Convective flow and heat transfer in the human respiratory system: 
A computational fluid dynamics approach 
CFD 解析による気道内流れ場と対流熱伝達予測

Nguyen Lu Phuong and Kazuhide Ito
IGSES, Kyushu University

Steady airway inhalation has been computed inside a human respiratory model using computational fluid dynamics (CFD). The flow patterns were yielded through visualization of planes in complex flow areas that depicted formations of turbulent flow primarily from the nasal cavity to trachea. In this study, the convective heat transfer coefficients under a wide range of inspiratory flow rates were calculated and validated with values in the literature.

1. Introduction

The controlled objectives for indoor environments have been expanded from the macrospace of the entire room to the microenvironment of the local domain, such as the occupied zone, the space around the human body, the breathing zone, and the respiratory system. An integrated numerical procedure using computational fluid dynamics (CFD), a multi-nesting method, has been proposed by Ito and Asanuma [1] for predicting infectious disease transmission in a hospital space from a building scale to the respiratory tract. Furthermore, a fundamental study of a human respiratory model has been described by Phuong and Ito [2].

This paper reports on the numerical prediction of airflow and temperature distribution in the human respiratory system as a function of the breathing rate. In addition, convective heat transfer phenomena in the airway model are discussed by estimating the convective heat transfer coefficients (CHT Cs) of each human airway segment under various breathing conditions, by CFD. The database of CHTCs in the human respiratory system will contribute to the development and improvement of the human thermal response model. Simulation results for CHTCs in this study are validated with in vitro measured data in the literature.

2. Methods

2.1 Computational model generation

Original respiratory tract data were obtained using a Toshiba 64 multidetector row computed tomography (MDCT) scanner. The subject was a nonsmoking Asian male, 35 years old, weighing 75 kg, with a height of 170 cm and a body mass index (BMI) of 21. A set of 785 cross sections (slices) were obtained. CT scans produce continuous slices of the respiratory tract and store the images as standard Digital Imaging and Communications in Medicine (DICOM) data, a format commonly used for the transfer and storage of medical images. Figure 1 shows the selective CT images of the human upper airways. The original set of CT images is converted into a file format compatible with Mimics® (Materialise NV), 3D imaging software that generates and modifies 3D surface models from medical images. Generation of a surface model from the 2D contour data began with the translation of the segmented, modified, and smoothed contour points into a data
series that was loaded into ANSYS preprocessing software packages GAMBIT and TGrid. GAMBIT and TGrid were used to modify the surface mesh and finally create a volume mesh of the model, respectively. Figure 2 shows a schematic of the human airway model after reconstruction from CT images.

2.2 Outline of CFD simulation

CFD simulations were performed to calculate airflow, temperature, and heat transfer profiles under a number of breathing conditions. Steady flow fields were analyzed using a low Reynolds (Re) number-type k-ε model (Abe–Kondoh–Nagano model). The value of turbulent kinetic energy at the inlet, i.e., at the nostril, was prescribed assuming a 10% turbulence intensity. A no-slip boundary condition was applied for the wall surfaces inside the airway model. The second-order upwind scheme was used for the convection term, and the SIMPLE algorithm was used.

In this study, airflow rates of 7.5 to 60 L/min, i.e., the transitions from a laminar to turbulent flow regime, were considered. In addition to the air supply function of the human nasal cavity, the heat and mass transfer conditions to regulate the heat release from the core of the human body through the respiratory tract are also important. Here, the air conditioning capabilities of the airway model in terms of its heat transfer were simulated for different flow rates. Room air at a temperature of 20°C was used as the inhaled ambient air, and the inner walls, which are covered by a layer of mucus, were assumed to be at a constant temperature of 37°C. The numerical and boundary conditions are summarized in Table 1, and Table 2 shows the cases analyzed as a function of the flow rate of inhaled air.

2.3 Convective heat transfer coefficient

The respiratory CHTC is a phenomenological constant relating the heat flux to the temperature difference between the free airstream and body surfaces or airway walls (in this research). The widely accepted expression for the CHTC ($h_c$) is as follows:

$$ h_c = \frac{Q}{(T_w - T_{in})} $$

(1)

Here, $Q$ implies that the heat flux unit is W/m², $T_w$ is the constant wall temperature, and $T_{in}$ is the air inhaled in the airway model in Kelvin.

The Nusselt ($Nu$) number is a commonly used dimensionless parameter that is defined as the ratio of convection heat transfer to conduction heat transfer for a given reference length. In equation form, the $Nu$ number can be expressed as

$$ Nu = \frac{h_c D_T}{k} $$

(2)

where $k$ is the thermal conductivity of the air (W/m-K), and $D_T$ is the diameter of the trachea (m). The $Nu$ number is often used to correlate experimental data with the convective heat transfer.

2.4 Experimental determination of convective heat transfer coefficient

Nuckols et al. [3] reported the detailed results of convective heat transfer experiments with a cast replica of human upper airways extending from the nose and mouth to the trachea. Considering inspiration only, Nuckols et al. expressed the experimental results in terms of $Nu$ number averaged over the entire cast from the nose to trachea as follows:

$$ Nu = 0.028(Re Pr)^{0.854} $$

(3)

The term on the right-hand side of equation (3) is a function of the Reynolds number, which is defined as the ratio between the inertial and viscous forces in a fluid. The $Re$ is often used to describe the regime of a particular flow: laminar, transitional, or turbulent. The $Pr$ in equation (3) denotes the Prandtl number, which is used to describe the relationship between the thermal and flow conditions. Specifically, the $Pr$ is the ratio of the molecular momentum diffusivity to the thermal diffusivity.

3. Results and Discussion

3.1 Airflow field and temperature

Some selected results from our numerical simulations for different inspiratory flow rates and inlet air temperatures are given in the following section. The corresponding $Re$ in the trachea for all cases analyzed are listed in Table 2.

(1) Airflow field

Figure 3 shows contours of the velocity magnitude in a vertical plane (y–z plane) for the inspiratory flow rates of 15 and 60 L/min, as representative cases. The airflow in the nasal cavity for both cases showed flow separation posterior to the narrowest valve and acceleration through the middle region of

<table>
<thead>
<tr>
<th>Table 1 Numerical and boundary conditions</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulent Model</td>
</tr>
<tr>
<td>Low Reynolds number-type k-ε model</td>
</tr>
<tr>
<td>(Abe–Kondoh–Nagano three-dimensional Cal.)</td>
</tr>
<tr>
<td>infix Boundary</td>
</tr>
<tr>
<td>$Q_{in} = 7.5, 15, 20, 30, 40, 60$ L/min,</td>
</tr>
<tr>
<td>$k_w = 3/2 (U_{in} \times 0.05)^2$, $k_i = \frac{34}{3} \times \frac{32}{3} \times T_{in}$</td>
</tr>
<tr>
<td>$T_{in} = 20°C$</td>
</tr>
<tr>
<td>Outflow Boundary</td>
</tr>
<tr>
<td>$U_{out} = \text{free slip}$, $k_{out} = \text{free slip}$</td>
</tr>
<tr>
<td>Wall Treatment</td>
</tr>
<tr>
<td>Velocity: no slip</td>
</tr>
<tr>
<td>Temperature: $T_{wall\ surface} = 37°C$</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2 Reynolds number in the trachea for different cases analyzed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inspiratory flow rate [L/min]</td>
</tr>
<tr>
<td>7.5</td>
</tr>
<tr>
<td>Reynolds number [-]</td>
</tr>
<tr>
<td>562</td>
</tr>
</tbody>
</table>
the nasal cavity because of the reduction in cross-sectional area. Wind drift generated by the centrifugal force was observed in the curved portion from the nasal cavity to the pharynx/larynx. At 60 L/min, the flow patterns in the nasal cavity and the pharynx were similar to those obtained at 15 L/min. However, a clear difference in the flow patterns in the laryngeal flow, which forms immediately after the glottis region, was observed.

(2) Temperature distribution

Figure 4 shows the temperature distributions in the human upper airways. At a flow rate of 15 L/min, a relatively low temperature was observed at the vestibule, and the air temperature in the lower airway region became well-mixed. Thus, the temperature distribution tended to be uniform from the pharynx to larynx. Moreover, the air temperature from the nostril was affected more rapidly at an airflow rate of 15 L/min compared to 60 L/min because the low flow rate induces a longer residence time for the air in the airways.

Figure 5(a) and (b) shows the temperature profiles in the mid-plane of the trachea and selected cross-sectional views at airflow rates of 15 and 60 L/min, respectively. In the posterior trachea, the low velocity caused higher temperatures Compared to the cross-sectional slices between both cases, the thermal effect tended to decrease in accordance with the increase in the airflow rate.

3.2 Heat flux and convective heat transfer coefficient

The wall heat flux distributions in the lower airway region are shown in Figure 6. In both representative cases, the relatively high wall heat flux appeared at posterior regions of the oral-pharynx where the temperature gradients increased. The lower sections of the airway exhibited low fluxes because the flow of the inhaled air and temperature gradient decreased in those regions. In the region at the trachea and bifurcations, a low heat flux value was confirmed in the case of 15 L/min because the inhaled air adjacent to the walls was equal to the
core body temperature. At 60 L/min, the values of the heat flux were still high even at the trachea and bronchial branches.

Figure 7 shows the regional Nu number in the human airway model versus the product of the Re and Pr numbers. Compared with the in vitro data obtained by Nuckols et al. [3], the results obtained in this study reproduced the qualitative trend of the heat transfer property. The quantitative discrepancy may stem from experimental uncertainties, e.g., the mean nonuniform bulk temperature of gas cannot be obtained within the model given the experimental conditions. Furthermore, the geometry of the human airway used in this study might differ from that used by Nuckols et al.

Figure 8 shows the predicted CHTCs at different inspiratory flow rates. The results showed that the mean CHTC \( h_c \) increased with the inspiratory airflow rate. The \( h_c \) in the nasal cavity was estimated relatively high because the narrowest air passage in the turbinate area enhanced the heat transfer phenomenon.

4. Conclusion

In this research, CFD analyses were carried out focusing on a wide range of airflow rates inside a human airway model, and the prediction results for the CHTC \( h_c \) were reported. The prediction results of the CHTC \( h_c \) based on CFD with a low Reynolds number-type k-ε model (Abe–Kondoh–Nagano model) reasonably approximated in vitro measured data from the literature.

In future work, we will carry out an in vitro experiment using a realistic prototype model of the human airways by using a particle image velocimetry (PIV) technique. The steady flow field will be measured to validate the simulation results.

Acknowledgement:

This project was supported by a Grant-in-Aid for Scientific Research (JSPS 21676005).

References:

