet: $P = P_{atm}$

At side wall: Isothermal, $T = T_w$

At top and bottom wall: Constant heat flux, $\frac{\partial T}{\partial y} = 0$

3. EXPERIMENTAL STUDY

The experimental setup consists of a model room (test section, in which, airflow distribution has been investigated; dimension is same as for CFD simulation, which is shown in Figure 1), blower for supply of air, a orifice meter for measuring flow rate which is attached with the main flow pipe, a 'U' tube Manometer which is attached with two sides of orifice meter and a bypass valve as shown in Figure 2. Air from blower passes through a flow straightener and then enters to the test section. The model room, made of glass (for flow visualization inside the chamber), is placed on a table. There is a bypass and flow control valve by which the flow can be control. The small holes at different positions on the top face of prototype have been made to fit the pitot tube. Velocities at different heights have been measured by moving the pitot tube vertically. It has been ensured that there is no air leakage from the chamber except inlet and outlet in the test time.



Figure 2: Experimental prototype with Pitot tube arrangement

After starting the blower by full opening of bypass valve and flow control valve, the certain flow rate was set by controlling the valves, which enables tests to be conducted at different Reynolds numbers. It can take some times to give uniform flow and attain steady-state condition. Smoke was used for this purpose, which ensured the steady state condition. Then pressure head was measured to get flow rate at inlet. With the help of pitot tube, dynamic pressure heads were measured to calculate velocity at different positions and different heights. The experiment was done for two different flow rates and the same were measured for each flow rate. Air flow was changed systematically by controlling the by pass valve so that the manometric head across the orifice varies uniformly. Inlet velocity to model room has been calculated by,

$$V = C_v \sqrt{\frac{2g(\rho_w/\rho_a - 1)\Delta h}{(D/D_c)^2 - 1}} \quad (7)$$

Where, $D = 38 \text{ mm}$, $D_c = 20 \text{ mm}$, $\rho_a = 1.2 \text{ kg/m}^3$, $\rho_w = 1000 \text{ kg/m}^3$, $C_v = 0.75$ and Δh is the manometer deflection of orifice meter.

Velocity at different position in the model room was calculated by,

$$V = \sqrt{2cgh \left(\frac{\rho_w}{\rho_a} - 1 \right)} \quad (8)$$

Where, $c =$ micrometer constant $= 0.02$ and $h =$ micrometer reading.

Since, the pitot tube is unable to give accurate results at low velocity, repeatability tests were conducted for two sets of operating parameters to test the accuracy of experimental results and average values were taken.

4. RESULTS AND DISCUSSION

Although test has been conducted for different positions, in this study, results and validations are presented for two inlet velocities (2.13 m/s , $Re_d = 5680$ and 3.13 m/s , $Re_d = 8300$) and two positions (see figure 1):

- Position 1: center of vertical plane parallel to inlet flow
- Position 2: center of vertical plane parallel to exit flow

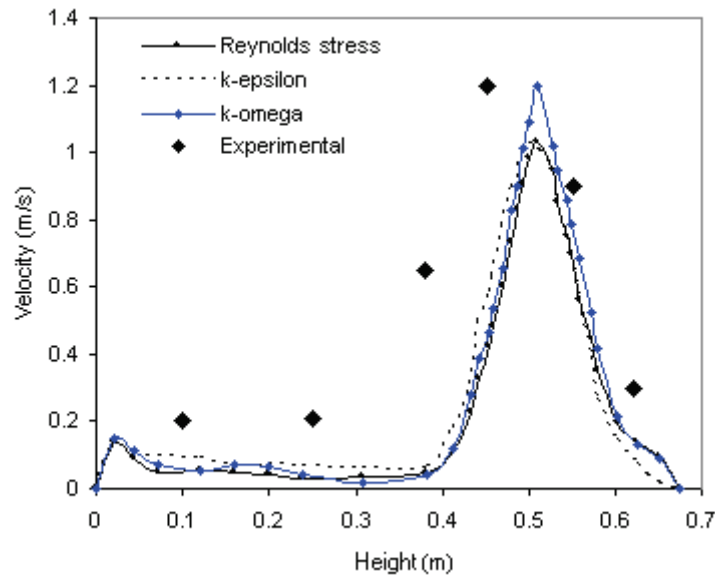


Figure 3: Velocity with height at position 1 for inlet velocity of 2.13m/sec

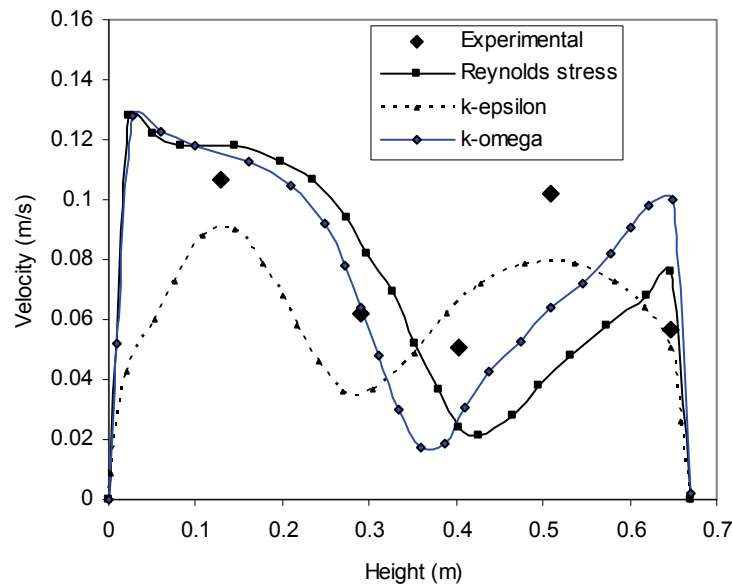


Figure 4: Velocity with height at position 2 for inlet velocity of 2.13m/sec

Figures 3 and 4 show the variation of velocities with respect to height at positions 1 and 2, respectively for the inlet velocities of 2.13 m/s, and Figures 5 and 6 show the variation of velocities with respect to height at positions 1 and 2, respectively for the inlet velocities of 3.13 m/s. Velocities obtained from numerical simulation with three turbulent models have been validated with experimental results. Maximum variation of velocity is obtained near the inlet. All the three turbulent model give very narrow results for the position 1 and give deviated results for position 2, although $\kappa-\varepsilon$ and $k-\omega$ give very narrow results compared to Reynolds stress model. $\kappa-\varepsilon$ and $k-\omega$ models give very closer results with experiment compared to Reynolds stress model. Maximum deviation of numerical result with obtained experimental data has been observed as 40%.

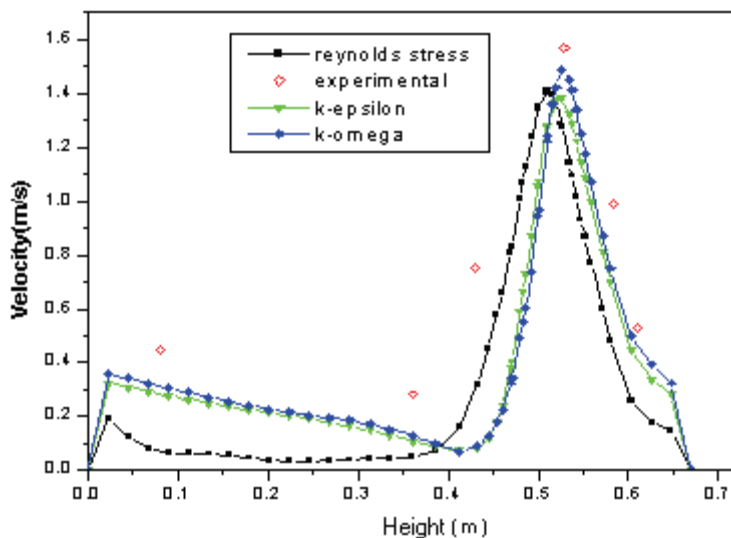


Figure 5: Velocity with height at position 1 for inlet velocity of 3.13m/sec

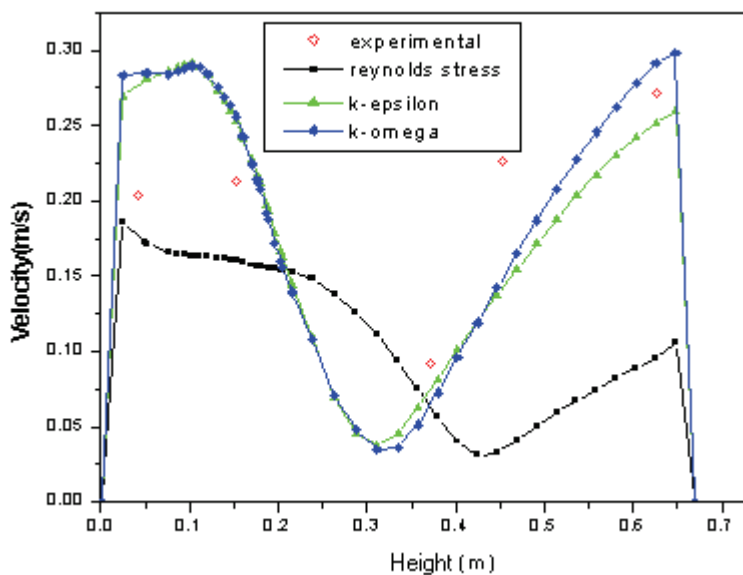


Figure 6: Velocity with height at position 2 for inlet velocity of 3.13m/sec

In this study, the temperature distributions are presented based on only the CFD simulation using three different models for inlet temperature of 21⁰C and side wall temperature of 35⁰C. Figures 7 and 8 show the variation of temperature with respect to height at positions 1 and 2, respectively for inlet velocity of 2.13 m/s. Although all the three models give narrow results at position 1, these results differ significantly at 2. These can be attributed that very low magnitude of velocity at position 2 gives more numeral deviation for different models compared to that of position 1. At position 2, $\kappa - \varepsilon$ and Reynolds stress models give more narrow results compared to $k - \omega$ model. The temperature variation gives very similar results also for inlet velocity of 3.13 m/s. However, $\kappa - \varepsilon$ and $k - \omega$ models give very narrow results, those are deviated significantly for Reynolds stress models at position 2 for inlet velocity of 3.13 m/s. $\kappa - \varepsilon$ model gives more uniform temperature distribution compared to others.

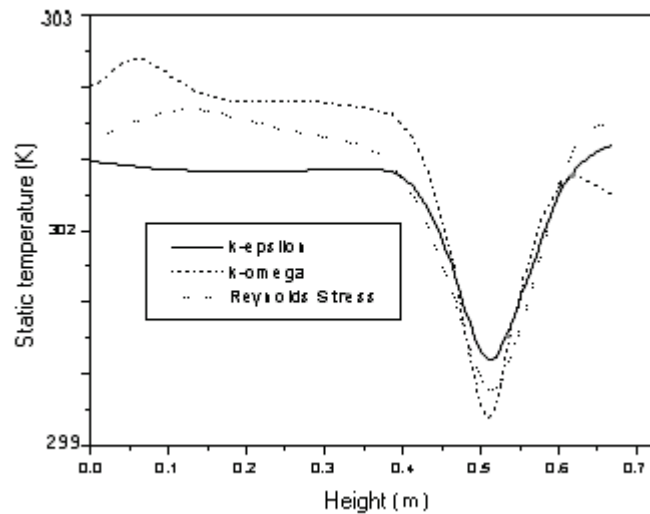


Figure 7: Temperature variation with height at position 1 for inlet velocity 2.13m/s

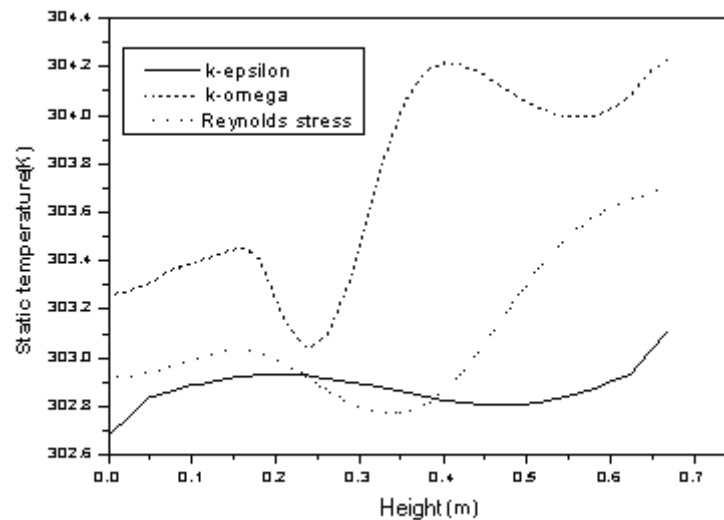


Figure 8: Temperature variation with height at position 2 for inlet velocity 2.13m/s

6. CONCLUSIONS

In the present study, airflow and temperature distributions of air-conditioning space have been investigated. For this purpose, an experimental study on flow pattern and velocity distribution in an enclosed space with single inlet and outlet has been conducted. The velocity and temperature distributions in the above enclosed space have been investigated using three dimensional CFD simulation and the results obtained from the simulation are validated with experimental data. From the experimental experience, it can be concluded that the Pitot tube arrangement is not suitable for very low velocity measurement. From the results and discussions, following conclusions can be drawn:

Results show the airflow inside the test chambers is turbulent. Maximum variation of velocity is obtained near the inlet. All the three turbulent models give very narrow results in the line of flow inlet and give deviated results in the line of flow outlet, although $\kappa - \varepsilon$ and $k - \omega$ give very narrow results compared to Reynold stress model. $\kappa - \varepsilon$ and $k - \omega$ models give very closer results with experiment compared to Reynold stress model. Maximum deviation of numerical result with obtained experimental data has been observed as 40%. Although all the three models give narrow temperature distribution at inlet plane, these results differ significantly at the outlet plane for both inlet velocities. Similar to the velocity distribution, $\kappa - \varepsilon$ and $k - \omega$ models give more narrow temperature distributions compared to Reynolds stress model.

NOMENCLATURE

c_p	specific heat capacity	(J/kgK)	Subscripts	
D	diameter	(cm)	a	air
g	gravitational constant	(m/s ²)	i	inlet
k	thermal conductivity	(W/mK)	t	turbulent
V, \bar{u} , \bar{v} , \bar{w}	velocities	(m/s)	w	wall, water
μ	viscosity	(kg/ms)		
ρ	density	(kg/m ³)		

REFERENCES

- Abanto, J., Barreo, D., Reggio, M., Ozell B., 2004, Airflow modeling in a computer room, *Building & Environment*, vol. 39, no. 12: p. 1393-1402
- Chen, Q., Xu, W., 1998, A Zero-equation turbulence model for indoor airflow simulation, *Energy & Buildings*, vol. 28, no. 2: p. 137-144.
- Eftekhari, M.M., Marjanovic, L.D., Holmes, M., Hanby, V., 2002, Air flow distribution for different wind pressure coefficient values in a naturally ventilated room, *ASHRAE Transactions*, vol. 108, no. 2: p. 1233-1240.
- Emmerich, S.J., 1997, Use of computational fluid dynamics to analyze indoor air quality issues, NISTIR 5997, National Institute of Standards and Technology, USA.
- Emmerich, S.J., McGrattan, K.B., 1998, Application of Large Eddy Simulation Model to Study Room Airflow, *ASHRAE Transactions*, vol. 104, no. 1: p. 1128-1137.
- Fan, Y., 1995, CFD modeling of the air and contaminant distribution in rooms, *Energy & Buildings*, vol. 23, no. 1: p. 33-39.
- Jouvray, A., Tucker, P.G., Liu, Y., 2007, On nonlinear RANS models when predicting more complex geometry room air flows, *Int. J. Heat Fluid Flow*, vol. 28, no. 2: p. 275-288.
- Nielsen, P.V., 1998, The Selection of Turbulence Models for Prediction of Room Airflow, *ASHRAE Transactions*, vol. 104, no. 1: p. 1119-1127.
- Posner, J.D., Buchanan, C.R., Dunn-Rankin, D., 2003, Measurement and prediction of indoor air flow in a model room, *Energy & Buildings*, vol. 35, no. 5: p. 515-526.
- Stamau, A., Katsiris, I., 2006, Verification of a CFD model for indoor air flow and heat transfer, *Building & Environment*, vol. 41, no. 9: p. 1171-1181.
- Williams, P.T., Baker, A.J., Kelso, R.M., 1994, Numerical calculation of room air motion-Part 1: Math, Physics, and CFD modeling, *ASHRAE Transactions*, vol. 100, no. 1: p. 514-530.
- Zhao, B., Li, X., Yan, Q., 2003, A simplified system for indoor airflow simulation, *Building & Environment*, vol. 38, no. 4: p. 543-552.
- Zhao, L., Zhang, Y., Wang, X., Riskowski, G.L., Christianson, L.L., 2001, Measurement of two-dimensional air velocity in a full-scale room using particle image velocimetry, *ASHRAE Transactions*, vol. 107, no. 2: p. 434-443.