

Influence of turbulence model on thermal plume in indoor air flow simulation

Citation for published version (APA):

Zelensky, P., Bartak, M., Hensen, J. L. M., & Vavricka, R. (2013). Influence of turbulence model on thermal plume in indoor air flow simulation. In *Proceedings of the 11th REHVA World Congress & 8th international Conference on IAQVEC (CLIMA 2013), "Energy efficient, smart and healthy buildings", June 16-19, 2013, Prague*

Document status and date:

Published: 01/01/2013

Document Version:

Accepted manuscript including changes made at the peer-review stage

Please check the document version of this publication:

- A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
- The final author version and the galley proof are versions of the publication after peer review.
- The final published version features the final layout of the paper including the volume, issue and page numbers.

[Link to publication](#)

General rights

Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

- Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
- You may not further distribute the material or use it for any profit-making activity or commercial gain
- You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the "Taverne" license above, please follow below link for the End User Agreement:

www.tue.nl/taverne

Take down policy

If you believe that this document breaches copyright please contact us at:

openaccess@tue.nl

providing details and we will investigate your claim.

Influence of Turbulence Model on Thermal Plume in Indoor Air Flow Simulation

Petr Zelenský^{#1}, Martin Barták[#], Jan Hensen^{**}, Roman Vavříčka[#]

[#]*Department of Environmental Engineering, Czech Technical University in Prague,
Czech Republic*

¹petr.zelensky@fs.cvut.cz

^{*}*Building Physics and Services, Eindhoven University of Technology, Netherlands*

Abstract

The paper deals with CFD modeling of heat sources in indoor environment and with the influence of turbulence model on the results of simulation. Simulations with a model of seated occupant acting as an indoor heat source were performed using various turbulence models. Air temperature distribution above a seated thermal manikin was measured using a thermal imaging camera. The results of simulations and experiment were mutually compared. The study indicates which turbulence models can be recommended for related types of applications.

Keywords – CFD simulation, simplified modeling, indoor heat source, turbulence model, thermal imaging

1. Introduction

Heat sources are very common in most indoor environments. They have an impact on indoor airflow as they generate thermal plumes – warm convective currents driven upwards by buoyancy forces. Thermal plumes can significantly influence air flow distribution indoors as well as indoor environment quality, possibly in a negative way [1–3]. Therefore, heat sources should be accurately incorporated in CFD simulations in order to simulate the indoor air flow and environment properly. CFD simulations can contribute to a more effective removal of heat gains from the indoor heat sources and consequently to a lower energy consumption of ventilation and air-conditioning systems.

An important part of CFD method is the modeling of turbulence. The selection of turbulence model can have a significant influence on CFD simulation results. In the current study we compare results from CFD simulations of buoyant air flow based on different types of two-equations turbulence models.

The results of simulations are also compared to the measurements by a thermal imaging camera, because just the inter-model comparison cannot indicate which turbulence model gives more realistic results. The study indicates which of the used turbulence models can be recommended for

related types of applications. This model will be used in a new approach to simplify models of heat sources, which is described elsewhere [4].

2. Turbulence Modeling in Indoor Airflow Simulations

Selection of the method to approximate turbulent processes of fluid flow is a very important part of CFD simulations. There are several approaches to deal with turbulence; they differ in complexity and accuracy [5].

The most computationally demanding is the direct numerical simulation (DNS method), in which the Navier-Stokes equations are numerically solved for instantaneous quantities. Close to this approach is the Large Eddy Simulation (LES method) which directly solves the large-scale turbulent motion (large eddies) and approximates the turbulent transport which occurs in small-scale eddies. Both methods are generally very accurate, but it is cumbersome to use them in solving more complex cases due to their high demand for computational power.

Reynolds decomposition and averaging of Navier-Stokes equations (RANS method) is a very usual approach to deal with turbulence in engineering applications. Only average flow quantities are solved directly and the effect of turbulent fluctuations is approximated on the basis of turbulence models. The accuracy of this method is lower than that of DNS or LES approaches, but the demand for computational resources is considerably reduced. The RANS method is usually sufficient in the CFD simulations for environmental engineering applications, as we are interested mainly in the prediction of mean flow and there is no interest in very detailed and accurate solution of turbulent processes.

A number of turbulence models have been developed for the RANS method. Commonly used models are summarized for instance by Zhai et al. [5]. In environmental engineering, the most frequently used turbulence models are the two-equation models such as k - ε and k - ω ; therefore these models were used for comparison in this study.

The standard k - ε model (herein after referred to as k - ε *Standard*) was proposed by Launder and Spalding [6] and it is suitable for flows at higher values of turbulent Reynolds numbers. Turbulence model according to Yakhot and Orszag [7] (k - ε *RNG*) gives slightly better results when simulating air flow in enclosed spaces. Turbulence model by Shih et al. [8] (k - ε *Realizable*) is suitable for approximating of turbulence in the environment with swirling flows, buoyancy flows and flows involving separation [5].

The model developed by Wilcox [9] (k - ω *Standard*) shows higher accuracy for turbulent flow near the wall with adverse pressure gradient. However, it is less robust in the areas of wakes and in the case of flow without effect of shear stress on the wall [5].

For the simulations in environmental engineering it is always necessary to consider the problem on hand as there is no universal model of turbulence.

Therefore, several simulations of thermal manikin acting as a heat source in an enclosed room were elaborated using different two-equations turbulence models. The results were mutually compared in order to determine the most appropriate model of turbulence for related types of applications.

3. Heat Source Model Geometry

We used a model of sitting thermal manikin as an example of heat source generating thermal plume in indoor environment. The selection of such exemplar was based on the fact that occupants are usually present in most indoor environments and it is not possible to effectively reduce their heat output. Moreover, their relative contribution to the total heat gains in various buildings is rising [3].

The computational model of thermal manikin resembling a sitting occupant is a detailed copy of metal thermal manikin assembled by Koiš [10] according to the previous prototype of the Centre for Indoor Environment and Energy at DTU in Lyngby, Denmark. The geometry of the manikin differs from the shape of a human body, see Fig. 1. However, such simplified models of occupants are frequently used in indoor environmental studies. Previous experiments of Zukowska et al. [11] indicated that simplification of thermal manikin geometry such as that used by Koiš (regarding the shape of human body) should not significantly affect the character of thermal plume formed above the manikin.

The manikin was placed in the middle of an enclosed room with the floor dimensions $4.5\text{ m} \times 4.5\text{ m}$ and ceiling height of 3 m. The room dimensions and no obstacles around the manikin provided enough space for the thermal plume to develop sufficiently. The same dimensions of an enclosure used Borges et al. [12] in their experiments focused on thermal plume above a human body.

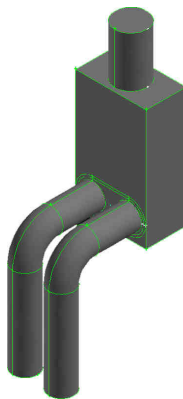


Fig. 1 Model of thermal manikin

4. Boundary Conditions and Other Simulation Parameters

CFD simulations were solved with the CFD software Fluent 6.3 using four different two-equation turbulence models: *k-ε Standard*, *k-ε RNG*, *k-ε Realizable* and *k-ω Standard*. All the turbulence models were applied in the mode without wall functions, which implies that the flow near the walls is solved without further approximations. On the other hand, the boundary layer mesh must be sufficiently fine to obtain reasonably accurate results.

The computational domain was meshed with orthogonal grid, only the areas with more complex geometry (close to the manikin) were meshed by tetrahedral and prismatic cells. The model of thermal manikin was surrounded by cells with edge dimension of 12.5 mm, which were followed up by cells with edge dimension of 25 mm and in more distant areas by cells with edge dimension of 50 mm which filled the remaining volume of the modeled room. The computational mesh was refined in the close vicinity of the thermal manikin surface in order to simulate heat transfer by convection properly. Dimensionless wall distance y^+ was less than 5 for the first five cell layers next to the manikin surface. The computational mesh was refined also in the close vicinity of the walls of the room.

The boundary conditions at the thermal manikin surface were considered as constant. The uniform heat flux from the surface was set to 57.3 W/m^2 , which gives the total sensible heat output of 90 W. The boundary conditions on the walls were identical for all the simulations in order to enable mutual comparison of the results. The surface temperature of the room walls was set to $19 \text{ }^\circ\text{C}$, their emissivity was 0.94 and the emissivity of the thermal manikin surface was 0.98. Heat radiation was simulated using the surface-to-surface (S2S) model [13].

The air flow was treated as unsteady and non-isothermal with the influence of thermal expansion but without effects of compressibility (so called Boussinesq approximation). The *Body Force Weighted* scheme was chosen for the discretization of pressure equation as it is recommended for solving buoyancy driven flows [13]. The convective terms in the solved equations were approximated using the second order upwind scheme. The pressure and velocity fields were coupled by the *SIMPLE* algorithm. The residuals reached within each simulation time step were in the order of magnitude of 10^{-5} or lower.

All the computational cases were evaluated in the same manner. They were simulated for a start-up period of 480 s after which the flow was considered as fully formed and the results were then recorded for further 120 s of simulated time with the time step of 1 s. The outcome of each simulation were 120 data files. The values of temperature, velocity and turbulence parameters were recorded at appropriate points. The final profiles of temperature, velocity and turbulence parameters were averaged over the time of 120 s.

5. Measurements by Thermal Imaging Camera

Air temperature distribution in the thermal plume was measured by a thermal imaging camera and compared with the simulations in order to indicate which turbulence model gives more realistic results. The set up of the experiment is shown in Fig. 2.

The metal thermal manikin was positioned in an enclosed experimental room with the ceiling height of 3 m, same as the height of the room modeled in CFD simulations. The floor dimensions of the experimental room $4\text{ m} \times 8\text{ m}$ were different from the floor dimensions of the room in CFD simulations ($4.5\text{ m} \times 4.5\text{ m}$). However, this difference should not affect the results of the experiment as the floor dimensions should not significantly affect lateral spreading of the thermal plume above thermal manikin.

Ambient temperature in the experimental room was $20.5\text{ }^{\circ}\text{C}$. Thermal manikin was heated up from inside by four light bulbs (placed inside its head, body and each leg), the heat was evenly distributed by forced flow from a fan placed inside the manikin. Total heat output of the manikin was 91 W.

A sheet of plain paper was stretched above the center of manikin's head. The temperature field was continuously transferred from the thermal plume to the paper by convection. Thermal images of the paper surface were recorded using the infrared camera Flir T620 at 1 Hz sampling rate for the total period of 120 s. The time-averaged horizontal temperature profiles were then evaluated at seven different heights above the manikin's head.



Fig. 2 Experimental chamber with thermal manikin

6. Comparison of Results with Different Turbulence Models

Velocity and temperature profiles in the thermal plume above the thermal manikin, simulated with different turbulence models, are compared in Fig. 3 and Fig. 4. The profiles are presented at different heights in vertical plane x - y (front view) intersecting the center of the thermal manikin.

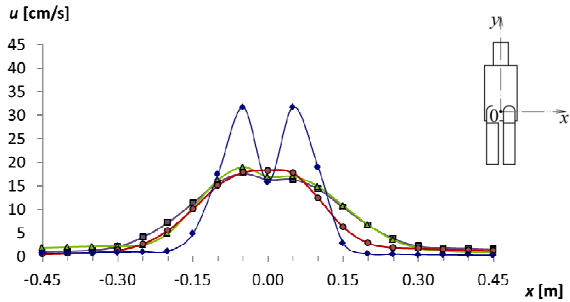
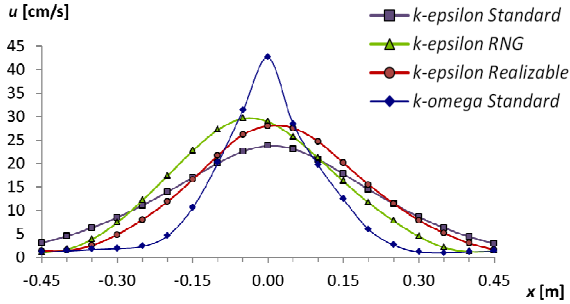


Fig. 3 Velocity profiles in vertical plane x - y , height $y = 0.975$ m (lower) and $y = 1.975$ m (upper)

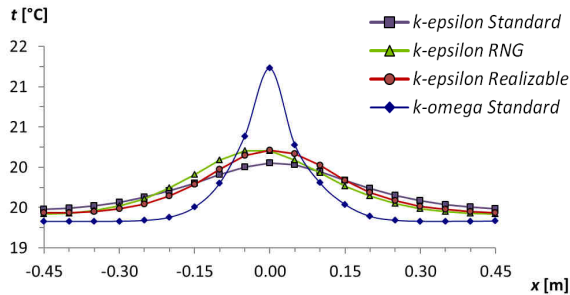


Fig. 4 Temperature profiles in vertical plane x - y , height $y = 1.975$ m

The velocity and temperature profiles obtained from the simulations with all kinds of $k-\varepsilon$ turbulence models are very close to each other. On the other hand, they are very different from the outputs of simulation with the $k-\omega$ *Standard* model, which predicts much higher velocities and temperatures in the thermal plume axis. Their magnitudes also decrease slower with the increasing height than in the case of $k-\varepsilon$ turbulence models.

Although the simulation results based on $k-\varepsilon$ turbulence models are very similar, they are not identical. The most different results were obtained using the $k-\varepsilon$ *Standard* model, in particular at higher levels above the heat source. The thermal plume in this case spreads more rapidly and its maximum velocity at higher levels is lower than in the case of other $k-\varepsilon$ turbulence models.

The velocity profile simulated with the $k-\varepsilon$ *RNG* model shows some asymmetry for the reasons that are not clear yet, see Fig. 3 ($y = 1.975$ m). One of the reasons could be slower oscillation or bigger amplitude of thermal plume wandering in the case with the $k-\varepsilon$ *RNG* model. Another possible reason could be tendency of thermal plume to permanently deviate from its axis to one side. These asymmetries have been noticed also in experiments published by other authors [14].

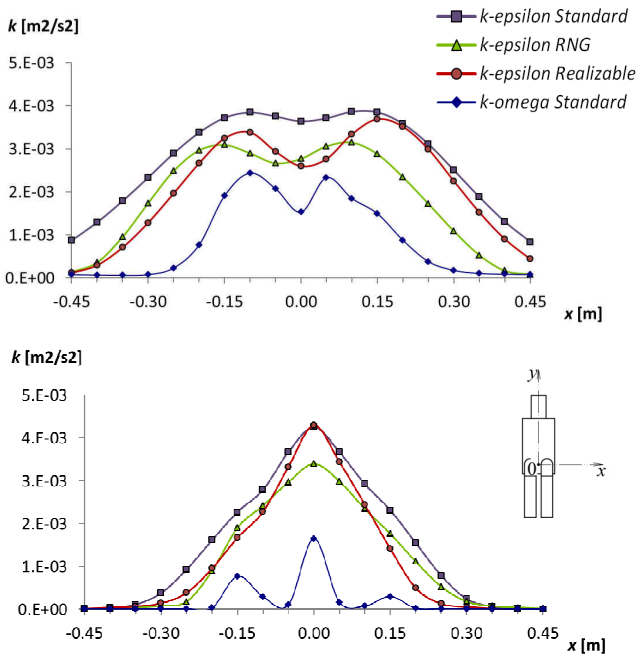


Fig. 5 Turbulence kinetic energy profiles in plane x - y ,
 $y = 0.975$ m (lower), $y = 1.975$ m (upper)

Fig. 5 compares the profiles of turbulence kinetic energy k as simulated with all the turbulence models. It indicates possible reasons for different behavior of the thermal plume when using different turbulence models. The higher is turbulence kinetic energy k the higher is intensity of turbulent mixing in air flow and the higher is the spreading rate of the thermal plume.

It is apparent that the $k-\varepsilon$ *Standard* model produces thermal plume with the highest turbulence intensity which causes the highest spreading rate of the thermal plume. On the other hand, the $k-\omega$ *Standard* model shows the lowest values of k among all the turbulence models. Thermal plume in this case is thus much narrower than in the simulations with $k-\varepsilon$ models.

7. Comparison of Simulations with Measurements

Temperature profiles obtained from the simulations were also compared with the experimental measurement by thermal imaging camera. For this comparison, it was necessary to use non-dimensional temperature profiles, because the ambient temperatures in the experiment and in the CFD simulations were different. Fig. 6 presents the non-dimensional profiles of air temperature defined at given height above the heat source as:

$$\frac{\Delta T}{\Delta T_{max}} = \frac{T(x) - T_{amb}}{T_{max} - T_{amb}} \quad (1)$$

where:

$T(x)$ is the temperature at the distance x from the vertical axis y ;

T_{max} is the maximum temperature in the plume at the given height;

T_{amb} is the ambient temperature.

From the comparison shown in Fig. 6 it follows that the results of simulations with all kinds of $k-\varepsilon$ turbulence models are in a reasonable agreement with the measurements, although at higher positions above the heat source, all the simulated temperature profiles are less flat (or less spread) than those which were measured.

The closest to the experimental data are the temperature profiles simulated with the $k-\varepsilon$ *Standard* model of turbulence. On the other hand, the temperature profile simulated with the $k-\omega$ *Standard* turbulence model was the most different from the measured one, being significantly narrower than the measured one.

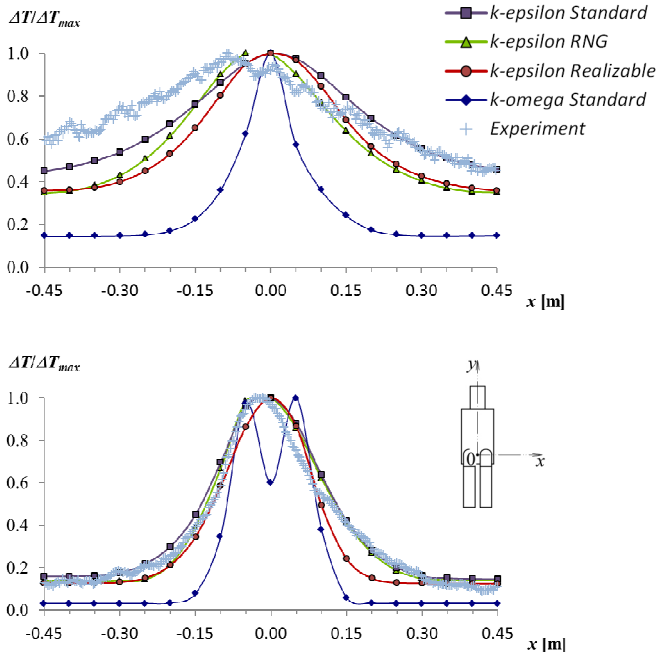


Fig. 6 Non-dimensional temperature profiles in vertical plane x - y , $y = 0,975$ m (lower) and $y = 1,975$ m (upper)

8. Conclusion

The study was focused on the influence of turbulence models on the formation of thermal plumes in CFD simulations of indoor air flow. The results from all the simulations were compared mutually and also with the measured temperature profiles.

The choice of a particular turbulence model affects the turbulent mixing and consequently the spreading rate of thermal plume, the horizontal velocity and temperature profiles and also the decay of the maximum velocity or the maximum temperature along the vertical distance from the heat source.

The turbulence models of k - ϵ type are much more appropriate for the simulation of thermal plumes, then the k - ω *Standard* model, which significantly underestimated turbulent mixing in the thermal plume and produced results which were too different from the measurements.

The k - ϵ *Simple* turbulence model produced results which showed the best agreement with the thermal imaging experiment. The use of this turbulent model can be recommended for the related type of applications, i.e. simulations focused on thermal plumes above indoor heat sources.

Acknowledgement

This project was supported by the research grant of CTU in Prague no. SGS12/179/OHK2/3T/12.

References

- [1] H. B. Awbi. *Ventilation of Buildings*, 2nd ed. London: Spon Press, 2003, p. 563.
- [2] V. Zbořil, A. Melikov, B. Yordanova, L. Bozhkov, and R. Kosonen. Airflow Distribution in Rooms with Active Chilled Beams. In: *Proceedings of the 10th International Conference on Air Distribution in Rooms – Roomvent 2007*, pp. 1–7, Helsinki, Finland, 2007.
- [3] D. Zukowska. Airflow interactions in rooms - Convective plumes generated by occupants [PhD Thesis]. Technical University of Denmark, 2011.
- [4] P. Zelenský, M. Barták, and J. L. M. Hensen. Simplified Representation of Indoor Heat Sources in CFD Simulations. In: *Proceeding of the 13th International Conference of the International Building Performance Simulation Association [in Review]*, Chambéry, France, 2013.
- [5] Z. J. Zhai, Z. Zhang, W. Zhang, and Q. Yan. Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 1 — Summary of Prevalent Turbulence Models. *HVAC&R Research*, vol. 13, no. 6, pp. 853–870, 2007.
- [6] B. E. Launder and D. B. Spalding. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Energy*, no. 3, pp. 269–289, 1974.
- [7] V. Yakhot and S. A. Orszag. Renormalization group analysis of turbulence. *Journal of Scientific Computing*, vol. 1, pp. 3–51, 1986.
- [8] T. Shih, W. Liou, A. Shabbir, Z. Yang, and J. Zhu. A new eddy viscosity model for high reynolds number turbulent flows. *Computers & Fluids*, vol. 24, pp. 227–238, 1995.
- [9] D. C. Wilcox. Reassessment of the scale-determining equation for advanced turbulence models. *AIAA Journal*, vol. 26, pp. 1299–1310, 1988.
- [10] G. Koiš. *Analýza proudů v klimatizovaném prostoru* [MsC Thesis, in Czech]. CTU Prague, Faculty of Mechanical Engineering, 2009.
- [11] D. Zukowska, A. Melikov, and Z. Popiolek. Impact of boundary conditions on the development of the thermal plume above a sitting human body. In: *Proceedings of the 10th REHVA World Congress „Sustainable Energy Use in Buildings“ – Clima 2010*, pp. 2–7, Antalya, Turkey, 2010.
- [12] Borges, C. M., Quintela, D. A., Brites, G. N., Gaspar, A. R., & Costa, J. J. Analysis of thermal plumes generated by a seated person, a thermal manikin and a dummy. In: *Proceedings of the 8th International Conference on Air Distribution in Rooms – Roomvent 2007*, 1, 253–256, Copenhagen, Denmark, 2007.
- [13] Fluent Inc. *Fluent 6.3 User's Guide*, 2006.
- [14] D. Zukowska, A. Melikov, and Z. Popiolek. Thermal plume above a simulated sitting person with different complexity of body geometry. In: *Proceedings of the 10th International Conference on Air Distribution in Rooms – Roomvent 2007*, pp. 191–198, Helsinki, Finland, 2007.