Air Flow Visualization for Fresh Produce Packaging by CFD Analysis

Hiroaki Kitazawa1*, Tatsuya Funaki2, Mitsuhiro Nakao3, Yasuhiko Ohshiro4, Masahiro Hiruta5 and Yutaka Ishikawa1

1 Food Engineering Division, National Food Research Institute, NARO, 2-1-12 Kan-nondai, Tsukuba, Ibaraki 305-8642, Japan
2 National Metrology Institute of Japan, AIST, 1-1-1 Umezono, Tsukuba, Ibaraki 305-8563, Japan
3 Graduate School of Science and Engineering, Kagoshima University, 1-21-24 Korimoto, Kagoshima, Kagoshima 890-8580, Japan
4 Ibaraki Prefectural Government, 978-6 Kasahara-cho, Mito, Ibaraki 310-8555, Japan
5 Industrial Technology Institute of Ibaraki Prefecture, 3781-1 Nagaoka, Ibaraki-machi, Ibaraki 311-3195, Japan

Received December 26, 2011; Accepted March 23, 2012

The purpose of this study was to clarify airflow in packaging containers with ventilation for fruit and vegetables and propose a favorable package shape for efficient ventilation. Here we investigated the influences of differences in the inlet air velocity through the ventilation port and diameter of the port on the air velocity distribution in a 1-layer packaging system container for strawberries employing computational fluid dynamic (CFD) analysis. It was suggested that the air velocity increases with a rise in the inlet air velocity or ventilation port size, i.e., the volumetric flow rate, but there were regions in which airflow could be hardly generated under any inlet condition. The results suggest that elimination of these regions is a key factor in developing a packaging container with a favorable shape for ventilation efficiency.

Keywords: air flow, computational fluid dynamics (CFD), packaging

Introduction

It is generally desirable to cool fruit and vegetables as soon as possible after harvest to prevent quality degradation (i.e., reductions of the quality of product appearance and hardness caused by physical damage) during distribution (Robbins and Moore, 1992; Maezawa and Akimoto, 1995). Pre-cooling of fruit and vegetables aims to achieve this, but it is sometimes performed for fruit and vegetables packed in packaging containers, for which the packaging container is required to have a function to maintain a homogenous airflow in the container. Preparation of containers with various shapes and repeating tests to identify the optimum shape meeting these conditions requires much time and labor. Moreover, it is difficult to measure the airflow velocity in a closed, narrow space.

Computational fluid dynamic (CFD) is a method to analyze fluid behavior in a space. CFD has been used in various fields including meteorology (Flaherty et al., 2006; Hanna et al., 2006), architecture (Awbi, 1989; Tominaga et al., 2002; Ota and Kondo, 2009), automobile engineering (Takagi, 1990; Fu et al., 2009), and biophysics (Kozu et al., 2010). It has recently been increasingly used for simulation of the air velocity and temperature distributions in greenhouses (Kacira et al., 1998; Kim et al., 2007) in the agriculture field and applications in the food processing field (Scott and Richardson, 1999, Hu and Sun, 2000; Boulet et al., 2010; Mondal and Datta, 2010), and the analysis is highly consistent with actual measurement. Applying this method to spatial airflow analysis in packaging containers for fruit and vegetables, efficiencies of ventilation and pre-cooling in packaging containers may be improved without having to prepare physical prototypes. Hence, there have been several reports on CFD analysis of changes in the airflow and/or temperature in packaging containers for fruits such as apples (Zou et al., 2006; Zou and Opera, 2007) and strawberries (Ferrua and Singh, 2009a; Ferrua and Singh, 2009b; Ferrua and Singh, 2009c; Ferrua and Singh, 2009d; Ferrua and Singh, 2011).

The aim of this study is to obtain basic information to improve ventilation efficiency in packaging containers for fresh produce. Here we investigated the influences of the air velocity and ventilation port diameter on airflow in a space...
simulating the inner shape of a packaging container for strawberries by simulation employing CFD analysis.

Materials and Methods

**Box shape for CFD analysis**  The analyzed packaging for strawberries was a corrugated carton (inner size: \(210 \times 290 \times 45.25\) mm) with a ventilation port (approx. \(15 \sim 16\) mm in diameter, \(d\)) in the center of each long side containing 2 trays packed with strawberries; this carton is actually used in the market (Fig. 1). Inside this packaging, the inner space is mostly divided into upper and lower regions because the margins of the 2 trays touch the inner wall of the box. A 3-dimensional shape simulating this space was prepared using a flow analysis preprocessor (GAMBIT 2.4.6, ANSYS Inc., PA, USA). Paying attention to the symmetric shape of the container, we applied the symmetry scheme to reduce the calculation load in the calculation described below and shorten the calculation time, and prepared a shape of the container cut into halves (Fig. 2). The \(d\) of the ventilation port was set at 13, 15, 17, 19, and 21 mm. The total number of meshes of each prepared shape was approx. 70000.

<table>
<thead>
<tr>
<th>Parts</th>
<th>Mesh size (mm)</th>
<th>Mesh type</th>
<th>Source</th>
<th>Attachment</th>
<th>Start size (mm)</th>
<th>Growth rate (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A internal space</td>
<td>8.0</td>
<td>Tet/Hybrid-TGrid</td>
<td>C</td>
<td>A</td>
<td>1.5</td>
<td>20</td>
</tr>
<tr>
<td>B internal space</td>
<td>8.0</td>
<td>Tet/Hybrid-TGrid</td>
<td>C</td>
<td>B</td>
<td>1.5</td>
<td>20</td>
</tr>
<tr>
<td>C internal space</td>
<td>1.5</td>
<td>Tet/Hybrid-TGrid</td>
<td>E</td>
<td>A</td>
<td>1.5</td>
<td>20</td>
</tr>
<tr>
<td>D inlet port</td>
<td>1.5</td>
<td>Tet/Hybrid-TGrid</td>
<td>E</td>
<td>B</td>
<td>1.5</td>
<td>20</td>
</tr>
<tr>
<td>E internal space</td>
<td>1.5</td>
<td>Tet/Hybrid-TGrid</td>
<td>F</td>
<td>A</td>
<td>1.5</td>
<td>20</td>
</tr>
<tr>
<td>F internal space</td>
<td>1.5</td>
<td>Tet/Hybrid-TGrid</td>
<td>F</td>
<td>B</td>
<td>1.5</td>
<td>20</td>
</tr>
<tr>
<td>G outlet port</td>
<td>1.5</td>
<td>Tet/Hybrid-TGrid</td>
<td>H</td>
<td>A</td>
<td>1.5</td>
<td>20</td>
</tr>
<tr>
<td>H internal space</td>
<td>1.5</td>
<td>Tet/Hybrid-TGrid</td>
<td>H</td>
<td>B</td>
<td>1.5</td>
<td>20</td>
</tr>
</tbody>
</table>

**Fig. 1.** Tested packaging for CFD analysis.
Top: outside, Bottom: inside. In actual markets, the diameter \((d)\) of ventilation port is approx. \(15 \sim 16\) mm.

**Fig. 2.** Box shape and structure, meshing scheme for CFD analysis.
Tet/Hybrid-TGrid means that meshes consist only of tetrahedral elements. Boundary conditions of all spaces and walls are fluid and no slip wall, respectively.
Analysis condition for CFD  Air was flowed into the container through the ventilation port on one side, and its flow in the container was reproduced using CFD analysis software (ANSYS Fluent 6.3.26, PA, USA). Zou et al. (2006) set the inlet velocity ($U$) at 0.9 m/s in CFD analysis of forced ventilation in a packaging container for apples. Referring to their report, we set $U$ at 0.3, 0.6, 0.9, 1.2, and 1.5 m/s. For the physical properties of air, the values at 15°C were adopted (viscosity ($\mu$): $1.7894 \times 10^{-5}$ Pa·s, density ($\rho$): 1.225 kg/m$^3$).

The Reynolds number ($Re$) in the flow field in the container can be presented by the equation below:

$$Re = U \cdot d \cdot (\mu/\rho)^{-1}$$  Eq. 1

The Reynolds number at the inlet port determined by Eq. 1 reached the maximum value, about 2150, when $U$ was 1.5 m/s and $d$ was 21 mm. The air flowed in through the inlet port is expected to form a turbulent field in a very limited region, but laminar flow, which is more simple for analysis, was assumed because the principle objective was to investigate the overall ventilation state in the container. In the simulation, steady state calculation was performed, and when the residual of numeric calculation concerning the air velocity was less than $10^{-3}$, the calculation was judged as converged.

For calculation, a personal computer (OPTIPLEX 740, Dell Japan, Kawasaki, Japan, CPU: AMD Athlon 64 X2 4000+ Dual core processor, RAM: 1.93 GB) was used.

Analysis items The influences of $U$ and $d$ on ventilation in the container were analyzed with regard to the air velocity distribution and flow direction, and the air velocity distributions at Lines 1-9 shown in Fig. 3 were calculated and organized. In addition, a dimensionless number, $R$, which is the ratio of the air velocity at each position ($U'$) to the inlet air velocity ($U$), was calculated using the equation below:

$$R = U'/U$$  Eq. 2

The volumetric flow rate, $Q$, of inlet airflow into a packaging container with ventilation can be presented by the equation below:

$$Q = A \cdot U$$  Eq. 3

**Fig. 3.** Position for calculating air velocity distribution.

Lines 1 and 4 exist on the symmetry boundary.
To investigate the volumetric flow rate ($Q$)-based ventilation efficiency using Eq. 3, the relationship with the mean $R$ at Lines 1–9 was calculated.

Furthermore, to investigate lateral diffusion of air flowed in through the ventilation port in the packaging container, $U'$ near the center of Line 1 (102.5 mm from the ventilation port) was divided into components in the X and Y directions, and the X velocity-to-Y velocity ratio representing the diffusion at this site was calculated.

**Results and Discussion**

With respect to vertical airflow on the symmetrical boundary in the analysis model, air flowed in through the inlet port and slowly diffused in the vertical direction in the container under all conditions (Fig. 4). There was a region in which no flow was generated near the wall on the inlet side. Figs. 5 and 6 depict the influences of $U$ and $d$ on overall flow in the entire space in the container. At $U$ of 0.3 m/s, airflow was limited near the symmetric axis at all values of $d$. At $U$
Visualization of Air Flow in Packaging

Fig. 6. Contour of air velocity distribution within tested packaging.
Left: upper layer, Right: bottom layer.
Figure shows the conditions of $U$ of 0.3, 0.9, and 1.5 m/s, and $d$ of 13, 17, and 21 mm.

However, there were regions with hardly flow generation near the end and center of walls on the ventilation and outlet port sides in the upper space under all analytical conditions.

The air velocity distribution ($U'$) on Lines 1 − 9 increased as $U$ increased at most positions (Fig. 7), but the flow decreased as $U$ increased at about 150 mm from the inlet port side on Line 2. This region corresponded to the position with little flow generation described above (Fig. 6). For this reason, it was thought that the flow near the region is obstructed by the air flow along the wall, but the details remain to be investigated. With respect the relationship between $R$ and $U$, a tendency similar to that in Fig. 7 was noted in most regions (Fig. 8), but on Lines 1, 4, 5, 7, 8, and 9 at $U$ of 0.3 m/s, marked reduction of $R$ was noted at a distance of 50 mm or

of 0.6 m/s, the flow slightly strengthened in the proximity, but the tendency was similar to that at $U$ of 0.3 m/s. At $U$ of 0.9, 1.2, and 1.5 m/s, air flowed in through the ventilation port firstly reached near the wall on the outlet port side and was partially emitted. On the other hand, air not emitted was generating a flow circulating along the wall and was merged into the flow near the center. This tendency became marked as $d$ increased. At $U$ of 0.9 m/s, a marked flow that makes a circuit in a space was generated when using a ventilation port with $d$ of 21 mm or greater for the upper space and of 17 mm or greater in the lower space. At $U$ of 1.2 m/s, $d$ was 15 mm or greater, and at $U$ of 1.5 m/s, flow making a circuit in the container was noted in both the upper and lower spaces at all values of $d$. 
Distance from inlet side wall (Lines 1 − 5) or symmetry boundary (Lines 6 − 9) (mm)

**Fig. 7.** The Effects of the differences of inlet velocity ($U$) and diameter of ventilation port ($d$) on the local velocity ($U'$).

Figure shows the conditions of $U$ of 0.3, 0.9, and 1.5 m/s, and $d$ of 13, 17, and 21 mm.
Fig. 8. The Effects of the differences of inlet velocity ($U$) and diameter of ventilation port ($d$) on the velocity-to-inlet velocity ratio ($R$). Figure shows the conditions of $U$ of 0.3, 0.9, and 1.5 m/s, and $d$ of 13, 17, and 21 mm.
more from the inlet port side. It was thought that $U'$ on these lines has a large tendency depending on $U$. For developing a packaging container, therefore, some thought should be given concerning the ventilation layout in order to avoid $R$ decreasing on the lines even if $U$ would be weak.

With respect to the air velocity distribution, the relationship between the mean $R$ and $Q$ was analyzed at various positions in the container using Eq. 2. On Lines 3, 5, and 7–9, the mean $R$ was affected greatly by $Q$ (Fig. 9), followed by those on Lines 1, 2, and 4. On Lines 1 and 4, dependence on $d$ was also marked, in addition to that on $Q$. The X velocity-to-Y velocity ratio near the center of Line 1 ranged between 0.02 and 0.08, indicating that most inlet air widely diffused in the lateral direction before reaching the outlet port side (Fig. 10). The lateral diffusion also did not decrease even when $U$ was increased. In contrast, at $d = 15 - 21$ mm, the ratio decreased as $U$ increased, suggesting that the lateral diffusion of air decreases as $U$ increases. These may have been presented as the above marked dependence of the mean $R$ on $d$ on Lines 1 and 4. On Line 6, the effect of $Q$ on the mean $R$ was weaker than those in the other regions, indicating that the ventilation condition may not be improved by increasing $Q$ in this region.

In the analyzed packaging, ventilation was heterogeneous, and it is likely that homogenous ventilation cannot be achieved by increasing $Q$ through the ventilation port. Elimination of the regions with hardly flow generation identified by this analysis is a short cut to improve ventilation in the

![Fig. 9. The relationship between volumetric flow rate ($Q$) and velocity-to-inlet velocity ratio ($R$) at each position.](image-url)
container. Based on these findings, we will analyze the influences of the position, size, and number of ventilation ports on airflow in the space and propose a modified packaging container with increased ventilation efficiency (i.e., cooling efficiency).

Acknowledgements The submission of GAMBIT 2.4.6 and ANSYS Fluent 6.3.26 was performed with the assistance of the Agriculture, Forestry and Fisheries Research Information Technology Center for Agriculture, Forestry and Fisheries Research, MAFF, Japan.

Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Cross-sectional area of ventilation port, m²</td>
</tr>
<tr>
<td>d</td>
<td>Diameter of ventilation port, m</td>
</tr>
<tr>
<td>μ</td>
<td>Viscosity, Pa·s</td>
</tr>
<tr>
<td>ρ</td>
<td>Density, kg/m³</td>
</tr>
<tr>
<td>Q</td>
<td>Volumetric flow rate, m³/s</td>
</tr>
<tr>
<td>R</td>
<td>Velocity-to-inlet velocity ratio</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
<tr>
<td>U</td>
<td>Inlet velocity, m/s</td>
</tr>
<tr>
<td>U'</td>
<td>Local velocity, m/s</td>
</tr>
</tbody>
</table>

References


