NUMERICAL SIMULATION OF TURBULENT HEAT TRANSFER IN INDUSTRIAL DRYING PROCESSES

Q. Ye*, K. Pulli* and A. Scheibe*
J. Domnick** and D. Gruseck**

* Institut für Industrielle Fertigung und Fabrikbetrieb, Universität Stuttgart
Nobelstr. 12, 70569 Stuttgart, Germany
** Hochschule Esslingen, Kanalstr. 33, 73728 Esslingen, Germany

ABSTRACT

Aiming to model drying processes in automotive industry, numerical investigations of turbulent heat transfer in a laboratory dryer and an industrial dryer have been carried out using CFD. Flat sheets, a 3-D driver's cab model, as well as more complicated geometries of substrates, for instance, a real car body were used in the simulations. The suitable turbulence model and appropriate grid resolutions were applied for the present industrial computations. The calculated heating-up behaviour on substrates was compared with experimental results. The distribution of local temperature gradients obtained in CFD calculations can be used to develop models for predicting paint film defects in drying processes.

1. INTRODUTION

In order to meet the increasing demands on the reproducibility and reliability of drying processes in car industry, a joint research project funded by the German BMBF has been launched in cooperation with different research institutes, four car manufacturers, four plant and paint manufacturers and two software houses. The aim of this project is to develop an experimentally verified simulation tool to predict the convective unsteady drying of paint films on complex, three-dimensional work pieces, e.g., car bodies.

Within this project different stages of experimental and numerical studies have been identified. In general, there are two parts of the investigation. One is the modelling of unsteady turbulent heat transfer on complex 3D objects; another is the modelling of the evaporating process of water born base coats on objects. The current paper only deals with the investigation on the prediction of turbulent heat transfer without paint film. It was decided to apply CFD (Computational Fluid Dynamics) based methods to predict the drying process.

In our previous stage of the research [1-2], assessments of available turbulence models used in CFD codes for turbulent heat transfer were carried out by means of suitable data sources, for instance, DNS (Direct Numerical Simulation) data, single jet impingement, as well as experimental data obtained in a laboratory dryer, in order to find out models with low sensitivity of heat transfer prediction to the grid spacing for flow calculations in drying processes of car industry. In this stage of the research the selected turbulence model and the suitable mesh resolution in the vicinity of the substrate were applied to the unsteady simulation of convective heat transfer of realistic turbulent flow in a laboratory dryer (Hygrex dryer) and Eisenmann dryer for car industry. A commercial CFD code (FLUENT 6.3) was used in the numerical studies. A 3D model of a driver's cab and a more complicated geometry of substrate, a real car body, were used in the simulations. The calculated heating-up behaviours on the substrates were compared with experimental results.

2. COMPUTATIONAL METHOD

The commercial state-of-the-art CFD code FLUENT based on the finite-volume method was applied in the present numerical investigations. Unsteady Reynolds-averaged Navier-Stokes equations with a turbulence model were solved. It is well known that predictions for near-wall sensitive parameters such as skin friction and Nusselt number are strongly influenced by the chosen turbulence model and the corresponding wall treatment [3-5]. Assessments of available models used in FLUENT6.3 for turbulent heat transfer were carried out in our previous research [1-2] by using simple turbulent flows, e.g., channel flow and single jet impingement. It was found that k-ε models with standard wall function and the v2f low-Reynolds-number model are quite sensitive to the near wall mesh spacing, although the former predicts quite good Nu-numbers for a coarse mesh (y+ > 30) and the latter can give the best Nu-number prediction for jet impingement if a very fine wall grid resolution (y+ < 5) is used. Against other models the sst-kω model can predict the heat transfer with reasonable accuracy but also can provide certain mesh insensitivity. Against other models the sst-kω model can predict the heat transfer with reasonable accuracy but also provide a certain mesh insensitivity, which is especially important for industrial applications. The sst-kω model will be therefore used in present complicated turbulent flow.

For the calculation of heat transfer it was found that the IR radiation could not be neglected. Here, a discrete ordinates (DO) radiation model was used with an angular discretization of 2 $\times$ 2. The emission coefficient of the electro-coated plate (light grey) was experimentally determined to be 0.95. The calculating procedure for the heating-up behaviour of work pieces that delivers important information for analysis of drying processes was as follows: At first, joint velocity and temperature fields were simulated using the steady solver in the presence of the substrates until the normalized maximum residuals of the mass, momentum and energy equations were below $1 \cdot 10^{-4}$. Subsequently, an unsteady calculation was performed to capture the real velocity field in the chamber. Finally, substrates were patched to the initial temperature of 25 °C. The calculation of the heating-up behaviour was carried out by solving the complete set of conservation equations using varying time steps between 10^{-4} s at the beginning and 0.5 s at the end.

3. SIMULATION OF TURBULENT HEAT TRANSFER IN THE HYGREX DRYER

3.1 Calculation using a flat sheet as a simple substrate

A Hygrex drying system LBT2500 produced by Hellmann-Hygrex GmbH which works based on the recirculation of hot air was used in the investigation. The dryer has a 0.72 $\times$ 0.72 $\times$ 0.60 m³ drying chamber. Originally, air is delivered from 25 mm nozzles located both on the roof and on the left side of the wall. However, in the present study only the nozzles on the left side of the wall, as shown in Fig. 1, were operated. A thin cataphoretically coated (film thickness approximately 20 µm) plate (KTL plate) with a size of 500 $\times$ 200 mm² and a thickness of 1 mm was used as substrate plate. The plate was attached to a specific holder (Fig. 1), enabling to install an arbitrary angle with respect to the inlet flow in the dryer. The vertical position of the flat plate with nozzle-to-plate distance L = 5.5·D was used in this paper. Detailed experimental and numerical investigations on flow and temperature fields with a horizontal flat plate in the Hygrex dryer can be found in our previous study [1].

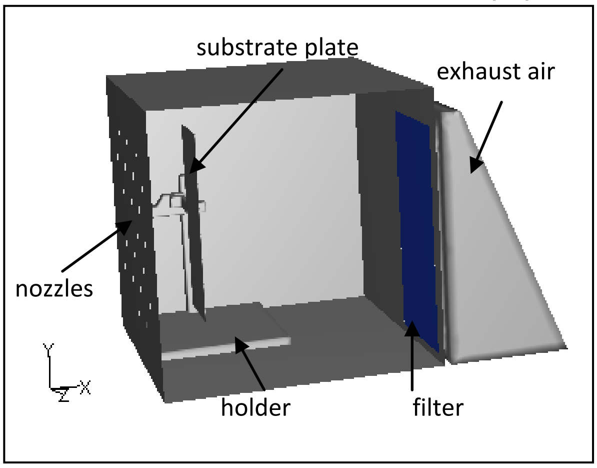

Figure 1 Hygrex dryer

A 3D computational domain with 1 million cell elements was created. Around the substrate plate several boundary layers with prismatic cells were constructed, providing a good resolution of the near wall flow field. Physically, a sufficiently fine grid close to the wall should be employed. As a compromise, an appropriately fine grid with 1 mm as the first interior node distance was used to reduce large computer storage and runtime requirements, especially for later applications using industrial dryers with complete car bodies.

As nozzle inlet, a mean velocity of 10 m/s and a temperature with a mean value of 85 °C were used. The turbulent intensity of 10 % in the nozzles was applied. The filter (see Fig. 1) at the chamber exit was calculated using a porous-jump model.

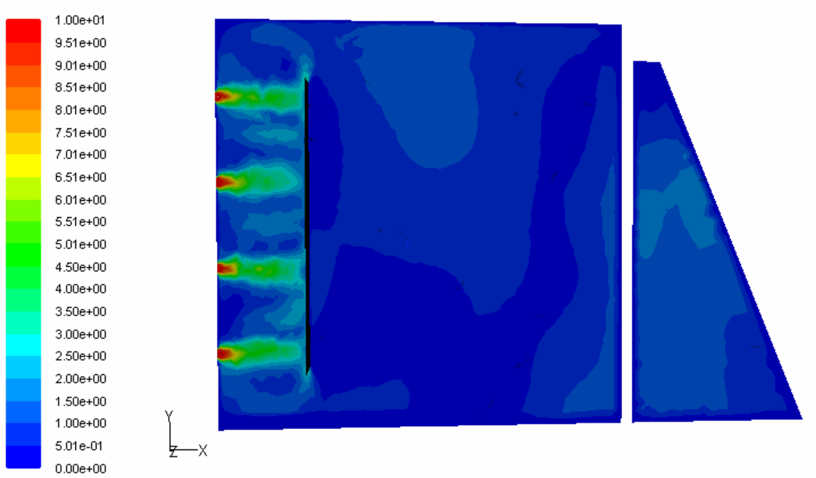

Figure 2 Velocity contours in a centre cross-section (z = 0)

Figure 2 shows the velocity contours in a centre cross-section. The flow between nozzles and the plate is characterized as multiple jet impingements. The velocity distribution and the non-dimensional wall distance y+ at wall-adjacent cells, as well as the temperature distribution on the plate are depicted in Fig. 3 (a-c). Circular velocity patterns with a stagnation point inside that are typical for jet impingement can be observed. The maximum temperature difference on the plate is about 10 K. Higher temperatures are located on the edge of the plate and near the stagnation points where the local heat transfer coefficient is higher. Detailed heating-up curves, for instance, at points 6 and 8 are shown in Fig. 4 and compared with experimental results that were obtained by means of thermocouples [1]. A quite good agreement between experiment and calculation was obtained.

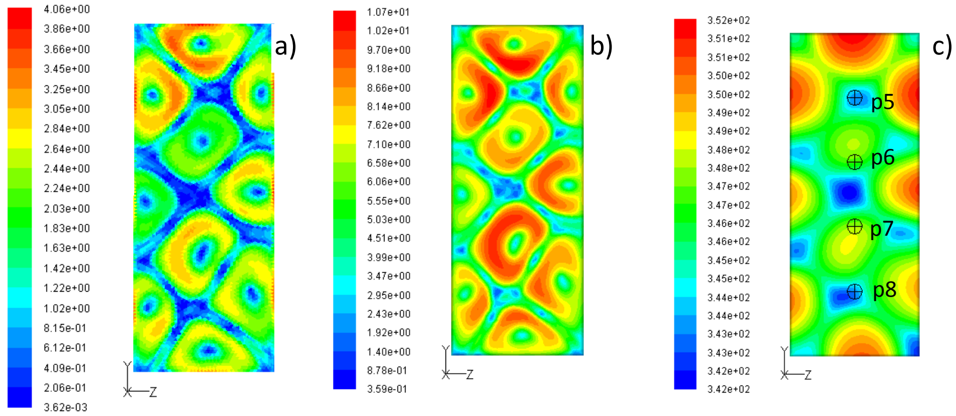

Figure 3 Contour distributions: a) Velocity contour (m/s) at wall-adjacent cells, b) non-dimensional wall distance y^+ on the plate, c) temperature distribution (K) on the plate at t = 96 s

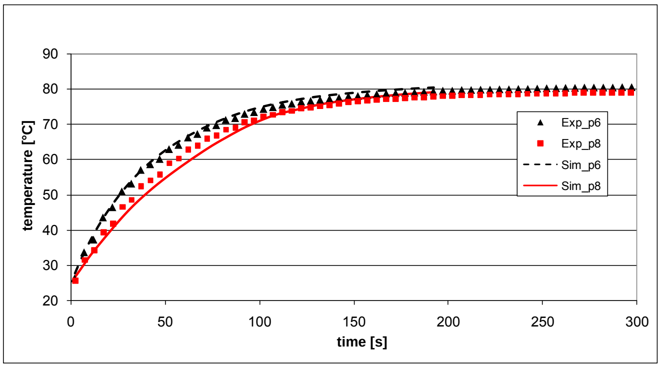

Figure 4 Comparison of heating-up curves on the KTL sheet

3.2 Calculation using a 3D model of a driver's cab

It has been mentioned above that for a given turbulence model the proper mesh resolution near the wall should be generated, in order to obtain the accurate prediction of the Nu number. For simple geometries, for instance, the flat plate used above, such mesh quality can be obtained easily by using mesh boundary layers on the object. However, for industrial applications it is quite difficult to achieve the desired grid that is suitable to the selected turbulence model in the whole wall region because of the complicated geometry of the objects and the complex flows.

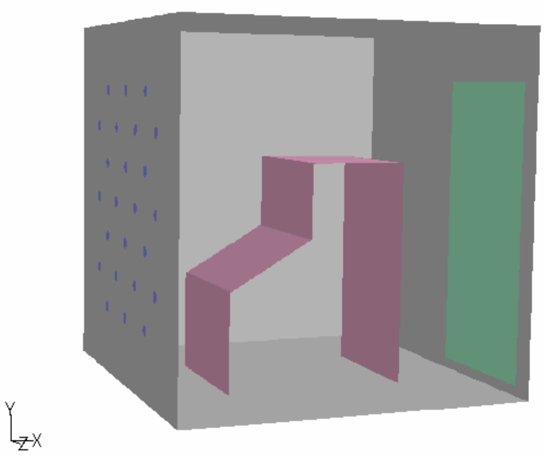

Figure 5 3D model of driver's cab

Further studies of model and mesh sensitivity to the prediction of heat transfer were carried out using a 3D model of a driver's cab.

The 3D model with a size of 0.373 × 0.3 × 0.45 m³, which was made by buckling a sheet into five surfaces, was put in the centre of the Hygrex dryer, as shown in Fig. 5. Two mesh cases, one with boundary layers of prismatic cells on both sides of the object (see Fig. 6(a)), the other with mesh boundary layers only on the outside of the object (Fig. 6(b)), were created around the 3D model. The first mesh layer was 1 mm away from the object surface. The same operating conditions as in the section above were used. The maximum y+ was 9 and was located in the wall surface close to the nozzle.

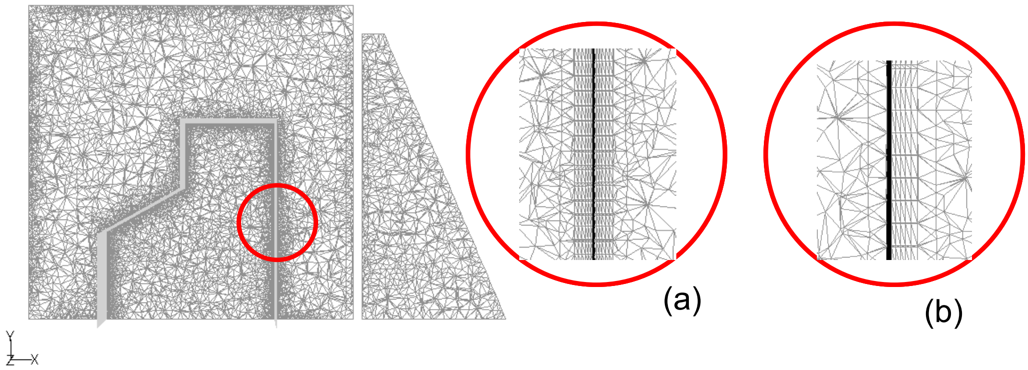

Figure 6 Grid in a cross section (z = 0) (a) Mesh boundary layers on both sides of the object (b) Mesh boundary layers on the outside of the object

The simulation results are shown in Figure 7. Obviously, the flow around the 3D model is more complicated than that in section 3.1. Inside the driver's cab the velocity is quite low. Higher temperatures are located on the surface that is vertically faced and close to the nozzles. A few measuring points for temperature were put on each surface, respectively. A detailed comparison between measured and simulated heating-up behaviour is depicted in Figure 8. Clearly, there is no significant difference between the heating-up curves with one or two side mesh boundary layers on the object for the present situation, where the flow outside the 3D model dominates the heat transfer process. Figure 8 shows also a good agreement between the simulation and the experiment.

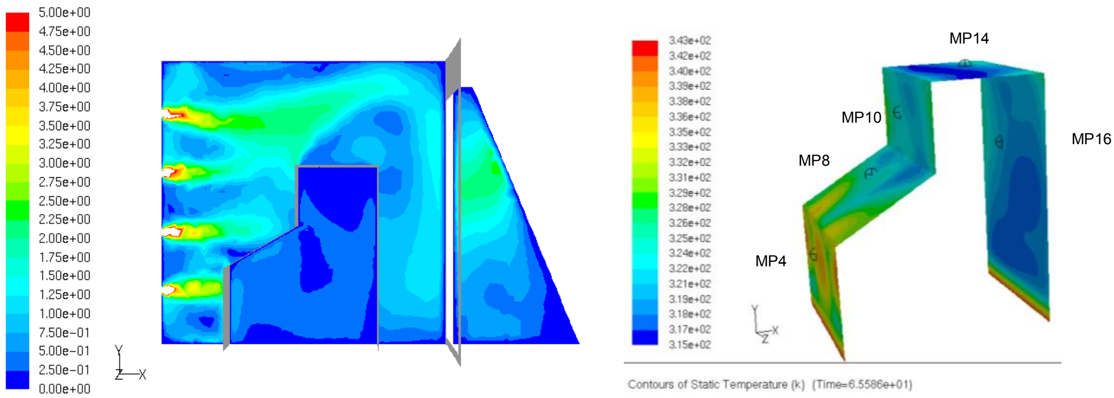

Figure 7: Simulation results in the Hygrex dryer (a) Velocity contours at z = 0 (b) Temperature contours on the surface of the 3D model

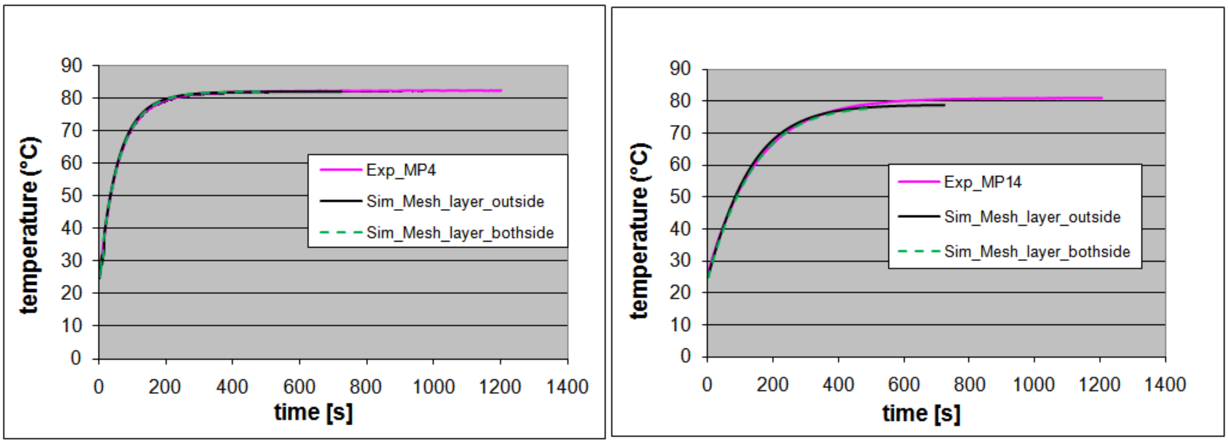

Figure 8 Comparison of heating-up curves on the surface of the 3D model

4. COMPUTATION OF FLOW AND TEMPERATURE FIELDS IN THE EISENMANN DRYER IN THE PRESENCE OF A CAR BODY

Numerical simulations of turbulent heat transfer in an industrial car dryer (Eisenmann dryer at the Fraunhofer Institute IPA) with a real car body (Golf 5) have been carried out.

4.1 Geometry model and grid generation

Figure 9 shows the draft of the Eisenmann dryer. The hot air with a temperature of 443 K delivered from the air inlets is distributed at first in different chambers and streams finally from numerous slots into the dryer chamber. In order to reduce large computer storage and runtime requirements, only the dryer chamber was used to create a computational domain with size $6 \times 5 \times 3.6$ m³. The boundary conditions at the velocity inlet in the slot channels were obtained from measurements. Since the inlet velocity in the slots is distributed asymmetrically on the two sides of the dryer chamber, a symmetrical model could not be used. Such an asymmetrical velocity distribution in the dryer chamber can be clearly observed from the simulation results in an empty chamber, as shown in Fig. 10.

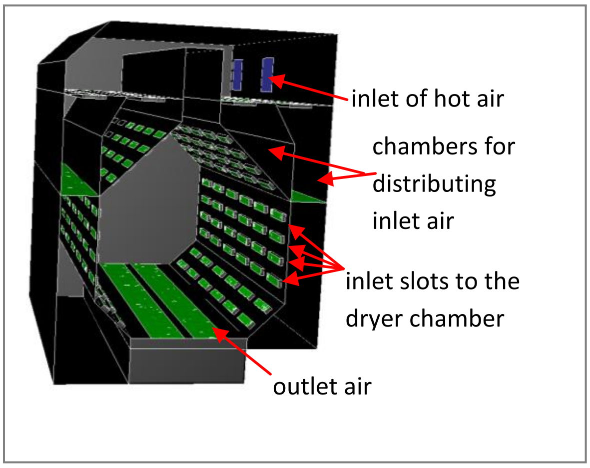

Figure 9 Eisenmann dryer

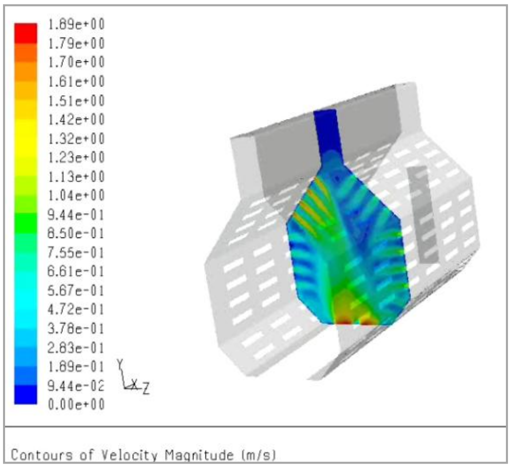

Figure 10 Velocity contours in a cross section in the empty chamber

A NASTRAN surface mesh of the car body which was the only available CAD data of Golf 5 for the present project was used. A car body volume mesh with tetrahedral cells was created by means of the ICEM-CFD program under the support of ANSYS-FLUENT and was then assembled with the volume mesh of the Eisenmann dryer, which resulted in a computational mesh with ca. 8 million tetrahedral cells (Figure 11). For some technical reasons mesh layers with prism cells on the outer skin of the car body were not created.

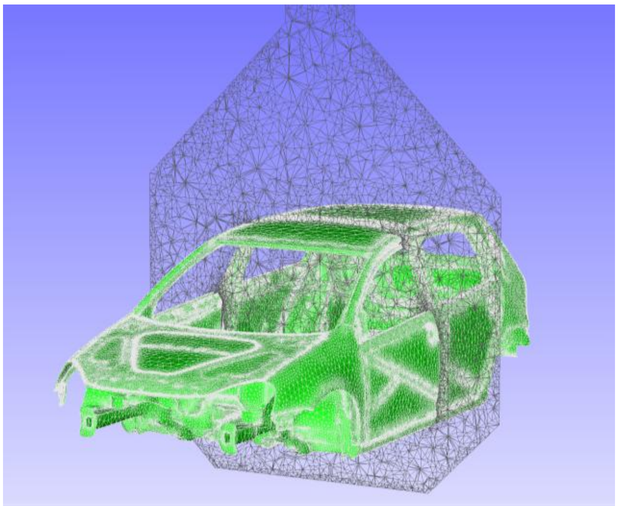

Figure 11 Mesh in a cross-section of the computational domain

4.2 Numerical method and simulation results

The shell conduction model in FLUENT was applied with corresponding shell thicknesses ranging from 0.7 to 2 mm. Altogether, there are 140 work piece regions with 10 classes of shell thicknesses in this car body model. The calculating procedure for the heating-up behaviour was similar to the case of the Hygrex dryer. After obtaining a steady solution for the flow and the temperature fields, the car body was patched to the initial temperature of 25 °C and an unsteady calculation with time step $\Delta t < 0.5$ s was carried out. The calculation was performed by using the NEC Linux Cluster at the High-Performance Computing Centre at the University of Stuttgart. A CPU time of 160 hours was needed for the unsteady simulation with a heating-up time of one minute by using 8 GB memory on a four CPU parallel processor.

The velocity vector field in a cross-section and temperature contours on the car body obtained from the unsteady simulation are shown in Figs. 12 and 13, respectively. Using the simulation results the local heat-up gradients of the car body can be studied. For a given time the temperature distribution on the outer shell of the car body is highly non-uniform because of the complicated shell structure and flow. The surface temperature on the fender, where the shell structure is relatively simple, is considerably higher than on other parts of the car body. In Fig. 14, the heating-up curve on the engine hood obtained from the simulation is compared with the experimental values. A good agreement was obtained. The numerical simulation yields local heat-up gradients on the car body that are useful information for studying the occurrence of paint film defects.

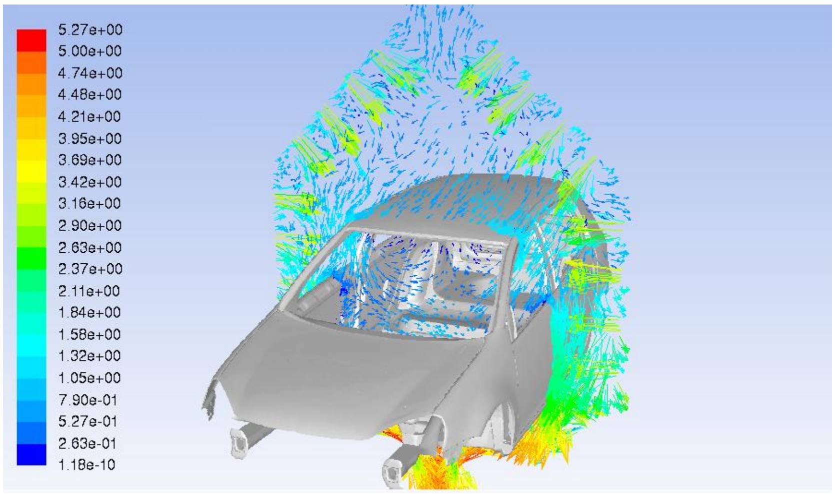

Figure 12 Velocity vectors (m/s) in a cross-section of the Eisenmann dryer

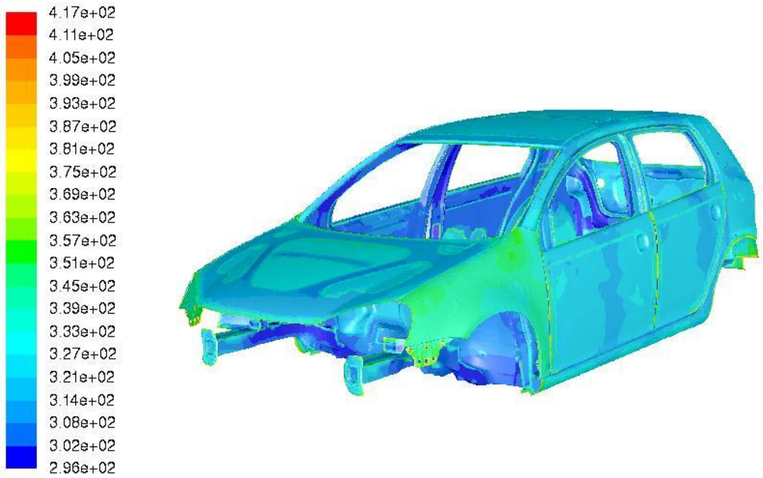

Figure 13 Temperature distribution (K) on car body surface at t = 20 s

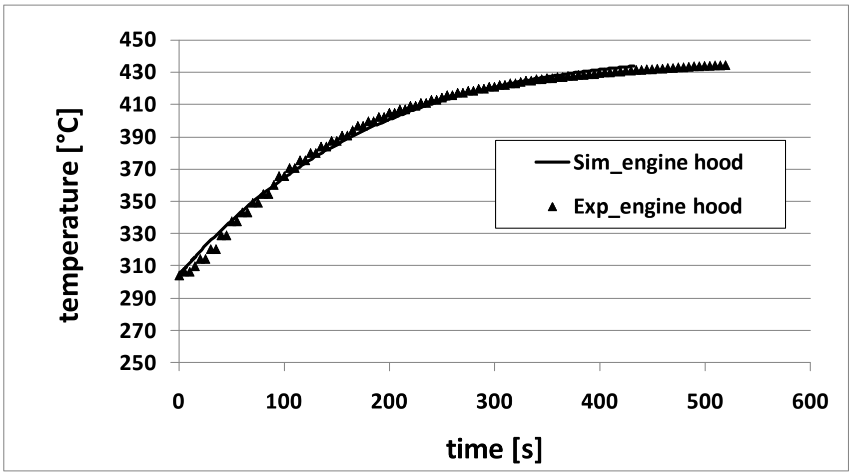

Figure 14 Comparison of heating-up curves on the car body

5. CONCLUSIONS

Aiming to model the drying process in paint shop dryers and ovens, numerical simulations of turbulent heat transfer have been carried out using FLUENT 6.3.26.

At first, calculations of turbulent heat transfer were carried out in a laboratory dryer (Hygrex dryer) with a flat plate and a 3D driver's cab model. The effect of the wall mesh resolution on the 3D model was studied. A good agreement between simulations and measurements was achieved using the sst-kω model and a reasonably fine grid spacing adjacent to the work piece. Numerical investigations were then performed using an industrial dryer (Eisenmann dryer) with a real car body, representing the typical situation of complex turbulent flow fields inside industrial dryers.

Simulation results contain important information, for instance, the distributions of local velocity, temperature, and especially of temperature gradients on work pieces, which are very useful for further investigations of the drying processes, including the effect of combined heat and mass transfer in an applied paint film and the prediction of paint film defects.

REFERENCES

[1] YE, Q., PULLI, K., SCHEIBE, A., J DOMNICK, and GRUSECK, D.: Numerical and experimental study of convective heat transfer with complicated turbulent flow in a laboratory dryer for turbulence model assessment. 6th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics. July 2008, Pretoria, South Africa.

[2] YE, Q., PULLI, K., SCHEIBE, A.: Prediction of turbulent heat transfer for industrial drying processes – Turbulence model assessment. NAFEMS World Congress 2009, June 16th-19th, Crete, Greece.

[3] BREDBERG, J., and DAVIDSON, L.: Low-Reynolds number turbulence models: An approach for reducing meshes sensitivity. Transactions of the ASME 126, pp. 14-21, 2004.

[4] POPOVAC, M., and HANJALIĆ, K.: Compound wall treatment for RANS computation of complex turbulent flows and heat transfer, Applied Scientific Research, 78(2), pp. 177-202, 2007.

[5] VIESER, W., ESCH, T. and MENTER, F. R.: Heat transfer predictions using advanced two-equation turbulence models. CFX Validation Report: CFX-VAL10/1002.

ACKNOWLEDGEMENTS

This research was financially supported by the German Ministry of Education and Research (BMBF). Part of the numerical simulations reported in this paper was performed at the High-Performance Computing Centre Stuttgart (HLRS).