Methodology for Simulating Air Leakages of an N95 Filtering Facepiece Respirator-A Pilot Study

Zhipeng Lei and Jingzhou (James) Yang*

Human-Centric Design Research Laboratory
Department of Mechanical Engineering
Texas Tech University, Lubbock, TX 79409
zhipeng.lei@ttu.edu, *james.yang@ttu.edu

ABSTRACT

Air leakages happen when a respirator cannot fit its user well. This paper presents a methodology to predict potential leakage locations for an N95 filtering facepiece respirator. In this pilot study, we assume that the respirator is impermeable. Geometry features of a large size headform and a 3M 8210 respirator were captured by a 3D scanner. Using finite element (FE) method, the interaction between respirator and headform models was modeled in order to obtain their deformed shapes and fitting positions. Using computational fluid dynamics (CFD) method, the air flow in the domain between the respirator inner surface and the headform frontal surface was simulated. Four situations, including the exhalation with nose, inhalation with nose, exhalation with mouth, and inhalation with mouth, were simulated separately. Results showed that the air leakage mostly happened at areas near the nasal bridge. It was also found that the pressure caused by the respiration was considerably low.

Keywords: air leakage, N95 face-piece filtering respirator, respirator fit and comfort.
DOI: 10.3722/cadaps.2012.43-53

1 INTRODUCTION

The N95 filtering facepiece respirator, which is widely used in threatening environments, protects wearers by preventing contaminant from human's respiratory system. The respirator and human face together create a micro-environment where the breathing air flows through. If a respirator cannot perfectly fit a human face, an air leakage happens. A study of the air flow in the micro-environment can give respirator designers and producers information about the respirator fit and leakage locations. CFD method uses numerical methods and algorithms to solve problems of fluid flows. In the micro-environment, CFD solver calculates the air flow in various conditions which cannot be achieved by experiments, and obtains detailed information that is impossible to be measured.

In several study, the air flow around a headform or a whole human body has been simulated by CFD method (Brohus et al., 1996; Abanto et al., 2004; Murakami, 2004; Zhu et al., 2006; Anthony et al., 2006). The procedure had four steps: (1) a 3d scanner captured the surface geometry of a manikin without the clothing and hair; (2) the computing domain was divided into two regions. The inner one that contains the geometry of the human body was meshed into unstructured grids and the outer one into structured grids (Sorensen et al., 2003); (3) respiration process, simplified to a steady inhalation process, was included in the simulation (Gao et al., 2004); (4) the air flow and heat transfer were calculated. However, these models did not contain the clothing, respirator, helmet and glass that are commonly used by human subjects.

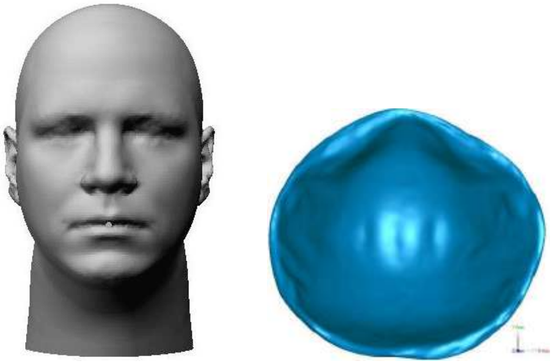
Facial features affect the flow field near the mouth of a breathing human face (Anthony et al., 2005). If a respirator and a headform are combined, the CFD model related to the chamber between the respirator and headform would become much more complicated. Butler (2007) investigated the fluid transportation in the environment near a respirator using the CFD method. Headform and respirator geometries were scanned separately and then were combined together in order to define the computational field. However, the chamber was formed by manually adjusting the alignment of the respirator with respect to the headform. This study of air flow near a respirator lacked a contact simulation between respirator and headform models for determining geometry features. Lei et al. (2010) studied the contact of N95 filtering facepiece respirators using the FE method. Although this study was mainly for determining contact pressure, it also generated deformed shapes and fitting positions of respirator and headform models, which can easily generate the chamber in air flow simulation.

Instead of the flow outside of the respirator, we are interesting the inner flow of a volume (chamber) formed between the respirator and headform surfaces. This paper presents a methodology to determine the leakage locations when an N95 facepiece filtering respirator is put on a headform. The alignment of the respirator and headform is obtained in a contact simulating using FE method. A flow volume covered by the deformed surfaces of the respirator and human face is defined and well meshed. An air flow in the flow volume is simulated by CFD method. The highly curved flow volume is meshed into small elements. We assume that the respirator is an impermeable wall. The exhalation and inhalation are simulated separately as steady fluid problems. Situations include exhalation with nose, inhalation with nose, exhalation with mouth, and inhalation with mouth. Through simulations, potential air leakage location can be determined and pressure changes on the respirator, caused by the breath, are also examined.

This paper is organized as follows: Section 2 introduces digital models and summary of the methodology. Section 3 briefly reviews the contact simulation process. Section 4 presents the CFD simulation. Finally simulation results are presented followed by conclusion.

2 MODELS AND METHODOLOGY

A large size headform, which is among five size headforms (large, medium, small, long/narrow and short/wide) created by the National Institute for Occupational Safety and Health (NIOSH) (Zhuang et al., 2005), is used in our simulation. An N95 filtering facepiece respirator, 3M 8210 respirator, designed to fit all users, is also used. Both the headform and the respirator are scanned using a 3D scanner, as shown in Fig. 1(a) and (b).



(a) (b)

Fig. 1: Scanned models of (a) large size headform; (b) 3M 8210 respirator.

The process of wearing a respirator involves in two biomechanical problems. The first problem is the contact between the respirator and the headform. The second problem is the air flow in the chamber between respirator and headform surfaces. Therefore, the methodology for predicting air leakage locations is based on the two biomechanical problems. The LS-DYNA software, a nonlinear FE solver, is used to solve the contact problem. FLUENT software, a CFD solver, is used to solve air flow problem. Although this paper focuses on the air leakage of the respirator, the geometry combination of the respirator and headform is hard to obtain unless we conduct a contact simulation. Thus, both contact and air flow simulations are conducted by coupled FE and CFD methods.

3 CONTACT SIMULATION BY THE FE METHOD

In this section, a contact simulation between a respirator and a headform is conducted using the FE method. This work is based on our previous models (Lei and Yang, 2010). Respirator and headform models are originated from 3D scanning surfaces of a 3M 8210 respirator and a large size headform. Referred to the structure of the N95 filtering facepiece respirator and the human head anatomy, FE respirator and headform models are built. Fig. 2(a) shows that the respirator model is comprised of multiple layers and two straps. As shown in Fig.2 (b), the headform model is divided into five parts (two areas for cheeks, one area for the upper forehead, one area for the chin, and one area for the back side of head). It has multiple layers that include a skin layer, muscle layer, fat tissue layer, and bone layer, as shown in Fig. 3.

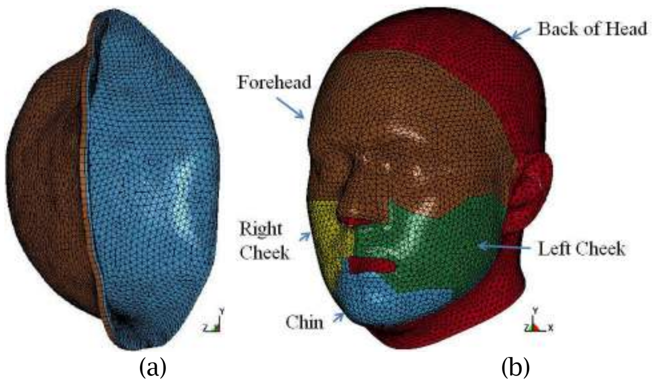
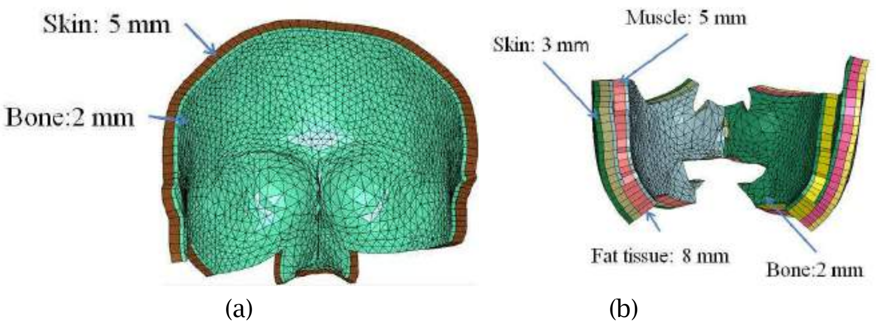
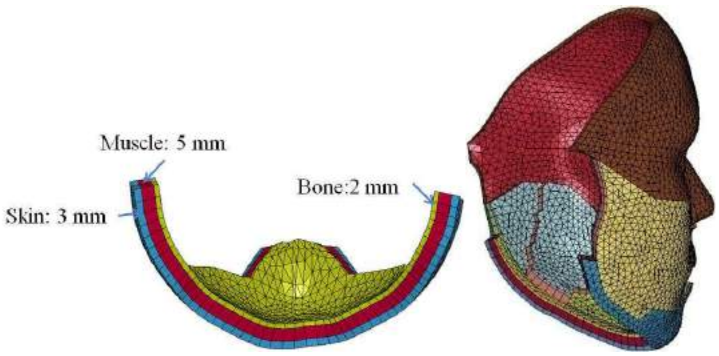


Fig. 2: FE models of (a) the respirator and (b) the head-form.





(c) (d)

Fig. 3: Layers in (a) forehead part; (b) left and right cheek parts; (c) chin part; (d) four front face parts together.

The whole contact simulation process has two stages. The first stage is to wrap the straps around the back of the headform and pull the respirator away from the face. The second stage is to release the respirator so that the respirator moves towards the face. Strap forces and contact interactions are generated between the respirator and headform. In the final state, the respirator contacts the human face, as shown in Fig. 4.

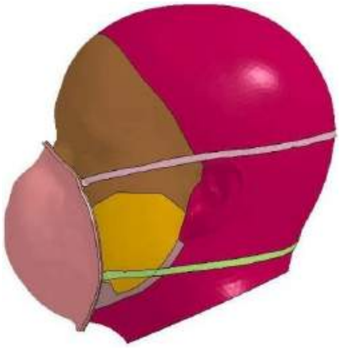


Fig. 4: The final state of the contact simulation.

4 AIR LEAKAGE SIMULATION BY THE CFD METHOD

This section conducts an air leakage simulation using the CFD method. First, the problem geometry is generated from the finial state of the contact simulation. Second, by meshing the chamber geometry, 3D grids are produced. Third, boundary conditions are defined and a CFD solver is set up. Finally, the solver calculates the air flow of respirator.

4.1 Chamber Geometry

The air flow field includes regions both inside and outside the respirator. But, in this study the air flow chamber between surfaces of the human face and respirator is considered. In the final state of the contact simulation, the respirator deforms and stays still on the headform. Since the breathing air is very light, we assume that it does not affect the position and shape of the respirator. So, our CFD simulation can use respirator and headform surfaces extracted from the final state of the contact simulation.

The inner surface of the respirator and the outer surface of the headform are extracted and stored as STL file format that describes unstructured surfaces consisting of triangles, as shown in Fig. 5. Respirator and headform surfaces are placed in the same global coordinate system, in which the z coordinate is in the normal direction to the headform frontal face, the x coordinate is along the lateral direction of the headform, and the y coordinate is along the vertical direction of the headform.

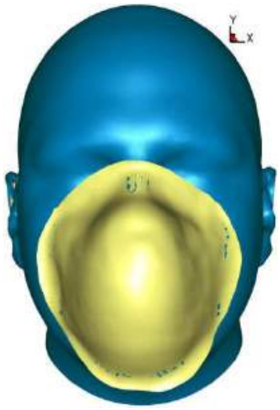
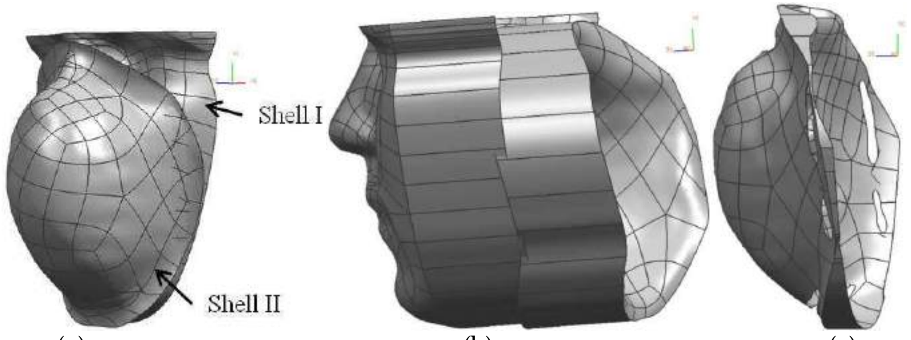


Fig. 5: Respirator and headform surface in STL format.

Because only cheek, nose and chin areas contact the respirator, other areas can be removed for computational efficiency, as shown in Fig. 6(a). Shell I is the trimmed human face and Shell II is the respirator surface. Then, Shell I is extruded 50 mm in the positive z direction and Shell II is extruded 50 mm in negative z direction. Fig. 6(b) shows two new bodies created by extruding Shell I and II. After conducting a Boolean operation, intersection between two bodies, we can obtain a flow volume which indicates the region of air flow inside the respirator, as shown in Fig. 6(c). The flow chamber has three surfaces including the human face, respirator and edge. Geometry features of deformed surfaces of the respirator and human face are maintained. The edge surface embraces other two surfaces, creating a closed domain. The flow volume (chamber) has a highly complicated geometry with a small volume of 0.000134 m^3 .



(a) (b) (c)

Fig. 6: CAD processing: (a) two shells; (b) the extending of two shells; (c) the flow volume.

4.2 Grid Generation

Because of the complex geometry of the flow volume, the grid generation is time consuming in the CFD modeling. The general procedure of the grid generation is that faces of the flow volume are firstly meshed; then the whole flow volume is meshed based on the meshed faces. Since unstructured grids method is the only option for our highly complicated geometry, triangles are chosen as face grids and tetrahedral cells are chosen as body grids.

The shape of the flow volume is like a shell with different thicknesses. In some locations of the shell, the thickness is very thin, since the human face is too close to the respirator. We need to find a proper element size that is small enough to mesh the shell in critical places. However, if the element size is too small, the computer does not have enough memories to perform the meshing. Respectively, there are 131114 triangular elements on the faces of the flow volumes and 980397 tetrahedral elements inside of the flow volume, as shown in Fig. 7.

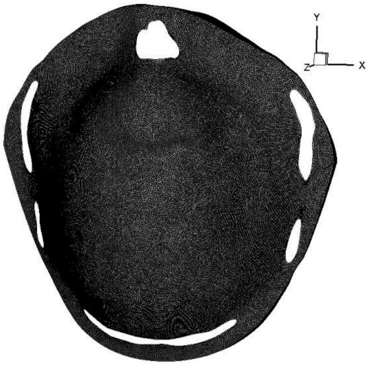


Fig. 7: the flow volume with unstructured meshes.

4.3 Air Flow Velocity and Turbulence

When a healthy adult is at rest, the average respiratory rate is between 12 to 20 breaths per minute (Beckett, 1995). Six to eight liters of respiratory minute volume are required by a normal human. Consider the respiratory rate is 15 breaths per minutes and respiratory minute volume is 6 liters. So, we can assume that each cycle gives two seconds of inhalation and two seconds of exhalation, as well as 0.5 L breathing air. The air flow rate of breathing is (0.5 L / 2 s =) 0.25 L/s. In this study the inhalation and exhalation are considered separately. In other words, there are two simulations with constant air flow rates. One is for the inhalation, in which a constant flow rate of 0.25 L/s comes into the respiratory system. The other one is for the exhalation, in which a constant flow rate of 0.25 L/s comes out from the respiratory system.

In reality, either nose or mouth is used for breathing. In the situation of the nose breathing, an area that contains the nasal vestibule is defined as the vent for breathing air. Because the vent surface is perpendicular to y coordinate, we calculate the projection of the vent in x-z plane and its area is 370.2 mm^2. In the exhalation simulation, the velocity at the vent is described in Cartesian components as $V_x = 0, V_y = -\frac{0.25 \times 10^{-3} m^3}{370.2 \times 10^{-6}} = -0.675 m/s, V_z = 0$. Meanwhile, in the inhalation stage, the velocity at the vent is $V_x = 0, V_y = \frac{0.25 \times 10^{-3} m^3}{370.2 \times 10^{-6}} = 0.675 m/s, V_z = 0$.

In the situation of the mouth breathing, an area that contains the mouth is defined as the breathing vent. The vent surface is perpendicular to z coordinate and its projection in x-y plane is 408.3 mm^2. Similar to nose breathing, the exhale velocity at the mouth vent is $V_x = 0, V_y = 0, V_z = \frac{0.25 \times 10^{-3} m^3}{408.3 \times 10^{-6}} = 0.612 m/s$, and the inhale velocity at the mouth vent is $V_x = 0, V_y = 0, V_z = -\frac{0.25 \times 10^{-3} m^3}{408.3 \times 10^{-6}} = -0.612 m/s$.

The Reynolds number Re is defined as $\mathrm{Re} = \rho UL / \mu$ for judging the turbulent effect, where ρ is density, U is a typical velocity in the problem, L is a length scale, and μ is dynamic viscosity. If the Reynolds number exceeds 1000, turbulence should be considered. Otherwise, the flow can be assumed as laminar flow without viscous effect. In our model, the density of the air is $1.225\, kg / m^3$, U is the velocity at the vent, L is the width of the vent and the dynamic viscosity of the air is $1.7894 \times 10^{-5} N \bullet s / m^2$. So, Reynolds numbers in the nose breathing and mouth breathing are about 500, which is bellow 1000. The air flow in our case is calculated as laminar flow.

4.4 Governing Equations and Boundary Conditions

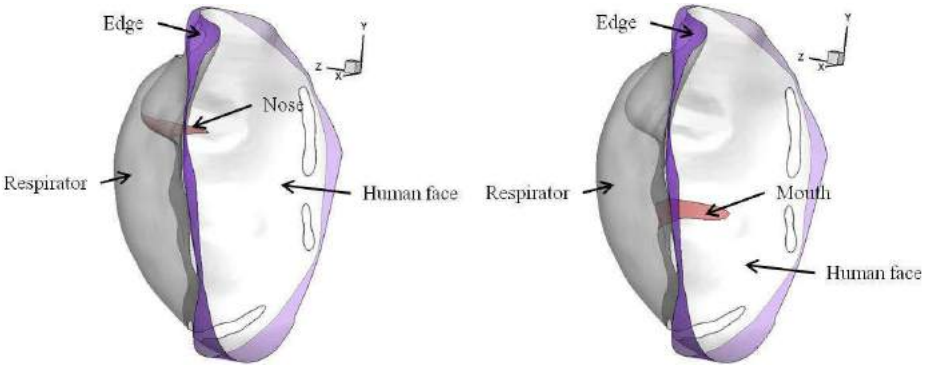
The air flow inside the flow volume is a steady incompressible laminar flow problem. Since the density of the air is very low, we ignore its gravitational body force. Further, there is no chemical reaction and no external body force. Governing equations that come from the mass conservation and the momentum conservation can be written as follows:

$$\nabla \bullet \vec{v} = 0 \tag{4.1}$$

$$\vec{v} \bullet \nabla \vec{v} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \vec{v} \tag{4.2}$$

The vector of velocity, $\vec{v}$, and the pressure, *p*, are unknown, while the density, ρ, and the kinematic viscosity, ν, are known. We define the standard atmospheric pressure, which can be referred as the pressure outside the flow volume, as 101325 Pa. The pressure, p, in the Equation (4.2), is the gauge pressure.

Our model contains 4 specific surfaces: the human face, respirator, vent (nose or mouth) and edge, as shown in Fig. 8. Boundary conditions are applied at these surfaces. Since human face and respirator surfaces do not deform during the air simulation, a wall boundary condition is applied to both surfaces. The velocity at the wall is 0 in all direction. The velocity inlet boundary condition is applied at the surface of the vent by defining a flow velocity. At the edge surface, the air directly contacts the outside of the flow volume. The velocity at this boundary is unknown, but the pressure is equal to the standard atmospheric pressure (the gauge pressure is 0). In FLUENT software, the pressure outlet boundary condition can be used to describe this specification. A summary of boundary conditions is shown in Table 1.



(a) (b)

Fig. 8: Boundary surfaces: (a) the nose breathing; (b) the mouth breathing.

Surface	Boundary condition	Mathematical expression
Human face	Wall	$\vec{v} = 0$
Respirator	Wall	$\vec{v} = 0$
Nose or, mouth	Velocity inlet	$V_y = -0.675\mathrm{m/s(exhale)},\ V_y = 0.675\mathrm{m/s(inhale)}$ or, $V_z = 0.612\mathrm{m/s(exhale)},\ V_z = -0.612\mathrm{m/s(inhale)}$
Edge	Pressure outlet	The Gauge Pressure: $p = 0Pa$

Tab. 1: Boundary conditions.

4.5 Solver

The finite volume method is adopted in the air flow problem. The basic idea of the finite volume method is to divide the computational domain into a large number of control volumes and to build an algebraic system for the control volumes (Chung, 2001). In previous section, the chamber geometry has been meshed into tetrahedral cells that are control volumes. One can integrate the Equation (4.1) and Equation (4.2) over each cell and discrete them using the average values of cell variables (velocity, pressure, density and viscosity) that are stored at the cell center. Because the velocity and pressure are the only unknowns and the density and viscosity are uniform over the domain, the pressure based solver in FLUENT software is well suited for our problem. A famous algorithms, named SIMPLE, which is based on the predictor-corrector approach, is selected.

Because the problem has complicated geometry and uses unstructured mesh, discrete schemes for the velocity, pressure and time need to be carefully selected. A second-order upwind scheme is used for the velocity discretization, since it gives more accuracy in tetrahedral grids. A PESTO! scheme for pressure interpolation, which is suited for the highly curved domain, is used in this simulation. Iterations during the simulation utilize an implicit time scheme, which is unconditionally stable with respect to time step size.

Before running the simulation, tone can provide an initial condition for the whole flow volume. Although our problem is a steady state flow and all variables are time independent, the initial condition gives a guess on how the calculation begins. In the initial state, the velocity within the computing field is uniformly assigned as the velocity at the vent (nose or mouth) and the gauge pressure is 0.

5 RESULTS

Totally four air flow simulations are conducted, including the nose exhalation, nose inhalation, mouth exhalation and mouth inhalation. The simulation stops whenever the velocity and pressure fields reach the steady state.

Compared with the velocity magnitude at the vent (nose or mouth) as 0.675 (nose) or 0.672 (mouth) m/s, velocity magnitudes in the computing domain is below 1.2 m/s, which implies that there is no big velocity jump in the calculation. In the flow field, the velocity vectors, including magnitudes and directions, are not uniformly distributed. Fig. 9 shows velocity vector fields of four simulations. Vectors with the highest velocity appear at areas near the human nasal bridge, and other vectors are centered in the area near the vent (nose or mouth). Blank areas in the flow volume indicate that there is no air movement.

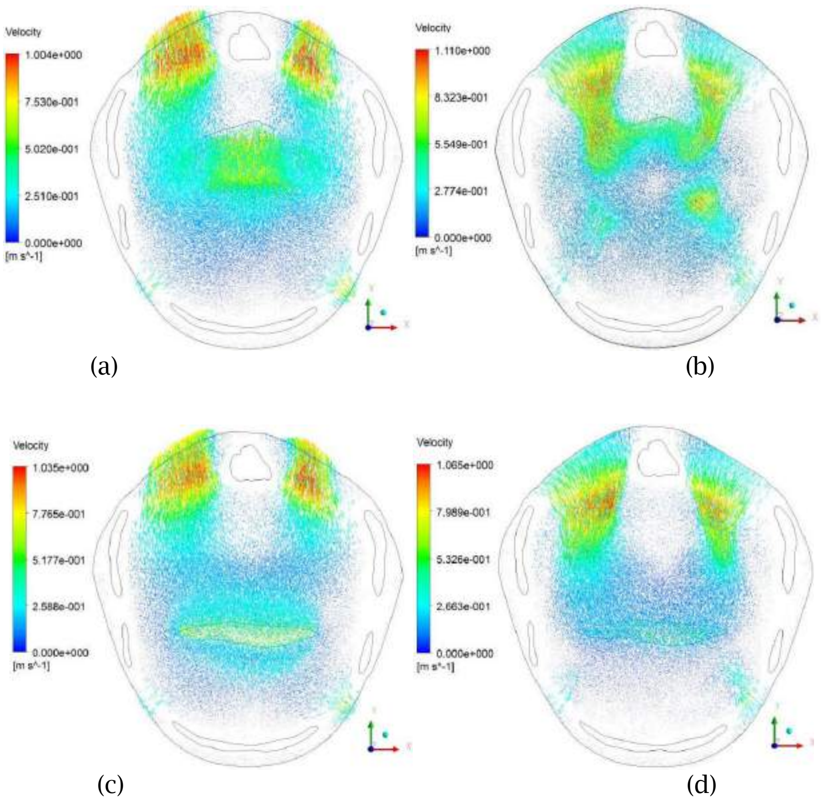
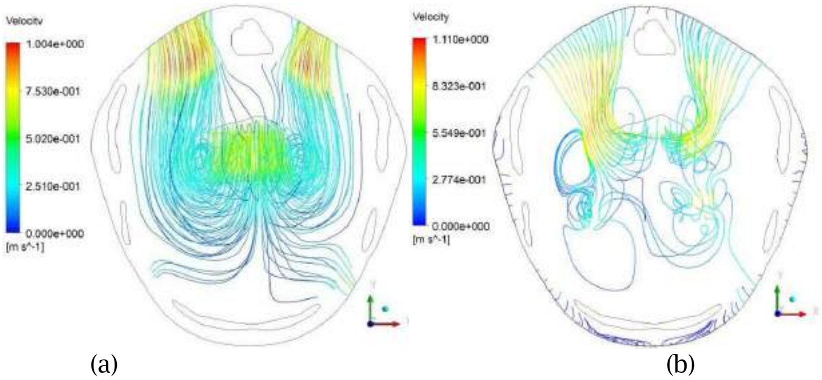
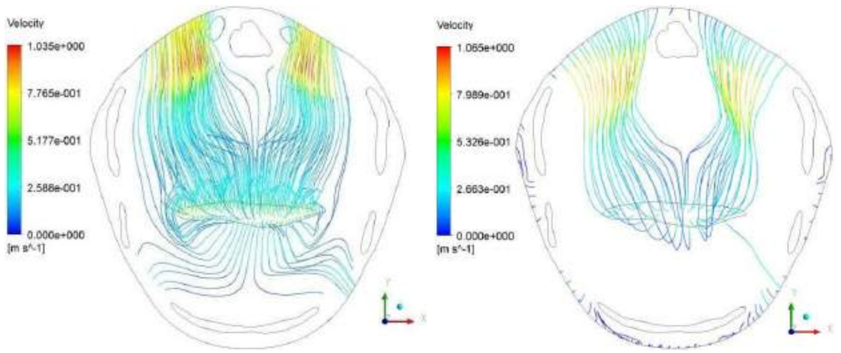


Fig. 9: Air velocity vectors: (a) the nose exhalation; (b) the nose inhalation; (c) the mouth exhalation: (d) the mouth inhalation.

In the exhalation, the air comes outward from the vent (nose or mouth) and escapes the flow volume through the edge. In the inhalation, the air comes into the flow volume through the edge and leaves through the vent (nose or mouth). Fig. 10 shows air streamlines in four simulations. Fig. 10 provides the information about air leakage locations. Areas of the edge near the human nasal bridge are locations that have potential air leakages. There are also some small leakages at areas of the edge near the bottom of the human cheek. Other areas of the edge do not appear air leakage.

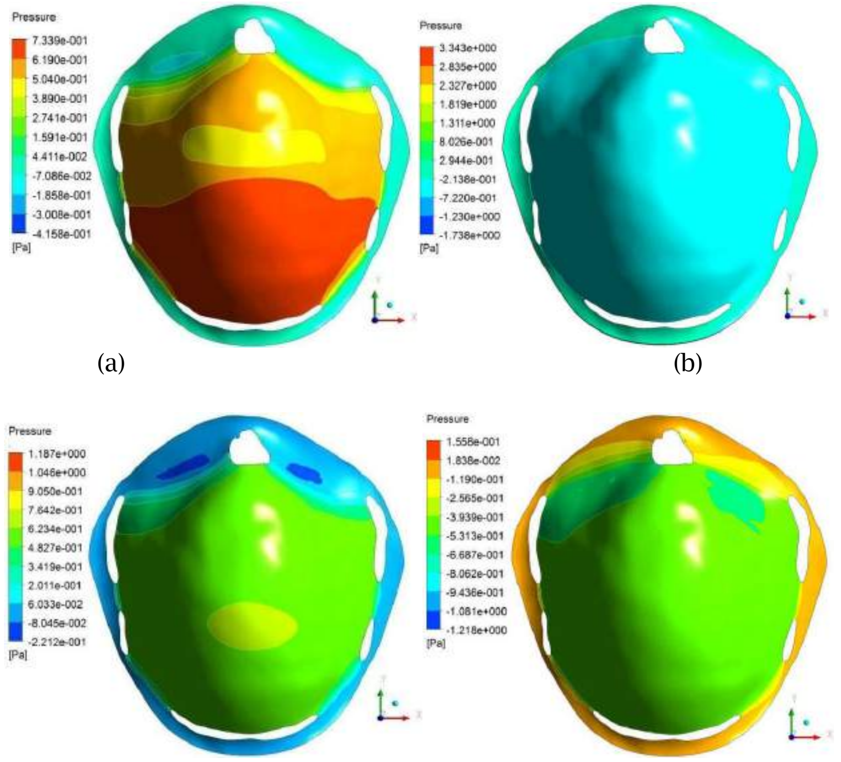




(c) (d)

Fig. 10: Air streamlines: (a) the nose exhalation; (b) the nose inhalation; (c) the mouth exhalation: (d) the mouth inhalation.

Fig. 11 gives the pressure contour of four simulations. The pressure on the respirator surface is very low with absolute values below 10 Pa. Thus, the pressure influence on the respirator can be ignored.



(c) (d)

Fig. 11: Pressure contour: (a) the nose exhalation; (b) the nose inhalation; (c) the mouth exhalation: (d) the mouth inhalation.

6 CONCLUSION AND DISCUSSION

This study presented a methodology to investigate the air leakage of an N95 filtering facepiece respirator. The methodology was based on combined FE and CFD methods. According to simulation results, air velocities were unevenly distributed within the flow volume. Coupled FE and CFD methods solved the difficulty of combining a respirator and a headform and determining their deformations. Massive air streamlines gathered at areas near the nasal bridge indicated air leaking locations. Results also showed that there are very low pressures on the respirator, proving our assumption that there was no deformation of the respirator during breathing.

In this pilot study, we had two assumptions: (1) the exhalation and inhalation were separated; (2) the respirator is impermeable. In real world, these assumptions are not true. In our future study, an unsteady velocity will be used to represent iterations of the exhalation and inhalation, and the condition that the air penetrates through the respirator will be added. Experimental validation will also be carried out.

REFERENCES

[1] Abanto, J.; Barrero, D.; Reggio, M.; Ozell, B.: Airflow Modeling in a Computer Room, Building and Environment, 39, 2004, 1393-1402. doi:10.1016/j.buildenv.2004.03.011.

[2] Anthony, T. R.; Flynn, M. R.; Eisner, A.: Evaluation of Facial Features on Particle Inhalation, Annals of Occupational Hygiene, 49, 2005, 179-193. doi:10.1093/annhyg/meh082.

[3] Anthony, T. R.; Flynn, M. R.: Computational Fluid Dynamics Investigation of Particle Inhalability, Journal of Aerosol Science, 37, 2006, 750-765. doi:10.1016/j.jaerosci.2005.06.009.

[4] Beckett, B. S.: Illustrated Human and Social Biology, Oxford: Oxford University Press, 1995, 78.

[5] Brohus, H.; Nielsen, P.: CFD Models of Persons Evaluated by Full-scale Wind Channel Experiments, In: Proceedings of Roomvent, Yokohama, Japan, 2, 1996, 137-144.

[6] Butler, K. M.: A Computational Model of Dissipation of Oxygen from an Outward Leak of a Closed-Circuit Breathing Device, National Institute of Standards and Technology Technical Note 1484, June 2007, 3.

[7] Chung, T. J.: Computational Fluid Dynamics, Cambridge: Cambridge University Press, 2001.

[8] Gao, N.; Niu, J.: CFD Study on Micro-environment around Human Body and Personalized Ventilation. Building and Environment, 39, 2004, 795-805. doi:10.1016/j.buildenv.2004.01.026.

[9] Lei, Z.; Yang, J.; Zhuang, Z.: Contact Pressure Study of N95 Filtering Facepiece Respirators Using Finite Element Method, Computer Aided Design and Applications, 7, Issue 6, 2010, 847-861.

[10] Lei, Z.; Yang, J.: Toward High Fidelity Respirator and Headform Models, 1st International Conference on Applied Digital Human Modeling, July 17-20, 2010, Miami, Florida.

[11] Murakami, S.: Analysis and Design of Micro-climate around the Human Body with Respiration by CFD. Indoor Air, 14 (Suppl 7), 2004, 144-156. doi:10.1111/j.1600-0668.2004.00283.x.

[12] Sorensen, D. N.; Voigt, L. K.: Modeling Flow and Heat Transfer around a Seated Human Body by Computational Fluid Dynamics, Building and Environment, 38, 2003, 753-762. doi:10.1016/S0360-1323(03)00027-1.

[13] Zhu, S.; Kato, S.; Yang, J.: Study on Transport Characteristics of Saliva Droplets Produced by Coughing in a Calm Indoor Environment, Building and Environment, 41, 2006, 1691-1702. doi:10.1016/j.buildenv.2005.06.024.

[14] Zhuang, Z.; Bradtmiller, B.: Head-and-Face Anthropometric Survey of U.S. Respirator Users. Journal of Occupational and Environmental Hygiene, 2, 2005, 567-576. doi:10.1080/15459620500324727.