Title	Evaluation of various turbulence models to predict indoor conditions in a naturally ventilated room
Author(s)	Hajdukiewicz, Magdalena; Geron, M.; Keane, MM
Publication Date	2013
Publication Information	Hajdukiewicz, M; Geron, M.; Keane, MM (2013) Evaluation of various turbulence models to predict indoor conditions in a naturally ventilated room The 11th REHVA World Congress & 8th International Conference on IAQVEC (CLIMA 2013) Prague, Czech Republic, 2013-06-16- 2013-06-19
Item record	http://hdl.handle.net/10379/4603

Evaluation of Various Turbulence Models to Predict Indoor Conditions in a Naturally Ventilated Room

Magdalena Hajdukiewicz[#1], Dr. Marco Geron[#2], Dr. Marcus M. Keane[#3]

[#]*IRUSE, Department of Civil Engineering and Ryan Institute*
National University of Ireland Galway, University Road, Galway, Ireland

[1]m.hajdukiewicz1@nuigalway.ie;

[2]marco.geron@nuigalway.ie; [3]marcus.keane@nuigalway.ie

Abstract

In recent years, Computational Fluid Dynamics (CFD) has become a popular tool in building simulation. However, developing reliable CFD models requires a high level of expertise in fluid dynamics and numerical techniques. Furthermore, choosing the right turbulence model is a crucial issue for an accurate CFD analysis. The objective of this work is to utilise Reynolds Averaged Navier - Stokes (RANS) models to predict airflow patterns and air temperature stratification inside an operating naturally ventilated study room, occupied by a person working on a laptop. The paper is a continuation of a recently published work on CFD model calibration; and explores the performance of various turbulence models to accurately simulate indoor conditions. This is done through a comparison of the simulation results with the measurements in a normally operating building. The results of zero equation, standard k-ε, RNG k-ε, k-ε EARSM, standard k-ω and SST k-ω turbulence models are qualitatively analysed and quantitatively evaluated against field measurements performed in a normally operating building. Based on the accuracy and computational stability of the simulations, recommendations are given for the most accurate turbulence model in predicting indoor conditions in a normally operating naturally ventilated room occupied by a person.

Keywords - CFD; field measurements; validation; turbulence model; Reynolds Averaged Navier – Stokes; RANS

1. Introduction

For the last 50 years CFD has progressively become more popular and accessible for research and industry sectors, mainly because of the development and advancement in computing processing power and the availability of commercial software. In spite of the user friendly interfaces and simplicity of the use of commercial codes (when compared to the academic codes), it is essential to ensure the CFD results are realistic [1]. The ability of CFD in dealing with complex flows within built environments has become very important to provide health and safety (e.g. Refs. [2]), thermal comfort for the occupants (e.g. Refs. [3]), test energy efficient designs (e.g. Refs. [4]), or apply required environmental conditions (e.g. Refs. [5]). Zhai [6] summarised typical CFD applications in building design, including site planning, natural ventilation studies, HVAC system designs or pollution dispersion and control. When compared to other methods for predicting ventilation performance in buildings, CFD was found to be the most popular [7].

The accuracy and reliability of CFD predictions is in general a big concern [8]. This problem is even more relevant when numerical simulations of bulding zones are carried out during typical building operating schedules. Many types of errors may be introduced in CFD simulations (i.e. discretisation errors, round off errors, iteration errors, physical modelling and human errors) and need to be considered. Moreover, the simulations of non-controlled environments exhibit uncertainties on the boundary conditions, which can have a strong effect on the reliability of the simulation results. The authors investigated this problem previously [9] by developing a formal calibration procedure to assess the effects of the uncertainty of the boundary conditions on model results in naturally ventilated rooms. In this paper, which is a continuation of the work carried out previously [9], an analysis of the accuracy and robustness of various Reynolds Average Navier - Stokes (RANS) turbulence models in a naturally ventilated space during a typical building operating schedule is carried out. The overall aim is to evaluate the effectiveness of different standard turbulence models in predicting flow patterns and temperature profiles in naturally ventilated room. This is done in a real-life scenario where only a limited number of measurement points are available.

2. Turbulence Modelling

Direct numerical simulations (DNS) of the Navier - Stokes equations are still too computationally demanding for typical engineering flows, even with the incredible level of computational power reached nowadays. Nevertheless, RANS models have shown the ability to accurately predict engineering flows in different enviroments. When RANS equations are utilised a new unknown variable is introduced (Reynolds stress tensor) and the system of equations is not “closed” anymore. Thus, there is a necessity to introduce turbulence models to “close” the system of equations. The turbulence models have to bridge the wide scale gap that exists between direct numerical simulations (DNS) and RANS models. It is evident how there can not be a single turbulence model or turbulence approach used to bridge this gap for different engineering flows. Consequently, the choice of the right turbulence model for the engineering flow of interest becomes crucial for creating a reliable and accurate CFD simulation. The performance of various turbulence models for modelling airflows in built environments was assessed by previous research. A comprehensive study [10] evaluated, in terms of accuracy and computational cost, the performance of eight turbulence approaches (RANS, large eddy simulation (LES) and detached eddy simulation (DES)) to simulate indoor airflow. The accuracy of models was evaluated by validating the CFD results with experimental data available in literature. Based on the study figure 1 summarises tested turbulence models that proved good and acceptable accuracy for various airflow types.

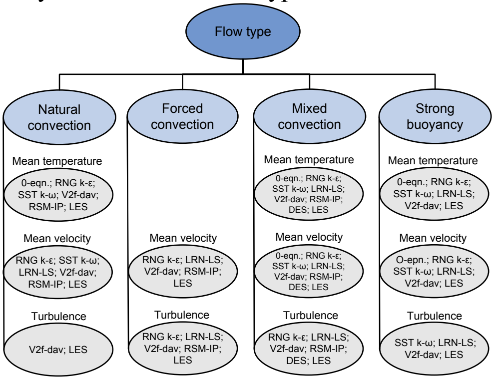

Fig. 1 The best suited turbulence models for various airflow types [10].

Previous research has provided recommendations regarding turbulence modelling in CFD analysis of natural ventilation systems. In general, good predictions for naturally ventilated spaces were achieved with both the k-ε and k-ω family [11], but more accurate results were observed when LES models were utilised. However, this can increase the computational cost, which could be demanding if a formal calibration procedure is utilised [9].

The objective of this paper is to analyse the accuracy and robustness of various RANS turbulence models in a naturally ventilated space during a typical building operating schedule. Six turbulence models are tested. Five of them are based on the Boussinesq eddy-viscosity approximation (zero equation, standard k-ε, RNG k-ε, standard k-ω and SST k-ω), while one (k-ε EARSM) is based on a non-linear constitutive relationship for the eddy viscosity. In the explicit algebraic stress model (k-ε EARSM) the constitutive relationship is derived from a simplified Reynolds stress equation [12]. Non-linear formulation can improve sensibly the flow predictions when strong curvature and rotation effects are present. The results of several turbulence models are qualitatively compared, and the ability of each model to capture indoor airflow phenomena is estimated. Moreover, the performance of the turbulence models is quantitatively evaluated using field measurements in a normally operating study room.

3. Study Room Description

The demonstrator used in this research is a naturally ventilated study room in the library building at the National University of Ireland (NUI) in Galway, Ireland (figure 2). The dimensions of the room are 2.70m (D) x 4.46m (L) x 3.10m (H). The external wall consists of windows. The internal wall, opposite to the external wall, contains a glass surface and an open door leading to an open reading space. The remaining two internal walls border with other study rooms, similar to the one modelled. The 3D geometry of the study room was created based on the technical documentation and site visits. Figure 2 shows the level of detail in the geometry of the modelled room, including the locations of air temperature (red) and air speed (green) sensors utilised in field measurements.

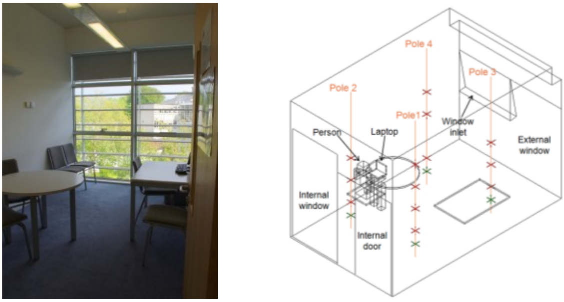

Fig. 2 Modelled study room

4. Field Measurements

A reliable CFD model should be validated with trusted experimental data. In order to support the calibration procedure, this research performed experiments in a normally operating study room exposed to outdoor conditions. The boundary conditions for the CFD model were provided by an automatic weather station installed at the NUI Galway campus and an air speed sensor placed at the centre of the window opening. In order to validate the CFD model a network of thirteen wireless air temperature sensors (red) and four air speed (green) sensors was deployed (figure 2). Characteristics of the sensor types utilised are reported in Refs. [13]. The air temperature sensors were deployed in four horizontal layers to observe the air temperature stratification inside the room. The air speed sensors were located at one horizontal layer near the floor level, where the highest air speeds were expected. All the sensors were deployed in locations where the measurements best described the influence of air speeds and air temperatures on the occupant’s thermal comfort. The accuracy of indoor air speed and air temperature measurements were ±0.01m/s between 0.05–1m/s and ±0.35^{o}C in a range between 0^{o}C and 50^{o}C respectively.

In order to gather data that supported the development and validation of the model, the experiment in the study room was performed on a cloudy day of April 10th, 2011. The external weather conditions, as well as indoor air speeds and air temperatures, were monitored throughout the day. The outdoor air entered the room through the open window. The open internal door allowed for airflow between the modelled room and the adjacent open plan space. The modelled study room was occupied by a sitting person working on a laptop. The measurements were taken during the typical operation of the building and maintaining steady conditions inside the room was difficult. Thus, the longest continuous period of the steady measurement conditions occurred over the 12 minutes at noon, when the outdoor and indoor conditions were relatively steady. This measurement period was chosen to be used in the CFD model. The average values of the measurements provided both, the boundary conditions for the model and the data utilised to validate the CFD results.

5. CFD Model

The CFD simulations of the study room in Nursing Library building were performed using the commercial software Ansys CFX v.12.1. The airflow and air temperature stratification were simulated in a naturally ventilated room occupied by a person. A RANS steady state model was utilised for each simulation carried out. A convergence factor of 0.01% of residuals, an energy conservation target of 1% and the monitor of air temperatures at the measurement points were utilised to deem the solution converged. The zero equation, standard k-ε and RNG k-ε models were developed with a high resolution discretisation scheme. Due to the convergence issues a blend factor of 0.75 was utilised in the standard k-ω and k-ε EARSM models, while a factor of 0.7 in the SST k-ω model. The maximum Y^+ along the walls and windows was 61.3 with an average value of 14.9, while for the laptop it was about 9.5. The Y^+ values for the person were slightly higher with a mean value of about 35. Those values were within the acceptable limits when wall functions were utilised. The step by step development of the CFD model, together with the grid independence verification, validation and calibration are available in the article published previously [9].

6. Results

Figures 3 and 4 present a qualitative comparison between the results obtained with the different turbulence models considered. In figure 3 the air temperature stratification in a vertical slice passing through the sitting person is shown. Figure 4 illustrates the velocity vector distribution for the same slice. The general airflow patterns and temperature distribution were similar for all turbulence models. However, differences between the various turbulence models were evident in the prediction of the recirculation zones underneath the window and behind the person. The zero equation model predicted a stronger downward velocity for the flow entering through the window and generated a smaller recirculation zone underneath the window inlet. Also, more cold air was engulfed in the recirculation zone when compared to the other turbulence models. The results achieved with the standard k-ε and RNG k-ε were qualitatively similar and no major differences could be highlighted between those two turbulence models. The k-ε EARSM model generated a smaller recirculation zone underneath the window inlet and a stronger fluid flow close to the floor than the standard k-ε model. The vortex behind the person was also different due to the interaction of the eddy with the stronger flow along the floor. The standard k-ω and SST k-ω turbulence models presented, as expected, strong differences between the evaluation of the recirculation zone underneath the window inlet and behind the person. The two models indicated the formation of two counter-rotating vortices behind the person, which were not calculated by other models.

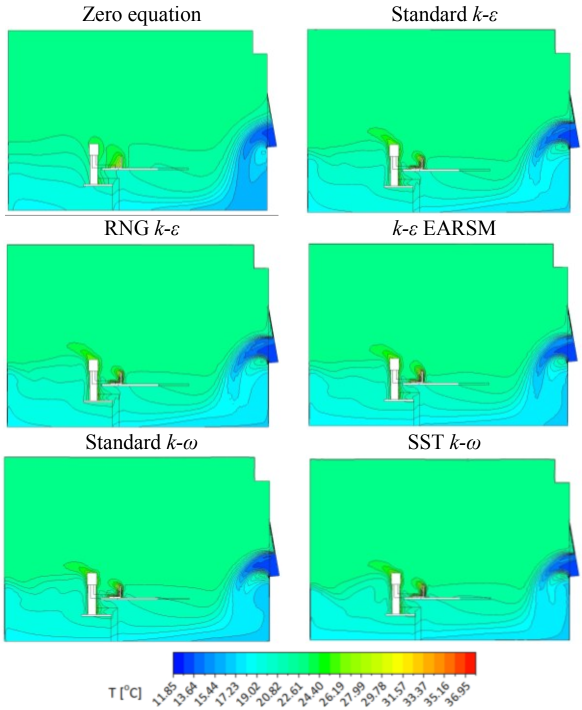

Fig. 3 Air temperature stratification inside the room

After a qualitative comparison between the different turbulence models a quantitative analysis was carried out in figures 5 and 6. Figure 5 compares the temperature profiles along the 4 vertical poles inside the room with the gathered experimental data. The experimental data error bars were evaluated considering the accuracy of the instruments and the standard deviation of the data analysed. The temperature profiles at pole 1 were almost overlapping and quite close to the experimental measurements. The temperature variation at the pole 2 presented a different profile between 1 and 1.5 m from the floor level at the recirculation zones' locations. Nevertheless, no turbulence model was able to predict the steep temperature reduction observed with the experimental data. It is believed that this could have been a consequence of the pole being positioned behind the person. In this case the person geometry approximation may have been simplified too much. The measured and simulated results at pole 3 were almost overlapping but all turbulence models failed to predict the air temperature at the lowest measurement level. All of the turbulence models tested accurately predicted the temperature profiles at pole 4. Generally, the air temperatures were overestimated at the upper sensor levels and underestimated at the lowest level by the investigated turbulence models. Measured data did not show as large air temperature variation as the CFD results.

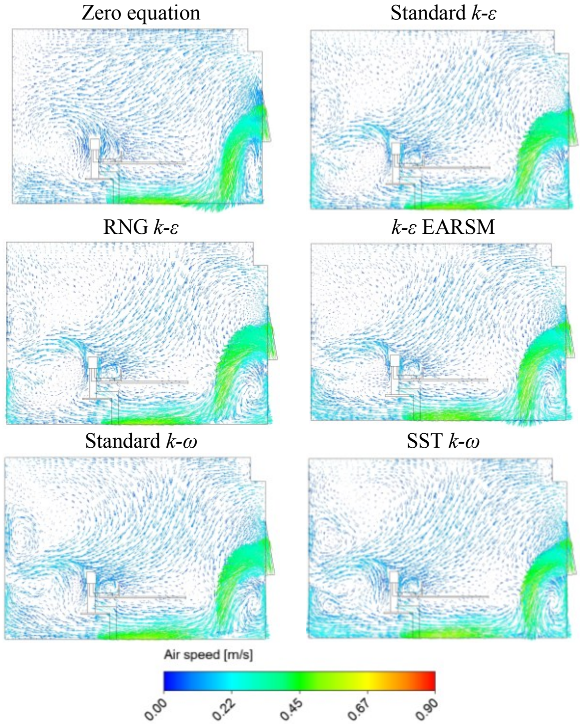

Fig. 4 Airflow distribution inside the room

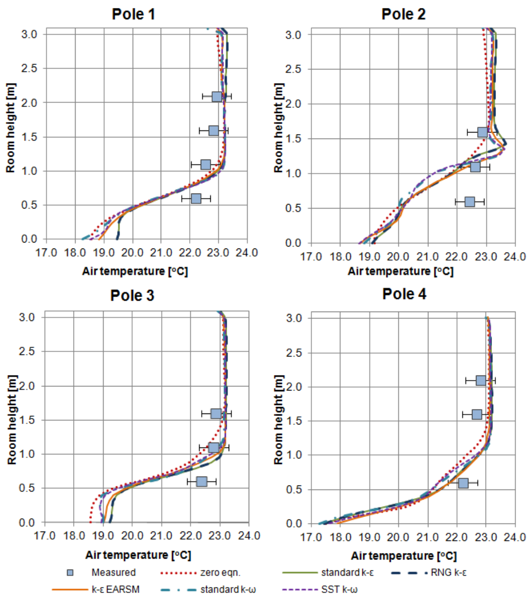

Fig. 5 Indoor air temperature profiles (range of measured data shown)

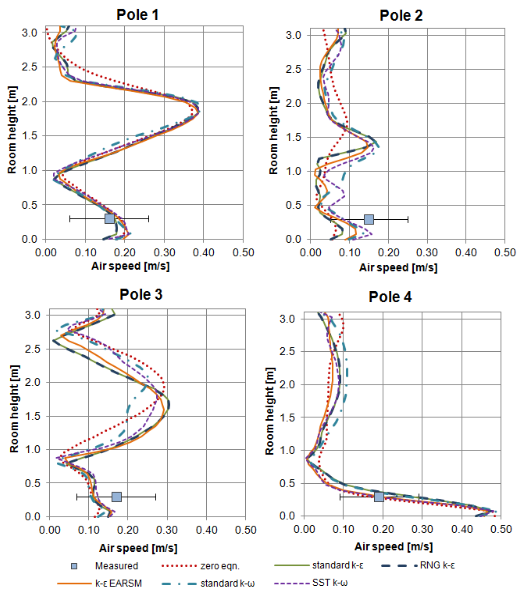

Fig. 6 Indoor air speed profiles (range of measured data shown)

The quantitative analysis was also carried out for the velocity profiles predicted along the four poles (figure 6). The different turbulence models presented an uneven distribution of the velocity profiles especially for the poles closer to the recirculation zones previously highlighted (pole 2 and 3). In general, the zero equation model showed a flatter profile, which is quite most evident at pole 2. The standard *k-ω* model also showed significant differences, from the other turbulence models predictions, on the velocity profile in proximity of the two recirculation zones. At the measurement locations all turbulence models behaved similarly and fitted within the error band of measured data. However, the distribution of the results at the pole 2 was more diversified and the *k-ω* family models appeared to match the air speed measurement best.

A quantitative comparison of the turbulence intensity (TI) is also carried out and reported in table 1. The TI plays an important role in the user comfort analysis, being associated with the "draftiness" sensation of the moving air.

Table 1. Turbulence intencities (TI) [%] at the air velocity measurement locations

Location	Measured	Stand. k-ε	Err%	RNG k-ε	Err%	k-ε EARSM	Err%	Stand. k-ω	Err%	SST k-ω	Err%
Pole 1	50.0	19.8	60.4	19.8	60.4	21.3	57.4	18.0	64.0	13.6	72.8
Pole 2	46.7	61.1	-30.8	61.1	-30.8	43.6	6.6	45.6	2.4	43.3	7.3
Pole 3	58.8	48.1	18.2	48.1	18.2	50.3	14.4	37.1	36.9	32.6	44.5
Pole 4	47.4	33.5	29.3	33.5	29.3	38.7	18.3	29.9	36.9	39.8	16.0

It can be seen that, generally, the TI is strongly underestimated at pole 1 for all turbulence models. The standard and RNG *k-ε* models predict the same values of TI and perform the worst, when compared to the other turbulence models. The *k-ε* EARSM model predicts the TI for the poles 2, 3 and 4 well. The standard *k-ω* model accurately predicts the TI at pole 1 but fails at the other locations; while at poles 1 and 3 the TI is predicted badly by the SST *k-ω* model.

7. Conclusions

This study significantly progresses recently published work on CFD model calibration [9] and presents a comparison between various RANS turbulence models used to predict airflow patterns and air temperature stratification inside a naturally ventilated study room. The performance of turbulence models in accurately simulating indoor conditions was evaluated against field measurements in a normally operating building.

From the qualitative point of view, the results showed some substantial differences in the airflow recirculation zones. However, those differences did not affect the quantitative comparison with the measurements. As opposed to a detailed CFD analysis, such as in biomedical or aeronautical sectors, built environment simulation allows only for limited number of sensors deployed in an operating building in order to validate model predictions. This paper analyses such circumstances and indicates that the zero equation model can give a good first approximation of indoor environmental conditions while simultaneously being computationally efficient and robust at the same time. The other turbulence models seemed to play a more significant role when recirculation zones were present. No differences were observed between the two k-ε models. However, different vortex structures were predicted by the k-ω family. In order to determine the most accurate model to predict the particular environment, the CFD simulation needs to be validated with data obtained from a larger quantity of measurement points.

8. Acknowledgment

This research was funded by the Irish Higher Education Authority (HEA) through the NEMBES Project.

9. References

[1] P.D. Gaspar and R.A. Pitarma. Evaluation of CFD codes by comparison of numerical predictions of an air-conditioned room case study. Advanced Computational Methods in Heat Transfer VIII, vol.5, 2004.

[2] C.C.K. Cheng, K.M. Lam, R.K.K. Yuen, S.M. Lo and J. Liang. A study of natural ventilation in a refuge floor. Building and Environment 2007, 42(9): 3322–3332.

[3] T. Catalina, J. Virgone and F. Kuznik. Evaluation of thermal comfort using combined CFD and experimentation study in a test room equipped with a cooling ceiling. Building and Environment 2009, 44(8): 1740–1750.

[4] A. Alajmi and W. El-Amer. Saving energy by using underfloor-air-distribution (UFAD) system in commercial buildings. Energy Conversion and Management 2010, 51(8): 1637–1642.

[5] K.C. Noh, H.S. Kim and M.D. Oh. Study on contamination control in a minienvironment inside clean room for yield enhancement based on particle concentration measurement and airflow CFD simulation. Building and Environment 2010, 45(4): 825–831.

[6] Z. Zhai. Application of Computational Fluid Dynamics in building design: Aspects and trends. Indoor and Built Environment 2006, 15(4): 305–313.

[7] Q. Chen. Ventilation performance prediction for buildings: A method overview and recent applications. Building and Environment 2009, 44(4): 848–858.

[8] AIAA. Guide for the verification and validation of Computational Fluid Dynamics simulations. American Institute of Aeronautics and Astronautics, 1998.

[9] M. Hajdukiewicz, M. Geron and M.M. Keane. Formal calibration methodology for CFD models of naturally ventilated indoor environments. Building and Environment 2013, 59(0): 290-302.

[10] Z. Zhang, W. Zhang, Z. Zhai and Q. Chen. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 2: Comparison with experimental data from literature. HVAC & R Research 2007, 13(6): 871–886.

[11] J.M. Horan and D.P. Finn. CFD reliability issues in the prediction of airflows in a naturally ventilated building. International Journal of Ventilation 2005, 4(3): 255–268.

[12] D.C. Wilcox. Turbulence modeling for CFD. California, U.S.A.: La Cãnada, DCW Industries, Inc., 3rd ed., 2006.

[13] M. Hajdukiewicz, M. Walsh and M.M. Keane. Formal calibration methodology for a CFD model of a naturally ventilated room. In: Proc. 12th International Building Performance Simulation Association Conference, Building Simulation 2011, Sydney, Australia, 14-16 November 2011.