Since January 2020 Elsevier has created a COVID-19 resource centre with free information in English and Mandarin on the novel coronavirus COVID-19. The COVID-19 resource centre is hosted on Elsevier Connect, the company's public news and information website.

Elsevier hereby grants permission to make all its COVID-19-related research that is available on the COVID-19 resource centre - including this research content - immediately available in PubMed Central and other publicly funded repositories, such as the WHO COVID database with rights for unrestricted research re-use and analyses in any form or by any means with acknowledgement of the original source. These permissions are granted for free by Elsevier for as long as the COVID-19 resource centre remains active.
A study on computational fluid dynamics modeling of a refrigerated container for COVID-19 vaccine distribution with experimental validation

Mingkan Zhang a,†, Jian Sun a, Brian Fricke a, Kashif Nawaz a, Kyle Gluesenkamp a, Bo Shen a, Jeffrey Munk b, Xiaobing Liu a

a Buildings and Transportation Science Division, Oak Ridge National Laboratory, One Bethel Valley Road, Oak Ridge, TN 37830, USA
b Electrification and Energy Infrastructures Division, Oak Ridge National Laboratory, One Bethel Valley Road, Oak Ridge, TN 37830, USA

ARTICLE INFO

Keywords:
CFD modeling
Vaccine distribution
Refrigerated container
CO2 concentration

ABSTRACT

A key issue with the distribution of vaccines to prevent COVID-19 is the temperature level required during transport, storage, and distribution. Typical refrigerated transport containers can provide a temperature-controlled environment down to −30 °C. However, the Pfizer vaccine must be carefully transported and stored under a lower temperature between −80 °C and −60 °C. One way to provide the required temperature is to pack the vaccine vials into small packages containing dry ice. Dry ice sublimes from a solid to a gas, which allows the allowable transport duration. This can be mitigated by transporting in a −30 °C refrigerated container. Moreover, because the dry ice will sublimate and thereby release CO2 gas into the transport container, monitoring the CO2 concentration within the refrigerated container is also essential. In the present work, a 3D computational fluid dynamics model was developed based on a commercially available refrigerated container and validated with experimental data. The airflow, temperature distribution, and CO2 concentration within the container were obtained from the simulations. The modeling results can provide guidance on preparing experimental setups, thus saving time and lowering cost, and also provide insight into safety precautions needed to avoid hazardous conditions associated with the release of CO2 during vaccine distribution.

1. Introduction

By the end of 2020, two types of COVID-19 vaccines were developed by manufacturers (Pfizer and Moderna) in the United States, both of which are based on mRNA and lipid nanoparticles. To ensure maximum potency of both vaccines, low-temperature transportation and storage are required. The Pfizer vaccine requires ultralow temperature storage (between −80 °C and −60 °C) [1], and the Moderna vaccine requires −30 °C storage. Typical refrigerated containers can only provide a temperature-controlled environment of about −30 °C. To safely transport the Pfizer vaccine, Pfizer designed a reusable package for transportation and storage that can maintain the vaccine at the target temperature for 10 days [1]. The package can accommodate between 1000 and 5000 doses and uses dry ice to maintain the proper storage temperature. However, distributing the vaccines to the general population in the United States is still challenging, especially for rural or suburban areas, where local towns, pharmacy chains, and hospitals may not have the infrastructure required to store the vaccine at the required temperature. In addition, tremendous stress has been placed on equipment suppliers due to the high demand for ultralow-temperature refrigeration equipment in a short amount of time. To address the challenge, the temperature distribution in the container must be assessed under ultralow-temperature conditions. Furthermore, because of the sublimation of dry ice from a solid to a gas, the CO2 concentration in the container must be studied to address the safety concerns in vaccine transport.

In addition to experimental approaches, computational fluid dynamics (CFD) modeling can be used to study the airflow, temperature distribution, and component concentrations (e.g., CO2) in refrigerated containers. Some previous studies of refrigerated containers used CFD...
modeling to investigate airflow and temperature fields in the storage and transportation of fruits and vegetables [2,3]. Umeno et al. [4] developed a 3D CFD model to study the temperature distribution within refrigerated containers, which validated the capability of CFD to predict the temperature distribution within refrigerated containers. Budiyanto and Suheriyanto [5] reported a CFD model of a refrigerated container. They validated their model with experimental measurement by comparing the temperatures of the container walls, from which a good agreement was obtained between the experimental measurements and the CFD modeling results. Using the validated model, Budiyanto and Suheriyanto [6] investigated the effect of air velocity on cooling rate in a refrigerated container. They found that the cooling time of a high cube refrigerated container with low-speed and high-speed fans were 40 and 28 min, respectively. Jiang et al. [7] used CFD modeling to design the internal structure of a refrigerated container to improve the distribution of cooling capacity. The previous CFD studies on refrigerated containers were reviewed by Dehghannya et al. [8].

Most of the previous CFD studies of refrigerated containers focused on containers designed for the storage and transportation of fruits and vegetables. CFD studies on ultralow-temperature containers are rare because of the lack of demand for ultralow-temperature storage. Moreover, CO₂ concentration in a refrigerated container has not been studied because of occupational health and safety concerns regarding exposure to large quantities of CO₂ do not need to be considered in fruit and vegetable storage applications. In the present work, a 3D CFD model for an ultralow-temperature refrigerated container was developed. The model can reveal the airflow and temperature distribution within the entire container. The CO₂ concentration was accounted for in the model to assess the risk associated with the sublimation of dry ice from the vaccine packages. The container wall panels and insulation materials were also modeled to link the container indoor air to the ambient environment. The modeling results can be used to provide guidance on preparing experimental setups, thus saving time and lowering cost. Moreover, the CFD results can provide insight into safety precautions needed to avoid hazardous conditions associated with the release of CO₂ during vaccine distribution.

2. Methods

A 3D CFD model of a refrigerated container was developed using a commercial software, ANSYS/FLUENT (version 17.2). The model was used to investigate the airflow, temperature distribution, and CO₂ distribution in the container.

2.1. Simulation domain and governing equations

Fig. 1 shows the schematic view of the refrigerated container model used in the present work, with \( L = 1.165 \text{ m} \), \( H = 2.56 \text{ m} \), and \( W = 2.28 \text{ m} \) to match dimensions of a real refrigerated container. A Cartesian coordinate system was employed in which \( x, y, \) and \( z \) are the directions for width, height, and length of the container, respectively. The origin of coordinate is at the lower corner under the refrigeration unit, which was located at the back of the container with the return air grille located near the ceiling of the container (Fig. 1). The air flows from the T-shaped floor beams, passes the entire container, and returns to the refrigeration unit through the return air grille at the back of the container. The supply air was distributed the length of the container through a series of T-shaped floor beams (Fig. 2a). Insulated sandwich panels formed the ceiling and walls of the container. The interior surface of the wall and ceiling panels consisted of a stainless steel sheet, and the exterior side was covered with an aluminum sheet (Fig. 2b). The insulating material between the steel and aluminum sheets is polyurethane foam (PUR). The PUR panel was 136 mm thick for the ceiling and 66 mm thick for the walls. The stainless steel and aluminum sheets were both 2 mm thick. The properties and thickness of the PUR, stainless steel, and aluminum are shown in Table 1 [9,10]. Twenty boxes filled with dry ice, with \( L_{\text{box}} = 0.54 \text{ m} \), \( H_{\text{box}} = 0.5 \text{ m} \), and \( W_{\text{box}} = 0.5 \text{ m} \), were deployed on the floor of the container to simulate the vaccine packages.

The governing equations considered for the fluid flow and heat transfer include continuity, momentum, and energy equations. Because CO₂ leaks from the vaccine boxes to the container, in the present work, two gases, air and CO₂, was modeled. A mixture model was employed to describe the mixture flow and heat transfer. The continuity equation of the mixture model is written as

\[
\frac{\partial \rho_m \bar{V}_m}{\partial t} + \nabla \cdot (\rho_m \bar{V}_m \bar{V}_m) = 0.
\]

where \( \bar{V}_m \) is the mass averaged (mixture) velocity and \( \rho_m \) is the mixture density, which are given by the following equations,

\[
\bar{V}_m = \frac{\sum_{i=1}^{n} \alpha_l \bar{V}_l}{\rho_m},
\]

and

\[
\rho_m = \sum_{i=1}^{n} \alpha_l \rho_l.
\]

In Eqs. (2) and (3), \( \alpha_l \) is the volume fraction of gas \( l \) and \( n \) is the number of gases. In the present work, \( n = 2 \), and \( l = 1 \) for air and \( l = 2 \) for CO₂.

The momentum equation is

\[
\frac{\partial (\rho_m \bar{V}_m)}{\partial t} + \nabla \cdot \left( \rho_m \bar{V}_m \bar{V}_m \right) = - \nabla p + \nabla \cdot \left( \mu_m + \mu_t \right) \left( \nabla \bar{V}_m + \nabla \bar{V}_m^T \right) + \rho_m \bar{g} + \nabla \left( \sum_{i=1}^{n} \alpha_l \rho_l \bar{V}_l \bar{V}_l \right).
\]
\[ \mu_l = \sum_{i=1}^{\infty} \alpha_l \mu_l \] (5)

\[ \vec{v}_{dr} \] is the drift velocity for gas \( l \), defined as

\[ \vec{v}_{dr} = \vec{v}_l - \vec{v}_\alpha \] (6)

In the present model, air and CO\(_2\) were treated as incompressible ideal gases. The flow was turbulent in all conditions evaluated, so a standard \( k-\varepsilon \) model was adopted in the model. Therefore, the turbulence dynamic viscosity can be written as

\[ \mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon} \] (7)

The details of the \( k-\varepsilon \) model can be found elsewhere [11].

The energy equation in the mixture model form is

\[ \frac{\partial}{\partial t} \left( \sum_{i=1}^{\infty} \alpha_l \rho_l h_l \right) + \nabla \cdot \left( \sum_{i=1}^{\infty} \rho_l \vec{v}_l h_l + p \right) = \nabla \cdot \left( \lambda_{eff} \nabla T \right), \] (8)

where \( \lambda_{eff} \) is the effective conductivity and \( h_l \) is the enthalpy for gas \( l \).

To simplify the model, some additional assumptions were employed in the model. Because the T-shaped floor was the supply air outlets, the temperature of the floor was the same as the supply air temperature. The refrigeration unit was assumed to be perfectly insulated in the model, so no cooling power was lost through the refrigeration unit. We also assumed that CO\(_2\) leaked through the top surface of the vaccine packages. Moreover, the air flowing into the return air grille was cooled down by the refrigeration unit and sent back to the container as the supply air. The container was also assumed to be perfectly sealed, so there was no exfiltration of container air or infiltration of ambient air.

2.2. Experimental temperature validation

The CFD model was validated by comparing the temperature results for the model with experimental temperature results. In the experiment, the refrigeration unit conducted with to provide \(-37.3 \degree C\) temperature air from supply air with 0.38 m\(^3\)/s (800 CFM) flow rate. The experiment was running until a steady state achieved. Thermocouples were deployed in the container. Fig. 3 shows the locations of the six thermocouples marked as point 1 to point 6 to measure the temperature distribution in the container. Some other thermocouples were located on the internal surfaces of the container. Since the experiments provided the temperatures of the internal surfaces of the container, which can be used as the boundary conditions of the CFD model, ambient temperature changes were not considered in the validation. A steady-state simulation was conducted using the CFD model under the same conditions as the experiment. In this validation, only the air flow was modeled; the CO\(_2\) concentration was not considered. Temperatures were compared from the point 1 to point 6 in the container, as well as at the return air grille.

Table 2 shows the comparison results. The temperatures from the CFD model were slightly lower than the experimental data, which may be due to the uncertainty in the testing. Nevertheless, the results show a reasonable agreement between experimental data and simulation results. Therefore, the model is considered validated and is appropriate to use for the study of the refrigerated container.

3. Results and discussions

The validated CFD model was used to simulate a refrigerated container...
container exposed to a sunny day from morning (07:00) to evening (16:00). Because of solar radiation, the temperature of the various exterior surfaces changed with time during the daytime. Fig. 4 depicts the temperature changes of three external surfaces of a refrigerated container on August 27, 2013 at Hakata Port, Japan [5,12]. Because of the lack of temperature data on the container door surfaces and the relatively small surface area of the doors compared with the other surfaces, the door surfaces were assumed to be perfectly insulated in the present case study. The simulation started at 07:00, when the air temperature in the container was \(33\) °C. Initially, the CO\(_2\) concentration was 409.8 ppm, which is the average atmospheric CO\(_2\) concentration [13]. After 07:00, CO\(_2\) started to leak from the vaccine packages to the container at a rate of 0.0832 kg/h per package based on the experimental measurement. The supply air was blown from the T-shaped floor to the container starting at 07:00 at a rate of 0.38 m\(^3\)/s (800 CFM) and a temperature of \(37.3\) °C [14]. In this simulation, other operations, such as defrosting and opening/closing of the door, were ignored to simplify the model. Fig. 3 depicts the directions of the surfaces and the two cross-sections located at the middle of the container.

3.1. Airflow pattern

Fig. 5 depicts the vectors of airflow velocity at 08:00 and 16:00 on the cross-section \(z = 5.825\) m (Fig. 5a, b) and \(x = 1.14\) m (Fig. 5c, d).

Fig. 4. Temperature changes of three external surfaces in a refrigerated container [5,12].

Fig. 5. Vectors of air flow velocity on the cross-section \(z = 5.825\) (a, b) and \(x = 1.14\) (c, d) at 08:00 and 16:00.
which is located at the middle of the container (Fig. 3). Fig. 5 shows that some air flowed downward in the cross-section at 16:00, whereas at 08:00, the airflow was always upward. This is because the side walls of the container had been heated up at 16:00, so the air close to the side walls was heated, thereby creating an upward airflow close to the side walls. To replenish this upward airflow, a downward flow was also generated at the middle of the container. As a result, as shown in Fig. 5d, circulating flows can be observed in the cross-section at 16:00. Both airflow patterns were very simple. After the air was injected from the T-shaped floor into the container, most of the air moved to the return air grille because of the low pressure at the grille. The flow speed increased on the way to the grille since the grille area was much smaller than the total area of the supply air inlets at the T-shaped floor. Because of the high temperature of the ceiling surface, the buoyancy force along with the initial air velocity also drove the airflow from the floor to the ceiling. The airflow departed from the T-shaped floor at a low temperature, exchanged heat with the side walls and ceiling, and entered the return air grille. Because there was no short circuit for airflow in the container, the airflow pattern readily maintained the container at a low temperature.

3.2. Temperature distribution

To investigate the temperature distribution in the container and the insulating panels, five vertical lines were defined, which are line 1 to line 4 (yellow lines in Fig. 3) and the center line (red line). The center line is the cross-line of the planes $x = 1.14$ m and $z = 5.825$ m, located in the center of the container. Fig. 6 shows the temperature profiles in the container, in which Fig. 6a represents the temperature at different locations, and Fig. 6b indicates temperature profile changes with time. Fig. 6a shows that the vertical temperature profiles shared the same trend in the container. Lower than $y = 0.25$ m, the temperature was constant because when the supply air entered the container, and because of the circulating flows, air preferred to move toward the side walls, leading to a low heat transfer blow $y = 0.25$ m. From $y = 0.25$ to 0.7 m, the temperature changed rapidly, from $37.3^\circ$C to $35.9^\circ$C indicating a heat transfer at the bottom of the circulating flows. Then, the slope of temperature was fixed until the air reached locations just beneath the ceiling. Because the ceiling was heated by solar radiation, a high slope region was created. Moreover, Fig. 6a also indicates that $T_{line1} < T_{line3} < T_{line2} < T_{line4}$ at the same height when $y > 1$ m. In other words, at the same height, the closer to the return air grille, the lower the temperature. This is because the flow rate at the locations close to the return grille was higher than for the locations farther from the return air grille.

Fig. 6b illustrates the temperature change during the day inside the container along the center line. It reveals that at 08:00, after the refrigeration unit operated for 1 h, most of the container had been cooled to a temperature lower than $-36.5^\circ$C. The temperature distribution at that moment is confirmed by Fig. 7a, which depicts the temperature distribution on the two cross-sections at 08:00. Subsequently, because of the solar radiation and higher ambient temperature, the container temperature started to rise, so the container temperature at 12:00 was much higher than at 08:00. Also, a constant temperature region below $y = 0.25$ m existed only at 16:00, whereas temperature kept increasing after leaving the T-shaped floor at 08:00 and 12:00 because the circulating flows had not been developed at 08:00 and 12:00. Therefore, at 08:00 and 12:00, supply air flowed directly upward and could not generate a constant temperature region. To visualize the temperature distribution near the vaccine packages, Figs. 7d, e, and f show the temperature profiles at the cross-section $y = 0.4$ m at 08:00, 12:00, and 16:00, respectively. These profiles indicate that at 08:00, near all the packages, the temperature was lower than $-36^\circ$C. Because of the temperature of the external surfaces, the temperature increased to about $-35^\circ$C at 12:00. At 16:00, the temperature near the packages was even lower than at 12:00 because of the circulating flows.

Fig. 8 shows the temperature profiles along the center line in the insulating panel from the top of the ceiling to the external surface of the insulating panel (the red line in Fig. 2a) at 08:00, 12:00, and 16:00. It reveals that at 08:00, the temperature profile line was wavy instead of straight. At 12:00, the ceiling temperature reached its highest value, and a temperature span of more than 85°C existed between the two ends of the ceiling panel. Therefore, the PUR panels played a critical role in keeping the temperature in the container low even on a summer day.

3.3. CO$_2$ concentration

In addition to the airflow and temperature distributions within the refrigerated container, the CO$_2$ concentration was also studied because of the use of dry ice in the vaccine packages. Fig. 9a shows the CO$_2$ concentration profiles along line 1 to line 4 at 16:00. The CO$_2$ concentration difference at different heights was very small in the container (300 ppm difference). Also, below $y = 0.25$ m, the CO$_2$ concentration decreased, indicating that the container CO$_2$ concentration was lower than CO$_2$ concentration of the supply air. This decrease was because air preferred to move to the side walls of the container due to the circulating flows, leaving a lower CO$_2$ concentration region in the container below $y = 0.25$ m. Therefore, the circulating flow acted as a cap to resist heat transfer (Fig. 7c) and CO$_2$ spreading (Fig. 10c). Because of CO$_2$ accumulation, the CO$_2$ concentration increased from 08:00 to 16:00, from about 10,000 to 85,000 ppm, as shown in Fig. 9b. Because of the lack of circulating flows, at 08:00, the CO$_2$ concentration was increasing from the door to the refrigeration unit (Fig. 10a and b) following the airflow direction. Previous study shows that CO$_2$ concentration above 1000 ppm could potentially be harmful to human health [15], so even at 08:00, the container was not safe to human health. Therefore, extra fresh air ventilation is essential to ensure the safety of people entering the
Fig. 7. Temperature profiles on the cross-sections, including $x = 1.14$ and $z = 5.825$ (a, b, c), and $y = 0.4$ m (d, e, f) at 08:00, 12:00, and 16:00.

Fig. 8. Temperature profiles along the center line in the insulating panel above the ceiling at 08:00, 12:00, and 16:00.
4. Conclusions

In this research, a 3D CFD model was developed to model a refrigerated container for COVID-19 vaccine distribution. The airflow, temperature distribution, and CO$_2$ concentration were investigated in the simulation of the container. The temperature distribution was experimentally validated. The insulated wall and ceiling panels were integrated into the model to study the effects of solar radiation on the temperature within the container. The size of the container and the properties of the insulating panels were obtained from a commercially available refrigerated container. The simulations were conducted by assuming that the container was located outside during a sunny summer day, in which the container surface temperatures were obtained from measurements of an operating refrigerated container on August 27, 2013 at Hakata Port, Japan. The simulation lasted from 07:00 to 16:00. Some key conclusions from this study are as follows:

1. At 16:00, in the container, a circulating flow pattern was observed. Such a pattern did not exist at 08:00.
2. The circulating flow acted as a limiting factor to resist heat transfer in the container when $y < 0.25$ m. Therefore, a constant temperature region existed when $y < 0.25$ m at 16:00. As a result, the temperature around the vaccine package at 16:00 was lower than at 12:00. However, at 08:00 and 12:00, such a constant temperature region did not exist in the container.
3. The PUR panels played a critical role in keeping the temperature in the container low even on a summer day. As a result, the vaccine packages are cold enough in the container during the summer day from 08:00 to 16:00.
4. The circulating flow also acted as a cap to resist CO₂ spreading in the container, leading to a low CO₂ concentration region when \( y < 0.25 \) m.

5. The CO₂ concentration was about 85,000 ppm in the container at 16:00 because of the accumulation of CO₂. Even at 08:00, the CO₂ concentration was above 10,000 ppm, which is much higher than what is safe for human health. Therefore, extra fresh air ventilation is essential to ensure the safety of people entering the container for off-loading the product.

In future work, some improvements can be made to the CFD model. Air leaking will be considered in the CFD model to make it closer to reality. Moreover, some specific processes will be added to the CFD model, e.g., door opening/closing and defrosting, since such processes may have significant impacts on the temperature distribution in the container.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgement

Funding for this research was provided by the US Department of Energy, Office of Energy Efficiency and Renewable Energy. The authors would like to thank Erika Gupta, Ed Vineyard, and Samuel Petty, program managers of the Building Technologies Office, for her support of this work. The authors would like to thank Nader Awwad, Chris Repice, Douglas Ayer, and David Brisson from Carrier Global Corporation for providing some test data. The authors would like to thank the support from Anthony Gehl, Gerald Barth, Margaret Smith, and Brandy Milun of Oak Ridge National Laboratory for preparing the laboratory test. The authors would like to thank the leadership team’s support from Oak Ridge National Laboratory’s Energy Science and Technology Directorate (Xin Sun, Lonnie Love, Marti Head, Melissa Lapsa, Joe Hagerman, Ron Ott, Richard Raines, Robert Wagner, and Yarom Polsky) and Carrier Global Corporation (Bruce Hoopes, Yu Chen, Stella Oggiu, and James Fan).

References

[1] Pfizer-BioNTech, COVID-19 VACCINE U.S. DISTRIBUTION FACT SHEET, Pfizer Inc, 2020. https://www.pfizer.com/news/hot-topics/covid-19_vaccine_u_s_distrib_ution_fact_sheet.pdf.

[2] S.M. Sajadiye, M. Zolfaghari, Simulation of in-line versus staggered arrays of vented pallet boxes for assessing cooling performance of orange in cool storage, Appl. Therm. Eng. 115 (2017) 337–349, https://doi.org/10.1016/j.applthermaleng.2016.12.063.

[3] Y. Konishi, F. Tanaka, T. Uchino, D. Hamasaka, CFD prediction of the temperature distribution within a refrigerated truck filled with fruit and vegetables during transport, Trans. Japan Soc. Refrig. Air Cond. Eng. 26 (2009) 159–165, https://doi.org/10.11132/tj Jae.26.159.

[4] J. Dehghannya, M. Ngadi, C. Vigneault, Mathematical modeling procedures for airflow, heat and mass transfer during forced convection cooling of produce, A Review, Food Eng. Rev. 2 (2010) 227–243, https://doi.org/10.1016/j.s12393-010-9027-z.

[5] M.A. Budiyanto, T. Shinoda, Nasruddin, Study on the CFD simulation of refrigerated container, in: IOP Conf. Ser. Mater. Sci. Eng., Institute of Physics Publishing, 2017, p. 012042, https://doi.org/10.1088/1757-899X/257/1/012042.

[6] M. Arif Budiyanto, N. Suharyanto, Analysis of the effect of inlet velocity on cooling speed in a refrigerated container using cfd simulations, CFD Lett. 12 (2020) 55–62, https://doi.org/10.37904/cfdl.12.12.5562.

[7] T. Jiang, N. Xu, B. Luo, L. Deng, S. Wang, Q. Gao, Y. Zhang, Analysis of an internal structure for refrigerated container: improving distribution of cooling capacity, Int. J. Refrig. 113 (2020) 228–238, https://doi.org/10.1016/j.ijrefrig.2020.01.023.

[8] J. Dehghannya, M. Ngadi, C. Vigneault, Mathematical modeling procedures for airflow, heat and mass transfer during forced convection cooling of produce: A Review, Food Eng. Rev. 2 (2010) 227–243, https://doi.org/10.1016/s12393-010-9027-z.

[9] S.K. Sari, N.W. Pratami, Cooling load calculation of cold storage container for vegetables case study C Campus-USl, Ngipik, in: 2018 Int. Conf. Inf. Commun. Technol. ICOIACT 2018, 2018, pp. 820–826, https://doi.org/10.1109/ICOIACT.2018.850726.

[10] Federation of European Rigid Polyurethane Foam Associations, Thermal Insulation Materials Made of Rigid Polyurethane Foam. https://pdf4pro.com/view/thermal-insulation-materials-made-of-rigid-polyurethane-foam-376aacc.html, 2012.

[11] ANSYS, ANSYS FLUENT 17.2 Theory Guide. 2017.

[12] T. Shinoda, M. Arif Budiyanto, Energy saving effect of roof shade for reefer container in marine container terminal, J. Japan Inst. Navig. 134 (2016) 103–113, https://doi.org/10.1016/j.jinat.103.

[13] R. Rebecca Lindsay, Climate change: atmospheric carbon dioxide, Climate.Gov. (2020) 1–5, https://www.climate.gov/news-features/understanding-climate/climate-change-atmospheric-carbon-dioxide (accessed February 19, 2021).

[14] J. Sun, M. Zhang, A. Gehl, B. Fricke, K. Nawaz, K. Gluesenkamp, B. Shen, J. Munk, COVID 19 Vaccine Distribution Solution to the Last Mile Challenge: Cryogenic Refrigeration to Facilitate the Maintenance of Ultralow Temperature, ORNL, 2021, ORNL/TM-2021/1903.

[15] K. Azuma, N. Kagi, U. Yanagi, H. Osawa, Effects of low-level inhalation exposure to carbon dioxide in indoor environments: a short review on human health and psychomotor performance, Environ. Int. 121 (2018) 51–56, https://doi.org/10.1016/j.envint.2018.08.059.