NUMERICAL INVESTIGATION OF HEAT AND MASS TRANSFER IN A REFRIGERATED TRUCK COMPARTMENT

T. LAFAYE DE MICHEAUX[(a)], V. SARTRE[(a)*], A. STUMPF[(b)], J. BONJOUR[(a)]

[(a)] Université de Lyon, CNRS
INSA-Lyon, CETHIL, UMR5008, F-69621, Villeurbanne, France
Fax: +33(0)4 72 43 88 11, e-mail: valerie.sartre@insa-lyon.fr*
[(b)] Carrier Transicold Europe
4 rue Joseph Monnier, Rueil 2000 - Immeuble, 92500 Rueil-Malmaison, France

ABSTRACT

This work aims at presenting a numerical model to simulate the temperature and velocity fields in back compartment of a refrigerated semi-trailer. The cavity is ventilated by means of a ceiling evaporator. The air flow and temperature distributions in this ventilated enclosure are analysed from a CFD model. This model focuses on the jet development in the empty cavity. The influence of heat flux through the walls on the air temperature fields is observed. A parametric study regarding the blowing velocity is also presented. It shows that changing the velocity does not noticeably modify the air flow structure inside the cavity. However when the length of the back compartment is increased, the flow structure changes. Finally the insulated wall is modeled and a new boundary condition is set with fixed external ambient temperature and convective heat transfer coefficient. Results show that the influence of external conditions on the temperature field is strongly reduced.

1. INTRODUCTION

The reduction of energy consumption for perishable foodstuffs transport is related to the optimization of the overall device: the refrigerated unit and the insulated truck body. Airflow and temperature fields in the truck body are critical parameters regarding long-distance food transport mainly when the body is full.

1.1. Heat transfer in refrigerated cavities

One of the main studied objects related to heat transfer in refrigerated cavities is household refrigerators. With roll bond evaporators placed behind the back wall, heat transfer phenomena are based on natural convection and radiation.

Laguerre *et al.* (2006) studied experimentally and numerically the temperature and velocity fields in a household refrigerator (0.5x0.5x1 m^3). The boundary condition of the numerical model was defined using an overall heat transfer coefficient between the inside wall and the external air. This coefficient was deduced by heating the inside air at a constant temperature to reach steady state. In a following study this boundary condition was changed to a convective external heat transfer (Laguerre *et al.*, 2009). Both of these studies assumed a laminar air flow in the cavity and use the Boussinesq hypothesis to take into account the buoyancy forces due to density variations. A good accuracy was observed with experimental and numerical results when the radiative heat transfer is taken into account.

Gupta *et al.* (2011) studied numerically a ventilated domestic refrigerator composed of two compartments for fresh and frozen food. An overall heat transfer coefficient was defined for the lateral wall boundary condition, taking into account conduction through the insulation material and the external convection. The calculation of the Richardson number allowed the authors to conclude that the buoyancy forces are of similar order of magnitude as the inertia forces. The Reynolds number indicated that the flow was laminar. Commercial CFD software was used to solve the mass, energy and momentum equations for steady and transient states. The numerical results were compared with temperature measurements, the discrepancies were within 7%.

1.2. Heat transfer in cold store and refrigerated truck

Heat transfer in refrigerated truck or cold store is based on mixed or forced convection. After having been sucked and cooled down through an evaporator, the air is blown inside the cavity. These configurations are different from those in buildings because the inlet and outlet sections are located on the same side of the cavity, creating a strong recirculation zone.

Kolodziejczyk and Butrymowicz (2011) studied numerically the heat and mass transfer in a cold store loaded with pallets containing carrots. A uniform heat flux of 5 W/m² was assumed through the walls. The turbulent model used was RNG k-ε. The pallets were modeled as a porous material and the natural convection was not taken into account because airflow velocities were important and temperature gradient small.

Mirade and Picgirard (2001) compared experimentally and numerically the airflow inside different industrials beef carcass chillers. Their 2D computational model for a loaded cold store was in good agreement with the experimental study. A parametric study was performed using various air flow rates and blowing angles in order to obtain the most favorable configuration.

Moureh *et al.* (2002) studied experimentally and numerically the development of the air jet in a 13.3 m long truck body in an isothermal configuration. By means of a scale model, two main configurations were considered: the blowing section was either located just under the ceiling or 20 cm below the ceiling (Moureh *et al.* 2005). In both cases, they concluded that after having been blown in the cavity, the wall jet directly attaches to the ceiling and separates 10 m away (Figure 1). The attachment phenomenon is due to the Coanda effect which results from the attraction of a jet by a convex wall. The separation from the ceiling is due to an adverse pressure gradient created by the opposite wall of the cavity. The separation creates a recirculation cell with smaller velocity magnitude. Their numerical study concluded that the Reynolds Stress Model is able to accurately predict the jet separation, and not the k-ε model.

The purpose of this numerical work is to study airflow and heat transfer in a compartment of a refrigerated truck body. This study may help to understand the influence of different parameters on the temperature field, such as heat flux through the walls, blowing velocity of the air jet, length of the truck compartment and insulation of the truck body.

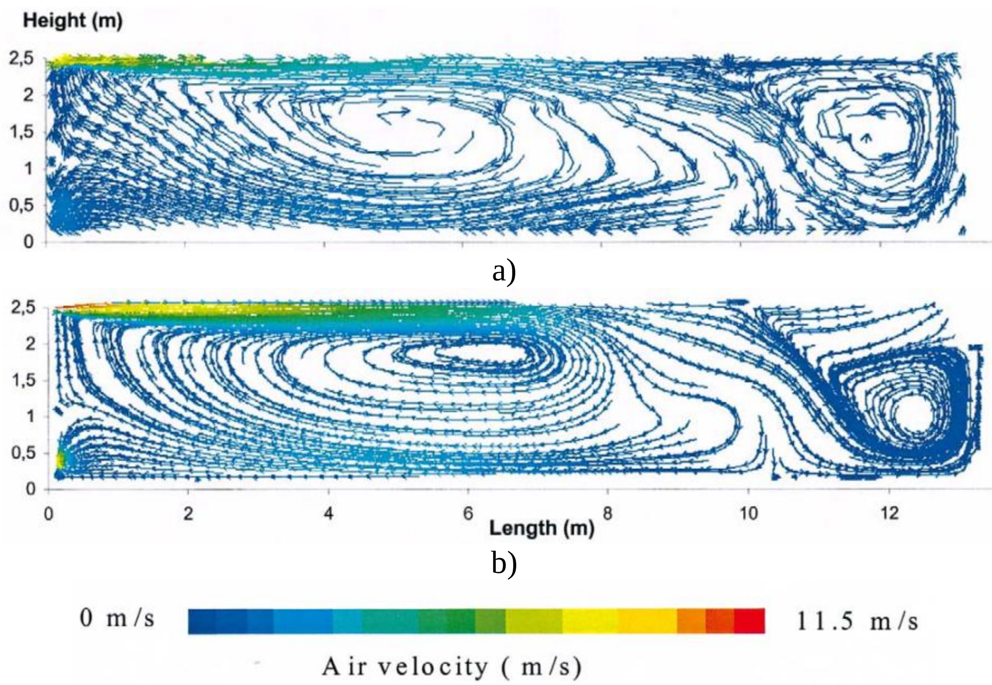


Figure 1: Velocity field in the refrigerated truck (Moureh *et al.* 2002)
a) Experimental results, b) Numerical results (RSM turbulence model)

2. NUMERICAL MODEL

2.1. Geometry, mesh, and numerical resolution

The present study focuses on the back compartment of a multi-temperature refrigerated truck. The simulation domain has been modified to simplify the geometry (Figure 2). A fictitious volume located at the evaporator air return is added in order to reduce the impact of the boundary condition on the flow pattern in the cavity (Figure 3).

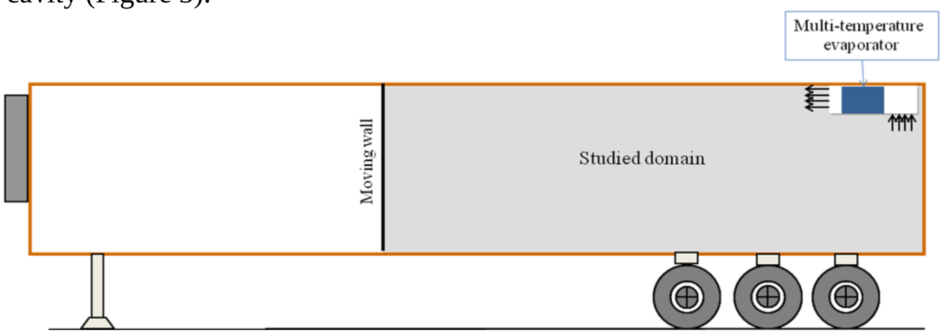


Figure 2: Studied configuration

Figure 3: Simulation domain

The simulations are performed in 2D and steady state, using a finite volume commercial CFD software (STAR CCM+ v6.04). The time-averaged Navier-Stokes differential equations, high-Reynolds numbers and incompressible flows for mass, momentum and energy conservations are solved. The turbulence model used is the RSM. Structured cartesian meshes are used close to the wall to predict the velocity with the relevant law of the wall. The *high y+ wall treatment* model available in the CFD software is used. Far from the wall, polyhedral meshes are used. Finer meshes are used near the inlet, and in the outlet ducts. The second-order upwind numerical scheme is used. The independency of the grid is verified for 6900 meshes. The convergence is considered to be reached when all the normalized residuals are below 10^{-4}.

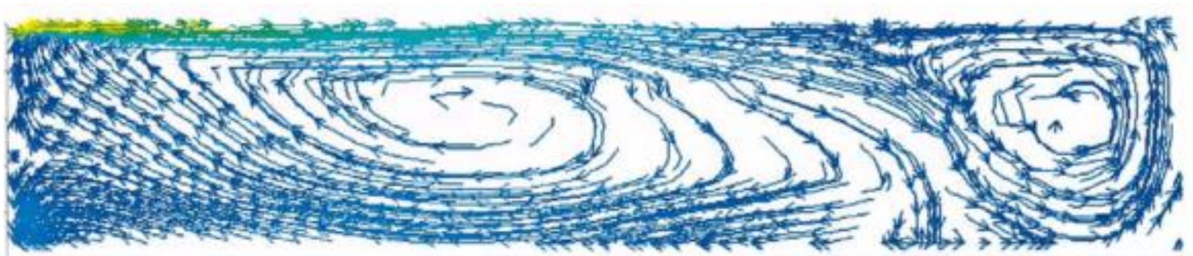
2.2. Boundary conditions, reference values

The boundary inlet condition is defined using a reference velocity of 2 m/s, corresponding to a typical blowing velocity in a ceiling evaporator. Turbulence at the inlet is defined with the turbulent kinetic energy (k_0) and with the energy dissipation rate (ε_0): $k_0=3/2(U^*I)^2$ where I=20% represents the turbulent intensity; $\varepsilon_0=C_\mu^{0.75}k_0^{1.5}/L$, where L representing the turbulent length scale is equal to the hydraulic diameter (0.2 m) multiplied by a coefficient depending on the geometry and on the CFD software. The blowing temperature is fixed to 273 K. A pressure value of 1013 hPa is fixed at the outlet. A uniform heat flux was imposed at all the walls with a reference value of 10 W/m². This value represents a homogeneous 20 K temperature difference through a truck body with an overall heat transfer coefficient (K-value) of 0.5 W/m²K experimentally measured by Clavier *et al.* (2011). The natural convection is not taken into account.

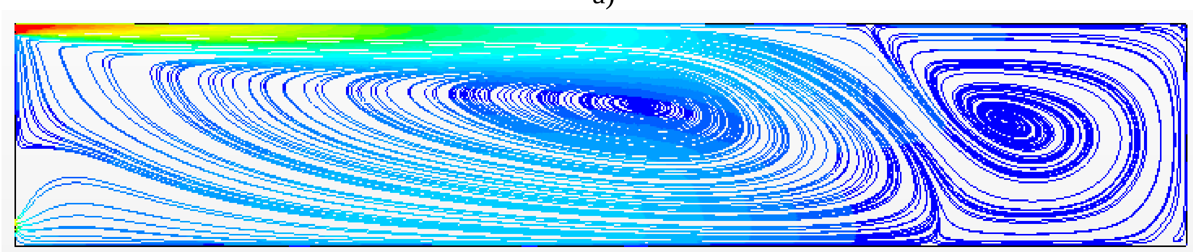
3. RESULTS AND DISCUSSION

3.1. Model validation

The methodology described in 2.1 was applied and validated with the isothermal experimental results from Moureh *et al.* (2002, 2005). The agreement between both results is acceptable (Figure 4). The turbulent length scale coefficient was obtained owing to this validation (0.12).



a)



b)

Figure 4: Velocity stream lines in the refrigerated truck,

a) Experimental results (Moureh *et al*. 2002), b) Numerical results, present study

3.2. Effect of the heat flux boundary condition

The influence of the truck body insulation is observed by means of a parametric study regarding the heat flux through the walls (Figure 5). When the heat flux is increased, the temperature inside the cavity also increases, especially close to the ceiling and floor. In the jet development zone close to the ceiling, a high heat flux creates a relatively cold zone (Figure 6 b). However a good temperature homogeneity is maintained in the middle of the cavity (Figure 6 a).

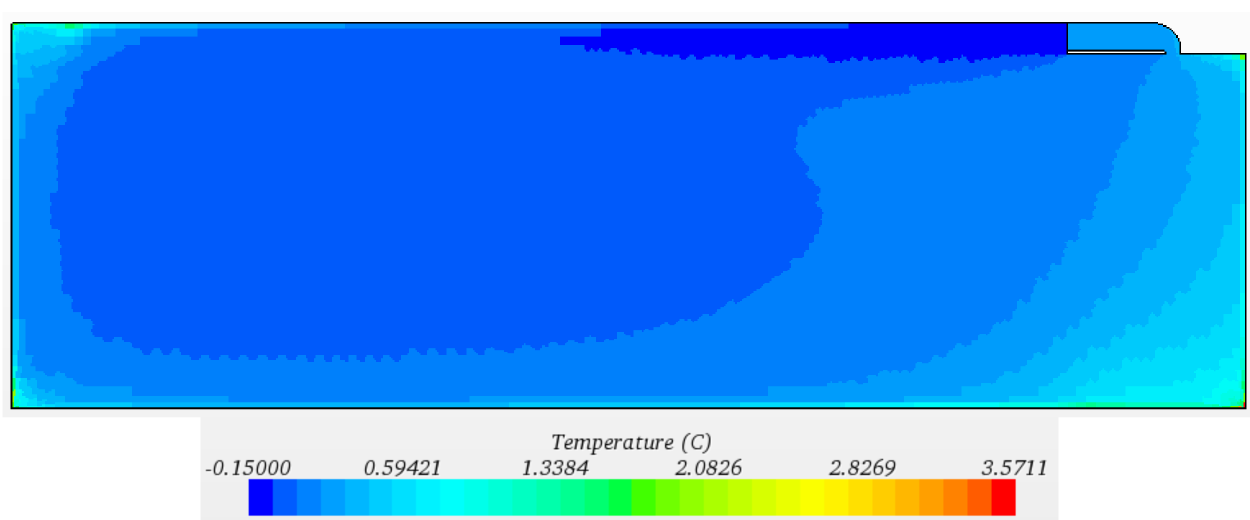


Figure 5: Temperature field of the compartment (heat flux: 10 W/m²)

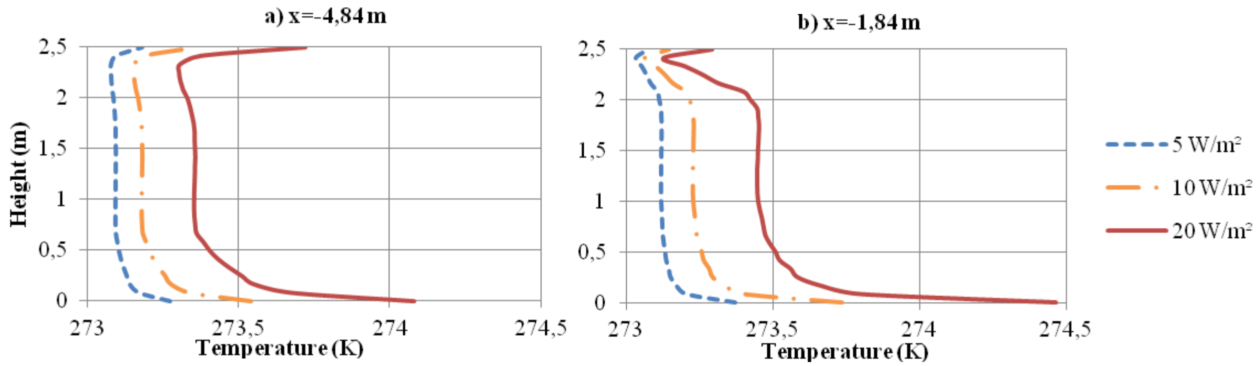


Figure 6: Influence of heat flux on vertical temperature profile.
a) 4.84 m and b) 1.84 m away from the blowing inlet.

3.3 The effect of blowing velocity

To observe the influence of the blowing velocity on the temperature and velocity profiles, the geometry has been meshed again in order to satisfy the law of the wall. The total height of the cartesian layers is kept constant in order to maintain the same mesh structure. Only the number and the growth rate (ranging from 1 to 1.2) of the layer are modified. By calculating the Reynolds number, it has been verified that the flow is turbulent for all cases. The dimensionless longitudinal velocity is depicted in Figure 7: When the blowing velocity is modified, no recirculation cell is observed, which would be noticed by a negative longitudinal velocity. The structure of the airflow does not depend on the blowing velocity.

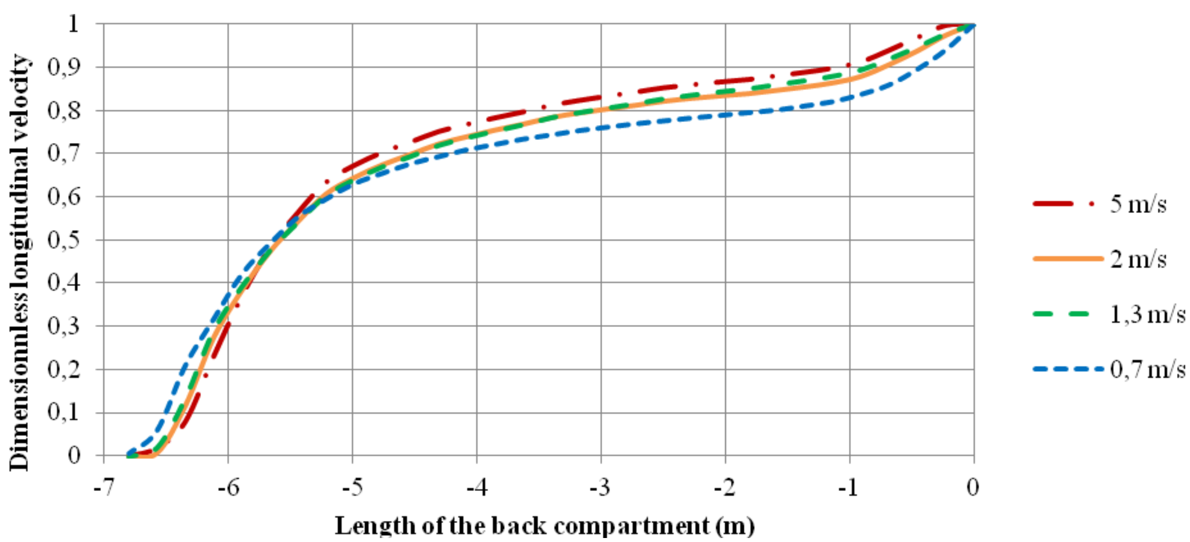


Figure 7: Longitudinal velocity on the jet axis along the truck enclosure (y=2.4 m).

The influence of the blowing velocity on the temperature profile is observed on Figure 8. By reducing the blowing velocity, the temperature profile becomes less homogenous close to the walls and in the middle of the cavity below the blowing jet. When the jet velocity is equal to 0.7 m/s, the first cartesian layer close to the wall is 9.6 cm thick which gives a less accurate profile in this regions.

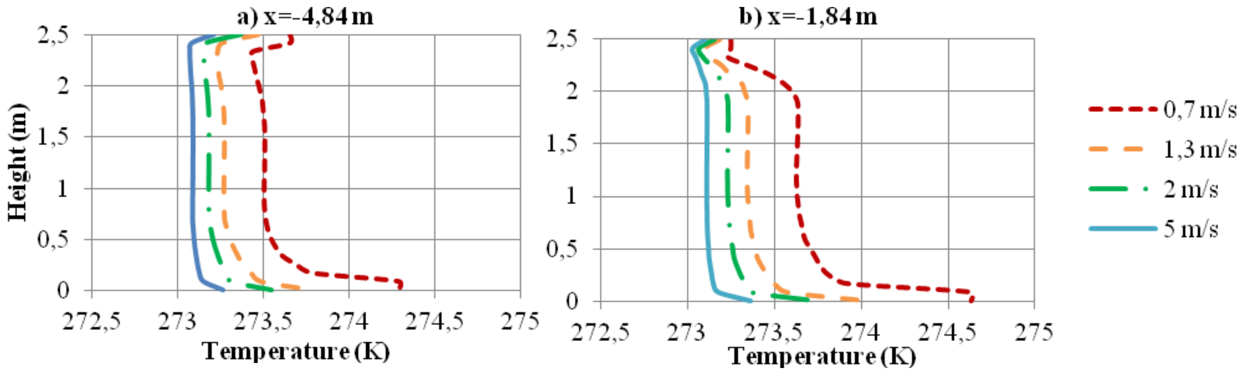


Figure 8: Influence of blowing velocity on vertical temperature profile.
a) 4.84 m and b) 1.84 m away from the blowing inlet.

3.4. Effect of the back compartment length

The inner wall in multi-temperature trucks is usually movable, which allows the transport company to increase or decrease the size of the back compartment. When increasing the length on the compartment, the flow structure is strongly modified (Figure 9). For a 13.3 m long cavity the jet separates from the ceiling 7.1 m away from the blowing section which creates an important recirculation cell. For 10 and 8 m cavities the jet separates from the ceiling at 7.8 m, resp. 6.45 m, but the induced recirculation cell is limited to the top corner.

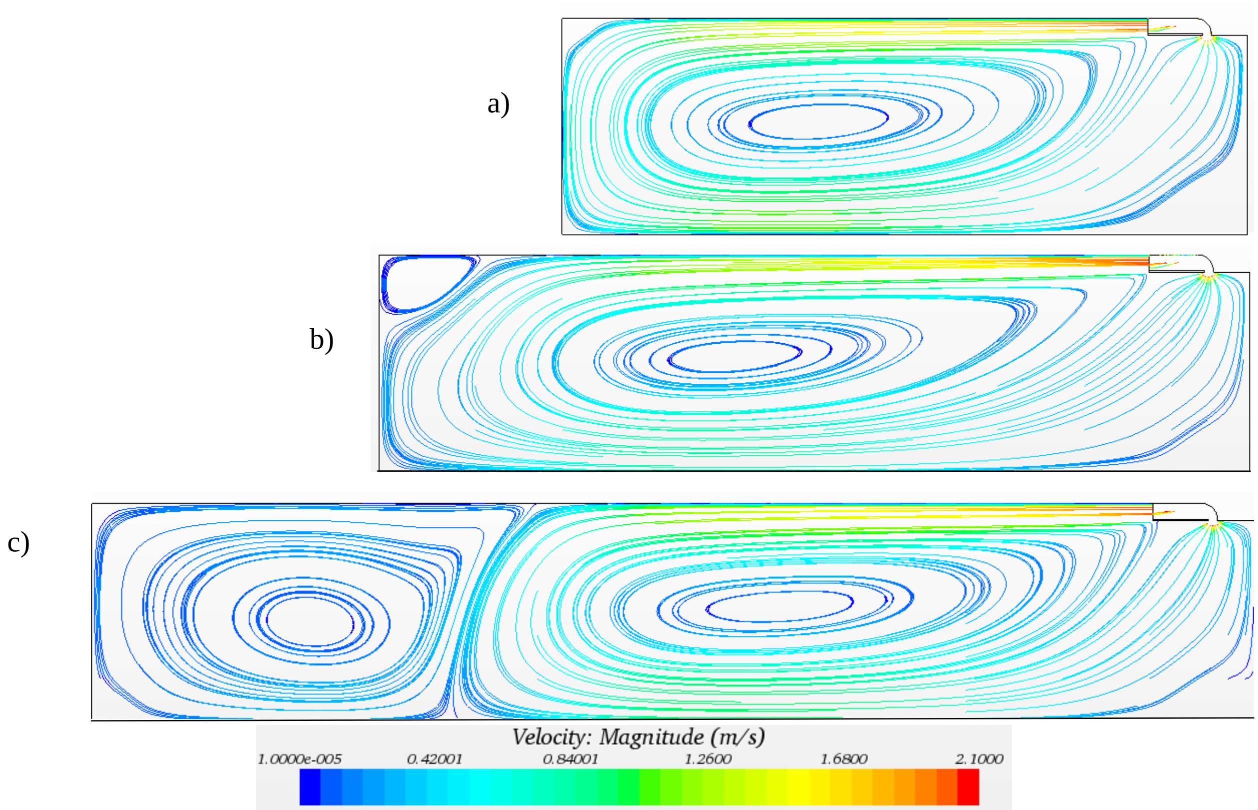


Figure 9: Velocity streamlines for three back compartment lengths: a) 8 m, b) 10 m, c) 13.3 m.

The airflow pattern observed with the 13.3 m long cavity has the same structure as that measured by Moureh *et al.* (2002 and 2005). However the separation point at the ceiling is not at the same location in both cases. According to Yu and Hoff (1999), this discrepancy cannot be explained by the blowing velocity which is 6 times higher in Moureh studies, because the flow structure is independent of the blowing velocity for turbulent flows. This difference rather may be due to the geometry: the position of the blowing and air return sections are not the same.

By comparing the separation point location between the 10 m and the 13.3 m long compartments, it can be observed that the jet is slightly more developed in the first case. This phenomenon is due to the Coanda effect occurring at the vertical wall, which attracts the air jet. Results also show that 10 m may be the maximal length allowing relevant air-renewal inside the truck body, because air velocity in the recirculation cell are 10 times smaller than in the main flow structure.

3.5. Insertion of the wall condition in the numerical model

The boundary conditions have been changed to observe the influence of the insulated wall on the temperature field inside the cavity. The walls have been meshed using polyhedrons; the 8.5 cm thick wall is divided at least into 4 cells. The thermal conductivity is fixed to 0.03 W/mK which is close to the conductivity of polyurethane foam. The compartment length is 8 m. The external convective heat transfer coefficient and the external ambient temperature are fixed.
The steady state temperature field is presented on Figure 10 for an ambient temperature of 20°C and a convective coefficient of 10 W/m²K. Results show an almost completely homogeneous temperature field inside the cavity. When changing the ambient external temperature to 40°C or the convective coefficient to 50 W/m²K, the temperature field does not change and is still homogeneous.
These results are really different from those presented in section 4.1. However by increasing the wall thermal conductivity, the air temperature field is similar (Figure 11). The thermal conductivity of the wall is therefore the main parameter. The ambient conditions, hence temperature and convective coefficient, do not have a high impact on the temperature field.

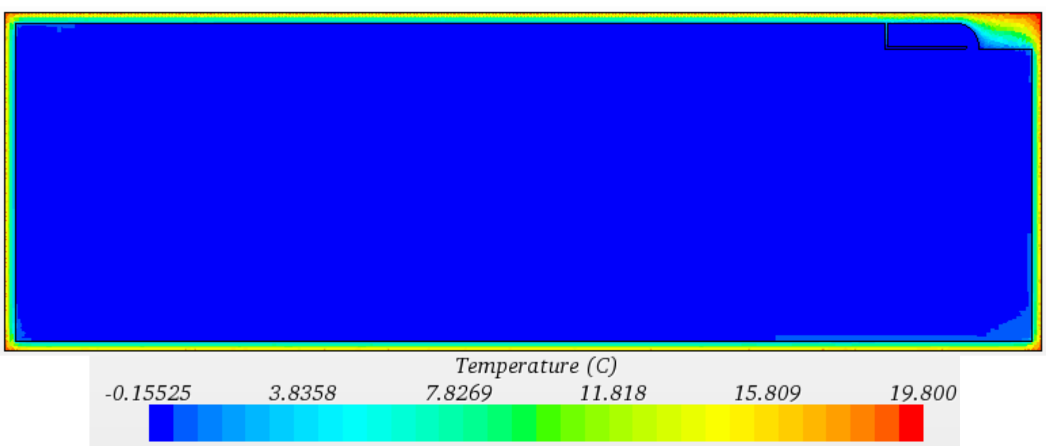


Figure 10: Temperature field of the compartment, including the insulated walls.
(T_{amb}=20°C, h_{ext}=10 W/m²K, λ=0.03 W/mK)

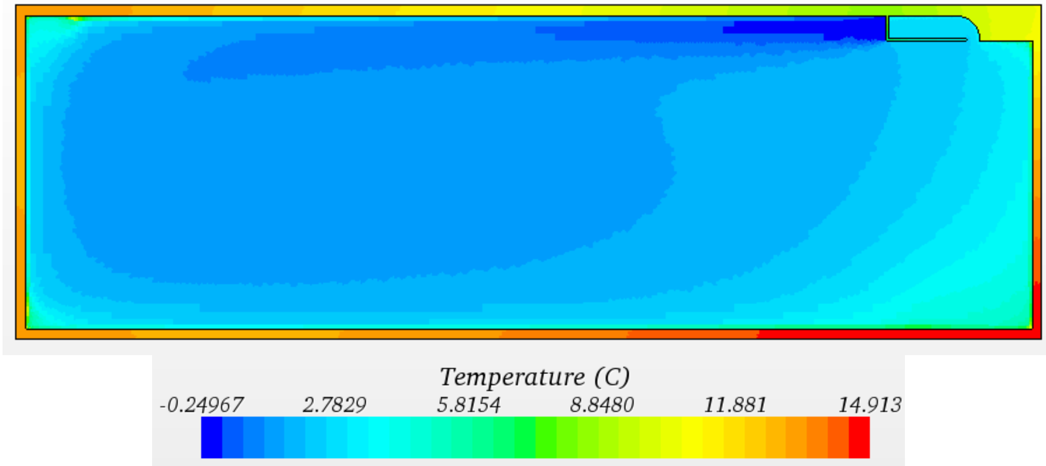


Figure 11: Temperature field of the compartment, including the insulated walls. (T_{amb}=20°C, h=10 W/m²K, λ=237 W/mK)

CONCLUSION

2D steady state simulations of heat transfer and air flow were carried out to study the behavior of a back compartment of a refrigerated truck using CFD software. The effects of the main parameters were studied. Using heat flux boundary condition, results show that this parameter strongly influences the temperature profile in the cavity. By changing the blowing velocity, the evolution of the temperature has been observed. A transient study for a loaded truck may be required to help designer to choose the proper blowing velocity. By studying the compartment length, the simulation highlights the evolution of the flow structure. The length of the back compartment should rather be limited to 10 m in order to get a proper air renewal in the entire cavity. Finally the boundary condition at the wall has been modified. It has been observed that the influence of environmental parameters is reduced, which will be graceful for an experimental validation of the model.

REFERENCES

1. Clavier F, Sartre V, Bonjour J. 2011, Infiltration heat load through the doorway of a refrigerated truck protected with an air curtain, *Proc. International Congress of Refrigeration*, IIF/IIR 783.
2. Gupta J K, Tulapurkar C, Khandelwal R, Gandikota M. 2011, Numerical modeling of refrigerator compartments, *Proc. International Congres of Refrigeration*, IIF/IIR 842.
3. Kolodziejczyk M, Butrymowicz D. 2011, CFD modelling of vegetable cold storage chamber, *Proc. International Congres of Refrigeration*, IIR/IIF 682.
4. Laguerre O, Ben Amara S, Moureh J, Flick D. 2006, Numerical simulation of air flow and heat transfer in domestic refrigerators, *Journal of Food Engineering*, *81*(1), 144–156.
5. Laguerre O, Ben Amara S, Remy D, Flick D. 2009, Experimental and numerical study of heat and moisture transfers by natural convection in a cavity filled with solid obstacles, *International Journal of Heat and Mass Transfer*, *52*(25-26), 5691–5700.
6. Mirade P S, Picgirard L. 2001, Assessment of air flow patterns inside six industrial beef carcass chillers, *International Journal of Food Science and Technology*, *36*, 463–475.
7. Moureh J, Menia N, Flick D. 2002, Numerical and experimental study of airflow in a typical refrigerated truck configuration loaded with pallets, *Computers and Electronics in Agriculture*, *34*, 25–42.
8. Moureh J, Flick D. 2005, Airflow characteristics within a slot-ventilated enclosure, *International Journal of Heat and Fluid Flow*, *26*(1), 12–24.
9. Yu H, Hoff S J. 1999, Airflow pattern similarity criteria for ceiling slot-ventilated agricultural enclosures under isothermal conditions, Transactions of the ASAE, 42 (2), 459–469.