Simulation of temperature distribution in refrigerated container 40 feet.

Vo Van Sim¹², Le Thi Kim Phung¹², Tran Tan Viet¹²*
¹ Faculty of Chemical Engineering, Ho Chi Minh City University of Technology (HCMUT), 268 Ly Thuong Kiet Street, District 10, Ho Chi Minh City, Vietnam
² Vietnam National University Ho Chi Minh City, Linh Trung Ward, Thu Duc District, Ho Chi Minh City, Vietnam

*Corresponding author’s e-mail: trantanviet@hcmut.edu.vn

Abstract. This study shows the method for predicting the temperature distribution and air flow in the box of a refrigerated container 40 feet, for two configurations (open and closed) of the rear door. Computational Fluid Dynamics – CFD (Computational Fluid Dynamics) is decoupled and used when the truck door is fully closed. The temperature, velocity vector and heat flow are simulated based on the finite element method. The simulation results are compared with the experimental data.

1. Introduction
In order to maintain the quality of agricultural produce in the cold, it is important to ensure an even temperature distribution inside the cold chambers. In practice, it is difficult to determine the temperature distribution based on experimental data because of some limitations such as experimental cost and technical problems with data collection. Computer based prediction and assessment tools are therefore an attractive alternative to survey the phenomenon of heat transfer in cold storage with low cost [1]. Heat transfer inside a container is the process of temperature change, including the process of convective heat transfer between the cold gas flow provided by the indoor unit with the air inside and the process of heat loss to the surrounding environment through the insulating walls along with time. The effective working level of a refrigeration system depends on factors such as the indoor unit’s parameters, thermal insulation materials and moisture insulation. However, the heat distribution of the internal fluid, in this case the cold gas flow, determines the quality of stable and efficient operation [2]. Considering the local heat factor, the gas flow vector is very important. With the goal of optimizing the distribution of the internal cold flow by controlling the flow rate, the inlet temperature enables the efficient operation [3]. In addition, the simulation using CFD allows the designer to predict the desired values when changing the influencing factors. This study shows the feasibility of using CFD to simulate non-isothermal turbulent flow inside a refrigerated container with temperatures ranging from -20 °C to 30 °C.

2. Simulation procedure
2.1. Geometry
The geometry of refrigerated container in this study is illustrated in Figure 1. It is modeled as a box with dimensions 11.548 m in depth (x-axis), 2.290 m in width (y-axis) and 2.575 m in height (z-axis). The cooling system is in the front part of the truck, as a box located in the middle of the roof. It has two round apertures on the bottom face to suck air and a rectangular one at the front where the cold air...
is blown into the main box [4]. Because the truck was simulated as the box which is symmetric through the (x, z) plane at mid-width, performing simulation for a half of the truck box is sufficient to describe the physical phenomena of the whole system.

| Parameters                      | Polyurethan | Inox 304 | galvanized-steel-sheet | Fluid      |
|---------------------------------|-------------|----------|------------------------|------------|
| Density (kg/m³)                 | 50          | 7850     | 7133                   | Ideal gas  |
| Heat capacity (J/kg.K)          | 1400        | 490      | 390                    | 1006.4     |
| Heat conductivity (W/m.K)       | 0.025       | 14       | 112.2                  | 0.0242     |
| Viscosity coefficient (kg/m.s)  | 150         | 2        | 0.5                    | 1.789e-5   |
| Thickness (mm)                  |             |          |                        |            |

**Figure 1.** Geometry of the truck box

### 2.2. Geometry

#### 2.2.1. Descriptive differential equations

In convective heat transfer, heat of the fluid changes with time and distributing coordinates. The derivative of temperature element includes derivative terms with respect to time and fluid coordinates.

**Energy Equation:**

\[ \frac{dT}{dt} = \frac{\partial T}{\partial t} + w \cdot \text{grad}(T) \] (1)

**Momentum Equation:**

Each element is subjected to three forces, including the gravity, pressure and friction. The force equation generates the motion:

\[ \rho \frac{dw}{dt} = \rho g - grandP + \mu \nabla^2 w \] (2)

**Continuity Equation:**

The equation represents the variation of fluid density in an uncompressed medium, the equation is as the following:

\[ \frac{\partial (w_x)}{\partial x} + \frac{\partial (w_y)}{\partial y} + \frac{\partial (w_z)}{\partial z} = 0 \] (3)

**Differential Equation for Heat Dissipation**
\[
\frac{\partial T}{\partial t} + w_x \frac{\partial T}{\partial x} + w_y \frac{\partial T}{\partial y} + w_z \frac{\partial T}{\partial z} = \frac{\partial}{\partial x} \left( k \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial z} \left( k \frac{\partial T}{\partial z} \right)
\]

(4)

- \(T\): temperature, °C
- \(w_x, w_y, w_z\): the velocity along x, y, z-direction
- \(k\): heat transfer coefficient, \(W\ m^2\ K^{-1}\)
- \(\rho\): density, \(kg\ m^{-3}\)
- \(\mu\): dynamic viscosity, \(kg\ m^{-1}\ s^{-1}\)
- \(\dot{f}\): time, \(s\)
- \(g\): 9.81 \(m\ s^{-2}\)

2.2.2. Conditions
Due to the symmetry in convection along y-direction, and the velocity field in the direction of \(y = 0\), the heat transfer is considered to change only in the two x and z directions, corresponding to the length and height of the container, respectively [5]. Then the function is described as \(t = f(x, z, \dot{f})\). This is unstable convective heat transfer with boundary conditions type 3.

**Initial conditions:**
- \(t = 0\), \(x = y = z = 0\), \(t_w = T = 30^\circ C\), \(w_x = w_y = w_z = 0\)
- \(x = 11.485, y = (0.895 – 1.395), z = (2.475 – 2.575)\)
- \(t_f = -20^\circ C, w_x = -2.5, w_y = w_z = 0\)

**Boundary conditions:**

\[
\frac{1}{\alpha} \frac{\partial T}{\partial t} = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial z^2}
\]

(5)

- \(\alpha\): thermal diffusivity, \(m^2\ s^{-1}\)

2.2.3. Mesh
Mesh is created to perform simulation as shown in Figure 2. The container is modeled as a box, the walls and volume are meshed into triangular tetrahedrons. The faces meshed for boundary conditions include 5 layers. The distances of the boundary layers from the inside to outside increases by a ratio of 1.2. Because of the symmetry of the geometry, the mesh model is halved. It means that the computation time can be significantly reduced but the calculation results are still guaranteed [4], [6]. Mesh parameters are presented in table 2 below.
Table 2: Mesh parameters

| Mesh type | Number of mesh elements | Criteria                  |
|-----------|-------------------------|---------------------------|
| Tetras    | 163998                  | Skewness < 0.9, Squish < 0.90 |
| Pentas    | 84290                   | Skewness < 0.9, Squish < 0.85 |
| Sum       | 248288                  |                           |

2.2.4. **Commercial CFD code**

The simulations were performed using the commercial CFD software package ANSYS FLUENT 18.0. Fluent uses the finite volume method to solve the governing fluid dynamics equations for energy transfer. These conditions are established for CFD simulation with a compressible flow mode. The process runs with the turbulent model (k-epsilon), fluid flow over the time (transient) and symmetry condition [6]. The computational domain was established by hexahedral and tetrahedral meshes. The total number of mesh elements was 84290 and 163998, respectively. First-order implicit time stepping was used with step size of 1s and thirty iterations were performed for each step time to achieve normalized residuals below 10^{-3} for all equations. The basic setting of CFD simulation is summarized in Table 2 and Figure 3.

![Figure 3: Iteration in CFD](image)

3. **Results and discussion**

3.1. **Results.**

**Door closed**

In the first phase of the study, the temperature field and air velocity distribution are simulated in the refrigerated container when the door is completely closed with the cold air flow supplied to the system of 2.5 m/s. As illustrated in Figures 4, 5, 7 and 11, the temperatures in the inner regions reach -18 °C at 2 hours and after 3 hours, the average temperature is below -19 °C. The state of the system gradually returns to stability. The temperature field, in general, has a similar distribution as the air velocity,
where the coldest region corresponds to the high velocity region and the hottest region corresponds to
the low velocity region in Figure 7.
The cold air generated by the refrigeration system enters the box at a temperature of \(-20^\circ C\) and is
heated when contacting with the high heat zone of the device wall (30°C). The hottest zone during
operation is always the upper part where the cold air obeys the convection phenomenon, the low
temperature tends to go below and the part close to the side of the truck where the insulation layer
minimizes heat exchange to the surrounding environment.

**Figure 4.** Temperature distributions along with time (door closed)

**Figure 5.** Air flow distribution
When the door is open, the refrigeration system is still operating, the hot air flow from the outside overflows into the heat exchanger due to the difference in temperature between the two air zones at 25000s. As soon as the door was opened, the gas closes for 10 minutes, as shown in Figures 8, 9, 10 and 11. The temperatures at the door and the upper part of the truck increase remarkably, the temperature
in this area is close to the temperature outside. In the lower half of the truck, the temperature remains around 5 °C, which could be explained by the vector-directed cold air flow as shown in Figures 9.

![Image of heat distribution](image1)

**Figure 8.** Heat distribution according to time when the door is open

### 3.2. Comparison of simulation and experimental results

The results of computational simulation are compared with the experimental results in the environment with equivalent conditions. The experiments were carried out in Ho Chi Minh City. At points as shown in figure 10, sensors were mounted and the temperatures were recorded. Results were recorded every minute. The comparison results were shown in Figure 13. After nearly 7 hours of running in closed mode, the temperature inside all points returned to -20 °C, the real supplied temperature was -22 °C and the system could reach -20 °C as desired after 2 hours. When the door was open, at points 3, 6 and 9, the temperature jumped to nearly the same level as the temperature outside. Especially at point 3, the temperature was at 30 °C.

![Image of velocity vector](image2)

**Figure 9.** Velocity vector when the door is open

![Image of investigated points](image3)

**Figure 10.** Location of investigated points
Figure 11. Temperature deviations at 9 points

Figure 12. Average temperatures at faces
To evaluate the generalization between experimental and simulation time is shown in Figure 13

![Figure 13. Comparison of simulation](image)

The average error of this process is 3%.

4. **Conclusions**

In this study, simulation by CFD is used to predict the heat field and the gas flow velocity for two configurations (open and closed) of the rear door. Inside the truck, the process of forced convective heat exchange is not stable over time. This matter is difficult to address using conventional numerical methods. The process is very important to help check the parameters for cold storage design and operation. It is found that the experimental and simulation results do not have significant deviations during the period of study. Moreover, Fluent is demonstrated as a particularly suitable tool for non-isothermal flow simulation with the above assumptions, and can be applied to optimize the location or technical specifications of the refrigeration system as well as to design various shapes and related materials.

**Acknowledgements**

We acknowledge the support of time and facilities from Ho Chi Minh City University of Technology (HCMUT), VNU-HCM

**References**

[1] Norton T, Sun D-W, Grant J, Fallon R, Dodd V. 2007 Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry. (Bioresour Technol) p 414

[2] Muhammad Arif Budiyanto, Sunaryo, Haris Fernanda, Takeshi Shinoda 2018 Effect of azimuth angle on the energy consumption of refrigerated container (International Conference on Power and Energy Systems Engineering)

[3] Björn Margeirsson, Sigurjon Arason 2008 Temperature monitoring and CFD modelling of a cold storage.
[4] Duy K. Hoang, Simon J. Lovatt, Jamal R. Olatunji, James K. Carson 2020 *Validated numerical model of heat transfer in the forced air freezing of bulk packed whole chickens* (International Journal of Refrigeration) p 93.

[5] Cleland, A.C., and R.L. Earle. 1984 *Freezing time predictions for different final product temperatures* (Journal of Food Science).

[6] R. W. Lewis, P. Nithiarasu and K. N. Seetharamu 2004 *Fundamentals of the Finite Element Method for Heat and Fluid Flow* (Fundamentals of the Finite Element Method for Heat and Fluid Flow, John Wiley & Sons)

[7] D. Westphalen, R. Zogg, A. Varon, M. Foran 2018 *Energy savings potential for commercial refrigeration equipment* (Final Report Prepared for Building Equipment Division Office of Building Technologies. U.S. Department of Energy e DOE).

[8] 2018 *Fluent I. Fluent release 18.*