# CFD MODELLING OF BENCHMARK TESTS FOR FLOW AROUND A DETAILED COMPUTER SIMULATED PERSON

N. Martinho<sup>1,2</sup>, A. Lopes<sup>2</sup> and M. Silva<sup>2</sup>

<sup>1</sup>Department of Mechanical Engineering, Polytechnic Institute of Leiria, Portugal email: martinho@estg.ipleiria.pt http://www.estg.ipleiria.pt <sup>2</sup>ADAI, Department of Mechanical Engineering, University of Coimbra, Portugal

**Summary:** The sensible heat losses, by convection and radiation, from a full scale Detailed Computer Simulated Person seated in a parallelipipedic room and the respective parameters, like air velocity and temperature, of the flow around the body were predicted by means of CFD simulations performed in accordance with the specified conditions of two Benchmark Tests from which experimental data is available for comparison. The accuracy of the CFD results was studied and compared in terms of physical approximation errors, namely inlet boundary conditions and turbulence models, and spatial discritization errors due to the number of grid elements used.

*Keywords:* Computer Simulated Person, CFD, Benchmark Tests, Thermal Manikins *Category:* Virtual manikins

## 1 Introduction

Based on the fact that different researchers and research centers around the world have developed different configurations, from geometry to turbulence modeling, to represent a Computer Simulated Person (CSP), Nielsen et al (2003) [1] and later Nilsson et al (2007) [2] introduced full scale benchmark tests focusing, the first one on the air flow around the CSP and the second on the different heat losses from the CSP or virtual manikin. The results obtained for the 2003 Benchmark were presented and compared at a workshop of the conference Roomvent 2004 by four different groups that took part in the CFD modelling exercise: Aalborg University - Denmark, University of Syracuse - USA, University of Tokyo – Japan and the Health & Safety Laboratory - United Kingdom [4]

One case of the 2003 benchmark test configuration is a sitting person facing a unidirectional flow field considered similar to the flow field in a mixing ventilated room (Fig. 1). The 2007 benchmark test was made identical to the mentioned earlier with the intention that some of the flow field data would be comparable between the two tests.

In this paper the two referred benchmark tests, for the sitting CSP, are modeled, by means of CFD simulations, according to the respective room boundaries and CSP conditions. By comparing these CFD results with the experimental ones for the respective modeled cases as well as with the numerical ones from reference [4], both in terms of flow parameters as well as in terms of the heat losses by the complex and detailed CSP, the aim of the study is to report the accuracy of the CFD results and how this accuracy is influenced by the used approximation in the modeling process, namely, boundary conditions, turbulence model and number of grid elements.

## 2 The Test Case

The sitting posture 2007 benchmark test is based on experiments carried out at the Aalborg University in Denmark in a wind tunnel with box shaped geometry with a window on the side and dimensions Length × Height × Width =  $2.44 \text{ m} \times 2.46 \text{ m} \times 1.2 \text{ m}$  (Fig. 1).



Fig. 1. Benchmark experimental setup for sitting posture (withdrawn from <u>www.cfd-benchmarks.com</u>)

The incoming air is distributed evenly over the full cross sectional area in front of the manikin. This unidirectional flow field is evacuated thru two circular exhaust openings behind the thermal manikin. The manikin is seated at a distance of 0.7 m from the inlet in the centre of the wind tunnel – Fig. 2.

For the 2007 (Manikin Heat Loss) benchmark test, air velocities were measured with hot-sphere anemometers in 5 levels in front (L1 - Fig. 3) of and behind (L4) the manikin at the room's width center. Air temperatures for different locations as well as in some points of the surrounding surfaces were also measured and reported. The air was supplied at a mean velocity of 0.27 m/s from

a surrounding laboratory hall with a mean temperature of 20.4  $^{\circ}\text{C}$  [2].



Fig. 2. Illustration of the sitting posture benchmark tests.

Measurements were performed with the detailed manikin '*Comfortina*', Nille's type, operated in constant surface temperature mode, at 34°C, without clothing in order to get fast and accurate heat loss levels [2].

For the 2003 (mixing ventilated room) benchmark test, the Manikin was operated in constant heat flux with an integral value of 76 W for all CSP surfaces (excluding knees). Air velocities were measured in 8 levels in front of (L1 – Fig. 3) and behind (L3 and L4) the manikin, for 3 different z - coordinate positions: in the center of the room (z = 0.6 m); 0.295 m to the right (z = 0.305 m) and to the left (z = 0.895 m) of the center. The air was supplied at mean velocities of 0.2 m/s and 0.5 m/s with a mean temperature of 22°C.



Fig. 3. Location of measured velocity and temperature profiles.

All experimental data is available in the <u>www.cfd-</u> benchmarks.com web site.

## **3 Numerical Methods**

## 3.1 Digital Models

The room's CAD model was built in accordance with the Benchmark dimensions (Fig. 2). For the CSP it was used a previously generated CAD model [3] for a female thermal manikin (*Maria*), also Nille's type and very similar to the one (*Comfortina*) used in the experimental tests.

Based on a picture of the experimental setup, obtained from <u>www.cfd-benchmarks.com</u>, the CSP CAD model sitting position was made as similar as possible (ex. body inclination and hands position) with the position used in the experimental tests. Figure 4 is a superposed picture of the experimental *Comfortina's* position and the used CAD model position for grid generation.



Fig. 4. Superposed pictures of Real and CAD manikin models.

From the CSP CAD model, a detailed, mostly structured mesh was generated for the respective surface parts. Figure 5 illustrates one of the most geometrical (CAD) complex parts and the resultant surface mesh.



Fig. 5. Head's CAD model and respective surface mesh.

The generation of the room's (air) hybrid volume mesh (Fig. 6), was then based on this surface mesh and on the room's CAD model (Fig.2).

### 3.2 CFD Simulations

A commercial CFD code (Ansys CFX) was used for the CFD simulations. All simulations were run until the residual convergence value (RMS) [7] for all quantities was below 1x10<sup>-5</sup>. The exception was the case where the k -  $\epsilon$  turbulence model was used once, for the flow near the manikin's hands, only values of the order of  $1x10^{-4}$  were achieved.



Fig. 6. Illustration of the 1.4 Million elements mesh (only some surface mesh parts are visible).

Both physical approximation errors and spatial discritization errors were considered for the study of the CFD results accuracy. In this context, the main following aspects were taken into account in the computed CFD simulations:

- The accurate geometrical representation of the Manikins geometry by the respective surface mesh;
- The resultant, grid dependent, y+ values near the CSP surface;
- The CFD results grid dependency, i.e., the number of mesh elements influence on the numeric results by testing three different meshes: a 1.1 Million elements, a 1.4 Million elements and a 1.8 Million elements;
- The use of inlet boundary uniform, as established in the benchmarks boundary conditions, and non-uniform profiles for velocity, turbulence intensity and temperature;
- The influence of the turbulence model by using two different *Two Equation Turbulence Models*, the widely used k - ε and the Shear Stress Transport (SST) [5];

Table 1 summarizes the set of simulations computed to estimate the respective CFD results accuracy for the referred different parameters, from which the influence in the predicted results was analyzed.

#### 3.2.1 Geometry and y+ values

The CSP digital model was represented by a total manikin's surface mesh of 95062 elements, leading to a

relation between the all manikin's CAD surface area and the respective surface mesh area of  $1.545 \text{ m}^2 / 1.543 \text{ m}^2$ = 1.001 - see example of a complex human part in figure 5. Taking into account this value and the fact that this number of elements is greater than the ones used by other authors [4], the CSP surface mesh was maintained constant for all simulations and considered to give a very good representation of the real manikin's surface.

Simul.	Grid El.	Turb. Model	Inlet Bound. Cond.	Objective (Study)
CFD1	1.1 M	SST	Profile1 2003	CFD Results Grid Sensitivity
CFD2	1.4 M	SST	Profile1 2003	
CFD3	1.8 M	SST	Profile1 2003	
CFD4	1.4 M	SST	Const.2003 0.2 m/s	Inlet Variables Profile influence
CFD5	1.4 M	SST	Const.2007 0.27 m/s	Inlet Variables Profile influence
CFD6	1.4 M	SST	Profile 2007	Comparison with EXP. data
CFD7	1.4 M	<b>k-</b> ε	Profile 2007	Turb. Model influence
CFD2	1.4 M	SST	Profile1 2003	Comparison with EXP. data
CFD8	1.4 M	SST	Profile2 2003	Comparison with EXP. data

Table 1. Summary of set of CFD simulations performed.

Related with the  $y^+$  values, reference [6] presents a generic *Heat Transfer Validation Test* that studies grid sensitivity for a series of different parameters and shows a very good agreement between experimental and numerical results for low values of  $y^+$  (less than 3) when *automatic near wall treatment* is configured as it was in the present work. In this sense, a grid refinement near the CSP model that leaded to maximum  $y^+$  values of the order of 5 (Fig. 7) was used for all CFD simulations.



Fig. 7. Example of y<sup>+</sup> values for the CSP surface (CFD4).

#### 3.2.2 Grid Sensitivity

Based on the referenced CSP surface mesh and on the room's CAD model, three successively refined grids, respectively 1.1 Million (CFD1), 1.4 Million (CFD2) and 1.8 Million elements (CFD3), were created to allow for a grid sensitivity study.

The most significant variable differences between the respective CFD results was encountered for velocity profiles at L4 (z = 0.6 m - center of the room) – Fig. 3, as shown in Fig. 8.



Fig. 8. Velocity profiles at L4 (z = 0.6 m).

Significant differences between CFD1 (1.1 M) and CFD2 (1.4 M) were estimated for points of higher velocity values, near the outlets flow, while the refinement for 1.8 Million (CFD3) elements leaded to no significant difference when compared to CFD2.

For all considered points (L1, L3 and L4 at the respective z values), the mean relative difference between CFD1 and CFD2 results was of 4.33%, with a standard deviation of 8.52%, while between CFD2 and CFD3 was of only 0.84%, with a standard deviation of 2.05%.

When considering the differences between CFD predictions and experimental data, **CFD1** conducted to a mean relative difference of 15.11%, with a standard deviation of 20.27%, **CFD2** reduced the mean relative difference to 13.59%, only less 1.5%, but with a standard deviation reduction to 13.01%. The mean relative velocity and standard deviation differences between **CFD3** and the experimental data of, respectively, 14.08% and 13.51% are very close to the ones for **CFD2**.

From the above results, **CFD2** predictions were concluded to be grid independent and the subsequent CFD studies, like the influence of the used turbulence model, were made with the 1.4 Million elements mesh (see Table 1).

#### 3.2.3 Influence of Boundary (Inlet) Variables Profiles

Although the inlet (front wall) variables values established in the Benchmarks [1, 2] are constant (uniform profiles), measured values at **L1** (Fig. 3), only 19 cm away from the inlet, show that actually they have non-uniform profiles.

In this sense, comparisons were made between CFD simulations where uniform - CFD4 and CFD5 - and nonuniform - CFD2 and CFD6 – profiles were used at the inlet boundary. The most significant differences between the predicted results and the respective measured values was found for temperature profiles, illustrated in figure 9, between CFD5 and CFD6, this last with inlet non-uniform profile values equal to L1.



Fig. 9. Temperature profiles at L2 and L4 (z = 0.6 m).

As it was expected, CFD predictions confirm a better agreement with experimental data when the inlet temperature profiles were considered. The mean relative difference between predicted and experimental values for the inlet constant values (**CFD5**) was of 2.66%, with a standard deviation of 2.23%, while, when considering the inlet variables profile (**CFD6**), the mean difference was reduced to 1.73% with a, coincidently, standard deviation value of 2.23%.



Fig. 10. Velocity profiles at L4 (z = 0.6 m).

For air velocity, differences between the referenced simulations at L4, were not so significant (Fig.10).

The turbulent kinetic energy ( $k_0$ ) and the dissipation of turbulent kinetic energy, ( $\epsilon_0$ ) profiles at the inlet where calculated from the measured turbulence intensity at **L1** by:

$$k_0 = 1.5 \cdot (u_0 \cdot I_0)^2$$
 and  $\varepsilon_0 = C_{\mu}^{3/4} \cdot \frac{k_0^{3/2}}{l_0}$ 

with the turbulence constant  $C_{\mu} = 0.09$  [7] and the turbulent length scale  $l_0 = 0.5$  m [1]. This way, instead of an uniform profile of turbulence intensity at L1 (near the inlet), the non-uniform profile illustrated in figure 11.was obtained. The L4 turbulence intensity profiles for simulations CFD5 and CFD6 are also illustrated in figure 11.



Fig. 11. Turbulence Intensity at L1 and L4 (z = 0.6 m).

For the 2003 Benchmark conditions the respective velocity results were similar when comparing the consideration, or not, of uniform profiles.

#### 3.2.4 Influence of Turbulence Model

Two *Two Equation Turbulence Models* were tested in the respective **CFD2** and **CFD7** simulations, the standard k -  $\varepsilon$  and the SST model [5], for the 2007 Benchmark conditions once turbulence intensity experimental data is available for this case.

As shown in Figure 12, no significant differences were found between the respective velocity profiles.



Fig. 12. Velocity profiles at L4 (z = 0.6 m).

In what respects to turbulence intensity, the SST model leaded to predictions in better agreement with the experimental data than the k -  $\varepsilon$  (Fig. 13), particularly at 1.1 m height where this discrepancy between the predicted and the experimental data is very high.

For the CSP heat flux predictions, the focus of the 2007 Benchmark, the SST turbulence model predictions were found to be much closer the respective experimental data than the k -  $\varepsilon$  turbulence model results (Fig. 15). It should be noticed that for flow regions near some CSP complex parts, namely the hands, no satisfactory convergence values of RMS where achieved, which may justify the great differences due to the convective heat flux calculated for these parts.



Fig. 13. Turbulence Intensity at L4 (z = 0.6 m).

## 4 Results

An example of the predicted radiative heat flux, corresponding to the 2007 Benchmark (**CFD6**), where the focus is on the CSP surface heat flux, is shown in figure 14. This result illustrates how the detailed representation of the human parts geometry and position, specifically with respect to other parts, can influence the respective radiative heat flux value and, consequently, the total heat flux to be compared with the experimental data.

The predicted CFD results, in terms of the CSP surface heat flux, and their comparison with the respective experimental data for the 2007 Benchmark test are summarized in figure 15.



Fig. 14. CSP Radiative Heat Flux (CFD6).



Fig. 15. Summary of the 2007 Benchmark Results.

Figure 16 illustrates the scalar velocity contours for the **CFD8** simulation, modeled with non-uniform inlet profile of mean velocity 0.5 m/s, and its similarity with the one presented by S. Murakami in Roomvent 2004 [4].

## **5** Conclusions

Physical approximation errors, like the consideration of uniform boundary conditions that are actually nonuniform, the turbulence modeling and near wall treatment, as well as spatial discritization errors, like the number of grid elements, can lead to significant differences between CFD predictions and, consequently, between predicted and experimental results. A presented example is the need of using detailed CAD models that realistically represent the real human body, instead of simplified ones, if the modeled CSP surface temperatures and heat fluxes results are intended to be used in studies where the aim is the calculation of local values like local thermal comfort parameters. By another way, the comprehensive study of the mentioned parameters influence on CFD predictions, can lead these last to results that are in good agreement with experimental data. In the particular studied cases, the integral heat fluxes for the respective body parts shoed agreement with experimental data, verv good demonstrating that predictions of local comfort parameters based on these values for the different body parts could be calculated with great accuracy.



Fig. 16. Scalar air velocity contours in the vertical mid plane: on top **CFD8**; below – Murakami (Roomvent 2004).

## References

[1] Nielsen, P. V., Murakami, S., Kato, S., Topp, Cand J.-H., Y. Benchmark Tests for a Computer Simulated Person, October 2003 Aalborg University, Indoor Environmental Engineering, ISSN 1395-7953 R0307.

[2] Nilsson, H.O., Brohus, H., Nielsen, P. V., "Benchmark Test for a Computer Simulated Person – Manikin Heat Loss for Thermal Comfort Evaluation", Aalborg University Denmark & Gavle University Sweden, 2007

[3] N. A. G. Martinho, M. C. G. Silva, J. A. E. Ramos, "Evaluation of Thermal Comfort in a Vehicle Cabin", Proc. Instn Mech. Engrs Vol 218 Part D: J. Automobile Engineering (2004), pp 159-165.

[4] Communications of the Workshop on Virtual Manikins, ROOMVENT 2004

[5] Menter, F.R., "Two-equation eddy-viscosity turbulence models for engineering applications", AIAA-Journal., 32(8), pp. 1598 - 1605, 1994

[6] Menter, F.R., "Heat transfer predictions using twoequation turbulence models", Ansys CFX Validation Report, Ansys Europe Ltd, 2004

[7] Ansys CFX Manuals, Ansys Inc, 2007