



This item was submitted to Loughborough's Institutional Repository (<https://dspace.lboro.ac.uk/>) by the author and is made available under the following Creative Commons Licence conditions.



CC creative commons
COMMONS DEED

Attribution-NonCommercial-NoDerivs 2.5

You are free:

- to copy, distribute, display, and perform the work

Under the following conditions:

BY: **Attribution.** You must attribute the work in the manner specified by the author or licensor.

Noncommercial. You may not use this work for commercial purposes.

No Derivative Works. You may not alter, transform, or build upon this work.

- For any reuse or distribution, you must make clear to others the license terms of this work.
- Any of these conditions can be waived if you get permission from the copyright holder.

Your fair use and other rights are in no way affected by the above.

This is a human-readable summary of the [Legal Code \(the full license\)](#).

[Disclaimer](#) 

For the full text of this licence, please go to:
<http://creativecommons.org/licenses/by-nc-nd/2.5/>

Simulation of human thermal responses in a confined space

Yi Zhang* and Tong Yang

Institute of Energy and Sustainable Development, De Montfort University, United Kingdom

*Corresponding email: YiZhang@dmu.ac.uk

SUMMARY

Latest human thermal comfort models, such as the IESD-Fiala model, are active multi-nodal thermal models that simulate physiological regulatory responses, e.g. changing metabolic rate and skin blood flow, shivering and sweating. Commercial CFD packages, such as ANSYS CFX, are widely used in studying transient thermal environment. The purpose of this study is to develop and demonstrate the method for integrating human thermal comfort models with the CFD environment for detailed transient simulations. Different integration strategies are discussed in this paper, as well as the technical problems with using detailed (clothed) 3-D model in the coupled simulation. It is highlighted that further research is required to exploit the full potential of the integrated model in environmental design.

KEYWORDS

Thermal comfort, IESD-Fiala model, CFD, Transient simulation, Integration

INTRODUCTION

Hypothetically, how would the human body respond to a confined space with adiabatic walls? Assuming the initial air temperature in the space is lower than the skin temperature, it would rise as a result of the accumulation of body heat. Gradually dry convective heat loss from the body would diminish as the air temperature approaches the skin temperature. Radiant heat transfer between body and surrounding walls is dependant to the wall temperature and the body surface temperature. For the hypothetical adiabatic walls, the surface temperature would remain a value between the body surface temperature and the air temperature, therefore the radiant heat loss from the body would disappear quicker than the convective heat loss. Depending on the size of the space and the metabolic rate, the body would soon rely on evaporation of sweat to maintain its core temperature. This would not last long either, since the humidity ratio in the space would be rising, too. Inside the body, the physiological regulatory system would cause the surface temperature to increase to maintain heat loss via convection and radiation, while sweat more. It would lose control, however, as soon as the body surface is completely covered with sweat. The question would then be for how long a person could survive in such a condition. We found we did not have the means to answer it.

With general advances in computing technology, especially in modelling and simulation, new tools have been developed for studying human thermal comfort. In the foreseeable future, engineers will be able to use digital thermal manikins to explore and experience the proposed thermal environment at the design stage. To achieve this, however, important developments in two directions are necessary. Firstly, a fully dynamic human thermal model with fine time resolution as well as identifiable individual characteristics is required. The aim is to be able to predict thermal comfort for a particular group of people in a wide range of environments and activities. Secondly, a seamless interface between the human model and the environmental model is needed. As well as to sense its environment, the human model should be able to react and influence the environment by heat and mass transfer, activities and control.

This paper uses a simple example to discuss the integration of an established human model with a commercial CFD package for dynamic simulation. The main objective is to identify the key issues in such attempts and provide outlooks for future development.

BACKGROUND

Computer models of human thermal comfort have been developed since 1960's. The latest advances provided great insight into the heat transfer process and the thermal physiology of the human body. Models can now predict human thermal responses in a wide range of thermal environments (Fiala, 1998; Zhang, 2003). Compared to the level details modelled inside the human body, heat exchange between the body and the ambient environment still relies on empirical and simplified methods. On the other hand, the technology of Computational Fluid Dynamics (CFD) has matured in recent years. It was marked by a range of commercial CFD software packages being widely adopted in both engineering and scientific works. It is natural to consider combining the power of both worlds, i.e. to integrate the human models with the CFD code.

Several attempts have been reported. The automotive industry has been using the CFD and human model to design comfortable cabin space. Since occupant's contribution to the thermal environment of the cabin of a car is considerably less than the weather, one-way link from the CFD tool to the human model (therefore the human model acts like a multi-sensor) is normally sufficient. Such examples can be found in (Han et al., 2001; Tanabe, 2004). For air flight cabin design and building (especially passive buildings) design, body heat from the occupants become significant, therefore a bi-directional link is necessary. Murakami et al. (2000) reported one of the early attempts for a full integration. A vase-shaped smooth figure was used to represent the human body in the CFD, which was coupled with Gagge's 2-node thermal regulatory model. The CFD code simulated the flow field for the given temperature boundary conditions. Meanwhile the 2-node model provided prediction of skin temperature distribution corresponding to the local sensible heat loss values calculated by the CFD. Steady state simulation of the coupled system was performed. Since the solution of the 2-node human model did not require iteration, an otherwise inefficient arrangement was used, i.e. the update of skin temperature was implemented in the out-most loop. Al-Mogbel (2003) replicated Murakami's approach with only a few changes. Firstly, a human figure consisted of 6 cylinders was modelled in FLUENT, an established commercial CFD package. Gagge's 2-node model was used, and the integration method appeared to be unchanged, except domain air temperature was chosen as the convergence criterion instead of the skin temperature in Murakami's study. This is questionable because domain air temperature is insensitive to local skin temperature distribution. The paper did not reveal the details of the implementation of the coupled model.

Neither simplified 3-D model, nor 2-node human model would satisfy the requirement for studying human response to heterogeneous environment. Tanabe et al. (2002) reported integration of the 65MN model with CFD and radiation code. Realistic 3-D model of an unclothed male body was used with the 65-node thermoregulatory model. Steady state results were shown to include the effect of solar (short wave) radiation. Convective heat transfer from the body, however, was calculated from empirical heat transfer coefficients, rather than from CFD simulation. As a result, the CFD code was mainly used to calculate the impact of human body on the environment. Other works include (Omori et al., 2004), in which CFD code coupled with Fanger's model was described. The paper was useful for validating CFD results against experimental results with a thermal manikin.

So far the approaches described in literatures were unsatisfactory in 3 ways. Firstly, the detailed 3-D models of human in the CFD environment represented the naked body, which is unlike the case in every day environment. It is perceivable that the shape and posture of the model may have a significant impact on the local convective and radiant heat transfer coefficients. Models of clothed body should be studied to quantify such impact. Secondly, evaporation at the skin surface is a complex process, which is connected with moisture transportation due to convection. In the studies where evaporation was considered, empirical methods rather than CFD were used. Most importantly, all studies we have found are steady state simulations. The requirement and challenge for the coupling method for a transient simulation is different than that for a steady state simulation. More considerations have to be given to the stability of the coupled model and the computational efficiency.

METHODS

In this paper we explore the new territory on two fronts – dynamic simulation with a clothed 3-D figure. A simple example is used to describe the method for integrating of human model with environmental simulation tools for the purpose of detailed transient simulation. A typical man (height=1.75m, weight=75kg, total skin area = 1.83m²) is standing in a box room (3x3 x2.5m). The 3-D model of the standing figure in a casual outfit, the Computational Thermal Manikin (CTM, Yang et al., 2007), was developed using a range of software including ANSYS ICEM CFD 11. The centre of the manikin is located at 1m to the right-hand side and the back walls. Since the box space has not any openings, the air flow of natural convection is driven by surface heat transfer. The surrounding walls were modeled with fixed temperature of 28°C, whereas the body surface temperatures is supplied by the IESD-Fiala model. Emissivity of all surfaces is set to 0.95.

In order to limit memory footage and reduce computing time, a “coarse” mesh was selected (Fig.1). There were 31006 surface mesh elements to form the detailed geometry of the manikin. Surface mesh size ranges from 0.002m to 0.109 m. In total 717,266 elements (263,937 nodes) were generated inside the fluid domain, including the prism layers near manikin body. The maximum value of y^+ (the dimensionless distance from the wall) was 2.02 and with an average value of 1.38. For accurate boundary layer simulation, $k-\omega$ based Shear Stress Transport (SST) model (Menter, 1994) was used. The discrete transfer model (Ansys, 2005) with 16 rays was used to model radiant heat transfer. Initial test showed that, compared to Monte Carlo method with 2 million histories, the discrete transfer model gives more consistent results at lower computational cost.

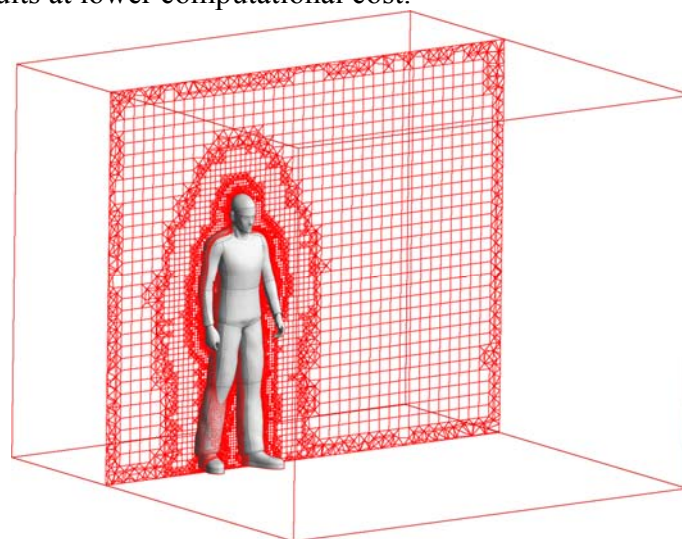


Figure 1. CFD model layout and mesh.

Methodology

To interface IESD-Fiala model with the commercial package ANSYS CFX 11, some modifications to the Fiala model are necessary. The objective is to replace the empirical calculations of environmental heat transfer in the Fiala model with CFX simulation results. The Fiala model treats total heat transfer between the human body and the environment as a sum of 7 components including conduction, convection, long and short wave radiation, and evaporation at the body surface, as well as respiratory convective and evaporative heat losses. In this example, conductive and short wave radiant heat transfer is insignificant. The respiratory heat losses are computed using the existing empirical equations in the Fiala model, except the condition of the inhaled air (domain average air temperature and moisture content) is obtained from CFX. We focus on the heat transfer due to convection (q_{conv}), long wave radiation (q_{rad}) and evaporation (E_{sk}).

Figure 2 shows the calculation of the total heat flux at the body surface (q_{sk}) in the Fiala model. The oval shapes identify variables/parameters, whereas the small circles represent equations/models. The mean ambient air temperature T_a , the air velocity v_a , and the surface temperature T_{surf} are used in calculating the convective heat transfer coefficient (h_{conv}) (see Fiala, 1998 for details). Convective heat flux q_{conv} is subsequently calculated. Similarly, the surface temperature T_{surf} and the mean radiant temperature of the envelope T_{env} are used to calculate the radiant heat flux q_{rad} . The evaporative coefficient U_{evap} is calculated from h_{conv} using Lewis analogy. The mean ambient vapour pressure p_a and the skin vapour pressure p_{sk} are then used to calculate evaporative heat loss E_{sk} . The surface heat flux q_{sk} is passed to the human model for calculating internal thermal state of the body and its regulatory response, which in turn updates T_{surf} and p_{sk} .

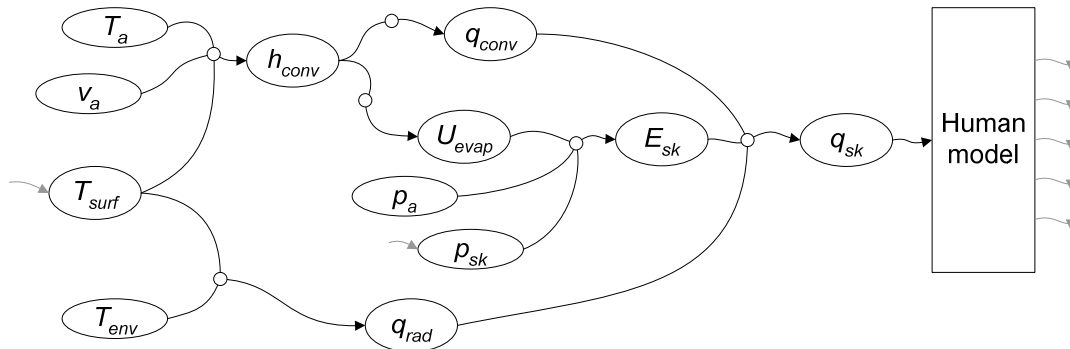


Figure 2. Calculation of environmental heat transfer in the Fiala model.

Figure 3 shows the data flow of the proposed CFD-based boundary calculation. Body surface temperature T_{surf} is fed to the CFD model, which solves the flow field as well as convective and radiant heat fluxes (q_{conv} and q_{rad}) at the surface. We did not implement a moisture transfer model in CFX; instead, moisture flux corresponding to the evaporative heat flux E_{sk} is added to the domain as a uniform volume source. Since h_{conv} is reversely calculated from q_{conv} , the evaporative heat loss E_{sk} is nevertheless affected by the CFD simulation result. Although omitted from the diagram, the CFD model also passes back mean air temperature (T_a), air velocity (v_a) and vapour pressure (p_a) in the domain, which subsequently affect the calculation of respiratory heat losses. It is worth noting that the Fiala model includes the boundary condition calculation in the solver iteration for determining the transient state of the human body. When CFD is involved in the calculation, a new iteration strategy is required to maximize the performance.

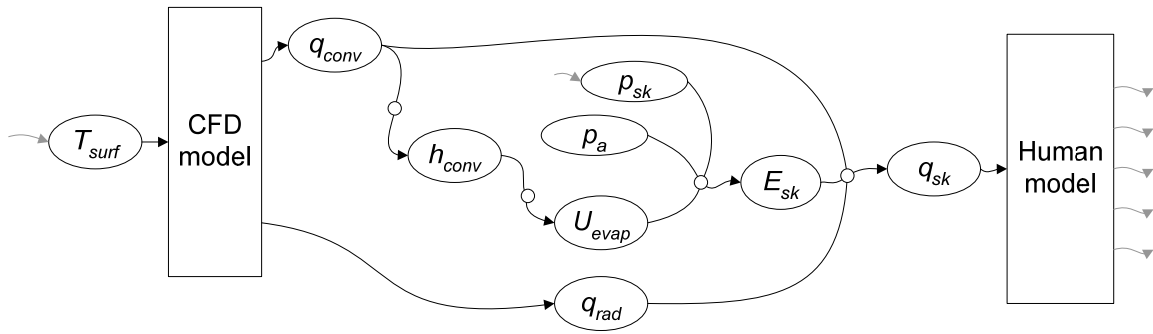


Figure 3. Environmental heat transfer calculation using CFD results.

Implementation

Problems arise with the attempt to couple two dynamic models, i.e. the environment model and the human model, with a small set of boundary parameters. How to solve the coupled system requires careful consideration to achieve optimum computational performance. The suitable strategies need to be thoroughly analyzed. In this paper we provide some pointers.

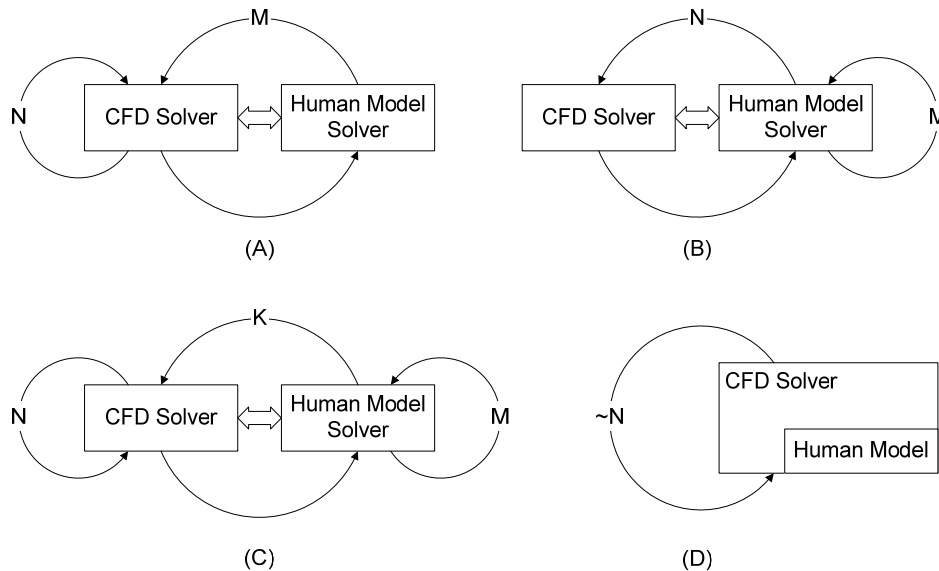


Figure 4. Implementation strategies.

Figure 4 shows 4 possible implementation strategies. Consider separate solvers are to be used for the CFD model and the Human model, figure 4A represents calling the CFD solver in each human model solver's iteration loop. Assuming the CFD solver requires N iterations to converge, while the human model solver requires M iterations, the total iterations the CFD solver has to perform equals $M*N$, which is highly expensive. Conversely, as in figure 4B where the human model solver is called from the CFD iteration loop, it is much more efficient as the increased cost on the human model solver is negligible. The disadvantage, however, is that significant effort is required to customize the CFD solver (e.g. using junction box functions in CFX), therefore the solution is "hardwired" to the particular software.

To maintain independence of the solvers as well as reasonable efficiency, the compromise is to introduce the third "supervisory" layer of iteration that alternatively calls both solvers (see figure 4C). If $K < M$, it is more efficient than strategy A. K can be minimized by reducing the physical time step size (Δt). Using strategy A as the reference, we tested two different settings for strategy C: $K=1$, $\Delta t=60s$ and $300s$, respectively. Table 1 summarises the results

from transient simulation for 60 minutes. It is clear that strategy C with a time step of 60 second is sufficiently accurate for our purpose. Strategies A, B, and C are “weak” coupling methods, since dynamic model are solved separately. The strategy D in Figure 4 represents the “strong” coupling approach that the differential equations of the human model being solved as part of system equation set by one solver for numerical stability and efficiency.

Table 1. Comparison of iteration strategies.

	Reference	Mean Error (SD)	
	Strategy A	Strategy C (K=1, $\Delta t=60s$)	Strategy C (K=1, $\Delta t=300s$)
Surface temperature T_{surf} ($^{\circ}C$)	31.6551	+0.0001 (0.0011)	-0.0067 (0.0084)
Total heat flux q_{sk} (w/m^2)	48.3609	-0.0522 (0.0586)	-0.2785 (0.5147)
Evaporative heat flux E_{sk} (w/m^2)	22.2557	+0.0157 (0.0267)	+0.2784 (0.2734)
CPU time / Physical time (hr/hr)	219	51	50

RESULTS

In the experiment, we chose constant temperature walls by assuming the thermal mass of the construction would stop wall surface temperature from rising during the experimental period. Initial temperature of wall surface and domain air was set to 28.0 $^{\circ}C$. Initial relative humidity was 60%. The human model (1.2 met, 1.7 clo) was assumed to be wearing the KSU outfit with street shoes (Fiala, 1998). The initial state was calculated from assuming the dynamic thermal sensation (DTS) value of 0.5, which equates slightly warm. Transient simulation was performed with the coupled model for 60 minutes. The results were compared with the prediction of the IESD-Fiala model with the same boundary conditions.

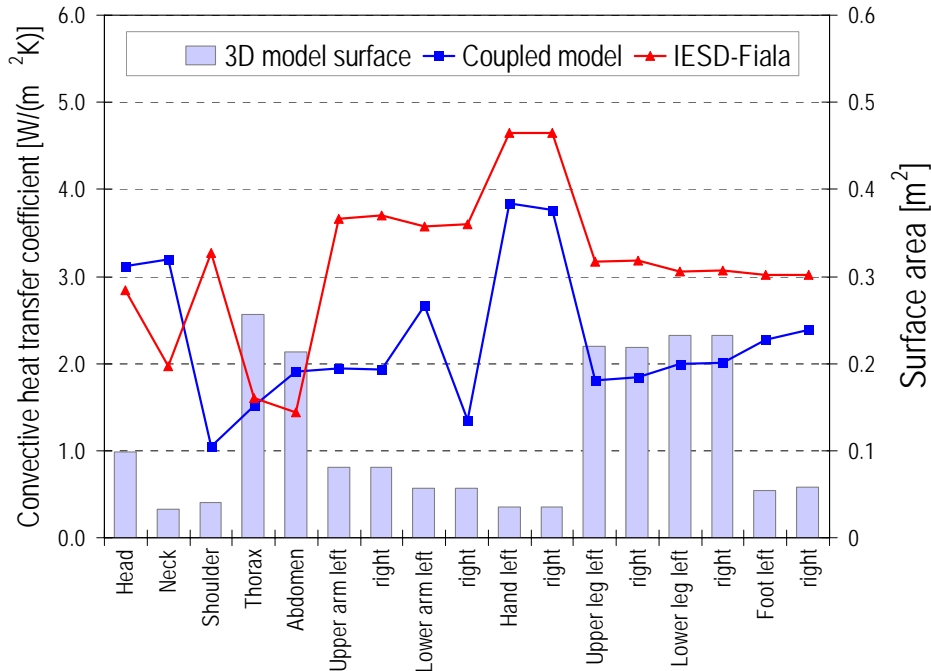


Figure 5. Local convective heat transfer coefficients at 60 min.

Figure 5 shows the predicted convective heat transfer coefficient (h_{conv}) at each body part at 60 minute. Significant difference is observed between the coupled model and IESD-Fiala. The calculation of h_{conv} in the Fiala model, which is based on empirical models from literature,

assumes minimal mean air velocity (v_a) to be 0.05 m/s. Whereas in the CFD model, domain average air velocity reached only 0.015m/s as the result of natural convection. Another difference is that, while the boundary condition for IESD-Fiala remained constant (28°C, 60%), the room condition was changing for the coupled model. Mean air temperature and relative humidity reached 28.25°C and 70.1%, respectively. It is interesting to note the asymmetry between left and right lower arms as a result of slightly lifted left arm. Figure 6 shows the cloth surface temperature distribution at 60 minute. Again, significant difference is present in clothed areas. Over time, both models predicted increase of skin temperature (Figure 7). The error was under 0.12°C. The coupled model predicted the person would feel significantly warmer than IESD-Fiala’s prediction.

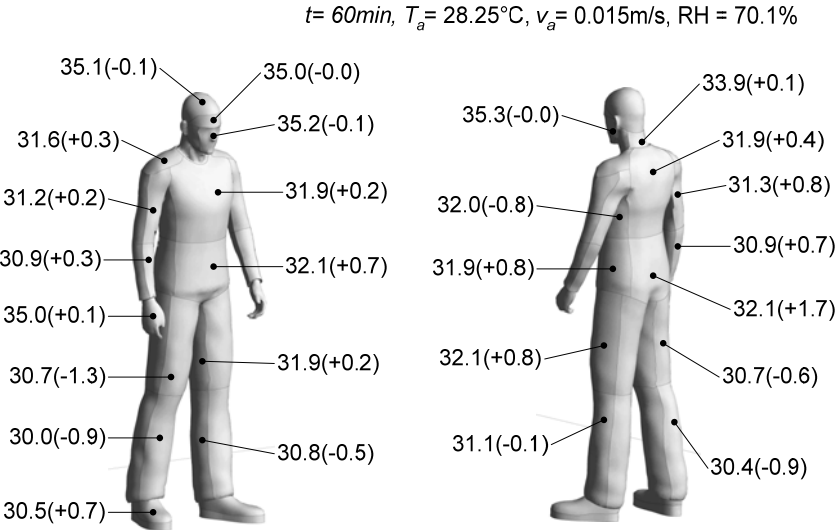


Figure 6. Prediction of cloth surface temperature at 60 min by the coupled model (°C). (Numbers in brackets are errors compared to the predictions of the IESD-Fiala model)

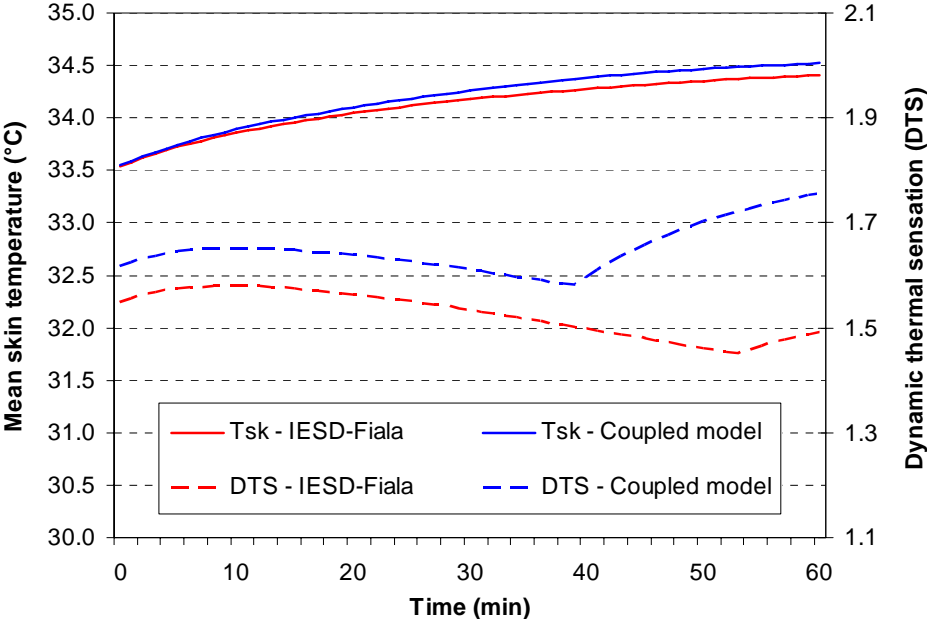


Figure 7. Change of mean skin temperature and dynamic thermal sensation over time.

DISCUSSION

In this paper we demonstrated transient simulation of the human body within an enclosed space, where air movement was driven by natural convection from the body. An established human thermal model was coupled with commercial CFD software to facilitate the simulation. Coupling method, especially the data flow between the models and implementation options, had been discussed. The simulation results showed that greater details of the interaction between human body and the environment can be revealed with the coupled model. The potential future applications for couple models like such are unlimited. The biggest challenge is, however, to validate these models. New experiments with human subjects, accurately controlled environment, and high resolution equipment may be required.

In our implementation, a number of simplifications had been made. Firstly, moisture transport from skin surface to the surrounding environment was calculated with empirical coefficients rather than using CFD. Secondly, heat and moisture transfer between the skin surface and the cloth surface was treated with much simplified thermal/moisture resistance values. Although in theory, realistic 3D model of clothed figure in CFD should provide more accurate simulation of convective and radiant heat transfer than simplified or nude models, the IESD-Fiala model treats clothing as a virtual layer without heat (or moisture) capacitance, which could cause extra error if area or shape of each surface segment is not correctly matched to the corresponding skin segment. Thirdly, time resolutions of the human model and the CFD model are significantly different (minutes vs. sub-seconds), which has an adverse impact on the coupled model. Further development and validation of the human thermal regulatory model with high time resolution are necessary. And finally, the coupled model is excessively expansive, especially when finer mesh with more accurate ray tracing method is used. Computational performance remains a barrier for future applications.

REFERENCES

- Al-Mogbel A. 2003. A coupled model for predicting heat and mass transfer from a human body to its surroundings. Proceedings of the 36th AIAA Thermophysics Conference, Florida, USA.
- Fiala, D. 1998. Dynamic Simulation of Human Heat Transfer and Thermal Comfort. PhD thesis. De Montfort University, Leicester, UK
- Han T., Huang L., Kelly S., Huizenga C., Zhang, H. 2001. Virtual Thermal Comfort Engineering, SAE Paper, 2001-01-0588, Mar. 2001.
- Menter F. R. 1994. Two-equation eddy-viscosity turbulence models for engineering applications, AIAA Journal 32 (1994) pp.1598-1605.
- Murakami S., Kato S. and Zeng J. 2000. Combined simulation of airflow, radiation and moisture transport for heat release from a human body. Building and Environment, 35(6): 489-500.
- Omori T., Yang J. H., Kato S. and Murakami S. 2004. Coupled simulation of convection and radiation on thermal environment around an accurately shaped human body. In Proceedings of the 9th international conference - RoomVent 2004. Portugal, pp.7.
- Tanabe S. 2004. Prediction of Indoor Thermal comfort in vehicle by a numerical thermoregulation model and CFD. In: Proceedings of the 9th international conference - RoomVent 2004. Portugal, pp.12.
- Tanabe S., Kobayashi K., Nakano J., Ozeki Y. and Konishi M. 2002. Evaluation of thermal comfort using combined multi-node thermoregulation (65MN) and radiation models and computational fluid dynamics (CFD). Energy and Buildings, 34(6): 637-646.
- Yang T., Cropper P. C., Cook M. J., Yousaf R. and Fiala D. 2007. A new simulation system to predict human-environment thermal interactions in naturally ventilated buildings, Proc. 10th International Building Performance Simulation Association Conference and Exhibition (BS2007), Beijing, China, pp.751-756.
- Zhang H. 2003. Human thermal sensation and comfort in transient and non-uniform thermal environment. PhD thesis, University of California at Berkeley, California, USA.