2020 - A computational fluid dynamics simulation

by Aqli Mursadin

Submission date: 09-Oct-2021 07:48PM (UTC-0400)

Submission ID: 1669683988

File name: 2020_-_A_computational_fluid_dynamics_simulation.pdf (312.46K)

Word count: 1430

Character count: 6924

A computational fluid dynamics simulation of exhaust gas flow through adsorbent

Abdul Ghofur[1,a)] **and Aqli Mursadin**[2]

[1, 2] Department of Mechanical Engineering, Lambung Mangkurat University, Jalan A. Yani km 35.500, Banjarbaru 70714, Indonesia

E-mail: a) ghofur70@ulm.ac.id

Abstract. The use of adsorbent has been effective for reducing exhaust gas emission from combustion engines. However, it may cause an increase in pressure drop. This, in turn, may result in a number of problems such as low flow rate and, therefore, low heat transfer rate throughout the exhaust pipe. This paper aims at simulating the behaviour of such a flow by looking at how pressure drop varies along the control volume. It is based on computational fluid dynamics (CFD) modelling and the solver is implemented using ANSYS 2019 R3. It considers a number of variations in the size of adsorbent particles and the use of induced air on the outside of the flow. Three levels of adsorbent particle size are used: 30 mesh, 40 mesh, and 50 mesh. As for the induced air, the variable is the gauge pressure having three levels: 0.5 bar, 1.0 bar, and 1.5 bar. Engine speed, exhaust flow speed, and ambient temperature and pressure are assumed to be constant. The result suggests that variations in adsorbent particle size and induced air pressure are responsible for significant variations in pressure drop distribution along the flow. Finally, some directions for future research are recommended.

1. Introduction

Air pollutants due to transportation in Indonesia, according to [1], consist of CO (75.50 percent), HC (18.34 percent), NO_x (8.89 percent), SO_x (0.88 percent), and solid particles (1.33 percent). These gases are known to be harmful to human health in particular, as well as to the ecosystem and the environment in general. CO, for instance, can cause death to human.

Fossil fuel vehicles are among the main sources of CO emission. This is even worse for old vehicles with deteriorating engine function [2]. Considering the danger posed by these pollutants in high concentration, it is very important to be able to control their emission. One way to do that is by controlling how exhaust gas is emitted to the atmosphere. Passing the exhaust gas through an adsorbent containing active carbon is proven effective in reducing the pollutant concentration [3].

A research into the use of naturally available materials of adsorbent for that purpose is currently underway [4]. It is based on the exploitation of peat soil contents of active carbon. The potential of this material is quite promising. As part of this research, the dynamics of the flow system through such adsorbent is also being studied to determine how it performs in terms of losses in flow energy. This behavior is critical when it comes to maintaining the engine's performance since high pressure drop can simply lead to backflow, low flow rate, low heat transfer rate throughout the exhaust pipe, and, eventually, damage to the engine.

The purpose this paper is to report a successful series of simulation of exhaust gas flowing from a vehicle engine out through some adsorbent. In addition, to see how the flow can be affected by diffusion (either natural or forced), the behavior of the control volume is further manipulated by inducing air from the outside with varying levels of pressure.

2. Methods

The problem is modelled as an exhaust gas flow through a porous material having a cylindrical shape of a certain length (i.e. 8 cm) and diameter (i.e. 3 cm). The behaviour of the flow is further controlled by inducing air at the outside of the adsorbent. One particular properties of the fluid is observed, namely, the pressure drop along the control volume. It is further assumed that engine speed (3000 rpm), exhaust flow speed (2 m/s), and ambient temperature and pressure are constant. The adsorbent is placed at the point where the flow is about to reach the outside air.

The CFD simulation is carried out using the software ANSYS 2019 R3. Both the flow and the adsorbent material are modelled using axisymmetry and finite volume meshing. Induced air is simulated by setting the outside pressure boundary values.

Hence, two input variables are considered for variation, i.e. size of adsorbent particles, and gauge pressure of induced air. Levels of the input variables are as follows:

- size of adsorbent particles: 30 mesh (595 μm), 40 mesh (400 μm), and 50 mesh (297 μm); and
- gauge pressure of induced air: 0.5 bar, 1.0 bar, and 1.5 bar.

The resulting pressure drop values will be observed and recorded. The distribution of these values will be visualized throughout the control volume.

3. Result and Discussion

Exhaust gas entering the adsorbent can have a gauge pressure of nearly 0. This is due to the model that the adsorbent is right at the end of the flow. It is not surprising that, at the outlet end of the adsorbent, the gauge pressure can be negative. At this point, of course, the flow can only leave the adsorbent if there is enough pressure buildup pushing from the upstream or if there is some effective pressure due to diffusion from the induced air.

Table 1 gives the summary of the output values (see also the flow pressure visualization in the Appendix). It can be seen that variation in pressure drop is related to variation in particle size. The pressure drop tends to increase as the size decreases. This has to be the case since the smaller the particle is, the more compact the material becomes and, therefore, the harder it is for the fluid to pass through.

Table 1. Output summary.

Adsorbent Particle Size (mesh)	Induced Air Pressure (barG)	Average Pressure Drop (10^{-3} barG)
30	0.5	0.982
	1.0	0.983
	1.5	0.979
40	0.5	2.736
	1.0	2.730
	1.5	2.764
50	0.5	3.852
	1.0	3.754
	1.5	3.782

On the other hand, variation in induced air pressure does not significantly lead to variation in pressure drop. There is no sign of any reliable pattern either between the two variables. For instance, an increasing series in induced air pressure values leads to a fluctuation in pressure drop and nothing more. While the boundary pressure applied through the induced air was certainly higher at any point than the exhaust gas pressure, it appears that the diffusion of the air through the porous medium (if it did take place) was not so effective in helping relieve the flow through the adsorbent. Hence, it appears that the flow is able to leave the adsorbent mostly due to some pressure buildup which is transient rather than its own steady-state energy.

There are, of course, several limitations in this simulation. It did not consider a particular direction of the induced air nor did it specifically put a measure to model the diffusion. This, however, should be a concern for future investigation.

4. Conclusion

A series of CFD simulation has been performed on exhaust gas flow through adsorbent. Two input variables are considered: size of adsorbent particles and gauge pressure of induced air. The output variable is pressure drop along the flow. The result suggests that pressure drop tends to increase as size of adsorbent particles decreases. Gauge pressure of induced air, on the other hand, does not seem to significantly affect the pressure drop.

It is recommended that the modelling should also accommodate the flow direction of the induced air. Another possible direction would be to include the heat transfer behavior of the adsorbent medium itself.

5. References

[1] Wardhana, W A 2001 Dampak Pencemaran Lingkungan (Yogyakarta: Penerbit Andi)

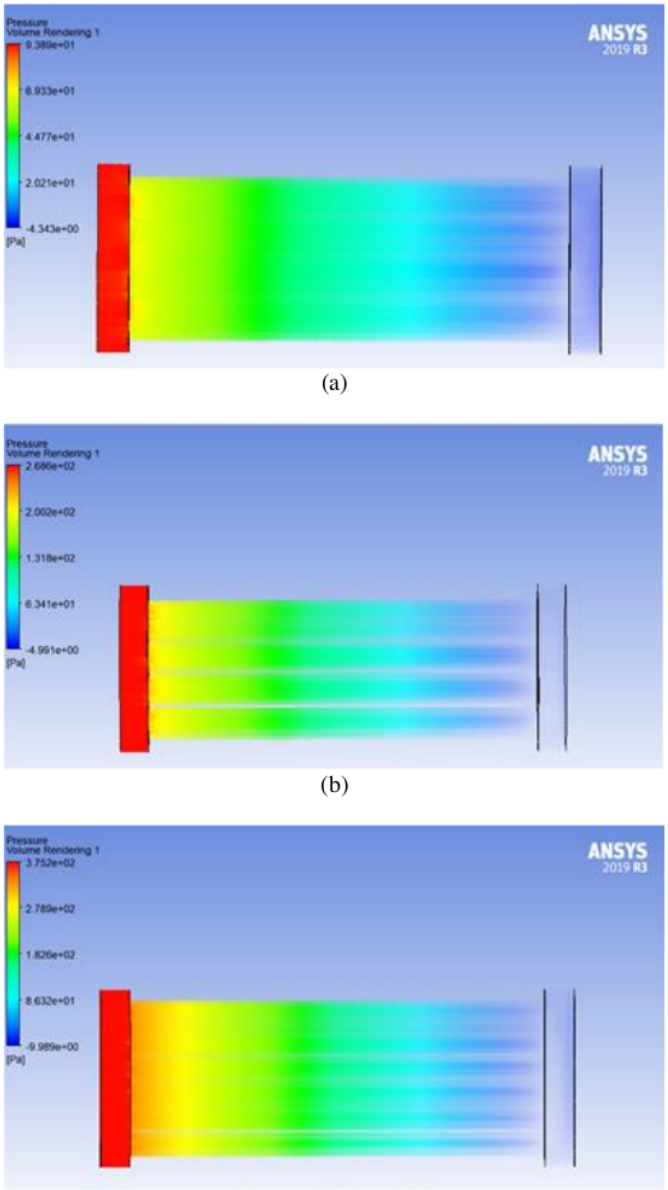
[2] Basuki, K T, Setiawan, B, and Nurimaniwathy 2008 Penurunan Konsentrasi CO dan NO_2 pada Emisi Gas Buang Menggunakan, Arang Tempurung Kelapa yang Disisipi TiO_2, paper presented at Seminar Nasional IV SDM Teknologi Nuklir

[3] Maryanto, D, Mulasari, S A, and Suryani, D 2009 KESMAS UAD vol 3 (3) p 198

[4] Ghofur, A, Mursadin, A, and Amrullah, A … Ongoing research project, …

[5] Soemardi, T P, Siswantara, A I, and Erwin Jurnal Makara, Teknologi vol 7 (2) p …

6. Appendix



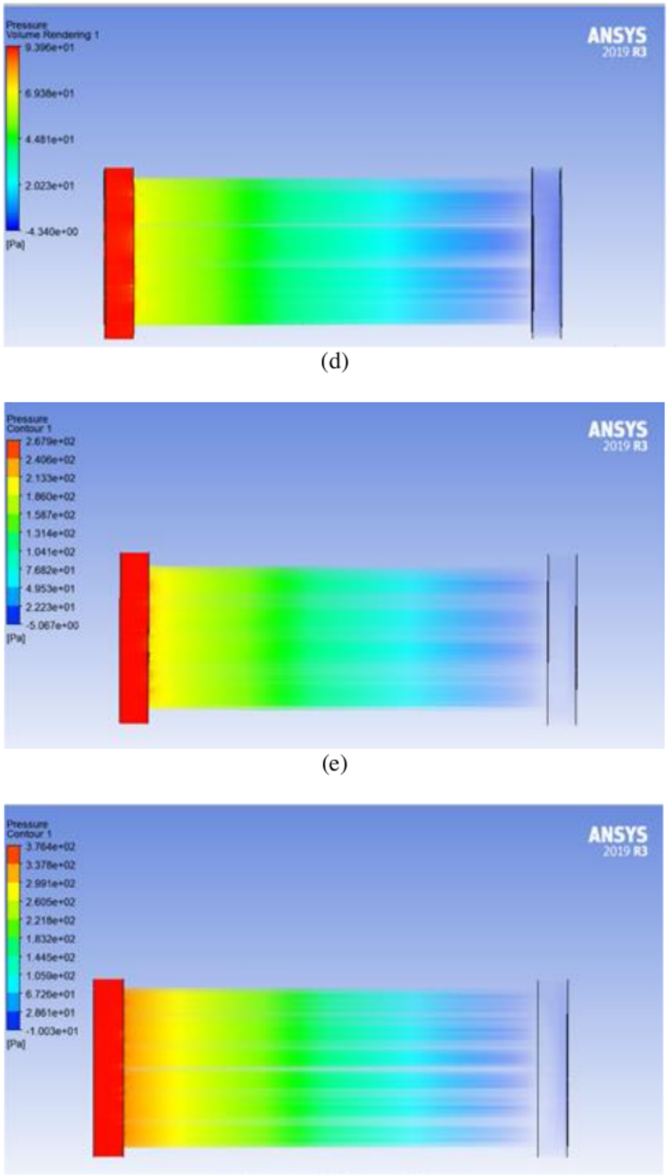
(a)

(b)

(c)

Figure A. Output visualization (*continued*).

(a) 0.5 barG, 30 mesh; (b) 0.5 barG, 40 mesh; (c) 0.5 barG, 50 mesh



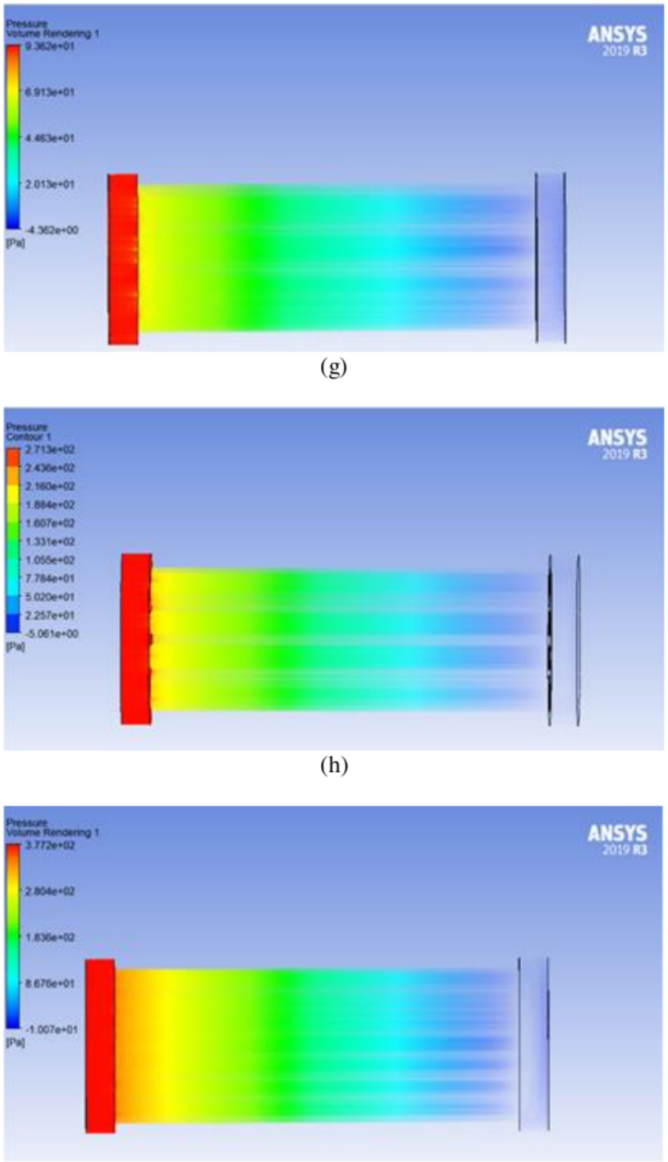
(d)

(e)

(f)

Figure A. (*continued*).

(d) 1.0 barG, 30 mesh; (e) 1.0 barG, 40 mesh; (f) 1.0 barG, 50 mesh



(g)

(h)

(i)

Figure A. (*continued*).

(g) 1.5 barG, 30 mesh; (h) 1.5 barG, 40 mesh; (i) 1.5 barG, 50 mesh

2020 - A computational fluid dynamics simulation

ORIGINALITY REPORT

0%	%	0%	%
SIMILARITY INDEX	INTERNET SOURCES	PUBLICATIONS	STUDENT PAPERS

PRIMARY SOURCES

Exclude quotes	On	Exclude matches	Off
Exclude bibliography	On		