Aero-thermal Simulation of a Refrigerated Truck Under Open and Closed Door Cycles

Alexandre Oury[1], Patrick Namy*[1], and Mohammed Youbi-Idrissi[2]
[1]SIMTEC, 8 rue Duployé, 38100 Grenoble, France
[2]Air Liquide, Centre de Recherche Paris-Saclay, 1 Ch. de la porte des loges, 78354 Jouy en Josas Cedex, France
*Corresponding author: patrick.namy@simtecsolution.fr

Abstract: This article presents two computational methods for predicting the distribution of temperature and air flow in the box of a refrigerated truck, for two configurations (open and closed) of the rear door. The first method, in which CFD and heat transfer are partially decoupled, is used when the door is closed and the cooling system turned on. The second (fully-coupled) method computes the natural convection occurring when the door is open after a cooling period, while the cooling system is switched off. The simulation results are compared to experimental values.

Keywords: Refrigerated truck, non-isothermal flow, forced convection, natural convection.

1. Introduction

Heat transfer inside a refrigerated truck is a key phenomenon that governs the temperature within the cooled compartment. The design of an efficient air-cooling system as well as the choice of appropriate thermal insulation materials requires a thorough understanding of the aero-thermal configuration of the refrigerated box. In this article, a numerical model implemented in COMSOL Multiphysics is described to simulate heat transfer phenomena occurring during the operation of a refrigerated truck. Aero-thermal simulations are carried out for two rear door configurations, closed or open, in order to describe periods of goods loading or unloading and periods of transport. The model is based on a coupling between CFD (turbulent k-ω or laminar approaches) and heat transfer. Comparison with experimental results shows a reasonable agreement in terms of temperature inside the box and at some specified points. This work demonstrates the feasibility of using COMSOL to simulate non isothermal turbulent flow.

2. Model description

2.1 Geometry

The geometry of the refrigerated compartment considered in the present study is given in fig. 1. It consists of a parallelepipedic box with dimensions ~6 m in depth (y axis) and ~2.5 m in both width (x axis) and height (z axis). The cooling system is included in a box located at mid-width on the roof, in the front part of the truck. It comprises two circular apertures on its bottom face for air extraction and a rectangular aperture on the rear through which refrigerated air is being blown into the main box (see fig. 2). The elements forming the cooling system, such as the heat-exchange system and the air fans, are not included in the model so that this box is treated as an empty volume.

Due to the presence of a (y,z) symmetry mirror at mid-width, the model is only implemented for a half of the truck box (as depicted in fig. 1b), which is sufficient to fully described the physical phenomena of the system.

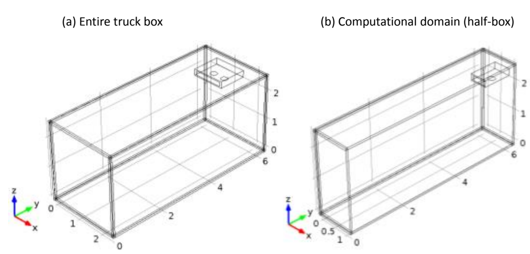

Figure 1. (a) Geometry of the real truck box and (b) half-box unit used as computational domain after symmetry considerations.

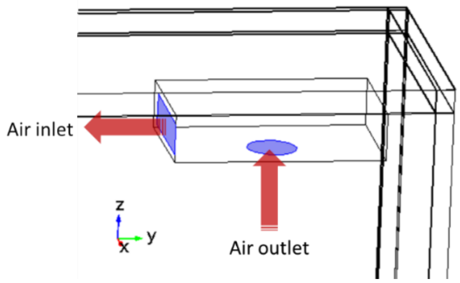

Figure 2. Air inlet and outlet positions of the refrigerating system in the half-box representation.

Table 1: Materials constituting each layer of the box walls.

Type of wall	**Constituting elements**
Lateral walls	Polyester, 2.5 mm Polyurethane A, 40 mm Polyester, 2.5 mm
Floor	Plywood plate, 24 mm Polyurethane A, 105 mm Plywood plate, 9 mm
Roof	Polyester, 3 mm Polyurethane A, 102 mm Polyester, 3 mm
Front wall	Polyester, 2.5 mm Polyurethane A, 80 mm Polyester, 2.5 mm
Rear wall	Polyester, 2.5 mm Polyurethane B, 55 mm Polyester, 2.5 mm

Table 2. Thermal properties of the materials constituting the box walls.

Material	**Conductivity ($W.m^{-1}.K^{-1}$)**	**Density ($kg.m^{-3}$)**	**Heat capacity ($J.kg^{-1}.K^{-1}$)**
Polyester	0.25	1380	1400
Polyurethane A	0.025	40	1400
Polyurethane B	0.025	50	1400
Plywood	0.15	400	1880

The walls of the box are made up of three layers consisting of a polyurethane foam sandwiched between two thin polyester-based layers, except on the floor where the inner and outer walls comprise an additional layer made of a plywood plate. The dimensions of each of the layers constituting the box walls are summarized table 1 and the physical properties of the materials are given in table 2.

2.2 Equations

2.2.1 Fluid dynamics model

- Door closed/refrigerating unit on:

When the rear door of the truck is closed, the fans and the refrigerating unit are both operating to provide coldness to the refrigerated compartment. Due to the high velocity of the air circulating near the ventilation system (~ 2 m/s for a flow rate of 1000 m^3/h), fluid dynamics is described in the full box by means of the RANS k-ω turbulence model, which is associated with the following four equations:

$$\rho\frac{\partial \boldsymbol{u}}{\partial t}+\rho(\boldsymbol{u}.\nabla)\boldsymbol{u} = \nabla.[-pI+(\mu+\mu_T)(\nabla\boldsymbol{u}+(\nabla\boldsymbol{u})^T) -\frac{2}{3}(\mu+\mu_T)(\nabla.\boldsymbol{u})I -\frac{2}{3}\rho k I] \quad (1)$$

$$\frac{\partial\rho}{\partial t}+\nabla.(\rho\boldsymbol{u})=0 \quad (2)$$

$$\rho\frac{\partial k}{\partial t}+\rho(\boldsymbol{u}.\nabla)k=\nabla.[(\mu+\mu_T\sigma_k^*)\boldsymbol{\nabla}k]+P_k-\rho\beta_0^*k\omega \quad (3)$$

$$\rho\frac{\partial w}{\partial t}\rho(\boldsymbol{u}.\nabla)\omega=\nabla.[(\mu+\mu_T\sigma_\omega)\boldsymbol{\nabla}\omega]+\alpha\frac{\omega}{k}P_k-\rho\beta_0\omega^2 \quad (4)$$

where p is the pressure, $\boldsymbol{u}$ the velocity field, ρ the air density, μ_T is the turbulent viscosity given by $\mu_T=\rho\frac{k}{\omega}$, k is the turbulent kinetic energy, ω is the turbulent dissipation rate and where σ_k^*, σ_ω, β_0 and β_0^* are the turbulence parameters of the model whose values are given in table 3.

- Door open/refrigerating unit off:

When the door is open, the refrigerating unit as well as the ventilation are both considered to be switched off. As a result, when the door is open after a cooling period, the air flows from the outside into the inside of the box, due to the temperature difference between the two media. In this case, fluid dynamics is simulated with the laminar flow Navier-Stokes equations, associated with natural convection:

$$\rho\frac{\partial \boldsymbol{u}}{\partial t}+\rho(\boldsymbol{u}.\nabla)\boldsymbol{u} = \nabla.\left[-pI+\mu(\nabla\boldsymbol{u}+(\nabla\boldsymbol{u})^T)-\frac{2}{3}\mu(\nabla.\boldsymbol{u})I\right] + (\rho(T)\text{-}\rho_0)\vec{g} \quad (5)$$

$$\frac{\partial \rho}{\partial t}+\nabla.(\rho\boldsymbol{u})=0 \quad (6)$$

In eq. (5), the natural convection term $(\rho(T)\text{-}\rho_0)\vec{g}$ is the driving force of the flow, which arises due to deviations of the local value of the density $\rho(T)$ from the reference value ρ_0. The latter is taken as the density of air at room temperature (20°C). The relations giving air density and viscosity as a function of temperature are given in table 4.

Table 3. Values of the turbulence parameters used in the CFD model describing the air flow when the door is closed (k-ω model).

σ_k^*	1/2
σ_ω	1/2
β_0	9/125
β_0^*	9/100
I_T	0.16 $Re^{-1/8}$ = 0.039
L_T	0.017 m

Table 4. Dependence of the air density and viscosity upon temperature.

Parameters	Value or expression
ρ	$\frac{pM}{RT}$ with *p*: pressure, *M*: molar mass of air, *R*: universal gas constant.
μ	-8.4x10^{-7} + 8.4x10^{-8} T - 7.7x10^{-11} T^2 + 4.6 x10^{-14} T^3 - 1.1x10^{-17} T^4

2.2.2 Thermal model

For both closed door and open door configurations, the general time-dependent heat transfer equation involving diffusion and convection is implemented in the air domain:

$$\rho C_p\frac{\partial T}{\partial t}+\nabla.\left[\rho C_p\boldsymbol{u}T-k\nabla T\right]=0 \quad (7)$$

where T is the temperature, k the thermal conductivity and C_p the heat capacity of air.
In the box walls, where heat is transported exclusively by diffusion, a simple diffusion equation is solved for:

$$\rho C_p\frac{dT}{dt}-\nabla.\left(k\vec{\nabla}T\right)=0 \quad (8)$$

2.3 Boundary conditions

The boundary conditions applied to CFD and heat transfer equations are summarized in table 5 for the two door configurations (open and closed).

The inner air and the walls are initially at room temperature. During the first period with the door closed, the refrigerating system operates to cool down the air circulating within the box. The temperature and the velocity U_0 of the air which is injected into the box are set in the model to time-dependent measurements obtained *in situ* with a dedicated experiment on a monitored truck.

No slip conditions are defined on every walls during that period. When the door is open, an open boundary condition is defined for both CFD and heat transfer equations on the rear of the box, allowing air and heat to get into the compartment.

Under open door condition, the refrigerating system as well as the fans stop working. Thus, the surfaces through which air was previously injected and extracted around the refrigerating box are turned to insulated boundary (no slip condition for the fluid and no heat flux).

A convective heat transfer condition is kept at all times on the outer surfaces of the box in contact with surrounding atmosphere. The external temperature T_{ext} is set to the values collected during the experiment. Symmetry conditions are set on the (y,z) symmetry plane splitting the real box in two.

Table 5. Boundary conditions applied in the CFD models for both open and closed door configurations.

<table>
<tr><th>Boundary</th><th>Door closed</th><th>Door open</th></tr>
<tr><td>Lateral</td><td rowspan="5">• CFD (inner boundary)
No slip condition: $\boldsymbol{u} = \boldsymbol{0}$

• Heat transfer (outer boundary)
Convective exchange with external air:
$-\boldsymbol{n}.(-k\nabla T) = h(T_{ext} - T)$</td><td rowspan="4">• CFD
No slip condition: $\boldsymbol{u} = \boldsymbol{0}$
• Heat transfer
Thermal insulation
$-\boldsymbol{n}.(-k\nabla T) = 0$</td></tr>
<tr><td>Floor</td></tr>
<tr><td>Roof</td></tr>
<tr><td>Front</td></tr>
<tr><td>Rear</td><td>• CFD
Open boundary:
$[-pI + \mu[(\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T) - \frac{2}{3}\mu(\nabla.\boldsymbol{u})I]\boldsymbol{n} = 0$
• Heat transfer
Open boundary:
$T = T_{ext}$ if $\boldsymbol{u}.\boldsymbol{n} < 0$,

$-\nabla T.\boldsymbol{n} = 0$ if $\boldsymbol{u}.\boldsymbol{n} > 0$</td></tr>
<tr><td>Symmetry plane</td><td>• CFD
$\boldsymbol{u}.\boldsymbol{n} = \boldsymbol{0}$
$\boldsymbol{K} - (\boldsymbol{K}.\boldsymbol{n})\boldsymbol{n} = \boldsymbol{0}$
$\boldsymbol{K} = [\mu(\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T) - \frac{2}{3}\mu(\nabla.\boldsymbol{u})I]\boldsymbol{n}$
• Heat transfer
$-\boldsymbol{n}.(-k\nabla T) = 0$</td><td>• CFD
Same as for door closed

• Heat transfer
$-\boldsymbol{n}.(-k\nabla T) = 0$</td></tr>
<tr><td>Air inlet*</td><td>• CFD
Inlet condition (ventilation on): $\boldsymbol{u} = -U_0.\boldsymbol{n}$
$k = \frac{3}{2}(U_0 I_T)^2$, $\omega = \frac{k^{1/2}}{(\beta_0^*)^{1/4} L_T}$
• Heat transfer
T(t) set to experimental values</td><td rowspan="2">• CFD
No slip condition (ventilation off):
$\boldsymbol{u} = \boldsymbol{0}$

• Heat transfer
Thermal insulation:
$-\boldsymbol{n}.(-k\nabla T) = 0$</td></tr>
<tr><td>Air outlet**</td><td>• CFD
Outlet condition with normal flow and blackflow suppressed:
$\boldsymbol{n}^{\boldsymbol{T}}.[-pI + (\mu + \mu_T)(\nabla \boldsymbol{u} + (\nabla \boldsymbol{u})^T) - \frac{2}{3}(\mu + \mu_T)(\nabla.\boldsymbol{u})I - \frac{2}{3}\rho k I]\boldsymbol{n} = -\widehat{p_0}$
$\widehat{p_0} \leq p_0$, $\boldsymbol{u}.\boldsymbol{t} = \boldsymbol{0}$, $\nabla k.\boldsymbol{n} = 0$, $\nabla \omega.\boldsymbol{n} = 0$
• Heat transfer
No diffusive flux
$-\boldsymbol{n}.(-k\nabla T) = 0$</td></tr>
</table>

*Air injected into the box. **Air extracted from the box by the ventilating system.

2.4. Mesh

The mesh created for the study is depicted in fig. 3. The box volume is meshed with tetragonal elements and five boundary layers are defined in the vicinity of the inner walls. A swept mesh with prismatic elements is created in the box walls, each of the wall layers containing three elements along the depth. The total number of elements is close to 130,000. A sensitivity study has been performed to ensure the independence of the solution on the mesh.

2.5. Resolution scheme and numerical details

The present model is used to simulate the air flow and the thermal behaviour of the truck box during an operating sequence composed of two steps: a first step (~ 3h) where cold air is injected into the box by the refrigerating system to cool down the box while the rear door is closed, followed by a second step (~ 10min) where the door is open and

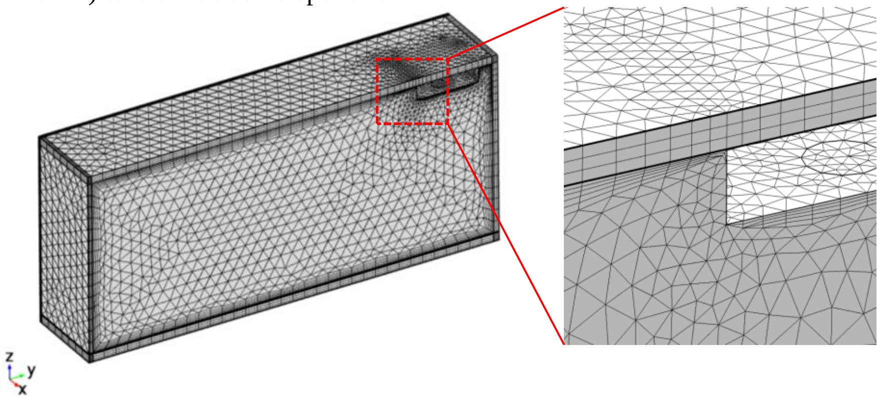

Figure. 3. Details of the mesh defined in the box and the walls

both the cooling system and the ventilation are switched off.

In order to minimize the degrees of freedom of the system, CFD and heat transfer were partially decoupled:

-For the first step (door closed), the turbulent CFD is solved independently of the thermal field, considering that the entire air volume is at a constant temperature of 20°C. The temperature field is then computed in a separate study, by injecting the velocity field calculated previously into eq. (7) and (8).

-For the second step (door open), natural convection is implemented by computing both laminar CFD and heat transfer equations together as described previously, setting the initial temperature to the value reached at the end of the closed door period.

Turbulent CFD, laminar CFD and thermal equations are all solved for with the fully-coupled solver, associated with the time-dependent BDF solver. The maximum time step is chosen large (200 s) during the closed door period since a quasi-steady state is reached. During the open door period, the maximum time steps are decreased significantly (1s-step at the very beginning of the period) to capture the rapid transient of hot air penetrating into the box ; the time steps are then increased again when the air velocity slows down as the temperature gets more uniform.

3. Results.

3.1. Door closed

The first stage of the study consisted in simulating the evolution of the temperature inside the refrigerated compartment together with the air velocity field for the first cooling period which lasts approximately 3 hours. During this phase, the door is maintained closed while cold air is injected into the box.

The air flow simulated in the box after 10 000 s is depicted in a streamline representation in fig. 4 and fig. 5, together with the local velocity (fig. 4) and temperature (fig. 5). At that time, the system is under quasi steady-state conditions. It can be observed that the velocity is maximum (> 1 $m.s^{-1}$) in the roof area facing the air inlet and along the door wall while it decreases sharply as the air continues its course into the rest of the box volume. The temperature field follows basically the same distribution as the air velocity, where the coldest areas correspond to high-velocity regions and hottest areas to low-velocity regions. Cold air that is produced by the refrigerating system enters the box at a temperature of ~ -27°C and is heated up by contact with the hotter box walls. The hottest region, where air is above 0°C, corresponds to a recirculation zone located in the bottom, closed the lateral wall

The present model was also used to calculate the heat power that is lost through the encasement of the box. Fig. 6 presents the evolution of the averaged heat losses simulated through each of the five walls of the half-box (lateral, top, bottom, front and rear). These were calculated by averaging the heat power that is extracted by convection (the $h(T_{ext} - T)$ term) on the outer boundaries of the box which are in contact with external air. A general trend is the global increase of the heat losses over time, as the air is cooled down inside the truck. It appears that most of the heat is being lost through the lateral and the rear walls, those two presenting similar loss profiles with a ~ 22 W/m²-plateau reached after 6000 s. On the opposite, a limited amount of energy is being lost through the floor, essentially because the latter is the most insulated, associated with the thickest materials. Such considerations can be advantageously accounted for in order to optimize the materials and the design of the box walls.

3.2. Door open

In the second stage of the study, the flow of air penetrating into the box as well as the associated temperature when the door is being open after the cooling period described previously were simulated. The cooling system and the ventilation are both considered to be switched off during this period, such that the only driving force for air is the natural convection arising as a result of the difference between inner and outer temperatures. Fig. 7 and fig. 8 show respectively the velocity and the temperature of the air entering into the box at different times (2, 10, 50 and 500 s) following the opening of the door. As it can be observed, hot air enters rapidly the box during the first seconds. When 50 s has been elapsed since the opening of the door, the simulated average air velocity has dropped in the box below 10 cm/s and is as low as 2 cm/s after 500 s. This has to be attributed to the prompt vanishing of the temperature difference between the outside and the inside of the box volume. Indeed, as it can be noticed on the thermal mappings, most of the box volume is at the outer temperature (around 25°C) after a short while (~10 s), except in the wall vicinity. The thermal inertia of materials keeps them cold for quite a long time, maintaining a certain coldness of the air surrounding the walls.

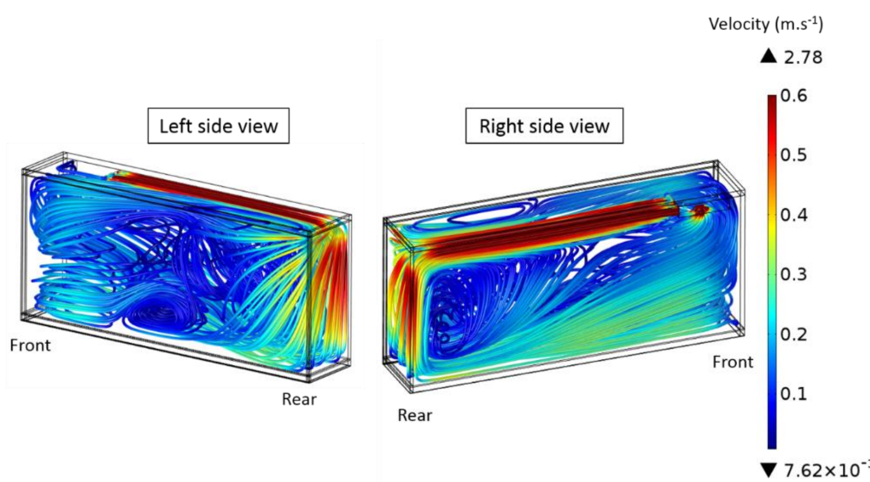

Figure 4. Streamlines representation of the air flow simulated after 10 000 s in the closed truck box, associated with the local velocity of air.

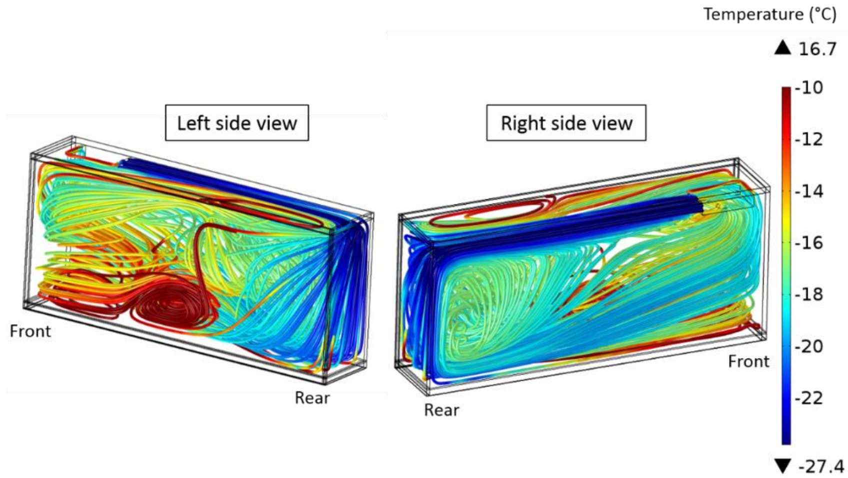

Figure 5. Streamlines representation of the air flow simulated after 10 000 s in the closed truck box, associated with the local temperature.

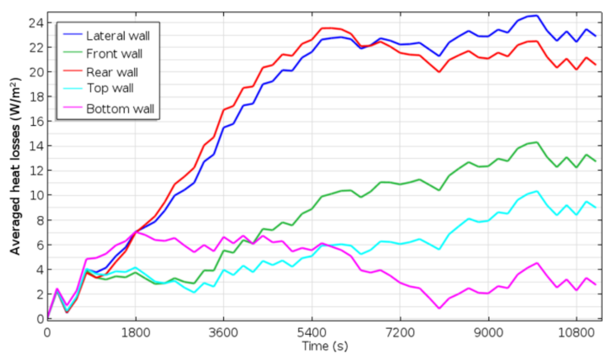

Figure 6. Evolution of the averaged heat losses simulated through the different parts of the truck box during the first cooling period (door closed).

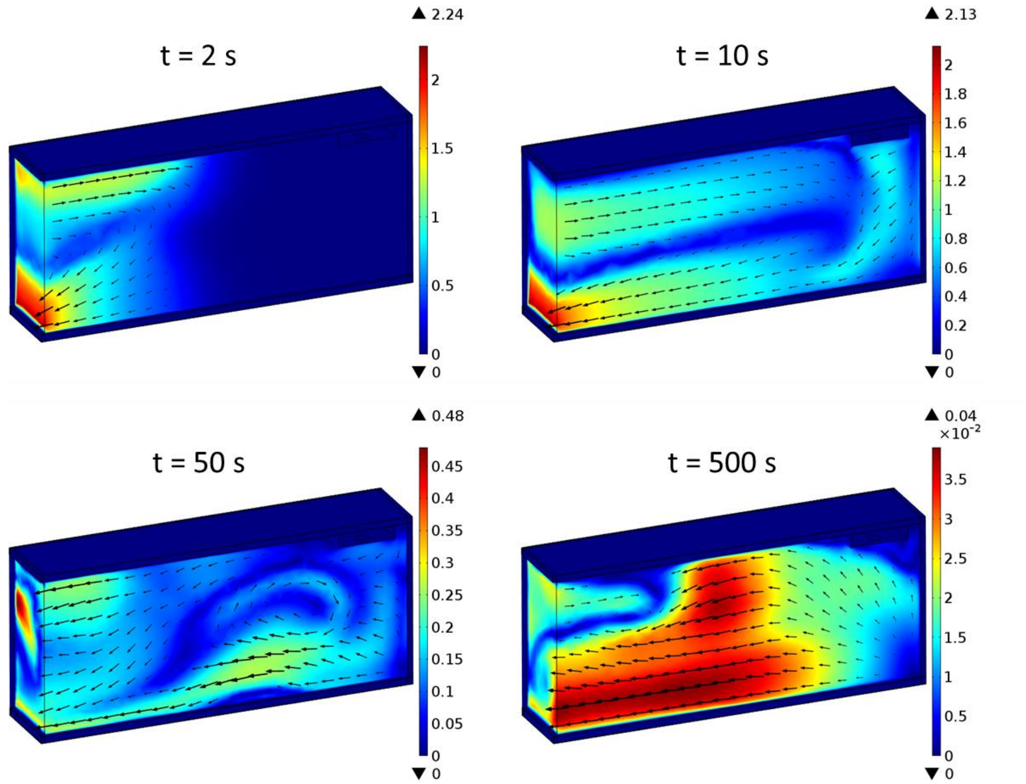

Figure 7. Evolution of the air velocity (in $m.s^{-1}$) in the truck box, at different times after opening the door.

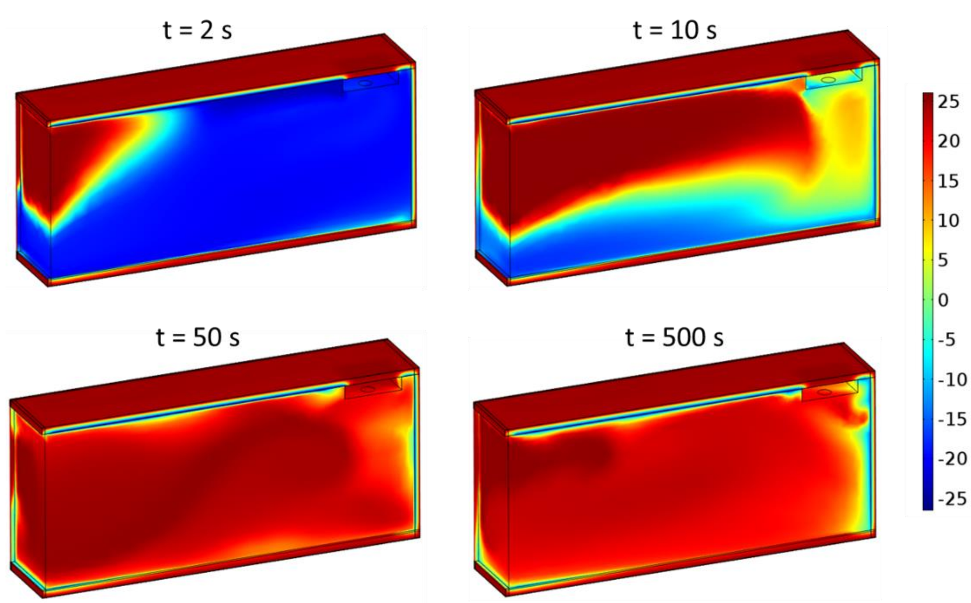

Figure 8. Evolution of the temperature of the air (in °C) in the truck box, at different times after opening the door.

3.3. Comparison with experimental measurements

The thermal predictions of the model were compared to experimental data obtained from a bench test in which the rear door of the truck was alternatively open and closed several times in a row. The temperature transient was experimentally collected by a sensor located close to the air extraction area, on the bottom face of the refrigerating system. The temperature transient simulated by the model in this area is given in fig. 9 together with that of the sensor. The experimental temperature decreases from 20°C to the target temperature -20°C (reached after ~4000 s) and then remains quite stable due to automatic thermal regulation. The first door opening can be noticed by the sharp increase in the temperature taking place at t = 11500 s. The door is maintained open until the temperature drops down again (cooling system turned on), etc.

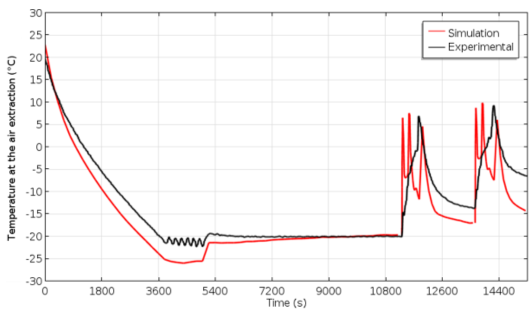

Figure 9. Comparison between simulated and experimental temperature transients measured around the air extraction area during the cooling period with the rear door closed.

It can be seen that the model predictions are in a reasonable agreement with the experimental temperatures. Some oscillations can however be observed in the temperature simulated during the two open door periods, which are not observed experimentally. This could be due to a difference between the location where the temperature is calculated in the model and to the actual position of the sensor. The intrinsic inertia of the sensor may also have a slight levelling effect on the measured temperature, while the present model simulates the instantaneous temperature of air.

4. Conclusion

In this article, we present two transient computational methods for predicting the temperature and the air flow in a refrigerated truck compartment for two configurations: (1) closed door/cooling system on and (2) open door/cooling system off. To simulate a cooling period with the door closed and the ventilating fans turned on, the CFD approach is based on the turbulent CFD equations for calculating the forced circulation of air inside the truck box, while the temperature is simulated with a heat transfer model which is partially decoupled to CFD and recalls the velocity field previously calculated. When the door is open after the cooling period and both the cooling system and the ventilating fans are switched off, the heat transfer equation is solved for together with laminar CFD equations in a fully coupled approach. The simulated temperature is compared to experimental data collected during a bench test, showing reasonable agreements. This study shows that Comsol is a tool particularly well adapted for simulating non isothermal flow with the above hypothesis. It can be used for instance to optimise the location or the power specifications of the cooling system as well as the design of the box walls and the associated materials.