Topic 3. Indoor and outdoor air quality, thermal comfort and health impact related to built environment

Application of CFD Predictions to Quantify Thermal Comfort for Indoor Environments

Tateh Wu[1,*], Andrew D. Clark[1], Gary L. Mitchell[2], Chao-Hsin Lin[1], and Raymond H. Horstman[1]

[1]Environmental Control Systems, Boeing Commercial Airplanes, Seattle, WA
[2]Fight Operation, Boeing Test & Evaluation, Seattle, WA

[*]*Corresponding email: ted.wu@boeing.com*

Keywords: CFD, Thermal Comfort, Indoor Environments

SUMMARY

A CFD simulation with thermal comfort calculations on a mechanically ventilated test room is conducted. Within the CFD-simulated indoor environment, thermal comfort across the volume was visualized. Special arrangements of the computational mesh around the body surface are used to investigate the effect of the local air velocity near the human body on thermal comfort. To obtain the overall thermal comfort indices for the human subject, weighting factors for the body segments were applied based on the skin blood flow. The effect of local air flow and heat transfer on the human body thermal comfort calculation is demonstrated. The local PPD of the back and pelvis is much higher for the 25.4 mm shell. With the given uniform ventilation inside the test room, the effect of local air flow and heat transfer on the overall thermal comfort is not significant. More applications of using the methodology outlined in this paper are needed.

INTRODUCTION

The HVAC system to achieve thermal comfort for indoor environments can be very complicated due to the variation of the intended function, energy consideration, and definition of thermal comfort, etc. To provide a thermally comfortable indoor environment, certain ranges of air temperature and air velocity magnitude are the most common parameters used by engineers when setting up the design requirements. Nevertheless, the factors, such as the metabolic rate of human activities, clothing level, thermal radiation, humidity, local air flow, local air temperature, etc., that could affect thermal comfort in indoor environments, as reported by numerous studies in the past (Fanger 1967; Fanger and Christensen 1986; Fanger et al. 1988; Fanger et al. 1985). Furthermore, the interactions among those factors to influence human's thermal sensation have been continuously studied for more than half a century (Schälin and Nielsen 2004; Hanzawa et al. 1987). As a result, various theories on the quantifying of thermal comfort in indoor environments based on human thermal regulatory mechanisms and the heat transfer between human bodies and their surroundings have been developed. Among them, Fanger's thermal comfort theory (Fanger 1967; Fanger 1982) has been widely applied to various indoor environments.

Computational Fluid Dynamics (CFD) has been utilized to provide the temperature and velocity fields for various indoor environments such as buildings, and surface and air transportation vehicles. Those CFD predicted data can be readily applied to the quantification of thermal comfort and hence, drive design requirements that account for the many interactions (Lin et al. 1992). In this study, a CFD simulation with a thermal comfort module on a mechanically ventilated test room was conducted. Thermal comfort within the test room was presented in terms of the Predicted Mean Vote (PMV) and Predicted Percentage of Dissatisfied (PPD), based on Fanger's theory (Fanger 1967; Fanger 1982). Special arrangements of the computational mesh around the numerical human subject in the test room were used to investigate the effect of the local air velocity near the human body on thermal comfort. Due to human physiology and thermoregulation which result in the non-uniformity of a human's skin temperature, theories on the related mechanisms have been proposed by researchers (Parsons 1993). To take the effect of skin temperature into consideration, a locally weighted PMV and PPD based on the blood flow at various portions of the human body to obtain the overall thermal comfort indices for the human subject is presented in this study.

METHODS

Description of CFD modeling

The CFD analysis was conducted using the commercial CFD code, FLUENT. The Computer Simulated Person (CSP, i.e. manikin) geometry was generated using a 3D scanned stereo lithography (STL) geometry file of Comfortina, a thermal manikin belonging to the Aalborg University (Nielsen et al. 2003). The model was split into 20 zones so that each could be individually assessed. The zones are shown in Figure 1 and based on the benchmark test for thermal comfort evaluation conducted by Aalborg University and Gävle University (Nilsson et al. 2007) with the exception for the additional neck zone.

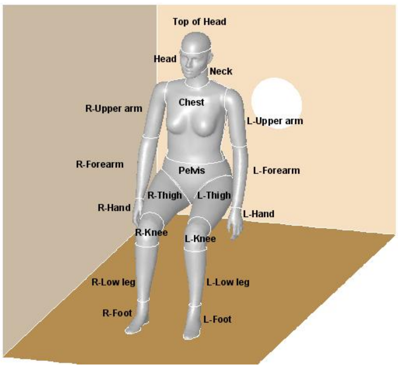
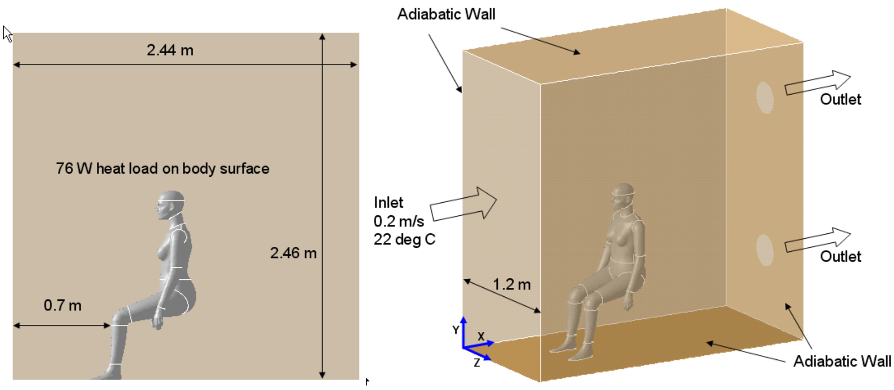


Figure 1. Various Portions of the Simulated Manikin



(a) Side View *(b) 3D View*

Figure 2. CFD Model of the Test Room with the Simulated Manikin inside

The manikin was placed in a box defined in the benchmark test (Nilsson et al. 2007) which had the dimensions 2.44 m x 2.46 m x 1.2 m, where Figure 2(a) shows a cross-section and Figure 2(b) shows a 3D view. The manikin was placed 0.7m from the front of the box when measured from the knee. The front of the box was defined as a uniform velocity inlet, which is considered similar to the flow field a person is exposed to in a mixing ventilated room (Nilsson et al. 2007). Centered on the aft wall are two 0.25 m diameter exhausts located 0.6 m from the floor and ceiling for which all the air is assumed to exhaust (i.e. no leakage).

The computational mesh for the manikin was generated using the ANSYS meshing package. The total volume mesh count for the model was 2.7 million cells. A cross-section of the mesh can be seen in Figure 3 and shows the cell count is highly concentrated around the manikin. Figure 3 also shows the magnified cross-section of the mesh where the boundary layer mesh can be seen with the surface mesh of the manikin. The boundary layer mesh has a total thickness of 25.4 mm and eight layers. Two surface mesh shells that are 12.7 mm (0.5 inch) and 25.4 mm (1 inch) away from the body surface are created for the body thermal comfort calculation. The surface mesh shells were separated so that each zone of the manikin was uniquely defined.

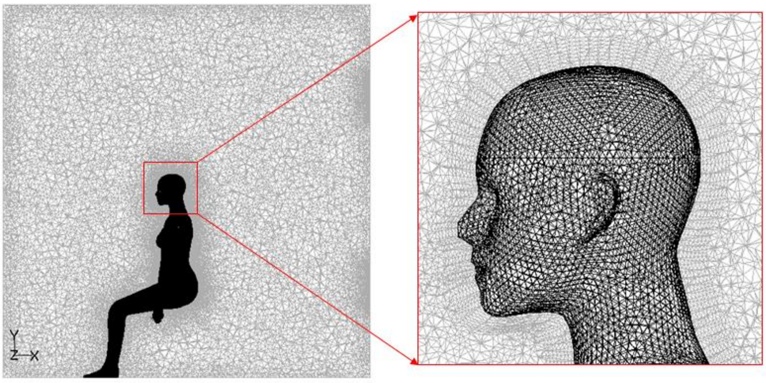


Figure 3. A Cross-Section of the CFD Volume Mesh (gray) and Surface Mesh of the Manikin (black)

With the exception of the knee, all surfaces of the manikin are set to a constant heat flux so that total heat load is 76W. All walls were set to a radiation emissivity of 0.85. The ceiling, sidewall and floor were defined as adiabatic walls. The forward face of the box was set to a uniform inlet of 0.2 m/s at 22 °C. Turbulence was modeled using the two equation turbulence model, k-ω SST, which in previous studies using the same benchmark model was found to lead to better agreement with the experimental data than the k-ε model (Martinho et al. 2008).

Thermal comfort calculation

As mentioned above, one method of evaluating thermal comfort is to use the predicted mean vote (PMV) and predicted percent dissatisfied (PPD) equations based on Fanger's theory (Fanger 1967; Fanger 1982). In this study, the calculation of thermal comfort within the test room and thermal comfort around the computer simulated person is demonstrated by using Fanger's theory with local air velocity magnitude, local air temperature, and local static air pressure predicted by the CFD simulation.

For the thermal comfort calculation on the body surface, special meshing arrangements were set up in the CFD model. Two of 12.7 mm (0.5 inch) thick volume meshes (prism layers) away from the manikin surface were created so that in addition to the body surface temperature, the local air velocity magnitude on the surfaces that are 12.7 mm (0.5 inch) and 25.4 mm (1 inch) away from the body surface are available for the body thermal comfort calculation. The effect of the local air velocity near the body on thermal comfort will be discussed in the next section.

RESULTS

In the CFD model, the sedentary numerical manikin is facing the inlet, where the incoming air with a uniform air velocity magnitude of 0.2 m/s at a temperature of 22 °C. On the back side of the test room, there are two exhaust openings. Figure 4 is a streamline plot showing the general airflow pattern around the manikin. The streamlines are colored by velocity magnitude ranged from 0 to 0.5 m/s. The airflow upstream of the manikin is very uniform as expected. The streamlines behind the manikin illustrate turbulent wakes that usually occurs downstream of a bluff body.

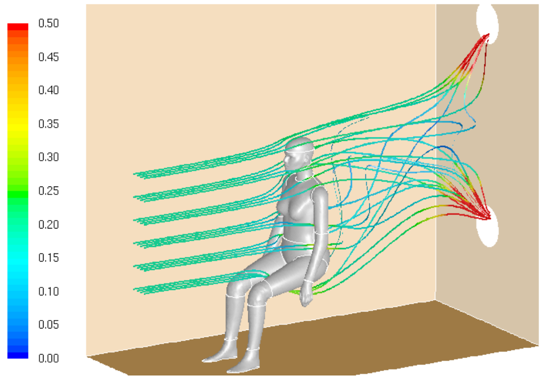
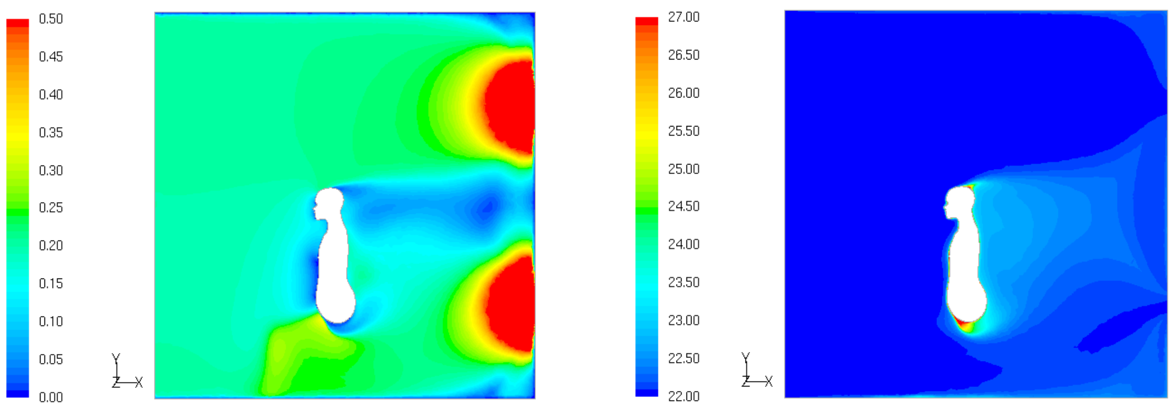


Figure 4. Streamlines inside the Test Room and Around the Simulated Manikin (Colored by velocity magnitude with a unit of m/s)

Air velocity and temperature contours are plotted on a vertical mid plane (at Z = 0.6 m) and shown in Figure 5 (a) and (b). The airflow between the two legs is moving faster and then has significant accelerations before exiting the two outlets. Within the test room, the temperature distribution is quite uniform in most of area. The air behind the manikin is a slightly warmer because the heat dissipated from the body is moving downstream. The temperature contours on the vertical mid plane (Figure 5(b)) also show high temperature gradients on the upper back of the head and below the pelvis due to the airflow is separated at the location of the former case and stagnant at the location of the latter case.



(a) Air Velocity Magnitude (m/s) *(b) Static Temperature (°C)*

Figure 5. Contours of Air Velocity and Temperature on the Vertical Mid Plane at Z = 0.6 m

As shown in Figure 6, the body surface temperatures excluding the knees are higher than the surrounding air temperatures. It can be seen from the side view that the front surfaces of the manikin are cooler than the rear body surfaces caused by the convective effect of the supply air direction. Similarly, the back the body surface temperature contours indicate a much higher temperature, around 32 °C, is predicted near the armpit area, as shown in Figure 6(b).

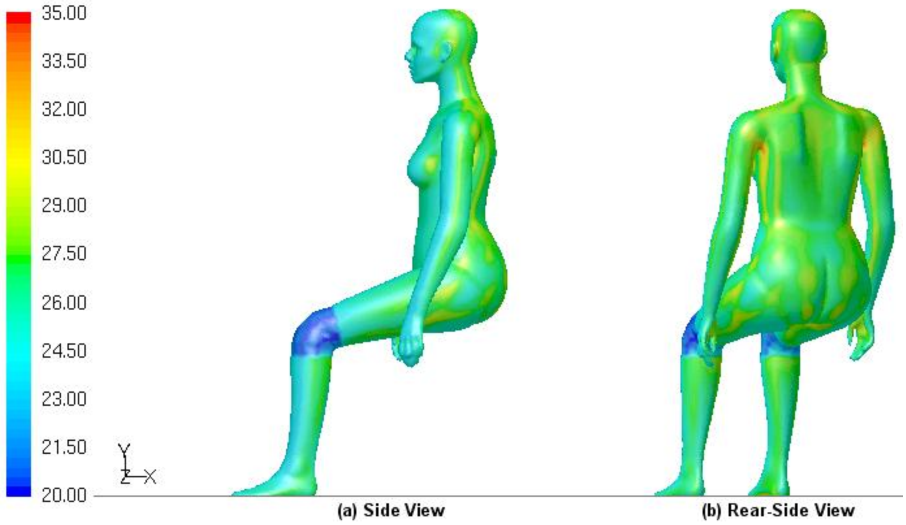
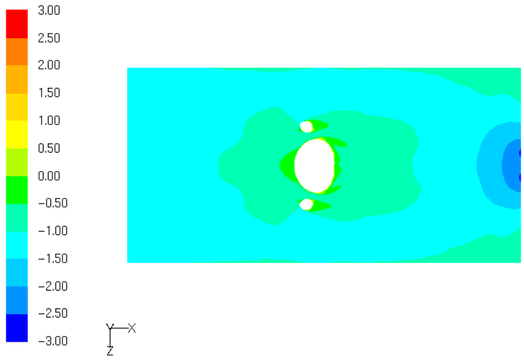


Figure 6. Contours of Body Surface Temperature (°C)

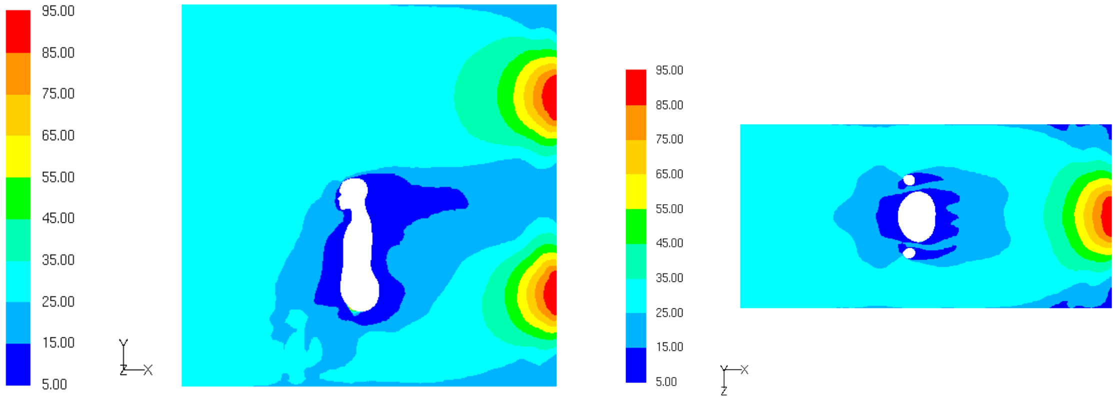
With the air velocity, air temperature, and radiation temperature predicted by the CFD simulation, a thermal comfort module implemented in the CFD post-processing was utilized to calculate the PMV and PPD thermal comfort indices within the test room. A clothing value of 1.0 representing typical business suit (Fanger 1982) is used. The physical activity level is assumed at a metabolic rate of 0.88 met that is equivalent to 43.741 kcal/(hr-m^2). The relative humidity is set at 50%. Figure 7 show the PMV contours on the vertical mid plane (Y=0.6 m) and the horizontal plane at 0.66 m above the floor (approximate at elbow height), respectively. The PMV plots indicate that the predicted mean vote around the manikin is between -1 and 0, which indicates the existence of a slightly cool thermal environment. Similarly, the PPD contours plots shown in Figure 8 indicate that most of area around the manikin is less than 15%, again an indication of the thermally comfort condition in the test room.

(a) Vertical Mid Plane at Z = 0.6 m



(b) Horizontal Plane at Y = 0.66 m (approximate elbow height)

Figure 7. Predicted Mean Vote (PMV)



(a) Vertical Mid Plane at Z = 0.6 m

(b) Horizontal Plane at Y = 0.66 m (approximate elbow height)

Figure 8. Predicted Percent Dissatisfied (PPD)

To further quantify the overall thermal comfort of the human body, the body surface temperature and the adjacent air velocity magnitude are output from the CFD model. Table 1 lists the average body surface temperature, the average air velocity magnitude at 12.7 mm (0.5 inch) and 25.4 mm (1 inch) away from the body surface for each of the human body portions indentified in Figure 1.

Table 1. Body Surface Temperature and Adjacent Air Velocity Magnitudes for Various Portions of the Simulated Manikin

Computer Simulated Person (CSP) Body Zone	Body Surface Temperature [C]	Air Velocity at 12.7 mm away from Body Surface [m/s]	Air Velocity at 25.4 mm away from Body Surface [m/s]
Top of head	26.2	0.158	0.169
Head	25.9	0.163	0.169
Neck	26.4	0.178	0.189
Chest	25.9	0.151	0.151
Back	27.6	0.183	0.191
L-Upper-arm	26.3	0.186	0.198
R-Upper-arm	26.3	0.186	0.197
L-Forearm	25.6	0.209	0.218
R-Forearm	25.6	0.211	0.219
L-Hand	25.7	0.214	0.225
R-Hand	25.6	0.217	0.231
Pelvis	26.9	0.183	0.195
L-Thigh	26.2	0.210	0.215
R-Thigh	26.2	0.209	0.221
L-Knee	21.5	0.179	0.186
R-Knee	21.5	0.179	0.186
L-Low-leg	25.5	0.193	0.207
R-Low-leg	25.5	0.194	0.208
L-Foot	25.8	0.201	0.217
R-Foot	25.7	0.201	0.217

Local and overall comfort calculation

The comfort calculation from Fanger was used to evaluate the local heat balance using the following:

$$PMV = \left[0.352e^{-0.042M} + 0.032\right]\begin{bmatrix} M - 0.35(43 - 0.061M - Pa) \\ -0.42(M - 50) - 0.0023M[44 - Pa] - 0.0014M[34 - Ta] \\ -3.4x10^{-8} f_{cl}\left[(T_b + 273)^4 - (T_a + 273)^4\right] - f_{cl}h(T_b - T_a) \end{bmatrix} \qquad (1)$$

where
M = metabolic rate = 43.741 kcal/m2-hr
Pa = (RH/100)(11.106-0.4174Ta+0.03696(Ta)^2) = vapor pressure [mmHg]
Ta= air temperature near surface [^{o}C]
Tb= body surface temperature [^{o}C]
fcl= clothing area factor 1.15
h= 8.95V$^{1/2}$ = heat transfer coefficient [kcal/hr-m^2-^{o}C]
V=Local air velocity [m/s]

$$PPD = 100 - 95e^{-\left[0.03353PMV^4 + 0.2179PMV^2\right]} \qquad (2)$$

Weighting factors for the body segments were applied based on the skin blood flow from Parsons (Parsons 1993). The PMV for the 12.7 mm (0.5 inch) "shell" was slightly warm, toward neutral at PMV=0.043 with a predicted dissatisfied equal to about PPD = 5.5 %. The PMV for the 25.4 mm (1.0 inch) shell was slightly cooler at PMV=-0.213 than that at the 12.7 mm (0.5 inch) shell and consequently, with a bit more predicted dissatisfaction equal to about PPD = 6.8 %. With the given fairly uniform air flow pattern inside the test room, the PPD based on the 25.4 mm shell is slightly higher than the 12.7 mm shell. However, it's interesting to note that the local PPD for the back and pelvis is significantly higher than other body portions for the 25.4 mm shell due to the local airflow and heat transfer. The complete comfort tabulation is shown in Table 2.

TABLE 2. Body Segment Comfort Calculation Using Comfort Equations with the Values Shown in Table 1.

Computer Simulated Person (CSP) Body Zone	Basal Blood Flow	Weighting Factor	PMV (12.7 mm)	PMV weighted (12.7 mm)	PPD % (12.7 mm)	PPD weighted % (12.7 mm)	PMV (25.4 mm)	PMV weighted (25.4 mm)	PPD % (25.4 mm)	PPD weighted % (25.4 mm)
Top of head	0.480	0.040	0.143	0.006	5.426	0.219	-0.150	-0.006	5.468	0.221
Head	0.480	0.040	0.134	0.005	5.374	0.217	-0.069	-0.003	5.100	0.206
Neck	0.480	0.040	-0.078	-0.003	5.125	0.207	-0.383	-0.015	8.049	0.325
Chest	0.700	0.059	0.053	0.003	5.058	0.298	-0.093	-0.005	5.180	0.305
Back	0.700	0.059	-0.295	-0.017	6.807	0.401	-0.761	-0.045	17.190	1.012
L-Upper-arm	0.125	0.011	-0.116	-0.001	5.279	0.056	-0.391	-0.004	8.187	0.086
R-Upper-arm	0.125	0.011	-0.146	-0.002	5.444	0.057	-0.399	-0.004	8.318	0.087
L-Forearm	0.125	0.011	0.148	0.002	5.452	0.057	-0.008	0.000	5.001	0.053
R-Forearm	0.125	0.011	0.141	0.001	5.413	0.057	-0.014	0.000	5.004	0.053
L-Hand	1.000	0.084	0.102	0.009	5.216	0.439	-0.086	-0.007	5.153	0.433
R-Hand	1.000	0.084	0.159	0.013	5.526	0.465	-0.047	-0.004	5.045	0.424
Pelvis	0.700	0.059	-0.226	-0.013	6.057	0.357	-0.558	-0.033	11.526	0.679
L-Thigh	0.475	0.040	-0.080	-0.003	5.132	0.205	-0.382	-0.015	8.047	0.321
R-Thigh	0.475	0.040	-0.041	-0.002	5.034	0.201	-0.386	-0.015	8.106	0.324
L-Knee	0.475	0.040	-0.057	-0.002	5.068	0.202	-0.081	-0.003	5.136	0.205
R-Knee	0.475	0.040	-0.055	-0.002	5.063	0.202	-0.081	-0.003	5.137	0.205
L-Low-leg	0.475	0.040	0.197	0.008	5.806	0.232	-0.001	0.000	5.000	0.200
R-Low-leg	0.475	0.040	0.204	0.008	5.863	0.234	0.007	0.000	5.001	0.200
L-Foot	1.500	0.126	0.118	0.015	5.287	0.667	-0.209	-0.026	5.905	0.745
R-Foot	1.500	0.126	0.151	0.019	5.471	0.690	-0.176	-0.022	5.645	0.712
Total	11.890	1.000		0.043		5.463		-0.213		6.796
Average			0.023		5.445		-0.213		6.860	

DISCUSSION

Using the CFD predictions on quantifying the thermal comfort level within the test room and for the human body is demonstrated. The first set of thermal comfort calculations within the test room is useful information for evaluating ventilation of indoor environments. For example, within a CFD-simulated indoor environment, thermal comfort across the volume can be visualized and indicate the area of concern. For the second set of thermal comfort calculations, the locally weighted PMV and PPD based on various portions of the manikin represents a more realistic human thermal sensation. The effect of local air flow and heat transfer on the human body thermal comfort calculation is demonstrated. Especially the local PPD of the back and pelvis is much higher for the 25.4 mm shell.

CONCLUSIONS

In this analysis, with the given uniform ventilation inside the test room, the effect of local air flow and heat transfer on the overall thermal comfort is not significant. More applications of using the methodology outlined in this paper are needed. It's recommended to use the same approach to evaluate thermal comfort for occupants in other CFD-simulated indoor environments that have a more complicated ventilation configuration.

REFERENCES

Fanger P.O. 1967. "Calculation of thermal comfort: Introduction of a basic comfort equation," *ASHRAE Transactions*, 73, Part II.

Fanger P.O. and Christensen N.K. 1986. "Perception of draught in ventilated spaces," *Ergonomics*, 29:2, pp. 215-235.

Fanger P.O., Melikov A.K., Hanzawa H., and Ring J. 1988. "Air turbulence and sensation of draught," *Energy and Buildings*, 12, pp. 21-39.

Fanger P.O., Ipsen B.M., Langkilde G., Olesen B.W., Christensen N.K., and Tanabe S. 1985. "Comfort limits for asymmetric thermal radiation," *Energy and Buildings*, 8, pp. 225-236.

Fanger P.O. 1982. *Thermal Comfort*. Robert E. Krieger Publishing Co.

Hanzawa H., Melikov A.K., and Fanger P.O. 1987. "Airflow characteristics in the occupied zone of ventilated spaces," *ASHRAE Transactions*, 93:1, pp. 524-539.

Lin C.H., Han T., and Koromilas C.A. 1992. "Effects of HVAC Design Parameters on Passenger Thermal Comfort," *SAE Technical Paper Series* 920264.

Martinho N., Lopes A. and Silva M. 2008. "CFD Modeling of Benchmark Tests for Flow Around a Detailed Computer Simulated Person," *The 7th International Thermal Manikin and Modeling Meeting* – University of Coimbra, Portugal.

Nielsen P.V., Murakami S., Kato S., Topp C., and Yang J.-H. 2003. *Benchmark Tests for a Computer Simulated Person*. ISSN 1395-7953 R0307. Indoor Environmental Engineering, Aalborg University.

Nilsson H.O., Brohus H. and Nielsen P.V. 2007. *Benchmark Test for Computer Simulated Person – Manikin Heat Loss for Thermal Comfort Evaluation*. Aalborg University and Gävle University.

Parsons K.C. 1993. *Human Thermal Environments*. Taylor & Francis, London, pp. 23-26.

Schälin A. and Nielsen P.V. 2004. "Impact of turbulence anisotropy near walls in room airflow," *Indoor Air*, Vol. 14, Issue 3, pp. 159-168.