

A NEW SIMULATION SYSTEM TO PREDICT HUMAN-ENVIRONMENT THERMAL INTERACTIONS IN NATURALLY VENTILATED BUILDINGS

Tong Yang¹, Paul C Cropper¹, Malcolm J Cook¹, Rehan Yousaf^{2,1} and Dusan Fiala²

¹Institute of Energy and Sustainable Development, De Montfort University,
Leicester LE1 9BH, UK

²Division of Building Physics (ftba), Universität Karlsruhe, Englerstrasse 7, D-76131
Karlsruhe, Germany

ABSTRACT

This paper describes the development of computational thermal manikins (CTMs) to be used in a coupled simulation environment to simulate the human thermoregulatory response in buildings. 3D graphic design and engineering tools have been used to create CTMs with different postures and clothing insulation levels. Computational fluid dynamics (CFD) simulations of a nude CTM in a space with displacement ventilation has shown good agreement with experimental data of measured convective and radiative heat transfer coefficients. Investigations of a clothed CTM in a space with natural ventilation has also been conducted and compared with published experimental data.

KEYWORDS

Computational thermal manikin, CFD, heat transfer, thermal comfort, model coupling.

INTRODUCTION

Computational fluid dynamics (CFD) is widely used by researchers to study the human occupancy factor in buildings. Various details of computational thermal manikin (CTM) have been used to study the microclimate around human occupants in buildings. The complexity of the geometry ranges from the simplified human shape to 3D scanned real manikins (Murakami et al 2000, Al-Mogbel 2003, Gao and Niu 2005, Sideroff and Dang 2005). In order to accurately predict the thermal plume around the human body, the overall and local thermal comfort/discomfort and air quality in the vicinity of human body, a detailed human geometry is necessary. Furthermore, personal differences, e.g. size, shape, clothing designs and activity levels also affect the thermal sensation for individuals (Havenith 2002).

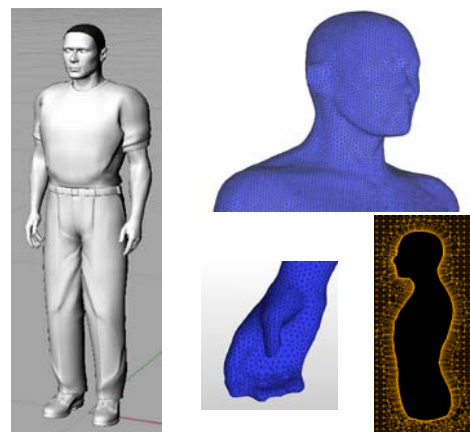
The objective of this study was to investigate the flexibility of designing a CTM with different postures and clothing insulation levels and explore the balance of geometric complexity and computational efficiency to predict the air flows of the personal micro-environment with CFD.

The work reported here is part of a large project on the development of a new coupled simulation system to simulate the human thermoregulatory response in naturally ventilated buildings. Coupled with a CFD simulation, the human thermo-physiological model, IESD-Fiala model (Fiala 2001), is adopted to simulate the metabolic heat production and the thermoregulatory control processes of the human body. The coupling technique of IESD-Fiala comfort model and CFD simulation are outlined. In naturally ventilated highly non-uniform indoor environments, detailed analysis of thermal comfort would be required not only for the whole body but also for individual body parts. The further studies of the subdivided CTM consisting 59 bodyparts will be the subject of another paper.

METHODOLOGY

Computational thermal manikin design

The virtual manikin represents a male subject with a height of 1.80m and a DuBois area of 1.83m². The 3D detailed human model was created using the 3D graphic design package, Poser 6 (e-frontier 2005),



(a) by Rhino

(b) by ICEM

Figure 1 The computational thermal manikin (CTM) modified in Rhino and meshed in ICEMCFD

for different postures and clothing levels. Some simplification was performed in Rhinoceros 3 (Robert McNeel & Associates 2005) engineering

design tools to remove unnecessary details of eye lashes, teeth and shoe laces, etc., then the manikin geometry was imported into ICEM CFD 10 (Ansys Inc. 2006) and put into a building enclosure with various ventilation openings configurations for CFD modelling. Figure 1 shows the details of the human body geometry and meshing.

For validation purposes, CFD predictions are compared with an experimental benchmark study for a nude standing CTM in a displacement ventilation environment (Nielsen et al 2003). The testroom dimensions are 3m×3.5m×2.5m as width, depth and height, respectively. There are two openings located centrally on opposite walls at two levels. The size of the lower level supply opening is 0.4m×0.2m and the higher level exhaust opening size is 0.3m×0.3. The manikin is placed 0.050m above the floor. The computational domain and ventilation opening settings are shown in Figure 2. Measure poles for data collection in the experiments are illustrated in Figure 3.

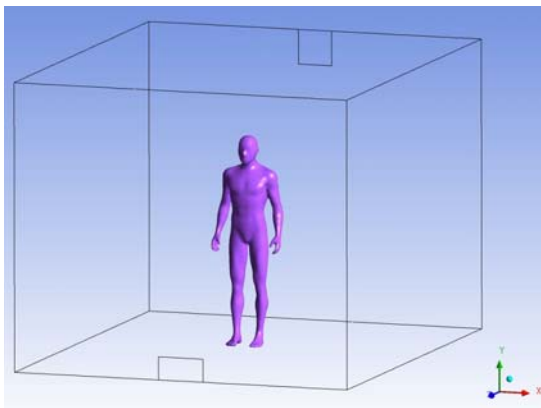


Figure 2 Computational domain and geometry specification of the benchmark case study with displacement ventilation

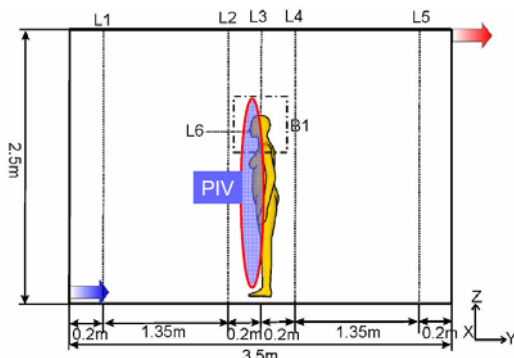


Figure 3 Measuring poles on the central section in the benchmark tests (Nielsen et al 2003)

Mesh configuration and boundary conditions

CFD solutions are obtained with Ansys-CFX 10, which is a general-purpose, unstructured-grid code

(Ansys Inc. 2005). CFX uses a Finite Volume method to model the equations for the conservation of mass, momentum and energy in terms of the dependent variables and pressure in their Reynolds time-averaged form. The solution variables are discretised on a co-located grid with a second order fully conservative vertex based scheme. The resulting linear equation system is solved with a fully coupled Algebraic Multi-Grid (AMG) solver.

In the present study the *SST* turbulence model, which is the combination of standard *k-ε* model (Versteeg and Malalasekera 1995) in flow far field and the *k-ω* model (Wilcox 1993) in the wall boundary layer, is applied with the high resolution second order discretisation scheme in combination with an automatic wall function. With a built-in lower limit for y^+ (the non-dimensional wall distance), the automatic near wall treatment ensures that the wall function is applied only in the log-law region, which allows for a consistent grid refinement near the wall.

Three levels of mesh resolution on the CTM surface were tested for grid independent solutions in line with best practice guidelines for CFD applications. The meshing configuration details are listed in Table 1.

Table 1 Meshing details

Mesh Name	Total number of elements	Min. size on CTM (mm)	Total number of prism layers	Max value of y^+
Grid A	351,000	8	3	8.95
Grid B	964,000	5	5	6.36
Grid C	2,500,000	3	10	3.49

CFD boundary conditions for the benchmark displacement ventilation case of a nude CTM

In the benchmark test, there is no exact ratio between the amount of convective heat transfer and radiative heat transfer between human body and the surroundings being stated. Srebric et al (2007) used a simplified CTM and recommended the convection to radiation ratio (C:R) 30:70 for CTMs. Boundary conditions used in the present study are shown in Table 2.

Computational domain and boundary conditions for the natural convection case of a clothed CTM

Considering a person wearing summer clothing, the clothing area factor f_{cl} was chosen as 1.15 (Fanger 1972 and Holmer et al 1999). Therefore, the surface area of the clothed CTM is 2.095m² and body height is 1.803m. The computational domain has the same width and depth as 3m (to omit horizontal aspect ratio effect) with a height of 2.5m. There are two openings at the floor level sized as 0.5m×0.25m;

Table 2 Boundary conditions for a standing nude CTM in the displacement ventilation case

CTM Height (m)		1.804
CTM A_{Du} (m ²)		1.830
CTM Feet distance from the floor (m)		0.050
Inlet	Temperature (°C)	22
	Velocity (m/s)	0.182
	Turbulence intensity (%)	40
	Turbulence length scale (m)	0.1
Outlet	Pressure boundary	Relative $p=0$
Wall boundaries		Adiabatic, emissivity=0.95
CTM boundaries		emissivity=0.98
Heat Flux on the CTM [W]	Convective	22.8
	Radiative	53.2
Heat flux on the floor [W]		10

another two openings of 0.25m×0.25m in size are located at the ceiling level. This configuration was designed to have little effect on the thermal plume generated by the human in the centre of the room. The clothed CTM is located on the centre of the floor as shown in Figure 4.

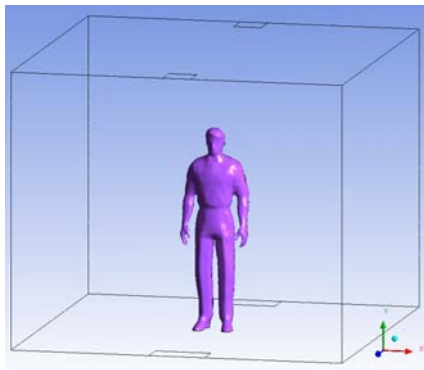


Figure 4 Computational domain of the clothed standing CTM in natural convection case

According to ASHRAE fundamentals handbook (1993), the mean surface temperature of a human body in the state of physiological thermal neutrality with normal indoor activity is 33.7°C. In the current test, this temperature boundary condition was used on the CTM. Air with temperature of 22°C was drawn in from the floor level openings merely due to the presence of human occupant. Surrounding walls were set as adiabatic boundaries.

RESULTS

The following CFD results are presented for displacement ventilation benchmark case study and natural convection test case.

Displacement ventilation case

Figure 5 shows the calculated velocity distribution for the room central cross-section for the DV case. Velocity and temperature profile along the datalines (L1, L2, L4 and L5) are shown in Figures 6 and 7 with the benchmark measurements, bkL1, bkL2, bkL4 and bkL5 namely.

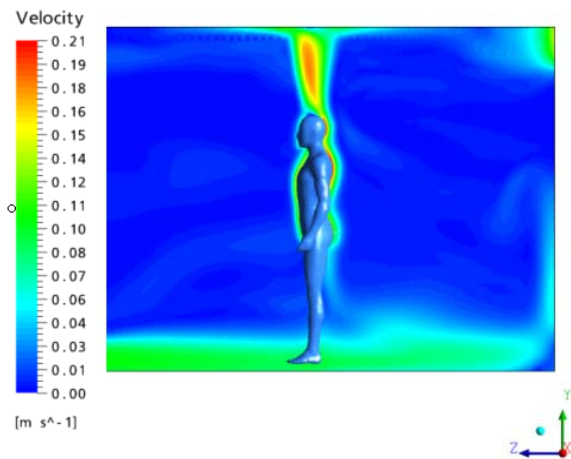


Figure 5 Velocity contour plot on the central vertical plane

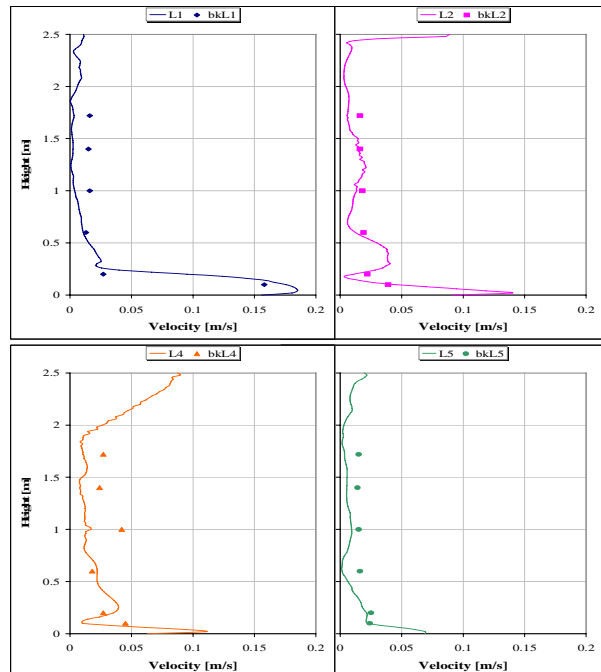


Figure 6 Velocity profile along datalines vs benchmark measurements

The maximum predicted velocity above the CTM head is 0.21 m/s in Figure 5, which agrees favourably with the experimental data of 0.23 m/s. In the benchmark study, the height and body surface area are 1.651m and 1.471m². The body height and

shape difference between the present CTM and the experimental work may have contributed to the velocity discrepancies close to the manikin (Figure6). The predicted mean convective heat transfer coefficient is $4.36 \text{ W/m}^2\text{K}$ in this study.

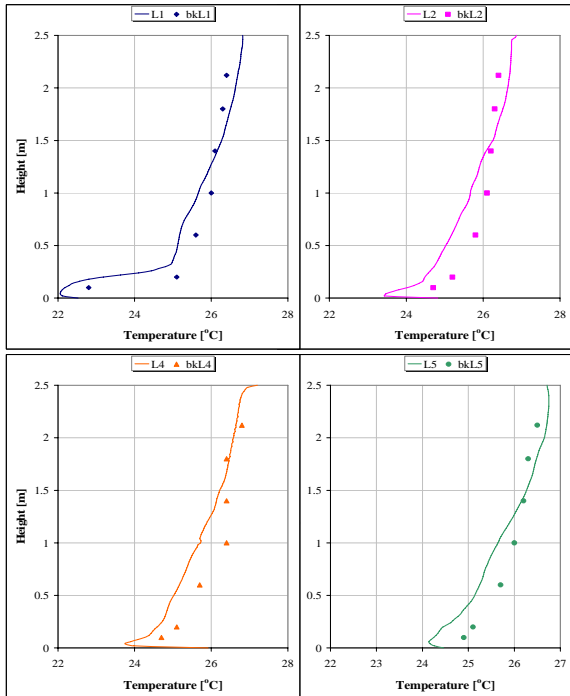


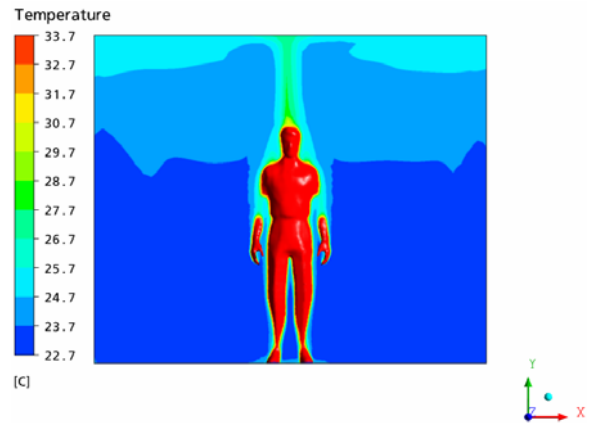
Figure 7 Temperature profile along datalines vs benchmark measurements

Previous experimental work obtained values in the range of 3.4 to $3.86 \text{ W/m}^2\text{K}$ in ambient air speed less than 0.1 m/s (Gao and Niu 2005). Other CFD predictions of values between 3.9 and $4.3 \text{ W/m}^2\text{K}$ were found in cases where the inlet velocity was less than 0.12 m/s . The difference between predicted and measured air velocity, air flow direction and turbulence intensity are thought to contribute to the discrepancy in convective heat transfer coefficients. Furthermore, RANS model cannot accurately reproduce the laminar to transition boundary flow around the human body.

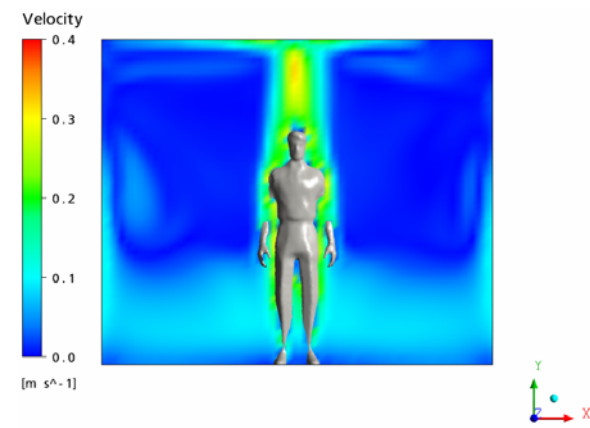
The mean radiative heat transfer coefficient is $4.61 \text{ W/m}^2\text{K}$ predicted in this study which agrees well with the generally accepted whole-body value of $4.7 \text{ W/m}^2\text{K}$. In addition to the shape and size difference, relative position and posture changes affect the view factors significantly in radiation studies (Yousaf 2007), which are reflected by the difference in temperature profiles in Figure 7. Above the height of 1.5 m , about the shoulder level, more turbulent mixing occurs, so the predicted temperature is closer to measured values.

Natural ventilation case

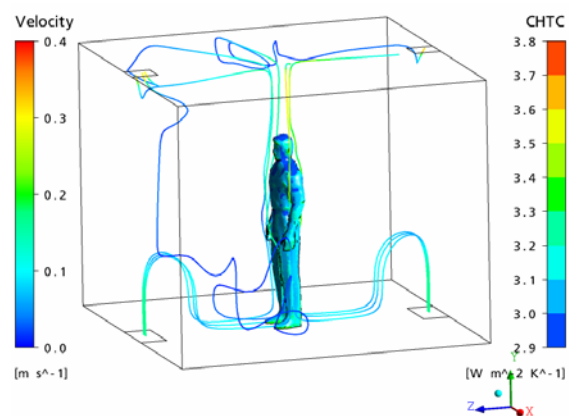
The predicted air flow field around the CTM and convective heat transfer coefficient distribution on the CTM surface are shown in Figure 8. The predicted area weighted average convective heat transfer coefficient (CHTC) on the CTM is 3.1 .



(a) Temperature distribution on central vertical plane



(b) Velocity distribution on central vertical plane



(c) Velocity streamline through openings and convective heat transfer coefficients on CTM

Figure 8 Air flow field around the clothed standing

CTM in natural convection case

The distribution pattern of CHTC shows that hands, feet, front face and ears region generally have higher

CHTCs than the central torso region. Lower CHTCs are found on the shoulders, where local stagnation of flow occurs in the region.

In reality, surface temperature distribution on human body is not uniform. The IESD-Fiala model is derived from a large body of experimental data by regression analysis, taking account of the environmental effects on human body and its responses. A coupled simulation system integrating the IESD-Fiala model with CFD software will provide closer predictions towards field measurements.

MODEL COUPLING

The IESD-Fiala human thermo-physiological model and the CFD model exchange data using files. This method was chosen because it is simple, robust and platform neutral. The IESD-Fiala model provides a file containing parameters such as surface temperatures and perspiration rates etc for each bodypart, which are used to provide boundary conditions for the CFD simulation. Using these boundary conditions the CFD solver is run to find a steady state solution. When convergence is achieved the solving process is suspended and parameters such as heat flux, mean air temperature and mean air velocity over each area of the body are returned to the IESD-Fiala model in a second file. The IESD-Fiala then uses this data to predict the body's responses and produces a new CFD boundary conditions file. The simulation process is then repeated as illustrated in Figure 9.

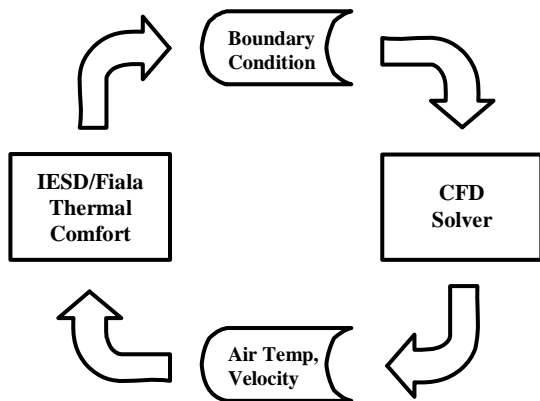


Figure 9 CFD solver and thermal comfort model interaction

The chosen CFD package (ANSYS CFX) provides facilities for extending the functionality of the solver by including user written FORTRAN code. CFX Expression Language (CEL) functions can be written to replace fixed values. In this application, CEL functions are used to apply boundary condition values provided by the IESD-Fiala model. This method allows the boundary conditions to be

changed, at the beginning of each coupled simulation cycle, before CFD solving is resumed after being suspended. Junction Box subroutines can also be called at specific points in the solver execution cycle, for example at the end of each iteration. This method is used to determine when convergence has been achieved, to output data required by the IESD-Fiala model and to load the new boundary condition values.

CONCLUSIONS AND FUTURE WORK

Heat transfer and air flow simulations around human occupants in indoor environments are highly complicated phenomena. Using detailed 3D CTM in an integrated CFD and human physiology model coupling system could be a robust design tool to investigate the human environment interaction dynamically, even taking account of various sizes, shapes, clothing designs and postures of the human occupants. The fine geometric features have great effects on radiation prediction.

CFD predictions of the air flow and temperature distribution around a nude CTM in a room with displacement ventilation has shown favourable agreement with experimental work. Predicted convective heat transfer coefficient is 4.36 W/m²K. It is at the upper end of most published experimental and CFD results, which range from 3.4 to 4.3 W/m²K. However, these were measured and simulated with inlet velocities less than 0.12m/s. The calculated radiative heat transfer coefficient of 4.61 W/m²K, agrees well with the generally accepted whole-body value of 4.7 W/m²K.

In the case of natural ventilation, the surrounding environmental parameters and inner human thermo-regulatory responses are dynamic and inter-related via the CTM skin/clothing surface. The effects of air flow on clothing insulation and on the convective and radiative heat transfer coefficients will be evaluated for each body segment of the clothed human body in the next phase of the work.

ACKNOWLEDGMENT

The authors would like to acknowledge the financial support received from the Engineering and Physical Sciences Research Council, UK via grant: EP/C517520/1. We would also like to acknowledge the technical supports provided by Ansys CFX.

REFERENCES

Al-Mogbel AM. 2003. A coupled model for predicting heat and mass transfer from a human body to its surroundings. 36th AIAA Thermophysics Conference. Orlando, Florida.

Ansys Inc. 2005. Ansys CFX 10.0 user manual.

- Anslys Inc. 2006. Ansys ICEMCFD 10.0 manual.
- ASHRAE. 1993. Fundamentals handbook, Chapter 8.
- de Dear RJ, Arens E, Zhang H. and Oguro M. 1997. "Convective and radiative heat transfer coefficients for individual human body segments", *International Journal of Biometeorology* 40:141-156.
- e-frontier Inc. 2005. Poser 6 manual.
- Fiala D, Lomas KJ. and Stohrer M. 2001. "Computer prediction of human thermoregulatory and temperature responses to a wide range of environmental conditions", *International Journal of Biometeorology* 45:143-159.
- Fanger PO. 1972. Thermal comfort - Analysis and applications in environmental engineering. New York, McGraw-Hill.
- Gao N. and Niu J. 2005. "CFD study of the thermal environment around a human body: A review." *Indoor Built Environment* 14(1): 5-16.
- Havenith G, Holmer I. and Parsons K. 2002. "Personal factors in thermal comfort assessment: clothing properties and metabolic heat production." *Energy and Buildings* 34(6): 581-591.
- Holmer I, Nilsson H, Havenith G. and Parsons K. 1999. "Clothing convective heat exchange--proposal for improved prediction in standards and models." *Ann Occup Hyg* 43(5): 329-337.
- 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.
- Nielsen PV, Murakami S, Kato S, Topp C. and Yang J.-H. 2003. "Benchmark tests for a computer simulated person." from http://www.civil.auc.dk/~i6pvn/cfd-benchmarks/csp_benchmark_test/Benchmark%20Tests_071103.pdf.
- Oguro M, Arens E, de Dear RJ, Zhang H. and Katayama T. 2001. "Evaluation of the effect of air flow on clothing insulation and on dry heat transfer coefficient for each part of the clothed human body." *J. Archit. Plann. Environ. Eng., AIJ*.(549): 13-21.
- Sideroff CN and Dang TQ. 2005. "CFD analysis of the flow around a computer simulated person in a displacement ventilated room." ASHRAE summer meeting, Denver, CO.
- Versteeg, HK and Malalasekera W. 1995. Introduction to computational fluid dynamics: the finite volume method. Harlow, Longman Scientific & Technical.
- Wilcox, DC. 1993. Turbulence modelling for CFD. California, DCW Industries, Inc.
- Yousaf R. 2007. "Radiation modelling," Ph.D. Interim Report 1, De Montfort University and University of Karlsruhe, Germany, 51 pages.