ICIC Express Letters
Part B: Applications
Volume **14**, Number **10**, October **2023**

ICIC International ©2023 ISSN 2185-2766
pp. 1079–1085

DESIGN OF INTAKE AND EXHAUST SHAPE OF AIR PURIFIER USING CFD

Younhee Kwon[1], Jaesung Kim[1,*], Daegi Kim[1] and Seok Chan Jeong[2]

[1]Department of Digital Manufacturing
Hanbat National University
125 Dongseo-daero, Yuseong-gu, Daejeon 34158, Korea
{ yxk47550; dg2471 }@hanbat.ac.kr; *Corresponding author: jaesungkim@hanbat.ac.kr

[2]Department of e-Business
Dong-Eui University
176 Eomgwang-ro, Busanjin-gu, Busan 47340, Korea
scjeong@deu.ac.kr

Received February 2023; accepted April 2023

Abstract. *In this study, the difference in air cleaning performance according to the shape of the inlet and outlet of the product was studied using numerical analysis. The fan boundary was introduced while the inside and outside of the air purifier are simulated simultaneously. The experimental results were used to validate a numerical simulation model, and 5% of error was obtained. Then, flow characteristics of 2 types of air purifiers were compared. As a result, it was confirmed that the product performance would change depending on the shape of the inlet and outlet by comparing the velocity of the flow fields.*
Keywords: Computational fluid dynamics, Air purifier, Fan boundary

1. **Introduction.** In much of the world, people spend on average 65% of their time indoors at home [1]. It is, therefore, important to understand the quality of air in homes, and how best to improve it. Technologies are rapidly being developed and adopted to mitigate indoor air pollution, and portable home air purifiers (HAPs) are one of the most effective technologies available to clean the surrounding air of harmful pollutants of both indoor and outdoor origin [2]. As the interest in air cleaning increases, various products are being launched. The shape and performance of the products being produced differ, and the shapes of the intake and exhaust are diverse. Despite the many reasons for the design and manufacturing of air purifiers, from an aerodynamic perspective, the intake and exhaust shapes can lead to changes in efficiency, performance, and even noise reduction. The application of computational fluid dynamics (CFD) in ventilation and indoor air science research has grown exponentially. As computer hardware capabilities have increased by a factor of 10,000 in the past 20 years, CFD has become an integral part of the research and development of complex ventilation systems in buildings [3]. In previous research, there have been many numerical analyses on the flow of room air purifiers. Eom et al. have conducted a case study on placement of portable air cleaner considering outdoor particle infiltration into an elementary school classroom [4]. Park et al. predicted the indoor fine dust concentration distribution in the residential space according to the conditions of each case of the air cleaning ventilation system using CFD [5]. In addition, many CFD studies have been conducted to improve the performance of the fan. Kim et al. visualized the flow inside an air purifier using CFD, and predicted the noise caused by airflow using computational aero-acoustics (CAA) [6]. Lee et al. studied the performance and internal airflow of the sirocco fan with a guide vane [7]. Burgmann and Janoske evaluated the efficiency of air purifier systems in classrooms under realistic conditions, i.e.,

DOI: 10.24507/icicelb.14.10.1079

including thermally driven flow effects [8]. There are no studies of the numerical analysis of the flow inside the air purifier or the flow in the room. This is because it is significantly difficult to simultaneously implement internal and external modeling, owing to the differences in the complexities of the purifier's internal and external designs. However, the flow characteristics through the air purifier may vary depending on the size or shape of the room; therefore, it is necessary to consider the inside and outside simultaneously.

The aim of this study was to analyze an air purifier's flow inside and outside in one flow field. The purposes were to evaluate the original design of the intake and exhaust shapes of air purifiers, and to suggest improvement of the design. The workflow of this study is shown as Figure 1. First of all, we modeled two types of air purifier inlet/outlet and formed the grid. Second, we presented a methodology for numerical analysis of the inside and outside of the air purifier simultaneously. Then, we perform numerical simulation and verified the analysis results by comparing them with the experimental results. Finally, we compared the numerical results of the two types of modeling.

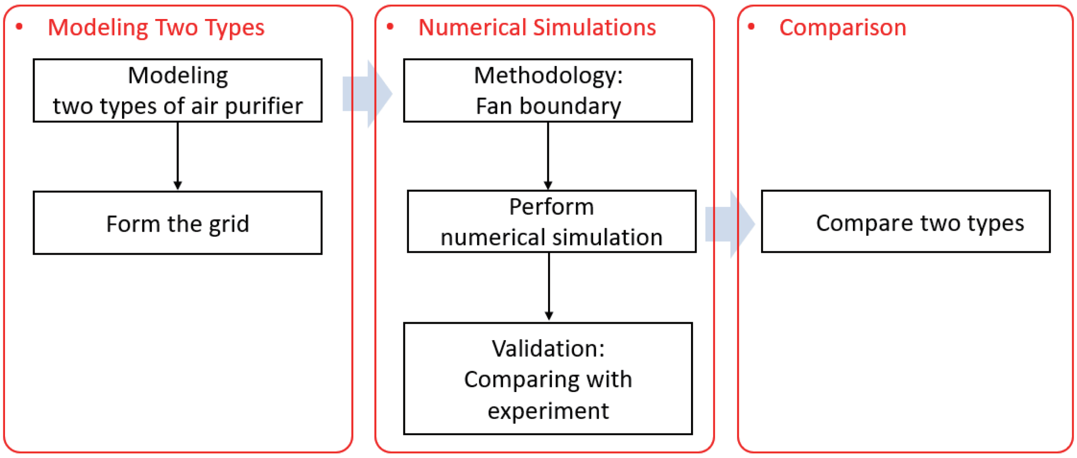

FIGURE 1. Workflow of this study

2. **Analysis Conditions and Numerical Techniques.**

2.1. **Modeling and case definition.** In this study, numerical analysis was performed by dividing the shape of the product inlet and outlet into two types. Type 1 is the shape of the entrance and exit of the conventional product, and type 2 is the shape that expands the length of the inlet and outlet. There are filter layers inside the purifier, but in this study, only the shape of the inlet and outlet is considered. Figure 2 shows the air purifier installed in the room. The air enters through the inlet at the bottom of the air purifier and exits through the outlet at the top with a fan installed inside the product. In type 2, the inlet and outlet are expanded when compared with type 1.

2.2. **Grid formation.** We applied inflated boundary layers to the grid near the wall, and reduced the grid size of the flow inside the air purifier for the accuracy of the numerical simulation. Hexahedra mesh and polyhedral mesh were properly mixed as shown in Figure 3. We performed a grid study to avoid domination of the grid. As a result, a grid was formed with 15.57 million nodes, 9.18 million elements, and the residual was converged to the 1e-5 level.

2.3. **Analysis conditions and techniques.** In this study, Ansys Fluent, a commercial code, was used to analyze the flow characteristics inside and outside the air purifier. Numerical analysis was performed using the three-dimensional incompressible RANS

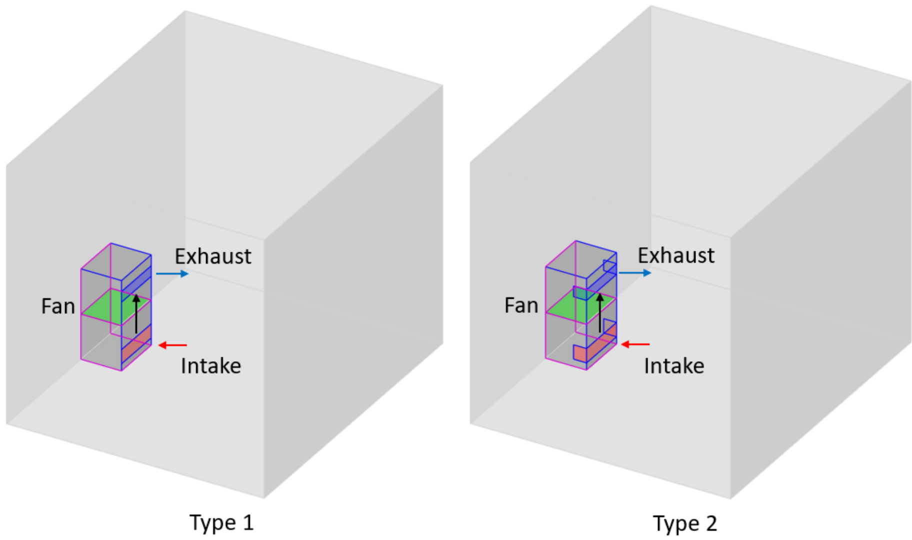

FIGURE 2. Modeling of 2 types

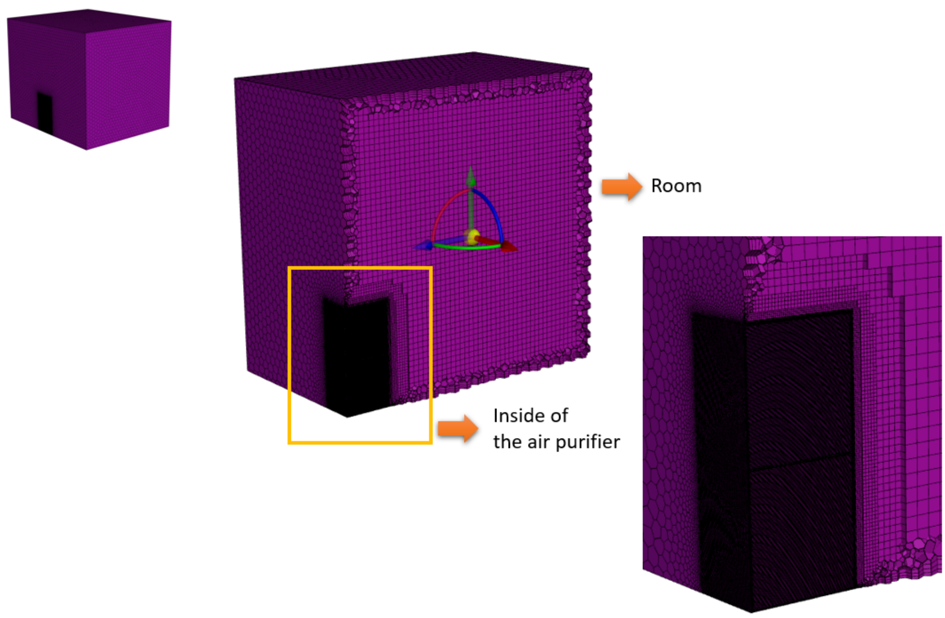

FIGURE 3. Cross section views with grids

(Reynolds Averaged Navier-Stokes) equation as a governing equation. The equation is as follows. (1) means the mass conservation equation.

$$\frac{\partial u_j}{\partial x_i} = 0 \tag{1}$$

In the above equation, u means the Reynolds-averaged velocity vector, and x_i means the i-th direction in space.

The momentum conservation equation is as the following Equation (2).

$$\rho \frac{\partial}{\partial t}(u_i) + \rho u_j \frac{\partial}{\partial x_j}(u_i) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \rho\overline{u_j' u_i'}\right] \tag{2}$$

In the above equation, p is the pressure, and μ is the dynamic viscosity. The last term in the equation means the Reynolds stress and includes the turbulence effect [9]. In order to properly predict this turbulence effect, a turbulence model is adopted.

In this study, to analyze the behavior of turbulence, the k-ε model was used. Normal analysis has the boundary condition of the "inlet" and "outlet", but in this analysis, the boundary condition of "inlet" and "outlet" is not used because the fan creates natural flow. In addition, the fan installed inside the air purifier was simulated by applying the "fan boundary". The reason is that if the fan is directly modeled, the density of the shape increases and the number of grids increases, and a transient analysis must be used to simulate the fan rotation, which takes too much time and computational resources. So, we took the "fan boundary" which can simulate the most important characteristics of the fan. The formula below is for the pressure jump used in the fan boundary. A fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. The relationship is as follows:

$$\Delta P = \sum_{n=1}^{N} f_n v^{n-1} \tag{3}$$

where ΔP is the pressure jump, f_n is the pressure-jump polynomial coefficients, and v is the magnitude of the local fluid velocity normal to the fan [10]. In the case of the fan boundary, the pressure jump and swirl of the fan are needed for input. The fan pressure jump value was input according to the fan performance curve, and the loss due to the filter or internal parts was considered. As a result of the analysis, the exit velocity has a 5% error with the experimental result. In addition to this study, there are examples of numerical analysis using fan boundary conditions. Kostek et al. simulated the vortex ring state, a major fluid characteristic around the helicopter rotor, using the fan boundary condition without modeling the helicopter rotor [11].

3. **Main Results.** Figure 4 shows the pressure distribution contour inside and around the fan. The blue part represents the low pressure where the flow enters, and the pressure increases as a pressure jump occurs in the red part past the fan boundary. Through the pressure distribution, the pressure jump, which is a characteristic of a fan, is simulated without a fan shape as a fan boundary, and their aspects are the same. However, it can be seen that the pressure range varies depending on the inlet/outlet design despite the same fan boundary condition being used. In the case of type 1, it can be seen that the pressure flow passing through the fan boundary increases significantly. In the case of type 2, where the inlet and outlet are widened, the pressure difference increases. The reason for this is shown in the dynamic pressure distribution of Figure 5. In the case of type 1, the dynamic pressure is distributed relatively lower than that of type 2. Low dynamic

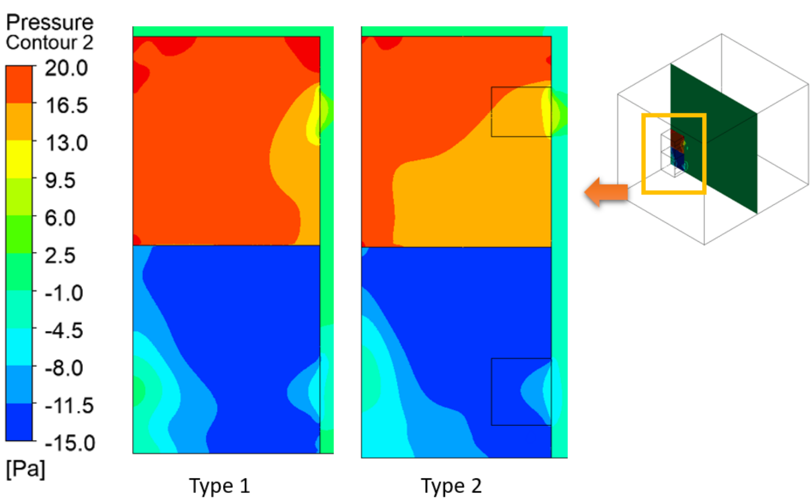

FIGURE 4. (color online) Pressure distribution in the cross section of the air purifier

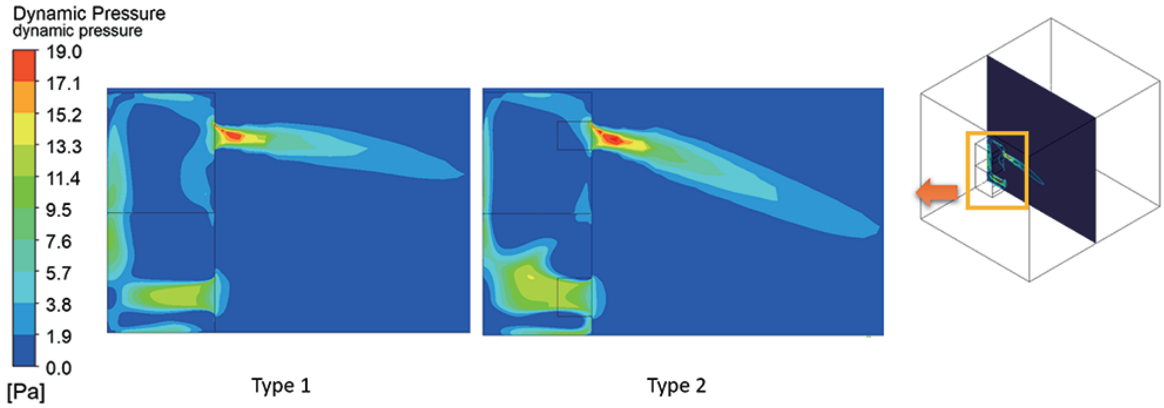

FIGURE 5. (color online) Dynamic pressure distribution in cross section

pressure means low speed. As the flow rapidly enters or exits the narrow inlet or outlet, the static pressure increases, not the speed, causing low efficiency and noise in fan.

Additionally, as a result of the analysis, the mass flow itself through the fan boundary differed in the two types. Figure 6 shows the difference in mass flow through the fan boundary and the degree of flow field circulation as an average velocity in the room. In the case of type 2, the mass flow increased by 40% compared with type 1, and accordingly, the overall flow field velocity in the room also increased by 27%.

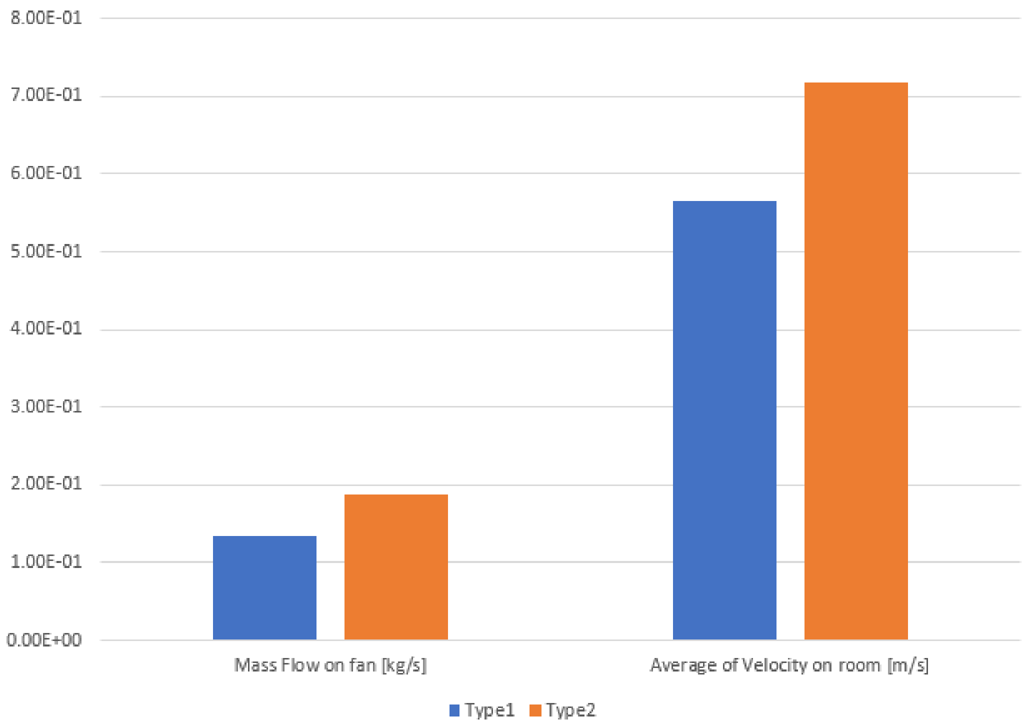

FIGURE 6. Mass flow through the fan boundary, flow velocity in the room

In Figure 7, the overall flow field volume was compared between the types. The overall flow rate was much higher in type 2, confirming that the circulation was relatively good. The flow rate is an important factor for good ventilation whereby it increases the ventilation factor in the room.

The above results indicate that the intake/exhaust design shape is directly related to product performance. This also means that the product efficiency can change depending on the product intake/exhaust shape under similar fan conditions.

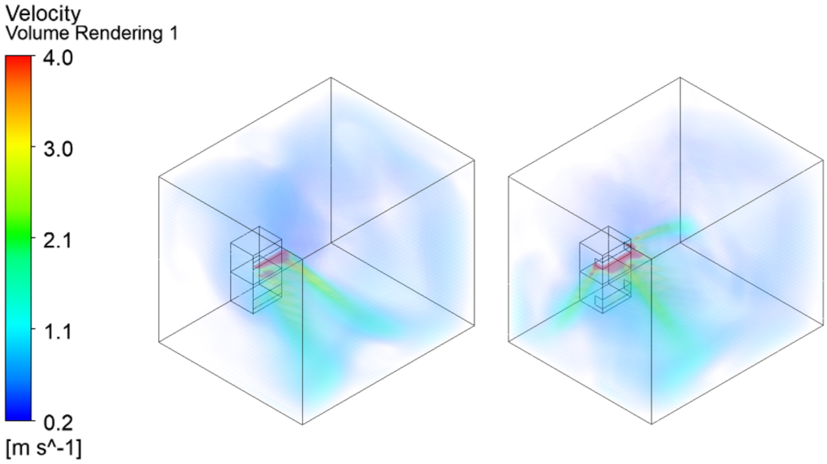

FIGURE 7. (color online) Velocity distribution in the room

4. **Conclusions and Discussions.** In this study, the difference in product performance according to the shape of the air inlet and outlet of a purifier was investigated using CFD. Both the inside and outside were included in the flow field and analyzed, and a fan boundary was used to simplify the internal flow. As a result of the comparison with the experiment to verify the validity of the fan boundary conditions, the error between the experiment and the analysis was 5%. In addition, when the inlet and outlet were expanded as in type 2, the mass flow through the fan increased by 40% and the room flow rate also increased by 27%. Therefore, the following conclusions are made: the fan boundary is adapted for this CFD process. This fan boundary is only applicable to an axial fan, not a centrifugal fan. In the case of a centrifugal fan, the flow direction is changed by the vanes, so it may have different results. In this study, only the possible significant effect on product performance of the air inlet and outlet shape is shown. The most effective air inlet and outlet may exist depending on the fan, internal component design, and room condition.

REFERENCES

[1] N. N. W. Klepeis and W. Ott, The National Human Activity Pattern Survey (NHAPS): A resource for assessing exposure to environmental pollutants, *Journal of Exposure Analysis and Environmental Epidemiology*, vol.3, pp.231-252, 2001.

[2] E. Cooper, Y. Wang, S. Stamp, E. Burman and D. Mumovic, Use of portable air purifiers in homes: Operating behavior, effect on indoor $PM_{2.5}$ and perceived indoor air quality, *Building and Environment*, vol.191, 2021.

[3] Y. Li and P. V. Nielsen, Commemorating 20 years of Indoor Air-CFD and ventilation research, *Indoor Air*, vol.21, pp.442-453, 2011.

[4] Y. S. Eom, B. R. Park, S. G. Kim and D. H. Kang, A case study on placement of portable air cleaner considering outdoor particle infiltration into an elementary school classroom, *Journal of Korean Institute of Architectural Sustainable Environment and Building Systems*, vol.14, no.2, 2020.

[5] H. G. Park, S. H. Park and J. H. Seo, Study on the indoor PM concentration changes by SA·RA location of air cleaning ventilation system in residential space, *Journal of Korean Institute of Architectural Sustainable Environment and Building Systems*, pp.105-115, 2019.

[6] J. S. Kim, U. C. Jeong, D. W. Kim, S. Y. Han and J. E. Oh, Optimization of sirocco fan blade to reduce noise of air purifier using a metamodel and evolutionary algorithm, *Applied Acoustics*, vol.89, pp.254-266, 2015.

[7] J. W. Lee, J. S. Lee, H. G. Lee and J. S. Cho, Study on the performance and internal airflow of the sirocco fan with a guide vane, *Journal of Fluid Machinery*, vol.22, no.3, pp.12-18, 2019.

[8] S. Burgmann and U. Janoske, Transmission and reduction of aerosols in classrooms using air purifier systems, *Physics of Fluids*, vol.33, 033321, 2021.

[9] H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics, the Finite Volume Method*, 2nd Edition, Pearson Education, 2007.

[10] ANSYS, Inc., *Ansys Fluent Theory Guide*, Cannonsburg, 2022.
[11] A. A. Kostek, K. Surmacz, M. Rajck and T. G. Grabowski, Application of fan boundary condition for modelling helicopter rotors in vertical flight, *The 22nd STAB/DGLR Symposium on New Results in Numerical and Experimental Fluid Mechanics*, vol.151, pp.355-364, 2021.