NUMERICAL SIMULATION OF THERMAL FLOW AND COMFORT IN AN OFFICE ROOM

Yuce B. E and Pulat E.*
*Author for correspondence
Department of Mechanical Engineering,
Uludag University,
Gorukle Campus, 16059 Bursa,
Turkey,
E-mail: pulat@uludag.edu.tr

ABSTRACT

In this study, airflow and temperature fields in an office room with a table and a sitting person are computed to predict the thermal comfort by means of percent dissatisfaction (PD). The office room is ventilated with a wall type air conditioning unit for winter condition and the flow is considered to be turbulent, steady, incompressible, and two-dimensional. Governing equations are solved by using commercial computational fluid Dynamics (CFD) code of ANSYS15 Fluent. Firstly, grid independency study is evaluated by using Standard k-ε (Std. k-ε) model with standard wall function approach, and then five RANS based two-equations turbulence models and near-wall modelling approaches are compared and validated by using well-known Annex20 test room. Although all k-ε based models exhibit similar performance, Std. k-ε model with std. wall function approach was preferred due to common usage and its numerical robustness. Computations are performed for nine different cases including three different inlet velocities (1.5, 2.0, and 3.0 m/s) and three different inlet temperatures (298, 300, and 303 K). Dissatisfied space increases with increasing inlet velocity for a specified inlet temperature, and decreases with decreasing temperature for a specified inlet velocity.

NOMENCLATURE

Ar	[-]	Archimedes number
g	[m/s^2]	Gravitational acceleration
L	[m]	Characteristic length
u	[m/s]	Inlet jet velocity
T	[K]	Temperature
Q	[W/m^2]	Heat loss from human body
V	[m/s]	Local mean air velocity
Tu	[%]	Local turbulence intensity
PD	[%]	Percent dissatisfaction

Special characters

β	[1/K]	Thermal expansion coefficient
k	[m^2/s^2]	Turbulance kinetic energy
ε	[m^2/s^3]	Turbulance kinetic energy dissipation rate
y^+	[-]	Dimensionless distance from the wall

Subscripts

s	Skin, surface
a	Air

INTRODUCTION

Suitable airflow is important in ventilated spaces to provide good air quality and thermal comfort for the occupants, to maintain specified thermal and flow conditions for industrial processes, and to ensure the efficient use of energy. The importance of ventilation-related energy use is increasing and may represent up to 50% of the total energy use of a building, particularly for certain typologies such as office buildings [1]. In addition, the productivity and efficiency of office staff are dependent on the conditions of thermal environment [2,3].

CFD is one of the primary methods used to assess indoor airflow [4-6], among other methods such as the building energy simulation (BES) approach and zonal models. In recent years, CFD has taken a prominent role in the simulation of indoor environment airflow problems [7,8].

Comfort generally can be evaluated as a balance between a variety of demands (D), and the capacities (C), to meet them. These D and C can be physical, physiological, or, psychological. In physical/physiological domains the ratio D/C is less than 20% is considered comfortable. A second level, with D/C ranging between 20% and 40%, is probably an optimal state for accomplishment of most work tasks. Thermal comfort is related to thermal environmental factors, such as temperature and air motion, and ASHRAE Standard 55 identifies the thermal and air motion environmental conditions [9,10]. There are six key factors that affect thermal comfort. Three of them are environmental such as air temperature, air movement, and humidity, and others are mean radiant temperature, body heat generation, and clothing [9]. ASHRAE 55 and another thermal comfort standard ISO 7730 [11] are both based on Fanger model, and currently they are the most frequently cited thermal comfort standards [12].

By considering progress in thermal comfort research, the focus of thermal comfort air movement was draught, defined as unwanted local cooling [13], and in this study, velocity and temperature distributions among environmental factors are computed to predict the thermal comfort by means of percent dissatisfaction (PD) [10] or draught rate (DR) [11] for an office room with a table and a sitting person.

ROOM MODEL

Modeled room is designed and constructed for thermal comfort research in the thermal technology laboratory of mechanical engineering department and two-dimensional physical model and boundary conditions of this office room with a table and a sitting person are given in Figure 1. Dimensions of the height and length of the room are 2.4 m, 3.7 m, and it is equipped with an Arcelik 4200 model split type air conditioning(AC) unit. AC unit has 3450-3800 W power and uses R22 as refrigerant. Detailed information about the room can be found in [14].

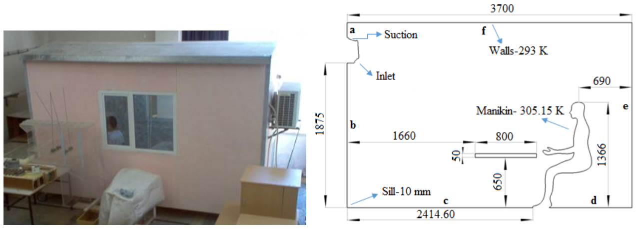

Figure 1 Test room and its two-dimensional model with boundary conditions (All dimensions are in mm)

NUMERICAL METHODS

Firstly, grid independency study is evaluated by using Standard k-ε (Std. k-ε) model with standard wall function approach by using three different cell numbers (4525, 18110, and 28500 cells), and then five RANS based two-equations turbulence models (Std. k-ε, RNG k-ε, Realizable k-ε, Std. k-ω, and SST k-ω, and four near-wall modelling approaches (Std, non-equilibrium, scalable wall functions, and enhanced wall treatment) are compared and validated by using well-known IEA Annex20 test room as seen from Figure 2.

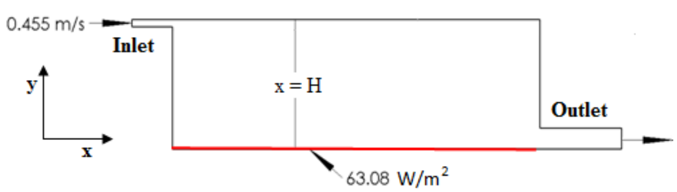

Figure 2 Geometry and measurement lines of IEA Annex20 room.

Air was treated under the Boussinesq assumption, and the Archimedes number is defined as,

$$Ar = \frac{\beta g L \Delta T}{u^2} \tag{1}$$

Dimensionless velocity, temperature and turbulence intensity profiles at all measurement lines as seen from Figure 2 are compared with experimental values for all turbulence models [15]. Although all k-ε based models exhibit similar performance, Std. k-ε model with std. wall function approach was preferred due to common usage and its numerical robustness.

Although excessive kinetic energy prediction at stagnation region and turbulence isotropy assumption are the main disadvantages of the Std. k-ε model, the model works surprisingly well in many types of flows [16] including ventilation and thermal comfort studies [17]. Comparison results only for this model is given in Figure 3.

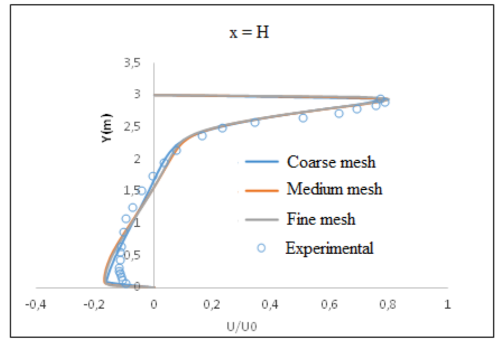

Figure 3 Comparison of dimenless x-velocity profiles with experimental values [18] at x = H=3m for Std. k-ε model with Std. wall function

Under the light of grid independency and validation study, test room is divided into 104590 hexahedral cells to resolve areas near the walls and around the table and human. Near-wall treatment is crucial in any turbulence model, and the most popular approach is the wall function approach due to its simplicity. Basically, the method requires that the first grid point near the wall be located in the fully turbulent region. The grid points must therefore not be chosen closer than y^+=11 but, on the other hand, the distance should not be much greater because the wall functions are not an exact description of the flow in the general situation [19]. Minimum, maximum, and mean values of y^+ values on the walls of model room shown in Figure 1 are given in Table 1. But, due to complex nature of room air movement, as mentioned by Voigt [20], it was not possible to design a grid with both satisfying resolution of the wall jet and $30<y^+<100$, as seen from Table 1.

Table 1 y^+ values on the model room walls

Walls / y^+	a	b	c	d	e	f
Minimum	2.36	1.25	11	3.2	0.7	0.3
Maximum	13.3	15.11	12	0.8	3.62	9.91
Mean	7.15	7.08	4.5	1.42	1.34	6.06

The k–ε model introduces two new variables turbulent kinetic energy and its dissipation rate (k and ε) into the system of equations. The values of k and ε come directly from their differential transport equations for the turbulent kinetic energy and the turbulence dissipation rate [21]. The governing equations were solved using the ANSYS 15 Fluent, which is based on the finite volume method. SIMPLE algorithm was used for the pressure-velocity coupling. Pressure and momentum were discretized by applying second order, and second order upwind respectively. First order upwind was applied for both turbulent kinetic energy and its dissipation rate. The convergence was achieved when the weighted residue of each of the governing equations was 10^{-6}.

No-slip boundary condition was applied all surfaces and other boundary conditions are shown in Figure 1. By considering mild winter condition in Bursa, wall surface temperatures are set to 293 K. Under thermal neutrality, by considering the heat loss from a human body (Q_m), the mean skin temperature may be estimated by the following Fanger equation [22],

$$T_s = (35.77 - 0.028Q_m) \quad (2)$$

But, Gao and Niu [23] reported that the body temperature is usually set in the range from 304 K to 306.7 K in numerical studies. So body surface temperature was set to 305.15 K in this study. For example, it was taken as 306.85 K in the study of Murakami et al [24].

Draft is defined as unwanted local cooling of the body caused by air movement, and percent dissatisfaction (PD) due to draft is determined according to ISO 7730 [11] by using following equation,

$$PD = (34 - T_a)(V - 0.05)^{0.62}(0.37VTu + 3.14) \quad (3)$$

where T_a is the local air temperature (20-26°C), V is the local mean air velocity(<0.5 m/s), and Tu is the local turbulence intensity in percent(10-60%) .

Since CFD program provides the information about velocity, temperature, and turbulence, PD is computed by integration of above equation into the software platform by using Custom Field Functions option.

RESULTS AND DISCUSSION

The nine cases shown in Table 1 are investigated. As seen from Table 2, velocity is increased from 1.5 to 3 m/s for fixed inlet temperature, and inlet air temperature is decreased from 303 K to 298 K.

Table 2 Investigated computational cases

Case no	Inlet vel. (m/s)	Inlet temp. (K)
1	1.5	303
2	2	303
3	3	303
4	1.5	300
5	2	300
6	3	300
7	1.5	298
8	2	298
9	3	298

The velocity vectors and temperature distributions obtained under specified boundary conditions are given in Figures 4 and 5 respectively. Total Percent Dissatisfaction and uncomfortable Percent Dissatisfaction distributions are also given in Figures 6 and 7.

At first sight, it is detected different flow pattern of the lowest inlet velocity (1.5 m/s) for all inlet air temperatures (303K case Ia, 300K case IIa, and 298K case IIIa) in Figure 4. The air jet is directed to upwards under the effect of the low pressure in the top of the AC unit. Since all inlet air temperatures are higher than wall temperatures, buoyancy force acts to tend the air flow downwards for all cases. But it seems that buoyancy has no any effect on flow pattern in low inlet velocity cases.

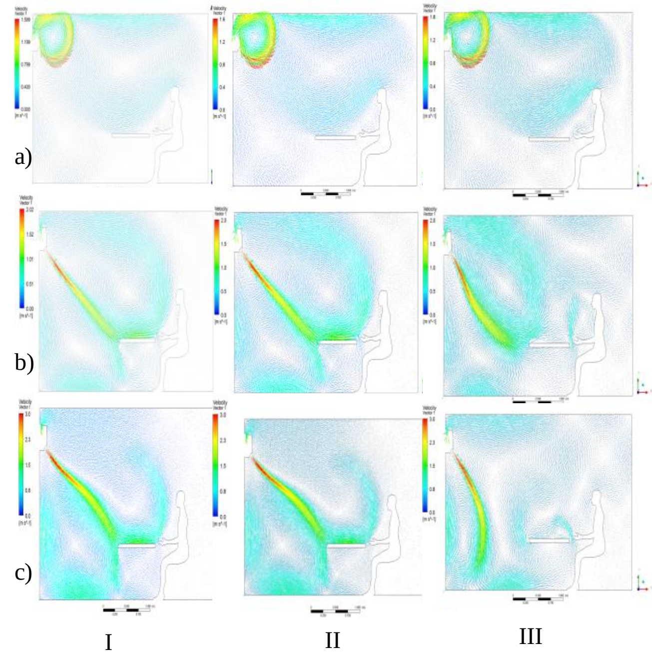

Figure 4 Vectoral velocity distributions for investigated cases a)1.5 m/s, b)2 m/s, c)3 m/s and I)298 K, II)300 K, III)303K

As seen from Figure 5, this "back-suction" of air prevents the warming of the air in the remainder of the room space.

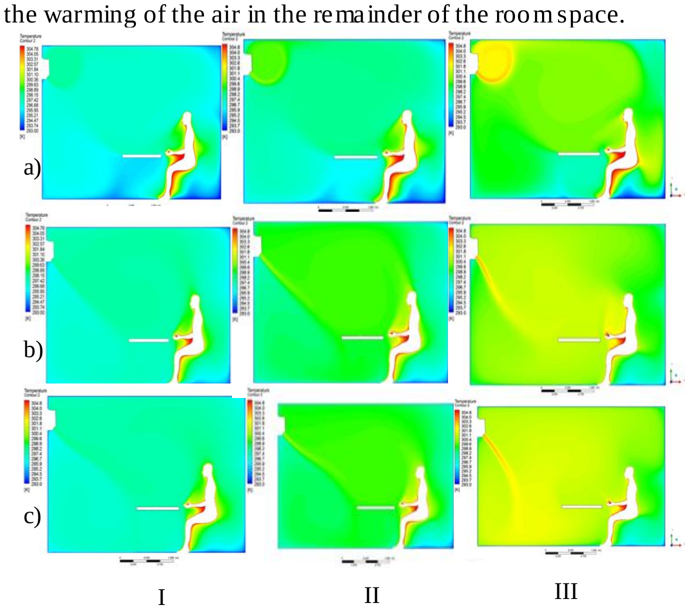

Figure 5 Temperature distributions for investigated cases a)1.5 m/s, b)2 m/s, c)3 m/s and I)298 K, II)300 K, III)303K

Although room temperatures increases with increasing inlet air temperatures, this phenomenon is not preferable from energy conservation point of view since heated air is discharged without diffusion inside the room. But, interestingly this situation causes increase in thermal comfort by decreasing dissatisfied regain around the human body, and dissatisfied regain decreases with increasing the inlet temperatures as seen from Figure 7 a.

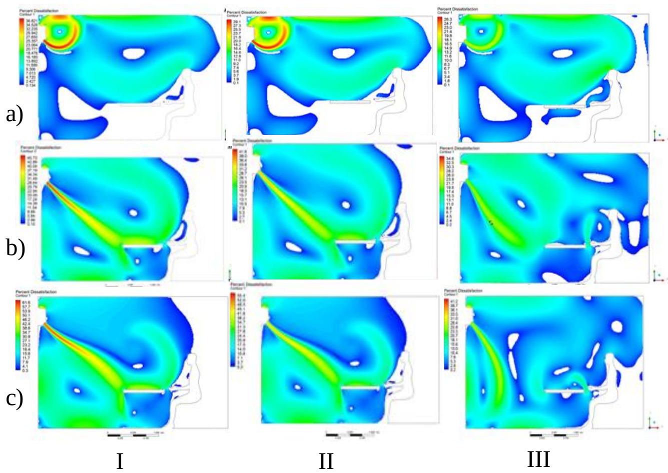

Figure 6 Total Percent Dissatisfaction (PD) distributions for investigated cases a)1.5 m/s, b)2 m/s, c)3 m/s and I)298 K, II)300 K, III)303K

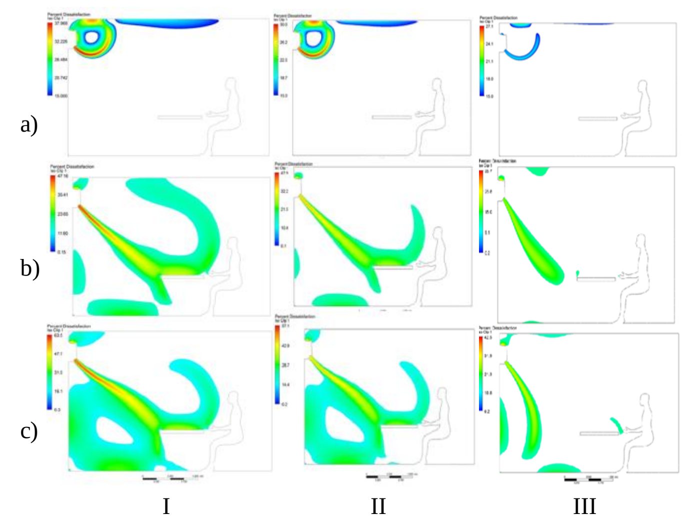

Figure 7 Uncomfortable (PD > %15) Percent Dissatisfaction distributions (PD) for investigated cases a)1.5 m/s, b)2 m/s, c)3 m/s and I)298 K, II)300 K, III)303K

For all other inlet velocity cases (Except the case IIIc), almost similar flow patterns are obtained. In all cases the highest temperatures are obtained around the human body since body surface temperature is higher than all considered inlet air temperatures. In case IIIc, mixing is enhanced, and as a result more uniform temperature distribution is obtained. In case IIIb, slightly different flow pattern is obtained, and temperature distribution is more uniform than the other seven cases except the case of IIIc. In low velocity cases (Ia, IIa, and IIIa), small cold plumes under the table and human body are formed, but the plume under the table disappears with increasing inlet air jet temperature. In contrast, the plume under the human body is persistent. However, the velocities of these plumes are low and there is no effect on PD distribution in the room as seen from Figures 6 and 7. In addition there is no any effect of slit of door sill on the flow and temperature distributions in all cases.

CONCLUSION

Airflow and temperature fields in an office room with a table and a sitting person for mild winter condition are computed to predict the thermal comfort by means of percent dissatisfaction (PD). For this purpose, Std. k-ε model with std. wall function approach was preferred due to common usage and its numerical robustness. For the lowest inlet velocity, the air jet is directed to upwards under the effect of the low pressure in the top of the AC unit. This back-suction phenomenon is not preferable from energy conservation point of view, but it causes increase in thermal comfort by decreasing dissatisfied region around the human body, and dissatisfied regain decreases with increasing inlet temperatures. Cold plumes obtained for the lowest inlet velocity cases does not affect the Percent Dissatisfaction (PD) distributions. For all other inlet velocity cases (Except the case IIIc), almost similar flow patterns are obtained. There is no any effect of slit of door sill on the flow and temperature distributions in all cases. Under the considered conditions, to avoid the back-suction effect, and simultaneously to decrease uncomfortable regions, medium speed and high temperature running case of AC unit is recommended by considering energy conservation for mild winter condition. However, need to experimental studies is obvious to verify back-suction phenomena.

Acknowledgement

This study is a part of M.Sc. Thesis of the first author accepted on 25.08.2015 by the Graduate School of Natural and Applied Sciences of Uludag University.

REFERENCES

[1] Working Group 16, Ventilation, good indoor air quality and rational use of energy, European Commission Joint Research Centre, Institute For Health and Consumer Protection, Physical and Chemical Exposure Unit, Report No. 23, EUR20741 EN, 2003

[2] Takaski, A., Tanabe, S.-i., Yanai, T., and Sasaki, M., Thermal comfort and productivity - Evaluation of workplace environment in a task conditioned office, *Building and Environment*, Vol. 45, 2010, pp. 45-50

[3] Tanabe, S.-i., Haneda, M., and Nishihara, N., Workplace productivity and individual thermal satisfaction, *Building and Environment*, Vol. 91, 2015, pp. 42-50

[4] Chen, Q., Computational fluid dynamics for HVAC: successes and failures, *ASHRAE Trans.*, Vol. 103-1, 1997, pp. 178–187.

[5] Zhai, Z., Application of Computational Fluid Dynamics in building design: Aspects and trends, *Indoor and Built Environment*, Vol. 15, 2006, pp. 305-313.

[6] Hoof, T.v., Blocken, B., and van Heijst, G.J.F., On the suitability of steady RANS CFD for forced mixing ventilation at transitional slot Reynolds numbers, *Indoor Air*, Vol. 23, 2013, pp. 236–249

[7] Pitarma, R.A., Ramos, J.E., Ferreira, M.E., and Carvalho, M.G., Computational fluid dynamics: an advanced active tool in environmental management and edu-cation, *Manage. Environ. Qual. Int. J.* Vol. 15, 2004, pp. 102–110.

[8] Nielsen, P.V., Computational fluid dynamics and room air movement, *Indoor Air*, Vol. 14 (Suppl. 7), 2004, pp. 134–143
[9] Goldman, R. F., Physical laws, scientific statistics, engineering allowances, and psychological validity, *ASHRAE HVAC&R Research*, Vol. 7, 2001, pp. 103-106
[10] ASHRAE Handbook-Fundamentals, Thermal Comfort, 2009.
[11] ISO 7730, Ergonomics of the thermal environment, 2005.
[12] Cheng, Y., Niu, J., and Gao, N., Thermal comfort models: A review and numerical investigation, *Building and Environment*, Vol. 47, 2012, pp. 13-22
[13] de Dear, R.J., Akimoto, T., Arens, E.A.., Brager, G., Candido, C., Cheong, K.W.D., Li, B., Nishihara, N., Sekhar, S.C., Tanabe, S., Toftum, J., Zhang, H., and Zhu, Y., Progress in thermal comfort research over the last twenty years, *Indoor Air*, Vol. 23, 2013, pp. 442–461
[14] Pulat, E., Numerical investigation of air distribution in a room ventilated with room air conditioning unit, *Proceedings of the National Air Conditioning Conference*, Antalya, Turkey, pp. 387-400, 18-20 November 2011.
[15] Yuce, B.E., Numerical simulations of airflow, temperature distribution, heat transfer, and thermal comfort in room ventilation, *M.Sc. Thesis*, School of Natural and Applied Sciences, Mechanical Engineering Department, Uludag University, Bursa, Turkey, 2012.
[16] Davidson, L., and Nielsen, P.V., Large Eddy Simulations of the flow in a three-dimensional ventilated room, *ROOMVENT'96 5th Int. Conf. on Air Distributions in Rooms*, Yokohoma, Japan, July 1996.
[17] Serra, N., and Semiao, V., Comparing displacement ventilation and mixing ventilation as HVAC strategies through CFD, *Engineering Computations*, Vol. 26, 2009, pp. 950-971.
[18] Nielsen, P.V., Flow in air conditioned rooms, *PhD Thesis from the Technical University of Denmark*, 1974.
[19] Sorensen, D.N., and Nielsen, P.V., Quality control of computational fluid dynamics in indoor environments, *Indoor Air*, Vol. 13, 2003, pp. 2-17.
[20] Voigt, L.K., Comparison of turbulence models for numerical calculation of air flow in an annex 20 room, Technical University of Denmark, DTU ET-AFM 2000-01, Lyngby, 2000.
[21] Launder, B.E., and Spalding, D.B., Lectures in Mathematical Models of Turbulence, Academic Press, London, 1972.
[22] Fanger, P.O., Thermal comfort: analysis and application in environmental engineering, Danish Technical Press, Copenhagen, 1970.
[23] Gao, N.P., and Niu, J.L., CFD study of the Thermal Environment around a human body:A review, *Indoor and Built Environment*, Vol. 14, 2005, pp. 5-16.
[24] Murakami, S., Kato, S., Flow and temperature fields around human body with various room air distribution, Part 1 - CFD study on computational thermal, *ASHRAE Transactions*, Vol. 103, 1997, Part 1, pp. 3-15.