Simulation and validation of Airflow Past Hand-Shape Mold Using OpenFOAM

K Loksupapaiboon¹,² and C Suvanjumrat¹,²,*

¹Department of Mechanical Engineering, Faculty of Engineering, Mahidol University, Nakhon Pathom, 73170, Thailand
²Laboratory of Computer Mechanics for Design (LCMD), Department of Mechanical Engineering, Faculty of Engineering, Mahidol University, Nakhon Pathom, 73170, Thailand

* Corresponding Author: chakrit.suv@mahidol.ac.th

Abstract
This paper aims to present a wind tunnel, aiming to examine the effect of airflow past the hand-shape mold especially airflow around the gap between the fingers of the mold. Part of the aim of this project is to investigate the behavior of airflow past the mold on the different angles, which was valuable to vulcanize rubber glove in the oven. The wind tunnel experiments were performed on the various angle of attack of hand-shape mold from 0 to 180 degrees under the airflow velocity of 5 to 20 m/s. The 3D simulations were carried out by using open source code software, OpenFOAM. The k-ε model was used to simulate the turbulent flow past hand-shape mold. The pressure-velocity coupling problem and the convection-diffusion term were solved by using a SIMPLE algorithm and upwind differencing scheme, respectively. The drag force by airflow on the rubber glove mold which obtained by the computational fluid dynamics (henceforth CFD) method was compared with the experimental data. The comparison between CFD simulation and the experimental data showed a fairly close agreement and the average error was less than 13.96%. Further research in this field would be of great help in developing a model to optimize the mold installation inside the rubber glove oven.

1. Introduction
Drying process is widely used to release moisture of food products, color or lacquer coating and rubber product. In the production of rubber glove, the baking is an important process for drying the liquid that coated on the hand-shape mold surface. This liquid consists of cleaning water, coagulant and compound rubber. The convection oven is the most favorite equipment used for drying liquid. One major issue found about the internal structure of the convection oven and the shape of the hand-shape mold is the wetness of the coating liquid on the mold surface remains. The wetness is associated with increased risk of a non-uniformity of air velocity around the gap between the fingers of hand-shape mold. This could be a contributing factor to prevent waste from the drying process. Scholars have long debated the impact of the uniformity of velocity and temperature of the air inside the oven, which
could affect the dryness of the products. Smolka et al. [1] studied the uniformity of airflow and thermal parameters inside the drying oven. These parameters could indicate the efficiency and homogeneity of the product being dried. The controlling of these parameters was very difficult in experimentation. The simulation methods were used to study and analyze the airflow instead. The CFD was employed to predict the airflow and thermal distribution that can reduce time and cost from trial and error. The simulation results also suggest that the uniformity around the products could increase the drying quality. Most of CFD software is commercial such as ANSYS-FLUENT, FLOW-3D, PHEONICS and COMSOL. This software has an expensive license cost. OpenFOAM (henceforth Open Source Field Operation and Manipulation) is one of open-source CFD software, which can be applied and developed codes for solving a fluid problem by using C++ language. It is used under GNU GPL (General Public License) without license cost [2]. The fluid flow problem together with heat transfer and turbulence flow behavior could be solved by OpenFOAM software [3-6].

The Reynolds-Averaged Navier-Stokes (henceforth RANS) equations are the set of average equations of motion over the time for the fluid flow that is primarily equations to describe the turbulent flow. RANS equations are so-called Reynolds stress, which is used to describe eddy effects, which is difficult to calculate. Many models were developed to calculate the Reynolds stress term and apply for the bluff body flow. The lift and drag force are important parameters from the CFD simulation for validating the turbulence model [7]. Kurec et al. [8] studied the airflow past a car with a rear wing, causing the down force on the car body. Four turbulence models were used to simulate the airflow. The force components that obtained from simulation were compared with experimental data. The results showed that the r-k-$\varepsilon$ and k-$k_l$-$\omega$ turbulence models gave the force component and were closer to the experiment. The k-$\varepsilon$ turbulence model was chosen to evaluate the effectiveness of airflow past complex structure i.e. the cyclist following by motorcycle.

The CFD results showed that the motorcycle following the cyclist decreased the drag force which was unwanted for aerodynamics benefits [9]. Suvanjumrat [10] has demonstrated that the lift and drag coefficient can be used to compare turbulence models for the flow past NACA0015. It appears that the Menter SST k-$\omega$ model with the SIMPLE algorithm and LUD scheme was the suitable model. This study also highlighted that k-$\varepsilon$ turbulence model can be applied to predict the airflow past hand-shape mold by using OpenFOAM software. The viscous flow was solved by using Semi-Implicit Method for Pressure Linked Equation (SIMPLE) algorithm and upwind differencing scheme. To validate the simulation model, the wind tunnel experiment is carried out. The results also showed that optimized angle can cause the good airflow past gap between fingers of hand-shape mold. There was an increase in the drying efficiency of the rubber glove. It is suggested that the CFD technique will be used to analyze the arrangement of hand-shape mold inside the batch type oven to avoid the blockage between molds.

2. Simulation Methods

2.1 Governing Equations

The governing equations of fluid dynamics consist of three fundamental physical principles: conservative of mass, momentum and energy. These equations are solved together to describe fluid flow behaviour. The generic form of the conservative equation for turbulent flow and incompressible fluid is as follows:

$$\frac{\partial \bar{\phi}}{\partial t} + \text{div}(\bar{\phi}U) = \frac{1}{\rho} \text{div} \left( \Gamma \text{grad} \left( \bar{\phi} \right) \right) - \frac{1}{\rho} \text{div} \left( \rho \bar{u'} \bar{\phi'} \right) + S_{\bar{\phi}} \quad (1)$$

where $\bar{\phi}$ is the mean general variable of flow properties, $\rho$ is the fluid density, $\Gamma$ is the diffusion coefficient, $\rho \bar{u'} \bar{\phi'}$ is the Reynolds stress term and $S_{\bar{\phi}}$ is the source term.
The Reynolds stress term was based on the fluctuation of turbulent flow. The calculation of Reynolds stress term requires extra equations or turbulence models which are derived from the mathematic model.

2.2 Turbulence Model

2.3 To identify the concept of the turbulence model, it is assumed that the Reynolds stress related to the eddy viscosity and the mean velocity gradient is known as eddy viscosity model, as can be seen from the following equation (2).

\[ \tau_{ij} = -\rho \bar{u}_i \bar{u}_j = \mu_t \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \rho k \]  

(2)

where \( \mu_t \) is the turbulent viscosity, \( \delta_{ij} \) is the Kronecker delta and \( k \) is the turbulence kinetic energy. The turbulent viscosity is written by:

\[ \mu_t = \rho C_{\mu} \frac{k^2}{\epsilon} \]  

(3)

where \( C_\mu \) is the dimensionless constant and \( \epsilon \) is the rate of dissipation of turbulence energy.

The standard k-\( \epsilon \) turbulence model is widely used to predict turbulent flow conditions. This model is the two equations model which consist of two extra transport equations, \( k \) and \( \epsilon \) equations [11].

\[ \frac{\partial \rho k}{\partial t} + \text{div} (\rho k \bar{U}) = \text{div} \left( \frac{\mu_t}{\sigma_k} \text{grad} (k) \right) + 2 \mu_t S_{ij} \cdot S_{ij} - \rho \epsilon \]  

(4)

\[ \frac{\partial \rho \epsilon}{\partial t} + \text{div} (\rho \epsilon \bar{U}) = \text{div} \left( \frac{\mu_t}{\sigma_\epsilon} \text{grad} (\epsilon) \right) + C_1 \frac{\epsilon}{k} 2 \mu_t S_{ij} \cdot S_{ij} - C_2 \rho \frac{\epsilon^2}{k} \]  

(5)

where \( S_{ij} \) is the velocity gradient yield which can be shown as follows:

\[ S_{ij} = \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \]  

(6)

The constant variables for the standard k-\( \epsilon \) turbulence model comprise \( C_\mu = 0.09, \sigma_k = 1.00, \) \( \sigma_\epsilon = 1.30, C_1 = 1.44, \) and \( C_2 = 1.92. \)

3. Experiment

The wind tunnel experiment, model WT300, was used to study the airflow past hand-shape mold. The structure and the equipment of the wind tunnel experiment are set out in Figure 1. The wind tunnel was installed the 2-axis load cell under the testing section. The hand-shape mold was connected to the 2-axis load cell inside the testing section. It was designed to adjust the angle of attack from 0 to 360 degree. The first angle of attack or 0 degree starts from the flow direction parallel to the hand-shape mold. The thumb side of mold was assigned to be attacked by the inlet flow. The angle of attack for the experiment consisted of 0, 45, 90, 135 and 180 degrees. Regarding the suction side of the wind tunnel, the grille was installed to decrease the turbulent behavior. The axial fan and the inverter were
used to control airflow velocity. The airflow velocity of the experiment including 5, 10, 15 and 20 m/s was prepared. The Reynolds numbers 38,265, 76,530, 114,795 and 153,061 were also chosen in the experiment. The airflow velocity was measured by pitot tube anemometer at the front of the mold. The drag force due to the airflow past hand-shape mold was measured by the 2-axis load cell under the steady-state condition. The sampling rate to record the drag force was 1 Hz along with the time of five minutes by using the data acquisition, MicroStrain SG-Link.

4. CFD model
The simulation of the airflow past hand-shape was considered to be a steady flow; therefore, the transient term was eliminated. The open-source code CFD software, OpenFOAM, was used to simulate the airflow.

The computational domain was created by the 3D model. It has the total length of 13H, the width of 10W and the height of 5W, where H and W are height and width of the hand-shape mold, respectively (Figure 3). In this study, the height and width of the hand-shape mold is equal to 250 and 120 mm, respectively. The 3D unstructured cell was used to divide the computational domain for the discretization. The finest cells are created on the mold surface, as shown in Figure 4. The best finest cell size which affected the number of total cells was carried out by a grid independence test. The drag force by airflow was used to arbitrate the convergence result. The range of the finest cell size varies from 1.0 to 1.5 mm with the increments of 0.1 mm. No significant differences were found between the grids independence test and the convergence. Furthermore, the computer that used in this research was out of memory when the finest cell size was less than 1.0 mm. Interestingly, the computer used in this research was out of memory when the finest cell size was less than 1.0 mm. However, the y+ of 2.31 was in the viscous sub-layer. These results suggest that it can be used to simulate the airflow past hand-shape mold in all cases with wall functions.
The boundary conditions were assigned to the domain including inlet, outlet and wall. The inlet was assigned to be the uniform flow according to the experiment while the outlet was assigned to be an ambient pressure. The outer surface of the domain and the surface of the hand-shape mold were assigned by no-slip wall condition. The angle between the hand-shape mold and X-Y plane (flow direction) could be adjusted according to the experiment. These simulation cases conducted a uniform inlet velocity according to the physical experiment composing 5, 10, 15 and 20 m/s. The angle of attack was also conducted on five angles including 0, 45, 90, 135 and 180 degrees. Further analysis showed that the drag force was investigated by 20 simulating cases. All CFD cases, the standard k-ε model governed the turbulent flow. However, the flow simulation with turbulence model was valid only in a fully developed turbulent flow which was far from the wall. In the near-wall region, the turbulence model could not be calculated. Therefore, two methods could be used to resolve this problem. The first method was created the first cell center keeping on the viscous sublayer ($y^+ \leq 1$). While the second one was wall functions which modeled the turbulent flow on the near-wall region. In this study, the wall functions were used to calculate the effect on the near-wall instead to generate the first cell center for making the $y^+$ equal or less than 1. The pressure-velocity couple problem and convective term of the governing equations were solved by the SIMPLE algorithm and upwind differential scheme, respectively [12]. The air which flows through the domain was considered to be an incompressible fluid with the density and dynamics viscosity of $1.225 \text{ kg/m}^3$ and $1.838 \times 10^{-5} \text{ kg/(m-s)}$, respectively.
5. Results and Discussion
The purpose of the study is to investigate the simulation of the airflow past hand-shape mold which is based on the CFD model. The results gained can be compared with the experimental data. The drag force countered the airflow was calculated by the pressure on the hand-shape mold. At a various angle of attack, it was compared with the measuring data by the 2-axis load cell. As can be seen in Figure 5, it shows the comparison of the graph of drag force on the various angle of attack when the airflow velocity increases. These results suggest that the more the drag force increases, the wider the angle of attack can be extended from 0 degrees to 90 degrees. Then, the drag force can be decreased when the angle of attack is increased to 180 degrees. The results of this study indicate that the CFD show significant difference than the experiment at the angle of attack of 0 and 180 degrees. It is exceptional for other angles since they show a significant difference than the CFD results. This behavior was caused by the structure of the cells of hand-shape mold on the trump side and little finger side were quite complex. These areas past by airflow at the angle of attack of 0 and 180 degrees. This problem can be resolved by decreasing \( y^+ \) on these areas. The comparison of drag force between the CFD and experiment had an average error of 10.73, 15.29, 14.54 and 15.29% along with the airflow velocity of 5, 10, 15 and 20 m/s, respectively. The results were satisfied with the accuracy of the CFD model.

At the angle of attack of 0 degrees, the inlet flow of 20 m/s past the hand-shape mold can be illustrated by color contour (see Figure 6 below). The airflow was not past through the finger gap of hand-shape mold. The blue color contour was appeared. Figure 7 shows the velocity contour of airflow past the finger gap of hand-shape mold at the angle of attack of 90 degrees. This study indicate that the airflow past the finger gap of hand-shape mold very well. In this regards, the airflow at a different angle of attack could be different from the airflow behavior around the gap between the finger. All in all, the suitable alignment of hand-shape mold inside the batch type drying oven should equal to 90 degrees for the most airflow past the gaps between the finger of the hand-shape mold.

6. Conclusion
In this investigation, the aim was to study the steady flow simulation of the airflow past a complex hand-shape mold. The study was carried out by using the open-source CFD software, OpenFOAM. The simulation results were compared with experimental data. These findings suggest that in general, the CFD model had good agreement with experiment. The average error of the CFD model was less than 13.96%. These findings have significant implications for the understanding of how the airflow behavior between the gap of the finger of hand-shape mold was different in the different angle of attack. The present study provides that the suitable angle of attacks was 90 degrees so that the open source software can be used to analyze the suitable alignment of hand-shape mold inside the batch type oven. This would be a fruitful area for further work.
Figure 5. The drag force comparison between CFD and experiment at various angle of attack and velocity.

Figure 6. The velocity contour of airflow past hand-shape mold at the angle of attack of 0 degrees by the inlet velocity of 20 m/s.

Figure 7. The velocity contour on X-Z plane of airflow past hand-shape mold at the angle of attack of 90 degrees by the inlet velocity of 20 m/s.
Acknowledgments
This research was granted by The Thailand Research Fund (TRF) and W.A. Rubberrmate Co., Ltd under Research and Researchers for Industry Program, Grant No. PHD61I0003.

References
[1] Smolka J Nowak A J and Rybarz D 2010 J. Food Eng. 97(3) 373
[2] Greensheilds C J 2019 OpenFOAM The OpenFOAM Foundation User Guide version 7 (UK: OpenFOAM Foundation Ltd.)
[3] Flores F Garreaud R and Munoz R 2013 Comput. Fluids 82 1
[4] Prasad B Hino T and Suzuki K 2015 Ocean Eng. 102 87
[5] Suvanjumrat C 2017 Engineering Journal 21(5) 225
[6] Zang B Vevek U S Lim H D and New T H 2018 J. Comput. Sci. 28 18
[7] Roberson E Choudhury V Bhushan S and Walters D K 2015 Comput. Fluids 123 122
[8] Kurec K Remer M Mayer T Tudruj S and Piechna J 2019 Int. J. Mech. Sci. 152 384
[9] Blocken B Toparlar Y and Andrianne T 2016 J. Wind Eng. Ind. Aerod. 155 1
[10] Suvanjumrat C 2017 Engineering Journal 21(3) 207
[11] Kaunder B E and Spalding D B 1973 Comput. Method Appl. M. 3 269
[12] Versteeg H K and Malalasekera 2007 An Introduction to Computational Fluid Dynamics The finite Volume Method 2nd edition (London: Pearson Education Ltd.,)