Study on the CFD simulation of refrigerated container

Muhammad Arif Budiyanto¹*, Takeshi Shinoda², Nasruddin¹

¹Departement of Mechanical Engineering, Universitas Indonesia, Kampus Baru UI Depok, Jawa Barat, Indonesia.
²Departement of Marine System Engineering, Kyushu University, Motooka, Nishi-ku, Fukuoka, Japan

*Corresponding author: arif@eng.ui.ac.id

Abstract. The objective this study is to performed Computational Fluid Dynamic (CFD) simulation of refrigerated container in the container port. Refrigerated container is a thermal cargo container constructed from an insulation wall to carry kind of perishable goods. CFD simulation was carried out use cross sectional of container walls to predict surface temperatures of refrigerated container and to estimate its cooling load. The simulation model is based on the solution of the partial differential equations governing the fluid flow and heat transfer processes. The physical model of heat-transfer processes considered in this simulation are consist of solar radiation from the sun, heat conduction on the container walls, heat convection on the container surfaces and thermal radiation among the solid surfaces. The validation of simulation model was assessed uses surface temperatures at center points on each container walls obtained from the measurement experimentation in the previous study. The results shows the surface temperatures of simulation model has good agreement with the measurement data on all container walls.

1. Introduction
The measure of energy consumption in the container port is an indicator of the container port performance [1]. The breakdown of total energy consumption identify the component of energy consumption in container port and will give direction in the effort of energy saving. In the recent development of green port technology the modern container port has converted their container handlings into the electrification system, so that all of energy source comes from the power supply from the national grid. Several study of energy mapping in container port has been conducted by Tran [2], Greencranes [3], and Shinoda & Budiyanto [4]. From the statistical data refrigerated container facility contribute the most power consumption in the container port [4].

The literature of research study about refrigerated container facility were also inadequate [5]. Research study was conducted by Jolly et al [6] conclude the power consumption of refrigerated container will depend on the setting of temperature inside the cargo hold. The study conducted by Fitzgerald et al [7] assumed the mean energy consumption of refrigerated container is 2.7 kW/TEU and has values variations around 60% due to various factors. Shinoda et al [8] conducted an measurement experiment of 40 feet high cube refrigerated container consumed about 7.2 kW/h. Looking from the inadequate research study in the refrigerated container, the estimation study by CFD simulation will give opportunity for further energy analysis in container terminal operation.

In order to estimate of energy consumption of refrigeration container, the cooling load of the containers assumed as the energy to be removed from the inside cargo and equal to the power consumption of the refrigeration system. CFD simulation give the prediction of surface temperature in
each container wall which is a part in the calculation of cooling load of refrigerated container. The 
objective this study is to performed CFD simulation of refrigerated in the container port in the 
transient condition. The simulation model will predict the surface temperature of container walls in 
various condition will the cooling load can be estimate.

2. Description of the simulation model
The heat transfer process in reefer container was simulated by means of a CFD simulation model. The 
PHOENICS software packages is utilized to simulate fluid flow and heat transfer [9]. In the present 
simulations, a cross sectional of container walls in Cartesian framework is use to simulate of the 
refrigerated container in storage yard. In order to reduce the computational time required by the 
solution of the CFD model, only middle section of reefer container is considered assuming the 
symmetry condition. The set of governing equations is solved by a finite control volume method 
(FVM) and described in the following section.

2.1 Set of governing equations
The simulation model is based on the solution of the partial differential equations governing the flow 
and convective heat transfer. The governing equations in the field of fluid flow based on 
incompressibility and heat transfer process were used of the generalized transport equations:

a. Continuity equation

\[ \nabla \cdot \mathbf{v} = 0 \quad (1) \]

b. Momentum equation

\[ \rho \left( \frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v} \right) = -\nabla p + \mu \nabla^2 \mathbf{v} + f_\beta \quad (2) \]

c. Energy Equation

\[ \int_{V} \rho C_p \left( \frac{\partial T}{\partial t} + \mathbf{v} \cdot \nabla T \right) dV = \int_{V} k \nabla^2 T dV + \int_{S} (q_b + q_s) dS \quad (3) \]

Where \( t \) is the time, \( \mathbf{V} \) is the velocity vector, \( \rho \) is the density, \( p \) is the pressure, \( \mu \) is the coefficient of 
viscosity, \( k \) is thermal conductivity, \( C_p \) is specific heat capacity and \( T \) is the local temperature. Natural 
convection was modelled using the Boussinesq approximation, which uses a constant density fluid 
model, but applies a local body gravitational force throughout the fluid that is a linear function of the 
volumetric thermal expansion coefficient \( \beta \) and of the local temperature difference. The buoyancy 
source is added to the momentum equation as follows [10] :

\[ f_\beta = -\Delta \rho g = -\rho_{ref} \beta (T - T_{ref}) g \quad (4) \]

Where \( \rho_{ref} \) and \( T_{ref} \) are the density and temperature at the boundary wall condition and \( g \) is the 
gravitational force. Internal energy from thermal radiation \( (q_R) \) is taken into account at the surface of 
the objective cell in FVM within the immersed solid (IMMERSOL) per unit volume as in equation:

\[ q_R = \varphi_R \varepsilon_R \varepsilon \sigma T_R^4 - T^4 \quad (5) \]

Where, \( \varphi_R \) is the radiation shape factor from a view of radiation material, \( \varepsilon_R \) is the emissivity of 
radiation material, \( \varepsilon \) is the emissivity of the objective solid body, \( T_R \) is the radiation temperature of the 
fluid and \( \sigma \) is Stefan Boltzmann constant \( (5.67 \times 10^{-8} \text{W/m}^2\cdot\text{K}^4) \). Particularly in equation (3), when the
solid area in the model such as the container walls and road are considered, the second term of the right side of equation is neglected, and equation (3) become the equation of heat conduction except the surface of the solid body. Energy of solar radiation ($q_S$) was considered in this model, and the detail of $q_S$ is described in the next section.

A Dirichlet boundary condition at the surrounding container is used to specify a uniform inlet velocity. A no-slip boundary condition for all walls is assumed for the container wall. The inlet air temperature varies with time and the walls are assumed adiabatic. In order to close the set of governing equations, we used a $k–\varepsilon$ turbulence model.

2.2 Solar radiation and sun geometry
In the PHOENICS software packages, the energy activation from the sun utilizes SUN object to compute the intensity and direction of solar radiation heat load upon surfaces within the domain [11]. It were able to import data from weather files to obtain solar radiation at a particular date and time, in accordance to the determined also location. The solar radiation consists of the direct, diffuse and reflected radiation incident on a given surface. These radiations and the angle of incidence ($\theta$) depends upon location on the solar geometry and also the orientation of the surface. In vertical surface, the basic parameters are defined in terms of altitude angle ($\gamma$) and azimuth angle ($\alpha$) [12]. Altitude angle is described as the angle between the sun rays its projection of sun’s rays onto horizontal planes. The solar azimuth angle is the angle in the horizontal plane measured from north to the horizontal projection of the sun rays. Figure 1 shows the illustration of variation of azimuth angle from sun geometry.

![Figure 1. Sun geometry.](image)

2.3 Geometrical model and initial condition
As mentioned in section before, the domain was assumed to be symmetric with regard to perpendicular planes through the middle section of reefer container. Middle section of container is the most representative to catch heat distribution through the surface wall. The model assumed as open system simulation which consist of four regions i.e. outside air, body of container, inside air and road concrete. Within the body container composed of three material of insulation are steel, polyurethane and aluminum [13]. The total dimension of the model is 112.3m² was constructed in Cartesian framework with the total gird is 59,200 cells. The time discretization was set to transient calculation in 12 hours divided into 168 time-step start from 6:00 until 18:00. The geometrical grid and illustration model shows in figure 2. The near wall uses power mesh ratio was introduced to satisfy the calculation particularly on the thin layer region.
In transient simulations initial values should be set for all variables to get convergence criteria. Physical domain set to be air with external pressure is 1 atmosphere and temperature is 23.39 degree Celsius. The energy activation utilize solar radiation from sun as heat source, in this case the IMMERSOL radiation model will be employed as energy equation [14]. Physical characteristic related to heat transfer was applied according to properties each material i.e. thermal conductivity and solar absorption factor. Initial value of simulation parameter and physical domain shows in table 1. Besides the energy activation, momentum flow quantities was set to complied the governing equations that had generated buoyancy effect. Wind speed was applied to the model as the momentum source. The value and direction of wind speed was imported from weather data which also contained solar radiation. For this purpose the initial value of weather data was deliberately taken from experimental measurement on summer condition [15].

### Table 1. Initial condition of simulation.

| Simulation Parameter   | Properties of Air          |
|------------------------|----------------------------|
| Dimension model        | Density 1.22 (kg/m³)       |
| Grid Cells             | Velocity 0.3 (m/s)         |
| Velocity & Pressure    | Specific Heat 1.006 (kJ/kg.K) |
| Energy Equation        | Initial temperature 23.39 (°C) |
| Gravitational Force    |                            |
| Buoyancy Model         |                            |
| Turbulence Model       |                            |
| Domain Material        |                            |
| Time dependence        |                            |
| Coeff. Wall Function   |                            |
| Iterations             |                            |
|                        |                            |

| Properties of Material | Thermal Conductivity |
|------------------------|----------------------|
| Aluminium              | 204 (W/m.K)          |
| Polyurethane           | 0.003 (W/m.K)        |
| Stainless steel        | 16 (W/m.K)           |
| Road                   | 1.13 (W/m.K)         |

2.4 Validation of simulation results

Validation of experiment and simulation model is conducted to produce an accurate results for further estimation of surface temperature on container walls. In this paper, preceding experimental result by Shinoda et al [4] is compared to the simulation results under similar condition. Figure 3 shows the
schematic of measurement setup of refrigerated container. Sensor devices used in the experiment are thermocouple, power meter, velocity meter and pyranometer. Thermocouple is the temperature sensor which installed inside and outside of container surfaces. On the inside surface attached 5 sensors at the middle surfaces i.e. floor, sidewall, ceiling and the center. On the outside surface attached 15 sensors each wall surfaces at the middle point including fan and compressor surfaces. Power meter employs to measure electricity consumption during experiment. Velocity meter also employs to measure wind speed and direction were placed in around the container location. Pyranometer is to measure solar insolation on surface object, this device placed at the top position on measurement location. The measurement data from all devices is recorded every minute during the daytime on summer season in 2013.

![Figure 3. Schematic view of measurement setup [4].](image)

The simulation result was validated by experimental measurement at specific points on the middle surface of container walls. The date of measurement data used in this validation is 27th August 2013. The parameter which use for validation is the temperature data in the center surface each wall i.e. ceiling, south, north, and bottom. The accuracy of the simulation was assessed by determining root mean square error ($E_{RMS}$). The equation for $E_{RMS}$ determination can be expressed as:

$$E_{RMS} = \sqrt{\frac{\sum_{n=1}^{N} (T_E - T_P)^2}{N}}$$

Where $T_P$ is simulated temperature and $T_E$ is measured temperature, $N$ is the number of population. In order to ascertain the simulation result appertain in the confidence level, 95% confidence interval was predetermined for each validation to ascertain the validity of the simulation model.

### 3. Results and discussion

This section will discuss the thermal analysis from the CFD simulation of the refrigerated container. The CFD simulation gives evidence of the physical mechanism of the heat transfer problem and provide the parameter directions either for the improvement of its simulation model and physical condition of the reefer containers.

#### 3.1 Validation of surface temperatures

The surface temperatures of simulation model was validated using experimental data of preceding study. Figure 4 shows the validation result in all surface of container. The simulation results were in agreement with the experimental data for the all surface, no significant differences were observed between experimental and simulated. The maximum $E_{RMS}$ is 2.25 occurs in ceiling surface, this instance due to the instabilities of the convergence criteria at the beginning of the calculation. As seen from the validation graph, a narrow gap found in the start point began disappear after some step calculation. Second highest $E_{RMS}$ occur in the north surface. This due to instabilities of temperature during experiment, where some indication appear in the temperature profile at certain time on all
around container body. Wind speed from seaside to land side caused a turbulent velocity on the ground surface, as a consequence natural convection is occurred from the ground to vertical wall of container, this phenomenon consistent with the result conduct by Inan [16]. The lowest $E_{RMS}$ is resulted in bottom surface of simulation condition, the reason is reasonable since the influence of wind velocity and solar radiation is very low in this surface area. From this validation, the result of surface temperature from the CFD simulation has good agreement with measurement data.

![Temperature Distribution](image)

**Figure 4.** Validation of temperature of simulation result with experimental data.

### 3.2 Temperature distribution on the container walls

Figure 5 shows temperature distribution on the container walls from the morning until the afternoon. This figure shows the temperature distribution occur on the container walls of refrigerated container with maximum temperature is around 45 degree Celsius at 12:00. The temperature distribution start from the road transfer to the south container walls through the foot of the container. During the morning until the afternoon the accumulated heat move from south walls to ceiling walls and peak temperature occurs on this walls. During the afternoon until evening heat move from ceiling walls into north walls. The heat mainly invade on ceiling and south walls. This instance caused by a strong load of solar radiation at the highest position of the sun located at the south of the model, this phenomenon consistent with research was conducted by Rodriguez [18]. In normal condition heat form the sun will invade through the outer wall of container, thus increase the temperature on the outer surface exposed by solar radiation and propagates to all around container surface. Heat transfer process in numerical calculation start from the release of the heat energy from heat source. Heat source comes from the heat flux of sun and then transferred to the simulation model object via air. Thermal diffusion occurs when the flux has reached a surface of the material of the model and then, heat will be absorbed, transferred or reflected depend on the characteristic of the material [19]. Due to the turbulence model, the reflected radiation will increase the temperature of air layer in surrounding the material and caused natural convection effect. The absorbed heat will be transferred within the material, when the heat exceeds the thermal capacity per unit area, heat transfer arises inside of insulation material through thermal conduction that flows into the inner surface. Finally, thermal diffusion occurs in the inner surface between heat conduction from insulation with cooling air in the inside of container.
Figure 5. Temperature distribution on the container walls of refrigerated container.

3.3 Estimation of cooling load
Estimation of the cooling load of refrigerated container is defined as benchmarking of thermal performance with the following equation [20]:

\[ q = K \cdot S \cdot (T_e - T_i) \]  

(7)
Here, value of $q$ represents cooling load through the wall (Watt), $K$ represents overall heat transfer coefficient of the container wall (W/m²K), $S$ represents area of the wall (m²) and $T$ represents temperature difference between inside and outside of container. Equation 8 is represent the total of heat flow from each side i.e. ceiling ($C$), south ($S$), north ($N$) and bottom side ($B$). The duration time of the calculation is from 10:00 until 16:00.

$$q_{Total} = q_C + q_S + q_N + q_B$$

Table 2. Calculation of cooling load of refrigerated container.

| Duration time   | 10:00-16:00 |
|-----------------|-------------|
| Azimuth Angle   | (degree)    |
| Ceiling         | 334.2       |
| $T$ Ceiling     | 49.4        |
| South (Side A)  | 43.8        |
| $T$ South       | 33.1        |
| North (Side B)  | 29.9        |
| $T$ Bottom      | 2246.7      |
| Heat load       | (Watt)      |

Table 2 shows the temperature in the ceiling surface has most contribution to heat penetration, followed by south surface. From this calculation obtained the cooling load of refrigerated container is about 2.2 kW. The temperature in the wall container is the main influence of the cooling load capacity. The heat flow which appear to the inner surface is equal with balance energy which required to keep chilled condition. This condition makes cooling load increase and as consequence the energy consumption of refrigerator also increase.

3.4 Future simulation: heat island problems

Container port mostly located in the shore of urban city areas. Buildings, roads, and other infrastructure replace open land and vegetation. Surfaces that were once permeable and moist become impermeable and dry. These changes cause urban regions to become warmer than their rural surroundings, forming an "island" of higher temperatures in the landscape [21]. Looking this phenomenon the future works of this simulation model will carry out the heat island problems occurs on the container port. Figure 6 shows the illustration of simulation model in the container storage yard model. The future simulation will provide an analysis of whole phenomenon in the container port by using the current simulation model.

Figure 6. Container storage yard for the future simulation of heat island problems.
4. Conclusions
The CFD simulation of refrigerated in the container port was done performed use PHOENICS software packages. The simulation result of one refrigerated container in the transient condition for one day operation has been validate using measurement data carried out by Shinoda et al (2014). From the comparison result shows the simulation results has good agreement with the measurement data and obtain maximum root mean square error is 2.55 C in the ceiling surface. From the simulation obtained the average amount of energy to remove heat load inside cargo from the effect of heat penetration due to solar radiation during one day operation is 2.2 kW. Furthermore, this result give a direction for estimation of energy consumption in a container port and can be used to predict the energy consumption in other location.

Acknowledgments
Authors would like to express our gratitude to Directorate Research and Community Engagement (DRPM) Universitas Indonesia in the providing of publication funding by PITTA 2017. Author also would like to thanks Department of Marine System Engineering, Kyushu University and Hakata Port Terminal Corporation for providing required data on this study.

References
[1] UNCTAD. Review of Maritime Transport 2015. *Unctad* 204 (2015). doi:10.1017/CBO9781107415324.004
[2] Tran TK 2012 Study of Electrical Usage and Demand at the Container Terminal *Deakin University*
[3] GREENCRANES. *Report on Port Container Terminals Energy Profile: Mapping of Port Container Terminals Energy Profile*. Valencia; 2013.
[4] Shinoda T and Budiyanto MA 2014 Energy saving effect of roof shade at reefer container storage yard *Proceedings of the International Forum on Shipping, Ports and Airports (IFSPA): Sustainable Development in Shipping and Transport Logistics* pp 455-460.
[5] Wilmsmeier G. Energy consumption and efficiency : emerging challenges from reefer trade in South American container terminals. *Fal*. 2012;(329):1-9.
[6] Jolly PG, Tso CP, Wong YW, Ng SM. Simulation and measurement on the full-load performance of a refrigeration system in a shipping container. *Int J Refrig*. 2000;23(2):112-126. doi:10.1016/S0140-7007(99)00044-4.
[7] Fitzgerald WB, Howitt OJA, Smith JJ, Hume A. Energy use of integral refrigerated containers in maritime transportation. *Energy Policy*. 2011;39(4):1885-1896. doi:10.1016/j.enpol.2010.12.015.
[8] Shinoda T and Budiyanto MA. Energy Saving Effect of Roof Shade for Reefer Container in Marine Container Terminal *The Journal of Japan Institute of Navigation*. 2016;134: 103-113.
[9] CHAM. PHOENICS-VR Reference Guide. In: Documentat. Concentration, Heat and Momentum Limited (CHAM); 2010. http://www.cham.co.uk.
[10] Sanchez E, Rolando A, Sant R, Ayuso L. Influence of natural ventilation due to buoyancy and heat transfer in the energy efficiency of a double skin facade building. *Energy Sustain Dev*. 2016;33:139-148. doi:10.1016/j.esd.2016.02.002.
[11] CHAM. PHOENICS Numerics;Modelling Capabilities. Concentration Heat and Momentum Limited (CHAM) 2015. http://www.cham.co.uk/products/modellingcapabilities.php. Published 2015. Accessed July 19, 2016.
[12] Bird RE. A simple, solar spectral model for direct-normal and diffuse horizontal irradiance. *Sol Energy*. 1984;32(4):461-471. doi:10.1016/0038-092X(84)90260-3.
[13] ISO 668. *International Standard*. Vol 2013. Switzerland; 2013. www.iso.org.
[14] Spalding B. *Trends, Tricks, and Try-Ons in CFD/CHT*. Vol 45. 1st ed. Elsevier Inc.; 2013. doi:10.1016/B978-0-12-407819-2.00001-3.
[15] Port H. *Outline of Hakata Port International Container Terminal*.; 2015.
[16] İnan T, Başaran T, Ezan MA. Experimental and numerical investigation of natural convection in
a double skin facade. Appl Therm Eng. 2016;106:1225-1235. doi:10.1016/j.applthermaleng.2016.06.124.

[17] Liu H, Chen Z, Chen B, Xiao X, Wang X. Studies on the temperature distribution of steel plates with different paints under solar radiation. Appl Therm Eng. 2014;71(1):342-354. doi:10.1016/j.applthermaleng.2014.06.031.

[18] Rodriguez-Bermejo J, Barreiro P, Robla JJ, Ruiz-García L. Thermal study of a transport container. J Food Eng. 2007;80(2):517-527. doi:10.1016/j.jfoodeng.2006.06.010.

[19] Çengel YA. Heat Transfer: A Practical Approach 2nd Edition. Mcgraw-Hill (Tx); 2nd edition (November 1, 2002); 2002. doi:10.1017/CBO9780511676420.004.

[20] Tanaka S, Takeda H, Iwata T, Takashi T, Michijin T. Architectural Environment Engineering. Inoue Shoin; 2006.

[21] Nuruzzaman MD. Urban Heat Island: Causes, Effects and Mitigation Measures - A Review. International Journal of Environmental Monitoring and Analysis. 2015;3(2):67-73