

Application of open-source CFD software to the indoor airflow simulation

Cong Wang^{*1}, Sasan Sadrizadeh^{1,2}, and Sture Holmberg¹

*1 Royal Institute of Technology
KTH, Brinellvägen 23
Stockholm 10044, Sweden*

*2 Lawrence Berkeley National Laboratory
1 Cyclotron Road, Berkeley
California 94720, USA*

**Corresponding author: congwang@kth.se*

ABSTRACT

The use of open-source CFD has been growing in both industry and academia. Open-source CFD saves users a considerable license cost and provides users with full transparency of implementation and maximum freedom of customization. However, it is often necessary to assess the performance of an open-source code before applying it to the practical use. This study applies one of the most popular open-source CFD codes – OpenFOAM to the indoor airflow and heat transfer prediction. The performance of OpenFOAM is evaluated and validated against a well-documented benchmark test. Various OpenFOAM built-in turbulent viscosity models are attempted within the framework of Reynolds Averaged Navier-Stokes Simulation (RANS) approach and the simulation results are compared to the experimental data. Among all models, the $k - \omega SST$ model has shown the best overall performance, whereas the standard $k - \epsilon$ model is the most robust one despite its deficiencies. The results of this study demonstrate the capability of OpenFOAM in the field of indoor air simulation and promote users' confidence in using OpenFOAM in their research work.

KEYWORDS

Computational Fluid Dynamics, OpenFOAM, indoor air flow, thermal manikin, turbulence models

1 INTRODUCTION

Accurate prediction of air movement, temperature and contaminant distribution as well as many other parameters of air flow in an enclosed environment is of paramount importance in assessing occupants' comfort and health. Due to the prohibitive cost of laboratory experiment, Computational Fluid Dynamics (CFD) has nowadays been increasingly utilized to simulate the indoor climate. Chen & Srebric (Chen & Srebric, 2000) summarized indoor and outdoor environment studies conducted by using CFD and concluded that CFD was a powerful tool to predict airflows in the built environment. Gao & Niu (Gao & Niu, 2004) summarized various issues regarding CFD studies of the thermal environment around a human body. Norton et al. (Norton et al. 2007) reviewed the applications of CFD in the ventilation design for the agricultural industry. Sadrizadeh et al. (Sadrizadeh et al. 2014) employed CFD to investigate the transport and dispersion of airborne bacteria in an operating theatre.

The majority of these research studies reply on closed-source and commercial CFD tools, such as ANSYS FLUENT, ANSYS CFX, STAR-CCM+, etc. Users benefit from using commercial CFD codes in many aspects. Commercial CFD codes are verified extensively by different user groups and validated against various benchmark tests. In addition, users can easily get support from the commercial CFD vendors when they are faced with code-related problems, an important advantage especially for inexperienced users. Therefore, users tend to gain confidence and feel proficient in using commercial CFD codes. Despite these merits, however, one major disadvantage of proprietary CFD packages is that they provide limited insight and serve as a black-box, that is, the implementation details are not transparent to users and users have limited or even no access to the code. Moreover, the use of commercial

CFD packages usually involves a considerable license cost and license fees increase with increasing use. As an alternative to commercial CFD packages, various open-source CFD codes have been developed and become available to the public. Most open-source CFD packages are free-licensed, which allows anyone to use the software and source code for free. Most importantly, open-source CFD tools offer users full transparency of technology and maximum freedom of customization, With Open-source CFD, users can easily implement a new technology and adapt the code to their own need. Among all, OpenFOAM is one of the most popular general-purpose open-source CFD codes, which is an object-oriented C++ tool box originally developed for the Finite Volume (FV) method (Jasak 1996; Weller et al. 1998). OpenFOAM has been reported to be adequate for a broad range of fluid dynamics applications (Jasak et al. 2007) and have good parallel scalability for both simple test cases (Axtmann & Rist 2016) and complex industrial cases (Lui 2015). OpenFOAM is also supplied with pre- and post-processing utilities, which permits a complete analysis and solution of problems within a consistent software package.

OpenFOAM has found its way into the automotive and railway industry to resolve aerodynamics (Blacha et al. 2016), aeroacoustics (Zörner et al. 2010; Wang 2013) and combustion (Migliaccio et al. 2009). Despite its established role in these fields, few applications of OpenFOAM to the indoor environment simulation have been found until recently. Ramechecandane et al. (2010) utilized OpenFOAM to investigate the particle dispersion with the presence of an inhomogeneous electric field. Konstantinov et al. applied OpenFOAM to the numerical simulation of the airflow and thermal comfort in a passenger car cabin (Konstantinov & Wagner 2016), a train cabin (Konstantinov & Wagner 2014), and an aircraft cabin (Konstantinov et al. 2014). Some researchers have developed innovative models and technologies of their own within the OpenFOAM framework. For instance, Liu et al. (2016) implemented a fast fluid dynamics (FFD) model based on OpenFOAM to simulate the indoor airflow. Similarly, Xue et al. (2016) proposed a new semi-Lagrangian-based PISO method implemented in OpenFOAM for fast and accurate indoor environment modelling.

One major disadvantage with open-source CFD is that their performance may not be well validated. Bugs or inappropriate implementation may exist in open-source CFD codes. As regards OpenFOAM, due to the limited number of applications in indoor airflow simulation, it is of value to assess the accuracy and robustness of OpenFOAM before applying it to practical use. Ito et al. (Ito et al. 2015a) validated several commercial CFD packages as well as OpenFOAM against a benchmark test that were performed on a non-isothermal flow in a 3D room, in which OpenFOAM achieved the closest match with the experimental data. Ito et al. (Ito et al. 2015b) further conducted a comprehensive analysis of cross-ventilation and floor-heating-induced natural convection using OpenFOAM and compared the simulation with measurements. Despite these existing studies, more extensive benchmark tests are still desirable to validate OpenFOAM as a useful tool in indoor environment simulation. This study aims to provide more validation evidence for OpenFOAM by performing a CFD simulation of airflow and heat transfer around a thermal manikin.

Thermal manikins have been broadly used in the indoor environment research in both laboratory experiment and numerical simulation. This study attempts to simulate a well-documented benchmark test of a thermal manikin (Nilsson et al. 2007) using OpenFOAM. Similar studies exist in the literature. Martinho et al. (Martinho et al. 2012) analyse the influence of various factors on the accuracy of CFD by comparing simulation results obtained from ANSYS CFX to the experimental data. Sadrizadeh & Holmberg (Sadrizadeh & Holmberg, 2016) used the same benchmark test to evaluate the performance of nine

turbulence models in ANSYS FLUENT. Ito et al. (Ito et al. 2015c) validated the simulation results obtained from Star-CD against the same benchmark test.

The complex state of indoor airflow and heat transfer requires appropriate turbulence treatment. Due to the low computational cost and moderate accuracy, RANS remains to be the dominant approach. Therefore, this study tests a number of OpenFOAM built-in turbulent viscosity models within the RANS framework against the experimental data.

2 METHODOLOGY

2.1 Experimental Setup

The experimental benchmark test on a thermal manikin was performed through cooperation between the Aalborg University in Denmark and Gävle University in Sweden (Nilsson et al. 2007). The experimental apparatus is a box-shape chamber with a dimension of 2.44 m × 2.46 m × 1.2 m. The chamber is made of 12-mm wood with a thermal conductivity of 0.15 W/(m·K) and the coefficient of convective heat transfer between the chamber wall and the outdoor air is 10 W/(m²·K) (Ito et al. 2015c). A sitting-posture manikin was placed along the centre line of the chamber, facing the inlet at a distance of 0.7 m away from the toe. The manikin was kept at a constant temperature of 34 °C without clothing or hair to facilitate the heat transfer. The incoming air was supplied over the full cross-section at a mean velocity of 0.27 m/s and mean temperature of 20.4 °C. Two circular exhaust outlets of a diameter 0.25 m were positioned on the rear wall with one 0.6 m away from the ceiling and the other 0.6 m away from the floor. Figure 1 shows the experiment rig and geometry of the simulation.

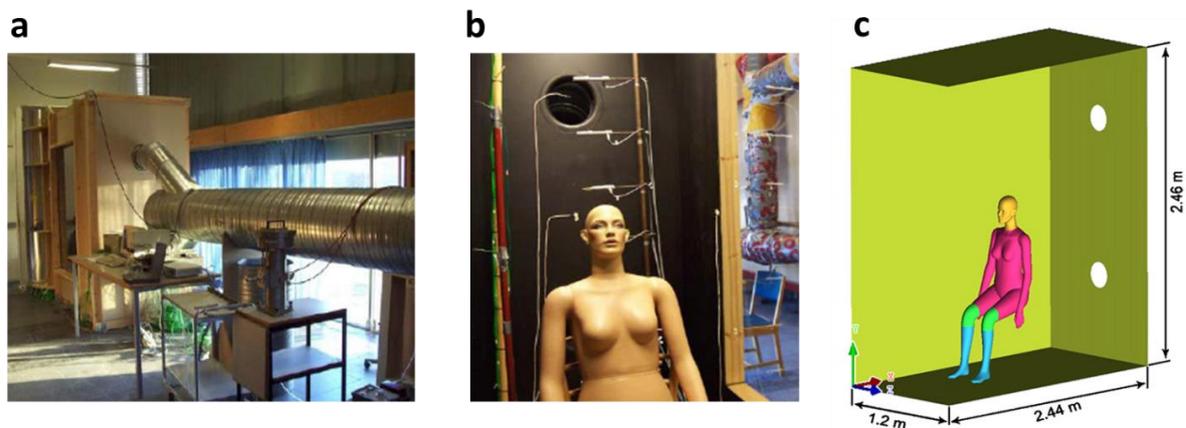


Figure 1: a) Experimental setup; b) Thermal manikin; c) CAD geometry for simulation.

The temperature and velocity distribution were measured at different locations along several vertical lines L1 – L4, as shown in Figure 2 a). In particular, the temperature is measured at L2 and L4, whereas the velocity is measured at L1 and L3. The vertical distribution of the temperature of the air is also measured 1.08 m ahead of the inlet, which has been used as inlet boundary condition for temperature in CFD simulation, as presented in Figure 2 b). The heat loss from the manikin to the ambient is also measured and documented.

2.2 Numerical Simulation

The numerical simulation is performed using OpenFOAM 2.3.1, which attempts to replicate the conditions in the experiment. The setup of the simulation is described in the section.

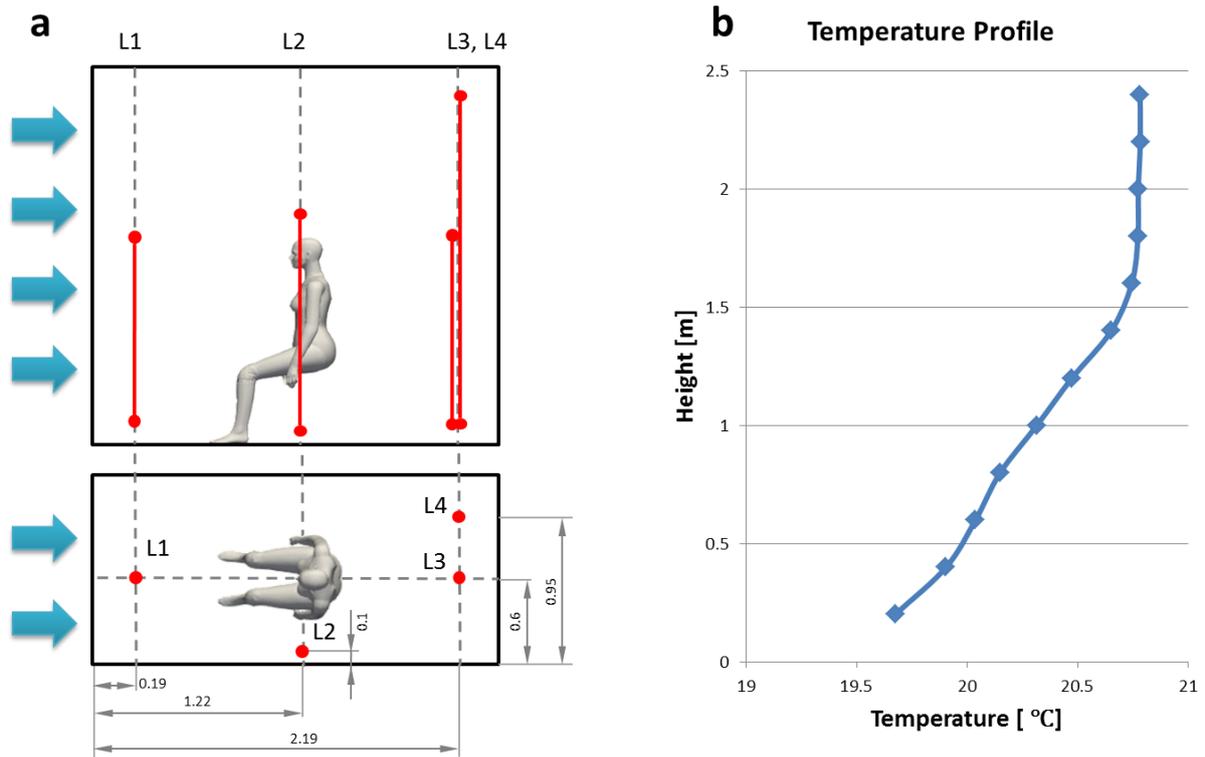


Figure 2: a) Locations of measurement lines; b) Temperature profile measured ahead of the inlet.

2.2.1 Mesh

The mesh is created using OpenFOAM native mesher – *snappyHexMesh*, which generates 3-dimensional hexahedra dominant meshes from triangulated geometries. The *snappyHexMesh* meshing utility adopts the cut-cell method, which is a top-down approach in contrast to the well-known bottom-up approaches such as the Delaunay or Advancing Front. A Cartesian mesh is firstly created as the base mesh and the hexahedra cells are then split and snapped to the geometry surfaces. Such an approach shows relative robustness in handling dirty geometries and favourable parallel scalability, and more importantly facilitates the automation of the meshing process. However, the cut-cell approach suffers from lack of control over surface meshes and limited robustness in boundary layer inflation. The limited capability of generating perfect boundary conformal meshes and failure in boundary layer addition are continually reported in the community of *snappyHexMesh* users.

The created mesh contains a total number of 1.4 million cells. The size of the cells of the base mesh is approximately 6.7 cm and the cells are continuously refined approaching the manikin and walls. To ensure the grid independence, the simulation is also performed on a coarser mesh and a finer mesh respectively.

2.2.2 Boundary Conditions

While a temperature profile presented in Figure 2 b) is used as the inflow boundary condition for temperature, the velocity is assumed to be 0.27 m/s uniform over the whole cross-section. The turbulence intensity and mixing length are specified at the inlet. Following Ito et al. (Ito et al. 2015c), the turbulence intensity is assumed to be 5% and the mixing length 0.16 m. The boundary condition for temperature at the surface of manikin is constant 34 °C and at walls makes the walls transfer heat with outdoor air at a temperature of 20.4 °C. For the calculation of radiation, the emissivity is set as 0.9 for all solid surfaces and 1.0 for inlet and outlet.

2.2.3 Turbulence Models

As pointed out in the previous section, RANS is the dominant approach in indoor air flow and heat transfer simulation. The RANS approach relies on the appropriate selection of turbulent viscosity models. Therefore, various turbulent viscosity models available in OpenFOAM are used in this study. Specifically, this study considers five two-equation models – the standard $k - \varepsilon$ model, the RNG $k - \varepsilon$ model, the realizable $k - \varepsilon$ model, the Launder-Sharma low-Re $k - \varepsilon$ model and the $k - \omega$ SST model; one four-equation model – the $\overline{v^2} - f$ model. The applications of these models are commonly found in the literature.

The standard $k - \omega$ SST model, which solves two semi-empirical transport equations for the turbulence kinetic energy (k) and the turbulence dissipation rate (ε), has a broad range of applications and has been reported to provide acceptable accuracy for simple flows. Derived using the renormalization group (RNG) method purely from the instantaneous Navier-Stokes equations, the RNG $k - \varepsilon$ model solves a slightly different equation for ε and has been found to exhibit better performance in indoor air flow simulations than the standard one (Chen 1995). Different from the standard and RNG $k - \varepsilon$ models, the realizable $k - \varepsilon$ model is ‘realizable’ in terms of the consistency with the physics of turbulent flow, which satisfies certain mathematical constraints on turbulence quantities. The Launder-Sharma model is the most widely used low-Re $k - \varepsilon$ model, which solves the transport equation for a modified dissipation rate and is supposed to account for the damping effect in the near-wall region. The $k - \omega$ SST model solves for the specific dissipation rate ω (also referred to as turbulent frequency) and combines the best behaviour of the $k - \varepsilon$ model in the freestream and the standard $k - \omega$ model in the near-wall region. The $\overline{v^2} - f$ model solves two additional equations for a velocity scale v^2 and a relaxation function f , which takes into account the near-wall anisotropy and non-local pressure-strain effects. The model has been successfully applied in the literature to the indoor ventilation design (e.g. Chen et al. 2013).

2.2.4 Solvers

There are a number of solvers available in OpenFOAM that are designed for heat transfer problems. As the buoyant effect is significant in this benchmark test and the temperature difference is not very small, the *buoyantSimpleFoam* solver is used to solve the fluid flow. *buoyantSimpleFoam* is a steady-state solver that directly solves compressible turbulent flows to account for buoyancy without imposing the Boussinesq approximation on the fluid.

To solve the radiative heat transfer, the finite volume discrete ordinate method (fvDOM) is employed, which solves the radiative transfer equation for a discrete number of finite solid angles using the finite volume method. In this study, the angular space 4π is discretized into 20 azimuthal angles and 7 polar angles, leading to a total number of 140 equations to be solved. The *buoyantSimpleFoam* solver together with appropriate boundary conditions automatically handles the coupling between the convective and radiative heat transfer.

3 RESULTS AND DISCUSSION

The velocity distribution at two vertical lines – L1 and L3, is presented for both the simulation and experiment in Figure 3. As a uniform velocity profile of 0.27 m/s is specified at the inlet in the simulation, the predicted velocity at L1 deviates very little from the inlet, as shown in Figure 3 a). However, small discrepancies can be observed between simulated and measured values, which may imply that a more precise boundary condition for the velocity should be applied to the inlet. Figure 3 b) shows that the velocity values at L3 predicted by different turbulence models are nearly identical, but slightly deviate from the experimental

results. All models tend to over-predict the velocity at lower height but under-predict the velocity at larger height and close to the floor.

Figure 4 presents the predicted and measured temperature profiles at L2 and L4. As with the velocity profiles, different turbulence models give rather similar temperature profiles. At L2, the temperature is underestimated for all locations except for the one close to the floor. At L4, all simulations overestimate the temperature at large height and underestimate the temperature at medium and low height. Similar to L2, the temperature at the floor level is significantly overestimated. This remarkable discrepancy can be ascribed to the deficiency of the inlet boundary condition for temperature. As can be seen from Figure 2 b), temperature measurement at the floor level is not available.

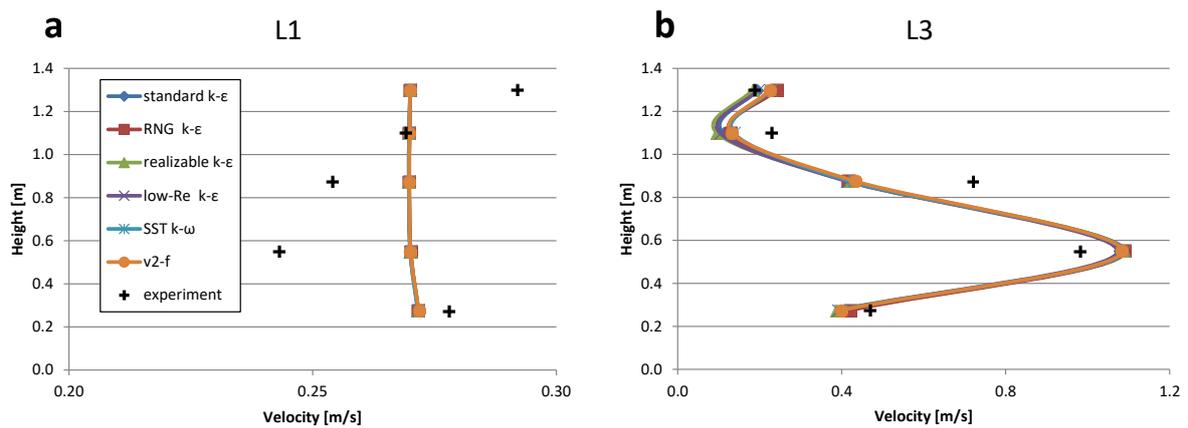


Figure 3: Velocity profiles at a) L1 and b) L3.

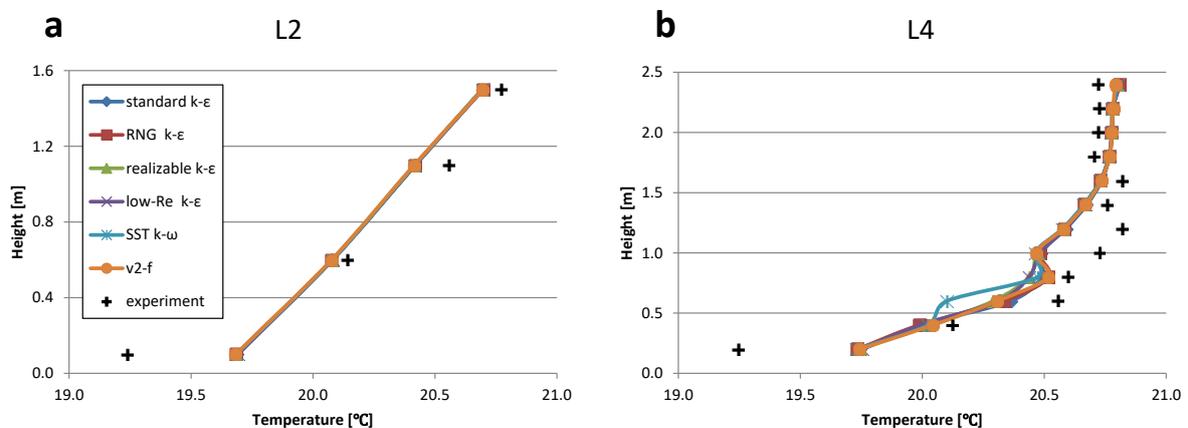


Figure 4: Temperature profiles at a) L2 and b) L4.

Figure 5 compares the predicted heat flux that the manikin releases to the surroundings with the experimental data. All turbulence models overestimate the total heat flux from the manikin. Not surprisingly, the most remarkable deviation occurs to the standard $k - \epsilon$ model, which over-predicts the heat flux by as high as 40 W/m^2 . The best match to the measurement belongs to the $k - \omega$ SST model, followed by the $\overline{v^2} - f$ model. The RNG $k - \epsilon$ model and realizable $k - \epsilon$ model perform fairly well and, as expected, exhibit superiority over the standard one. Although the performance of the Launder-Sharma low-Re $k - \epsilon$ model is better than the standard $k - \epsilon$ model, it is not as good as that of other models. It is noteworthy that the values of radiative heat flux predicted by different simulations are nearly identical,

ranging from 36% (standard $k - \epsilon$ model) to 43% ($k - \omega SST$ model) of the total heat flux, which is consistent with Martinho et al. (Martinho et al. 2012) who finds out that the radiative heat flux accounts for approximately 40% of the total heat flux. The different total heat fluxes result mainly from the different values of predicted convective heat flux. As the measurement does not distinguish the radiative heat flux from the convective one, however, it is hardly possible to evaluate the accuracy of convective and radiative heat flux prediction individually.

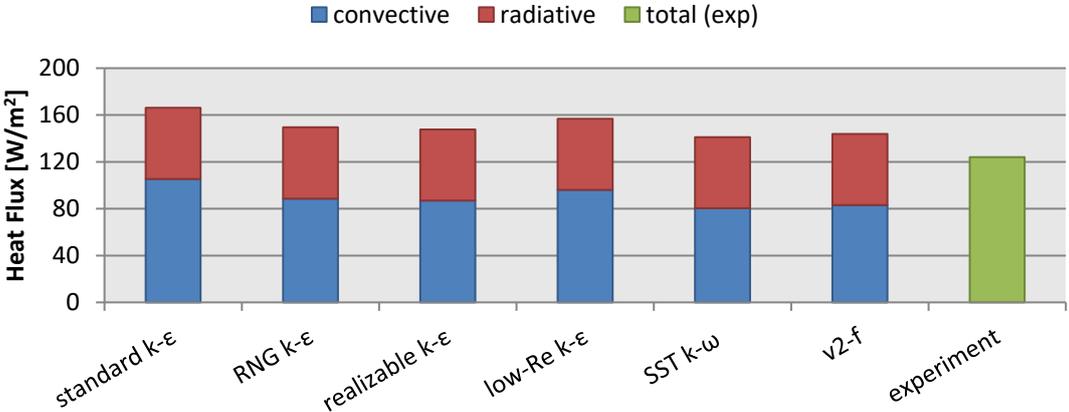


Figure 5: Predicted and measured heat fluxes from the thermal manikin.

Figure 6 presents the distribution of the convective, radiative and total heat flux over the manikin surface predicted by the $k - \omega SST$ model. As can be seen from Figure 6 b), the radiative heat flux is higher where the manikin ‘sees’ the surroundings than where the manikin ‘sees’ itself. The radiative heat transfer between different parts of the manikin is much lower than that between the manikin and the surroundings. Thus, the distribution of the radiative heat flux over the manikin surface justifies the utilization of fvDOM that takes into consideration the real human geometry and posture.

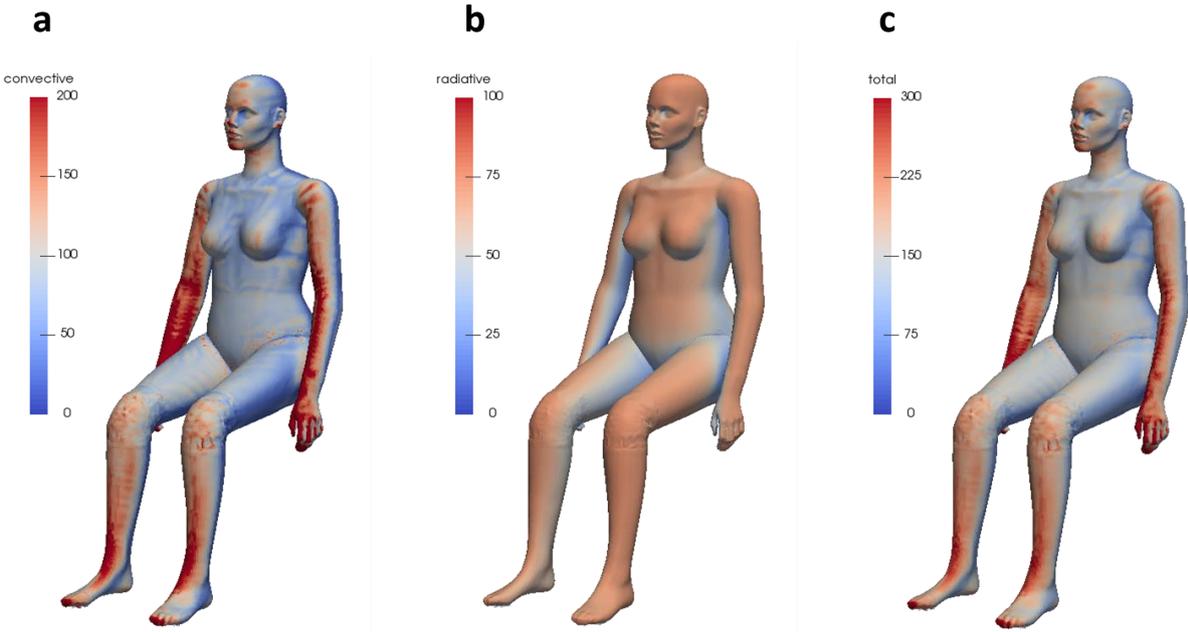


Figure 6: a) convective heat flux, b) radiative heat flux, and c) total heat flux over the manikin surface [W/m²].

Despite deviations from the experimental data, the simulation results obtained by OpenFOAM are comparable to the results obtained by commercial codes in the previous studies (Sadrizadeh & Holmberg 2016; Ito et al. 2015c). In the simulation, the standard $k - \varepsilon$ model has shown the best robustness and easiest convergence among all models used in the simulation. The $\overline{v^2} - f$ model has demonstrated significant convergence difficulties and has also been found to be sensitive to the initial conditions and the types of near-wall treatment. As it resolves two additional equations, the $\overline{v^2} - f$ model is also more computationally expensive than any other model. Despite the better accuracy of the RNG and realizable $k - \varepsilon$ models, their convergence is not as good as the standard one. Therefore, it is beneficial to start the simulation of the RNG $k - \varepsilon$, realizable $k - \varepsilon$, or $\overline{v^2} - f$ model from an established simulation of the standard $k - \varepsilon$ model. The $k - \omega SST$ model has shown the best trade-off between robustness and accuracy, which provides good accuracy without incurring significant computational or convergence difficulties.

4 CONCLUSIONS

In this study, we investigate the predication capability of an open-source CFD code – OpenFOAM, in the applications of indoor airflow and heat transfer simulations. For the ease of validation, the simulation is performed on a well-documented benchmark experimental test, conducted collaboratively by the Aalborg University in Denmark and Gävle University in Sweden. Six OpenFOAM built-in turbulent viscosity models have been attempted and their performances are compared and evaluated against the experimental data. The velocity and temperature distribution predicted by different models are rather closed to each other, but noticeable discrepancies with experimental data are observed. As regards the heat flux from the manikin to the surroundings, the $k - \omega SST$ model shows the best match with the measurement, followed by the $\overline{v^2} - f$ model. Despite the relatively good accuracy, the $\overline{v^2} - f$ model is more computationally expensive and less robust than any other model used in the simulation. The RNG $k - \varepsilon$ model and the realizable $k - \varepsilon$ model overcome some of the shortcomings of the standard $k - \varepsilon$ model and offer acceptable accuracy.

Despite the discrepancies that exist between the simulation and experiment, the performance of OpenFOAM is comparable to that of commercial codes used in previous studies in the literature. This study enhances our confidence in applying OpenFOAM to more involved studies in the future. It should, however, be emphasized that OpenFOAM has a steep learning curve due to its limited documentation. Proficient use of OpenFOAM and, in general, open-source CFD, requires more demanding competence of users than do commercial codes.

5 REFERENCES

- Axtmann, G., & Rist, U. (2016). Scalability of OpenFOAM with Large Eddy Simulations and DNS on High-Performance Systems. In *High Performance Computing in Science and Engineering '16* (pp. 413-424). Springer, Cham.
- Blacha, T., Gregersen, M. M., Islam, M., & Bensler, H. (2016). *Application of the Adjoint Method for Vehicle Aerodynamic Optimization* (No. 2016-01-1615). SAE Technical Paper.
- Chen, H. J., Moshfegh, B., & Cehlin, M. (2013). Investigation on the flow and thermal behavior of impinging jet ventilation systems in an office with different heat loads. *Building and Environment*, 59, 127-144.

- Chen, Q. (1995). Comparison of different k- ϵ models for indoor air flow computations. *Numerical Heat Transfer, Part B Fundamentals*, 28(3), 353-369.
- Chen, Q., & Srebric, J. (2000). Application of CFD tools for indoor and outdoor environment design. *International Journal on Architectural Science*, 1(1), 14-29.
- Gao, N. P., & Niu, J. L. (2005). CFD study of the thermal environment around a human body: a review. *Indoor and built environment*, 14(1), 5-16.
- Ito, K., Inthavong, K., Kurabuchi, T., Ueda, T., Endo, T., Omori, T., ... & Matsumoto, H. (2015a). CFD Benchmark Tests for Indoor Environmental Problems: Part 1 Isothermal/non-isothermal flow in 2D and 3D room model. *International Journal of Architectural Engineering Technology*, 2(1), 1-22.
- Ito, K., Inthavong, K., Kurabuchi, T., Ueda, T., Endo, T., Omori, T., ... & Matsumoto, H. (2015b). CFD benchmark tests for indoor environmental problems: Part 2 cross-ventilation airflows and floor heating systems. *International Journal of Architectural Engineering Technology*, 2(1), 23-49.
- Ito, K., Inthavong, K., Kurabuchi, T., Ueda, T., Endo, T., Omori, T., ... & Matsumoto, H. (2015c). CFD benchmark tests for indoor environmental problems: Part 3 numerical thermal manikins. *International Journal of Architectural Engineering Technology*, 2(1), 50-75.
- Jasak, H. (1996). Error analysis and estimation for finite volume method with applications to fluid flow.
- Jasak, H., Jemcov, A., & Tukovic, Z. (2007). OpenFOAM: A C++ library for complex physics simulations. In *International workshop on coupled methods in numerical dynamics* (Vol. 1000, pp. 1-20). IUC Dubrovnik, Croatia.
- Konstantinov, M., Lautenschlager, W., Shishkin, A., & Wagner, C. (2014). Numerical simulation of the air flow and thermal comfort in aircraft cabins. In *New Results in Numerical and Experimental Fluid Mechanics IX* (pp. 293-301). Springer International Publishing.
- Konstantinov, M., & Wagner, C. (2014). Numerical simulation of the air flow and thermal comfort in a train cabin. In *Proceedings of the second international conference on railway technology: research, development and maintenance*. Pombo, J.,(Editor) Civil-Comp Press, Stirlingshire, United Kingdom, paper (Vol. 328).
- Konstantinov, M., & Wagner, C. (2016). Numerical simulation of the thermal comfort in a model of a passenger car cabin. In *New Results in Numerical and Experimental Fluid Mechanics X* (pp. 383-393). Springer International Publishing.
- Liu, P. (2015). Scalability Performance Analysis of OpenFOAM on Modern HPC Clustering Technologies. In *The 3rd OpenFOAM User Conference 2015, Stuttgart, Germany*.
- Liu, W., Jin, M., Chen, C., You, R., & Chen, Q. (2016). Implementation of a fast fluid dynamics model in OpenFOAM for simulating indoor airflow. *Numerical Heat Transfer, Part A: Applications*, 69(7), 748-762.
- Martinho, N., Lopes, A., & da Silva, M. G. (2012). Evaluation of errors on the CFD computation of air flow and heat transfer around the human body. *Building and Environment*, 58, 58-69.
- Migliaccio, M., Beatrice, C., Briani, M., & Fraioli, V. (2009). *Numerical simulations of diesel engine combustion by means of OpenFOAM CFD ToolBox* (No. 2009-24-0002). SAE Technical Paper.

- Nilsson, H.O., Brohus, H., Jensen, M.K. & Nielsen, P.V. (2007). *Benchmark Tests for a Computer Simulated Person - Manikin Heat Loss for Thermal Comfort Evaluation*. www.cfd-benchmarks.com [1 May 2017].
- Norton, T., Sun, D. W., Grant, J., Fallon, R., & Dodd, V. (2007). Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: A review. *Bioresource technology*, 98(12), 2386-2414.
- Ramechecandane, S., Beghein, C., & Allard, F. (2010). Modeling fine particle dispersion in an inhomogeneous electric field with a modified drift flux model. *Building and Environment*, 45(6), 1536-1549.
- Sadrizadeh, S., Tammelin, A., Ekolind, P., & Holmberg, S. (2014). Influence of staff number and internal constellation on surgical site infection in an operating room. *Particuology*, 13, 42-51.
- Wang, C. (2013). Numerical Investigation of Flow Induced Noise in a Simplified HVAC Duct with OpenFOAM. Master Thesis. Royal Institute of Technology, Sweden
- Weller, H. G., Tabor, G., Jasak, H., & Fureby, C. (1998). A tensorial approach to computational continuum mechanics using object-oriented techniques. *Computers in physics*, 12(6), 620-631.
- Xue, Y., Liu, W., & Zhai, Z. J. (2016). New semi-Lagrangian-based PISO method for fast and accurate indoor environment modeling. *Building and Environment*, 105, 236-244.
- Zörner, C., Islam, M., & Lindener, N. (2010). The aerodynamics and aeroacoustics of the new Audi A8. *ATZ worldwide*, 112(6), 22-29.