International Journal of Innovations in Engineering and Technology (IJIET)
http://dx.doi.org/10.21172/ijiet.113.14

Optimization Study for Improving Flow Uniformity of Diesel Particulate Filter through CFD Analysis

Yeom Gyuin1, Han Danbee2, Yoonji Kim3, Nam Seungha4, Baek Youngsoon5

[1,2,3,5] *Department of Environment-Energy-Engineering, Suwon University, Gyeonggi-do Hwaseong-si, Rep. of Korea*

[4] *Ceracomb Corporation, 246-5, Onchen-daero 1122beon-gil, Asan-si, Chungcheongnam-do, Rep. of Korea*

Abstract- As the amounts of particulate matter (PM) and NOx have been rising in the exhaust gas of diesel vehicles, regulations on the exhaust gas emissions, such as EUROVI, have been reinforced. The only measures to respond to such regulations depend on the diesel oxidation catalyst (DOC) and diesel particulate filter (DPF), which are exhaust gas after-treatment systems. DPF removes PM after filtering it through pores in the ceramics, and then oxidizing it with CO2 and O2, whereasDOC removes NOx through a catalyst coated on a ceramic support. One of the methods used to improve the efficiency of DOC and DPF is to improve the flow uniformity of the exhaust gas. Under high flow uniformity, the service life of a catalyst is extended because the exhaust gas is uniformly introduced to the DOC and the local filtration of PM can be reduced for DPF.In this study, the flow uniformity of the exhaust gas was simulated under various conditions to improve the efficiency of PM and NOx removal using ANSYS Fluent 18.2, a commercial computational fluid dynamics (CFD) code. The simulation conditions represented a pressuredrop(=ΔP) effect according to the length of the out cone at the end of the DOC and DPF, and the flow uniformity based on the diameter according to the baffle installation distance. The simulation results revealed that the pressure drop decreased by approximately 80 Pa as the length of the out-coneincreased to 70, 140, and 210mm, and that the flow uniformity was the highest (approximately 97%) at a baffle installation distance of 60 mm and a diameter of 60 mm.

Keywords– Particulate matter, Diesel oxidation catalyst, Diesel particulate filter, Computational fluid dynamic, Pressure drop, flow Uniformity

I. INTRODUCTION

As regulations on the emissions of diesel engines have been reinforced as of late, the development of various diesel exhaust after-treatment systems, such as a diesel oxidation catalyst (DOC), diesel particulate filter (DPF), and selective catalytic reduction (SCR), is required[1]. Because diesel after-treatment systems generally process hazardous substances using a catalyst, it is important that the exhaust gas be uniformly introduced to the supportto improve the catalyst activity. Under a non-uniform flow, a filter crack may occur in the DPF because high heat is generated at the position where the matter collected during the regeneration process is locally accumulated, and the NOx reduction efficiency of DOC decreases because the flow concentrated at the center affects the performance of the catalyst. Therefore, a uniform introduction of the exhaust gas significantly affects the performance of the after-treatment systems, and it is therefore necessary to improve the flow uniformity of the exhaust gas for increased efficiency [2].

Lemme et al. [3] and Johnson et al. [4] attempted to obtain a more uniform velocity distribution through experiments on the flow velocity distribution inside a catalytic converter. Because the simulation accuracy has gradually increased and the time and experimental costs have been reduced owing to advances in computers and software, such experimental studies have been replaced with analyses based on computational fluid dynamics (CFD). Kim et al. [5], Anthony [6], and Herman et al. [7] analyzed the flow velocity distribution and pressure drop according to the inlet and diffuser angle of the catalytic converter, the cell densities of the DOC and DPF, and the curvature of the inlet exhaust pipe through simulations using a numerical analysis. Currently, many researchers are also analyzing the flow uniformity and pressure drop of the exhaust gas using CFD.

In this study, to improve the efficiency of particulate matter (PM) and NOx reduction systems with a 12-inch converter using CFD, a simulation onthe flow uniformity of the exhaust gas at the inlet was conducted. Thepressure drop and flow velocity distribution were analyzed according to the various lengths of the out cone, and the flow uniformity was analyzed according to the size and installation distance of the baffle.

II.GOVERNING EQUATIONS AND FLOW UNIFORMITY

2.1 Governing equations

In this study, internal pressure and flow analyses were conducted using Fluent 18.2, a commercial CFD code. For an analysis of the pressure, flow velocity, and flow uniformity of the exhaust gas inside the PM and NOx reduction system, a simulation was conducted based on the three-dimensional Navier-Stokes equation, and the standard k-ε model was used to consider the effects of turbulence on the exhaust gas. The porous ceramic structures of the DOC and DPFwere calculated using the porous material model in Fluent.

The following continuity equation (1), momentum equation (2), and porous material model equation (3) were used for the analysis.

$$\frac{\partial p}{\partial t} + \nabla \cdot (\rho \vec{v}) = S_m \qquad (1)$$

$$\frac{\partial}{\partial t}(p\vec{v}) + \nabla \cdot (\rho \vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\bar{\tau}) + \rho\vec{g} + \vec{F} \qquad (2)$$

$$S_i = \sum_{j=1}^{3} D_{ij}\,\mu v_j + \sum_{j=1}^{3} C_{ij}\frac{1}{2}\rho|v_j|v_j) \qquad (3)$$

Where, υ is the flow velocity (m/s), P is the pressure (Pa), ρ is the density of the fluid (kg/m3), D_{ij} is the coefficient of viscous resistance, C_{ij} is the coefficient of inertial resistance, $D_{ij} = \frac{1}{\kappa_i}$ and $C_{ij} = 2\beta$.

2.2 Flow uniformity

The flow uniformity of the exhaust gas significantly affects the purification efficiency and service life of the exhaust gas post-treatment system. In this study, the flow uniformity of the exhaust gas (= σ) was analyzed using equation (4), which was proposed by Weltens. A flow uniformity of 1 (100%) represents a perfectly uniform flow distribution, and a flow uniformity of close to zero (0%) represents an extremely local flow. In general, a uniformity of higher than 0.9 (90%) is required for the design of an exhaust gas purification system [8].

$$\sigma = 1 - \frac{1}{2}\sum_{i=1}^{n}\frac{|V_{avg} - V_i|A_i}{V_{avg}A^{1.00}} \qquad (4)$$

where, $A^{1.00}$ is the cross-sectional area of the DOC inlet, A_i is the cross-sectional area of cell i, V_i is the local velocity at cell i, and V_{avg} is the average velocity.

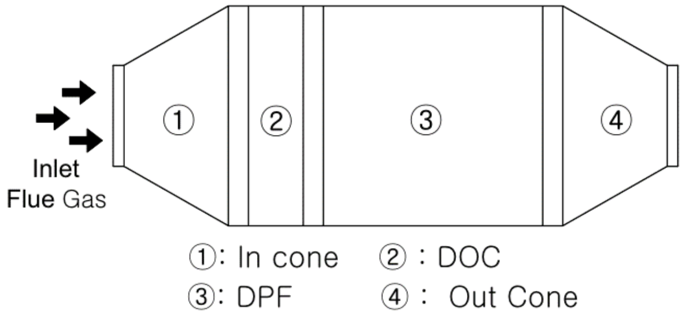
III. EXPERIMENTAL METHOD

The PM and NOx reduction system consists of an in-cone, DOC, DPF, and out-cone, as shown in Figure 1. The exhaust gas has a velocity of approximately 41 m/s and a temperature of 310 °C. The unburned gas is removed in the DOC through the oxidation reaction, and the PM is filtered while passing through the DPF. Table 1 shows the specifications of the 12-inch DPF and DOC.

Table -1 Specifications of DOC and DPF.

Items	Diameter	Length	CPSI	OFA	Bulk Density
DOC	304.8 mm	76.2 mm	400	70–76%	0.45 g/cc
DPF	304.8 mm	304.8 mm	200	60–68%	0.40 g/cc

*OFA, Open frontal area; CPSI,cell per square inch



3.1 Cross section of the PM and NOx reduction system.

For a uniform flow of the exhaust gas into the DOC and DPF under these conditions, the effects of the system inlet on the pressure drop was measured according to the length of the out-cone, and the effects of the baffle size and installation distance on the average flow velocity, flow uniformity, and pressure drop were simulated by installing a baffle in the in-cone. During the simulation,the pressure drop was calculated using an in-cone length of 140 mm and varying out-cone lengths of 70, 140, and 210 mm. Table 2 shows the baffle sizes and installation distances.

Table -2 Simulation values for the baffle size and installation distance.

Specifications		Baffle installation distance (mm)		
Baffle diameter	30 mm	30	60	90
	60 mm	30	60	90
	90 mm	30	60	90

IV. RESULTS AND DISCUSSION

4.1. Effects of the presence of the baffle on the exhaust gas flow

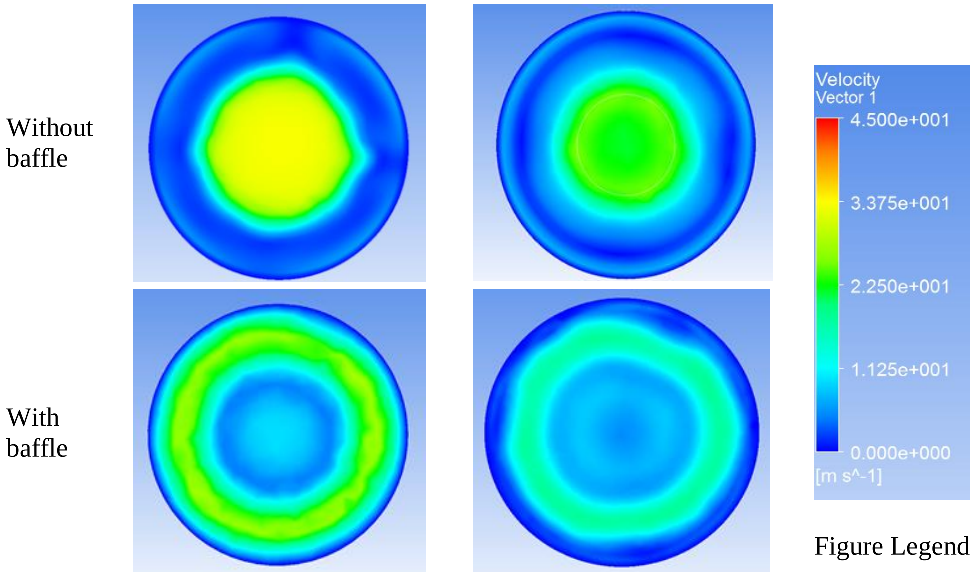
To analyze the effects of the baffle presence on the exhaust gas flow, a baffle with a 60 mm diameter was installed at 60 mm from the exhaust gas inlet, and the simulation was conducted, the results of which are shown in Table 3 and Figure 2. As Table 3 indicates, the pressure at the system inlet was not affected by the baffle installation, although the average flow velocity at the DOC inlet was approximately twice as fast (16.2 m/s) when there was no baffle. This appears to be because a fast flow velocity was introduced at the center when no baffle was applied, but the flow velocity distribution was distributed from the center to the periphery when the baffle was installed, as shown in Figure 3.

Table -3 Effects of the presence of the baffle on the pressure and flow velocity.

Category	Pressure (Pa)	Average flow velocity (m/s)	
	System inlet	DOC inlet	DOC outlet
Without baffle	25,560	16.2	6.01
With baffle	25,590	7.1	6.03

100 mm from the inlet 158 mm from the inlet

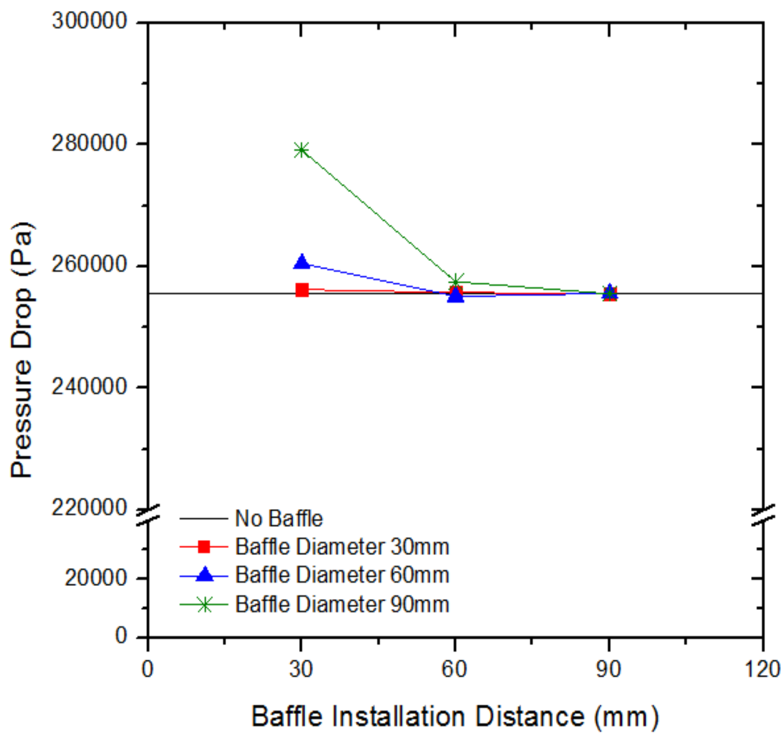
http://dx.doi.org/10.21172/ijiet.113.14



Cross-sectional flow velocity distribution according to the presence of a baffle.

4.2. Effects of the baffle installation distance on the pressure drop

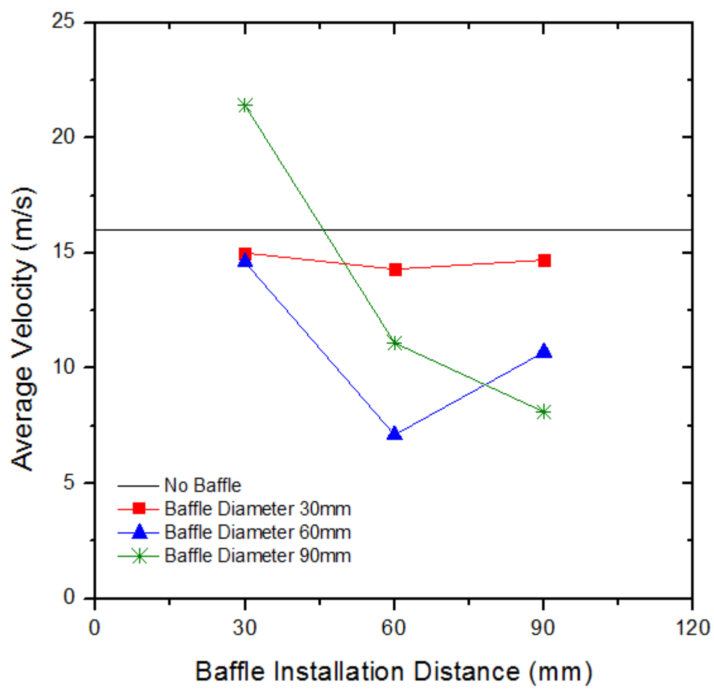
Figure 3 shows the effects of the baffle size and installation distance on the pressure drop. As can be seen from the figure, at an installation distance of 30 mm, the pressure drop increased with an increase in the baffle size. This appears to have occurred because, as the baffle size increased, the flow of the fluid was interrupted more,and therefore the pressure in the flow path was increased. In addition, at installation distances of longer than 60 mm, the pressure drop was constant at approximately 25,500 Pa. It is therefore necessary to keep the pressure drop constant by setting the baffle installation distance to longer than 60 mm.



Effects of the baffle installation distance on the pressure at the system inlet.

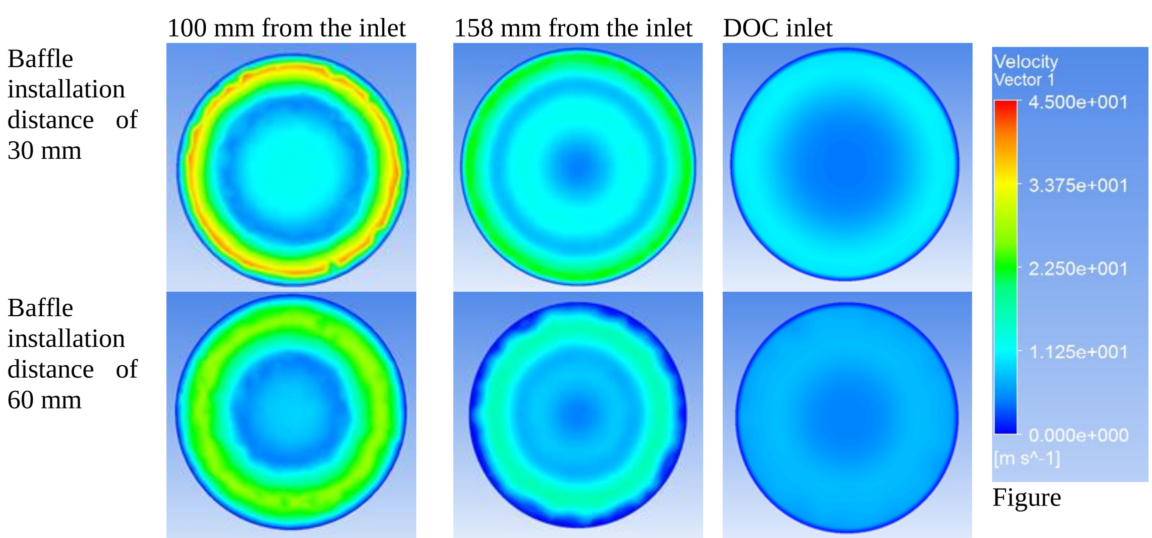
4.3. Effects of the baffle installation distance on the average flow velocity

Figure 4 shows the effects of the baffle installation distance on the average flow velocity. As can be seen from the figure, the average flow velocity was 16 m/s without a baffle, but was approximately 15 m/s at all distances for a baffle diameter of 30 mm. For a baffle diameter of 60 mm, the average flow velocity was approximately 15 m/s at an installation distance of 30 mm. It decreased to 7.1 m/s at 60 mm, and then increased to approximately 11 m/s at 90 mm. As shown in Figure 5, the flow velocity distribution was uniform at a 60 mm distance, but was not uniform at 30 or 90 mm. This appears to have occurred because the flow path between the baffle and inlet was short for an installation distance of 30 mm, and thus the flow velocitydistribution was wide and non-uniform, and because the flow path between the baffle and the DOC inlet was short for a distance of 90 mm, and thus the flow velocity distribution was wide and non-uniform. In addition, for a baffle diameter of 90 mm, the average flow velocity decreased as the installation distance increased, resulting in a flow velocity of approximately 8 m/s at 90 mm.



Effects of the baffle installation distance on the average velocity at the DOC inlet.

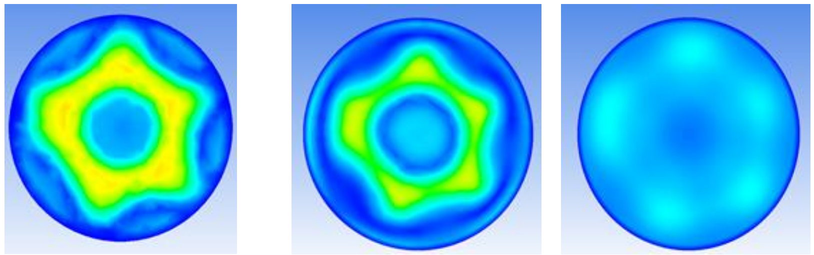
Flow velocity distribution measurement position



Figure

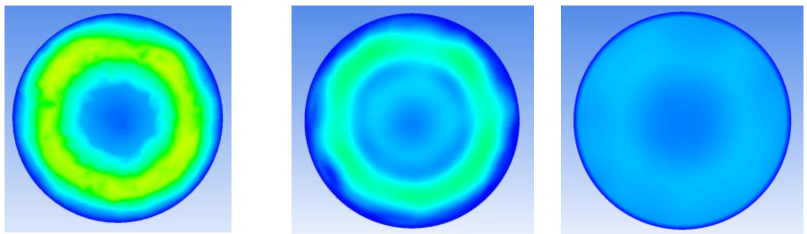
http://dx.doi.org/10.21172/ijiet.113.14

Baffle installation distance 90 mm



Legend

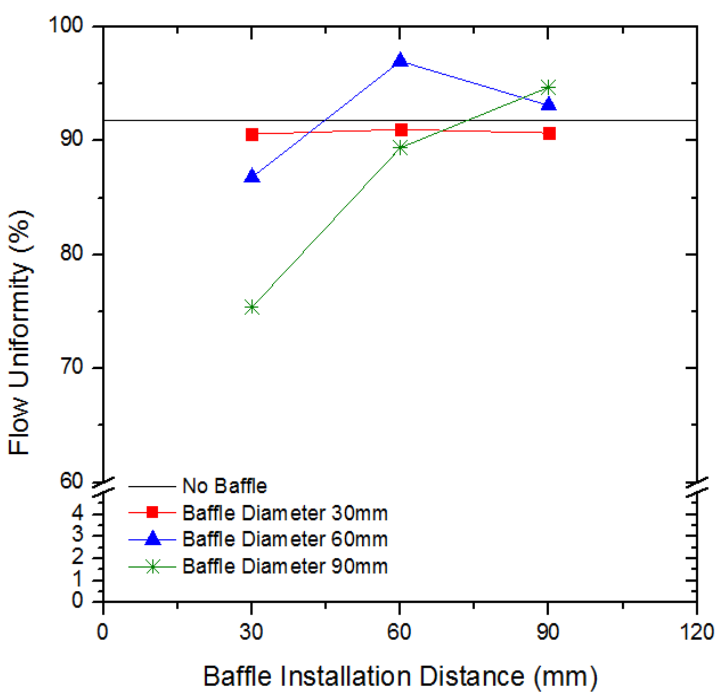
Baffle diameter 90 mm



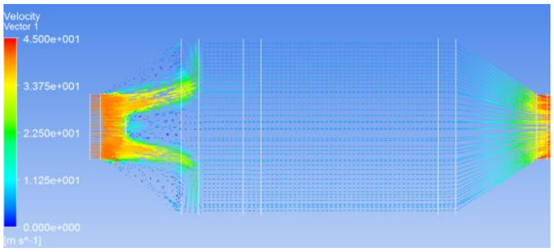
Flow velocity distribution for a baffle diameter of 60 mm according to the installation distance. *Flow velocity distribution range of 0–45 m/s.

4.4. Effects of the baffle installation distance on flow uniformity

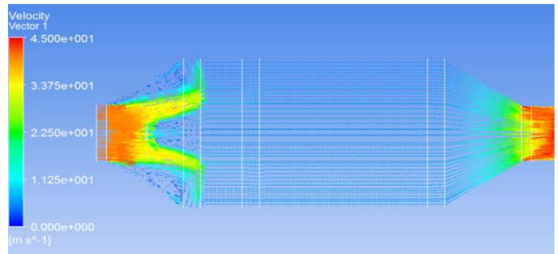
Figure 6 shows the effects of the baffle installation distance on the flow uniformity. As can be seen from the figure, the flow uniformity was 91.8% without a baffle, but was approximately 91.0% at all distances for a baffle diameter of 30 mm. For a baffle diameter of 60 mm, the flow uniformity was higher than 91.0% at installation distances equal to or longer than 60 mm. In particular, the flow uniformity was the highest (97.0%) at a distance of 60mm. As shown in Figure 9, the flow velocity distribution was the most uniform at a 60 mm distance, whereas the overall flow velocity distribution was not uniform at a 90 mm distance. This appears to be because the distance between the baffle and the DOC inlet was narrow, and thus a vortex occurred in the fluid flow in the back of the baffle. In addition, for a baffle diameter of 90 mm, the flow uniformity was approximately 94.7% at an installation distance of 90 mm. Figures 7–9 show the flow velocity distribution at installation distances of 60–90 mm according to the baffle diameter.



Effects of the baffle installation distance from the system inlet on the flow uniformity.

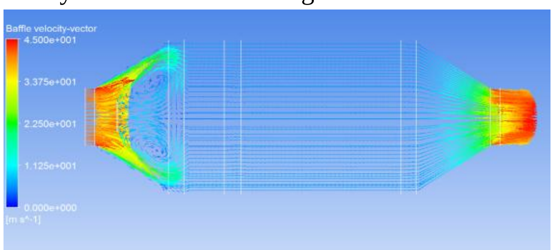


Installation distance of 60 mm

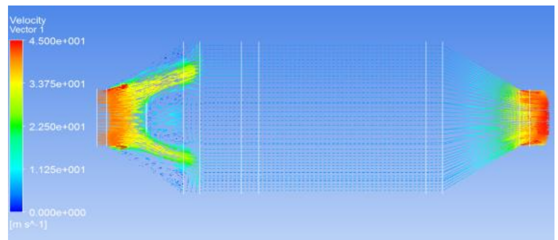


Installation distance of 90 mm

Flow velocity distribution according to the installation distance of a 30-mm diameter baffle.

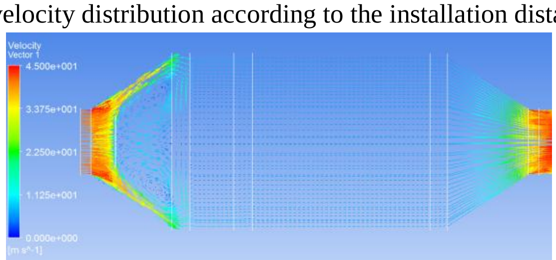


Installation distance of 60 mm

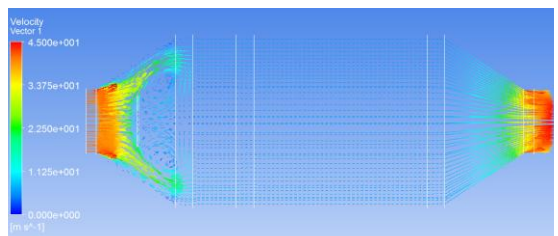


Installation distance of 90 mm

Flow velocity distribution according to the installation distance of a 60-mm diameter baffle.



Installation distance of 60 mm



Installation distance of 90 mm

Flow velocity distribution according to the installation distance of a 90-mm diameter baffle.

4.5. Effects of the length of the out-cone on the backpressure

Table 4 shows the pressure based on the position according to the length of the out-cone. As shown in the table, the pressure at the DOC inlet showed a tendency to decrease as the length of the out-cone increased, whereas the maximum pressure difference was approximately 160 Pa, indicating that the length of the out cone had almost no effect on the pressure drop.

Table -4 Pressure based on the position according to the length of the out cone

Category		Average pressure (pa)		
		Inlet	DOC inlet	Outlet
Out cone length (mm)	70	25,620	25,970	710
	140	25,560	25,830	580
	210	25,460	25,810	550

V. CONCLUSION

When an exhaust gas with a flow velocity of 41 m/s and a temperature of 310 °C was introduced into a 12-inchdiameter particulate matter (PM) and NOx reduction system with a diesel oxidation catalyst (DOC) and a diesel particulate filter (DPF), the pressuredrop and flow uniformity were simulated under various baffle installation and out-cone length conditions. The following results were obtained.

1. Regarding the effects of the presence of the baffle on the exhaust gas flow, the flow velocity was approximately twice as fast (16.2 m/s) compared to when no baffle was applied.This is because a fast flow velocity was introduced at the center when no baffle was applied, whereas the flow velocity distribution was dispersed from the center to the periphery when a baffle was installed.
2. Regarding the effects of the baffle installation distance on the pressure drop, the pressure drop was constant at approximately 25,500 Pa when the installation distance was equal to or longer than 60 mm. Therefore, it is necessary to set the baffle installation distance to equal or longer than this distance.
3. The experiment results of the average flow velocity according to the baffle size and installation distance revealed that the lowest average flow velocity (approximately 7 m/s) was observed when the baffle diameter was 60 mm and the installation distance was 60 mm. This velocity was approximately twice as low compared to when no baffle was used.
4. For the flow uniformity according to the baffle installation distance, it was approximately 91.8% without a baffle, but was the highest (approximately 97.0%) when the baffle diameter was 60 mm and the installation distance was 60 mm.
5. Regarding the effects of the length of the out-cone on the pressuredrop in the PM and NOx reduction system, the pressuredrop showed a tendency to decrease as the length of the out-cone increased, whereas the maximum pressure difference was only 160 Pa, indicating that the length of the out-cone had almost no effect on the pressure drop.

VI. ACKNOWLEDGEMENT

This study was conducted as a project titled “Development of Large Size Monolith Diesel Particulate Filter,” which is part of a top global environmental technology development project supported by the Korea Environmental Industry & Technology Institute under the Ministry of the Environment. We would like to express our sincere gratitude to Ceracomb Co. for their support.

VII. REFERENCE

[1] B.K. Choi and J.D. Cho, “Study on the Improvement of Uniformity of Inlet Velocity in Exhaust After-treatment System for System for Heavy Duty Engine,” KSAE, pp. 357-360, 2002.
[2] S.K. Jeong, W. Lee, G.S. Lee, K.H. Kim, S.H. Bae, and H.S. Kim, “A Study on Flow Characteristics in Diesel Particle Filter for Heavy-duty Diesel Engine,” KSAE, pp. 280-284, 2006.
[3] C.D. Lemme and W.R. Givens, “Flow through Catalytic Converters: An Analytical and Experimental Treatment,” SAE 740247, 1974.
[4] W.C Johnson and J.C. Chang, “Analytical Investigation of the Performance of Catalytic Monoliths of Varying Channel Geometries Based on Mass Controlling Conditional,” SAE 740196, 1974.
[5] M.C. Lai and J.Y. Kim, “Three-dimensional Simulations of Automotive Catalytic Converter Internal Flow,” SAE 9100200, 1991.
[6] A.J. Baxendale, “Computational Fluid Dynamics in Exhaust System Design and Development,” 94 Interanional E/G Design, Sterling Publication, Ltd., pp. 126-130, 1994.
[7] H. Welten et al., “Optimization of Catalytic Converter Gas Flow Distribution by CFD Prediction,” SAE 9300780, 1993.
[8] J.Y. Ahn, J.H. Ku, J.K. Park, and J.W. Kim, “A Study on the Pressure Drop and Flow Characteristics depending upon the Inlet-Outlet Geometry of Catalytic Converter,” KSAE, pp. 81-86, 2007.