Analysis of Swirling Flow on Cyclone Separator Using CFD to Reduce Particulate Matter of Diesel Engine

S. Nurcholik[1], L. Adnyani[1], D. Sa'adiyah[2], L. Rahmah[2], R. Revari[3]
[1]Departement of Naval Architecture, Kalimantan Institute of Technology, Balikpapan Email:dlukha@itk.ac.id
[1]Departement of Naval Architecture, Kalimantan Institute of Technology, Balikpapan Email:luhputria@itk.ac.id
[2]Departement of Mechanical Engineering, Kalimantan Institute of Technology, Balikpapan Email:devysetiorini@itk.ac.id
[2]Departement of Mechanical Engineering, Kalimantan Institute of Technology, Balikpapan. Email:03131005@itk.ac.id
[3]Departement of Engineering, Indonesia Power, Semarang. Email:rezarevari@gmail.com

Abstract
Particulate matter (PM) is one of the component in flue gas of diesel engine. As one of air pollution, PM needs more attention, because its existence irritates respiratory. The using of cyclone as additional part of diesel engine, can reduce PM concentration before released to surrounding. However, the shape of cyclone affects the percentage of PM due to turbulence and length of track. In this paper, the prediction of swirling flow in different type of cyclone separator will be conducted by using Computational Fluid Dynamics (CFD). The analysis will focus on particle movement inside the cyclone separator and its turbulence phenomenon. There are four types of cyclone separator which will be observed: Perry's method, Stairmand's method, and the modification of each methods. All of these cyclone separator will have the same velocity inlet, and will be simulated using RNG k-ε model as the turbulence modelling. The simulation shows that the Stairmand's method has the best turbulence and can collect most of PM.

Keywords: Cyclone separator, Simulation, CFD, Turbulence, Particulate matter.

Abstrak
Particulate matter (PM) adalah salah satu komponen yang terdapat pada gas buang hasil dari mesin diesel. Sebagai salah satu jenis pencemar udara, PM membutuhkan perhatian khusus, karena keberadaannya yang mengganggu sistem pernapasan. Penggunaan cyclone sebagai bagian tambahan dari mesin diesel, dapat mengurangi konsentrasi PM yang tersebar ke sekliling. Bagaimanapun, bentuk dari cyclone separator mempengaruhi persentase dari PM yang dikarenakan oleh turbulensi dan panjang lintasan. Pada tulisan ilmiah ini, prediksi dari pergerakan partikel pada cyclone separator dengan berbagai macam tipe akan diamati menggunakan Computational Fluid Dynamics (CFD). Analisis yang dilakukan akan berfokus pada pergerakan partikel di dalam cyclone separator dan fenomena turbulensi yang terjadi. Terdapat empat tipe cyclone separator yang akan diamati, yaitu : Perry, Stairmand, dan modifikasi dari masing-masing metode. Semua cyclone separator memiliki kecepatan partikel yang sama,, dan akan disimulasikan menggunakan model turbulen k—ε RNG. Hasil simulasi menunjukkan bahwa metode Stairmand memiliki turbulensi yang terbaik dan dapat mengumpulkan paling banyak PM.

Kata Kunci: Cyclone separator, simulasi, CFD, turbulensi, particulate matter

1. Introduction

Diesel engines are still dominant used for transportation due to its high efficiency, performance, and durability. However, Diesel engine contribute to produce pollution such as NOx, SOx and Particulate Matter (PM). PM has serious effect that caused human dieses such as cancer, asthma, and lung diseases. Development of PM is the effect of formation carbon soot and hydrocarbons condensation (Luo et al. 2015) which produce inorganic ion, organic carbon, element carbon (Gen, 2016) from combustion of fuel rich region (Savic at all, 2016).

Several methods have been observed to reduce PM of diesel engine such ultra-high fuel injection pressure and low temperature carbon. However, this method need some additional tool to the combustion chamber. Cyclone is another method which can be applied to the engine due to it has no moving parts and relatively low initial cost (Fung, 2014).

The trapping ability of cyclone can be influenced by physical properties, dispersity, and the shape of PM. Whereas the structure and geometrical parameters also have an impact on the trapping ability (Holbrow, 2013). Furthermore, the efficient trapping is affected by properties of gas such as chemical composition, temperature, and velocity (Sobolev 2011).

Cyclone design and optimization can be obtained by Computational Fluid Dynamic (CFD). CFD simulation can predict flow field characteristic and particle trajectories inside a cyclone separator (Hsiao et al 2009). Viscous flow inside a cyclone separator divided by two types: primary and secondary. The flow is the swirling flow in the cyclone and fortex finder. While secondary flow consists of secondary eddies interacting between the primary flow and processing fortex core (sun, 2017). According to the flow, the exhaust gas containing particulate enters tangentially in to the cyclone body forming a spiral flow finally exit trough the opened center body. in other hand, the PM would fall down to the exhaust collector in the bottom of cyclone (Buekers, 2012).

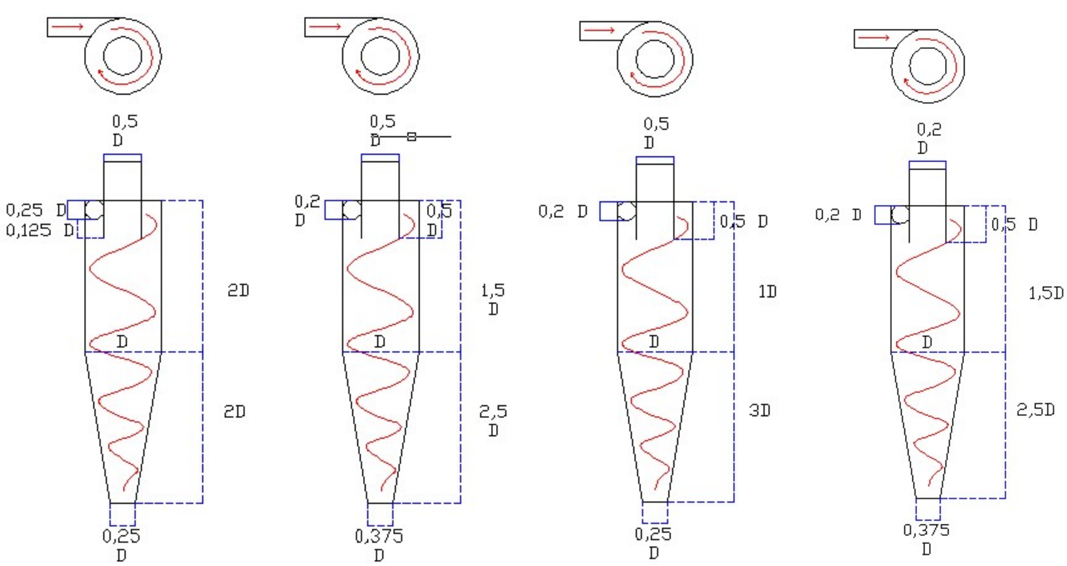
The aim of this paper is to predict swirling flow in the different cyclone design of Stairmind (1949), Perry (1999), modification 1, and modification 2. The analysis will focus on turbulence phenomenon by particle movement inside the cyclone.

2. Methods

2.1. Cyclone separator types

The three-dimenstional of cyclone separator is drawn in accordance with the type of each cases. Four types of cyclone separator was investigated in this paper : Perry's method, Stairmand's method, and the modification of each methods. The third and fourth type namely modification 1 and modification 2, which each of the types are modification of Perry's method and Stairmand's method, respectively. The shapes and dimensions of each types can be seen in Figure 1.

:



(a) (b) (c) (d)

Figure 1. The Shape and Dimension of : (a) Perry's Method (1999); (b) Stairmand's Method (1983); (c) Modification 1; and (d) Modification 2.

The geometry then will be meshed before simulated. The meshing using tet/hybrid as can be seen in Figure 2. Boundary conditions in this cyclone separator will determined as velocity inlet for inlet and outflow for outlet. The diameter of inlet adjusted with the diameter of exhaust, 3 cm.

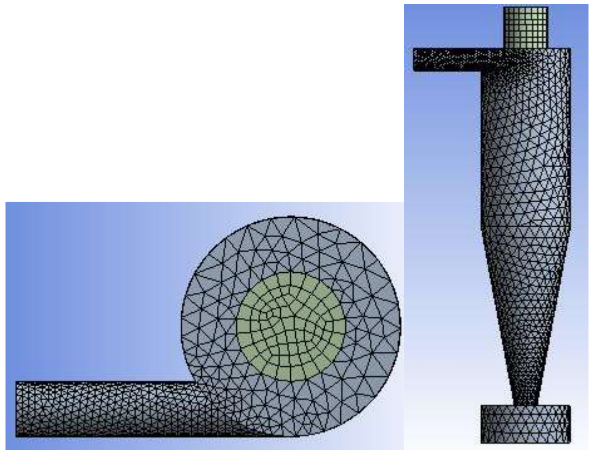


Figure 2. Meshing Example of Perry's Method

2.2. Computational Approach

The simulation of cyclone separator is using Computational Fluid Dynamics (CFD). The simulation parameter is as shown in Table 1. The numerical simulation will not added energy equation, because there is no heat transfer during process.

Table 1. Simulation Parameter

Data	**unit**	
Flue gas flow rate	kg/s	7.34×10^{-3}
Flue gas density	kg/m³	8.64×10^{-1}
PM diameter	µm	0,5
PM density	mg/m³	20
Inlet diameter	cm	3

The turbulence model for this simulation is using Renormalization Group (RNG) k-ε model, which is a model based on model transport equations for the turbulence kinetic energy (*k*) and its dissipation rate (ε). This turbulence model allow the effect of swirl in turbulence. The transport equation for this turbulence model are given as follows :

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b + \rho\varepsilon + Y_M + S_k \quad (1)$$

And

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (2)$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. The quantities α_k and α_ε are the inverse effective Prandtl numbers for k and ε, respectively. S_k and S_ε are user-defined source terms. The turbulent (or eddy) viscosity, μ_t, is computed by combining k and ε as follows:

$$\mu_t = \rho C_\mu \frac{c^2}{\varepsilon} \quad (3)$$

where C_μ = 0.0845, derived using RNG theory. The model constants $C_{1\varepsilon}, C_{2\varepsilon}, C_\mu, \sigma_k$ and σ_ε have the following defaults values: $C_{1\varepsilon} = 1.42$, $C_{2\varepsilon} = 1.68$. The RNG model provides an option to account for the effects of swirl or rotation by modifying the turbulent viscosity appropriately. The modification takes the following functional form :

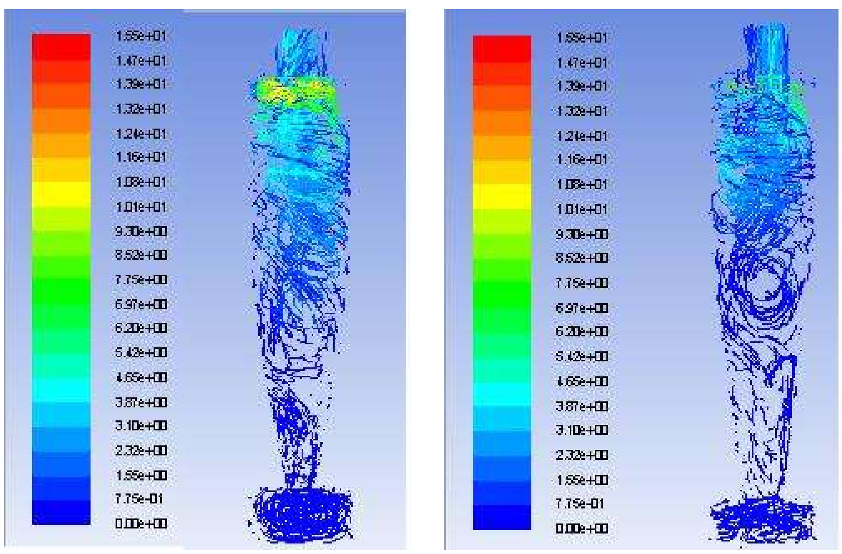
$$\mu_t = \mu_{t0} f\left(\alpha_s, \Omega, \frac{k}{\varepsilon}\right) \tag{4}$$

where μ_t is the value of turbulent viscosity calculated without the swirl modification. Ω is a characteristic swirl number, and $\alpha_s$is a swirl constant that assumes different values depending on whether the flow is swirl-dominated and three-dimensional flows when the RNG model is selected. For midly swirling flows, α_s is set to 0.007.

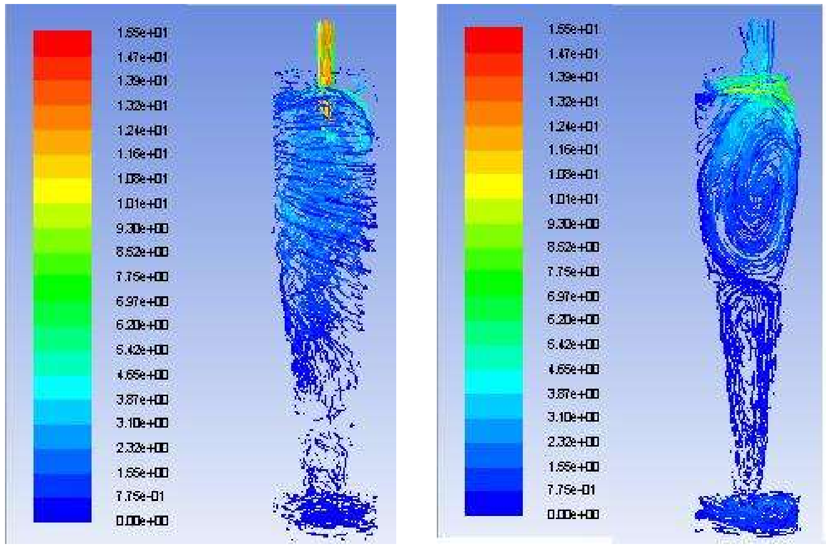
The investigation will focus on swirling flow in cyclone separator. The the particle pathline based on velocity will indicate the turbulence phenomenon.

3. Result and Discussion

The particle pathline for each type can be seen in Figure 3. The gradiation of color in the left side of figure represent the velocity, dark blue for 0 m/s to red for 15.5 m/s. The concept of cyclone separator is using centrifugal force to separate the particulate matter. PM with high density will fall to lower body of cyclone, while clean gas will lifted toward outlet. However, the shape and the length of body effect the behavior of PM inside cyclone.



(a) (b)



(c) (d)

Figure 3. The Particle Pathline Based On Velocity of : (a) Perry's Method; (b) Stairmand's Method; (c) Modification 1; (d) Modification 2.

From Figure 3 (a), the particle pathline shows high velocity and swirling flow near inlet area in Perry's method cyclone. It means this cyclone type has high turbulence. High turbulence will hold PM to high area, only a few of PM will fall to lower body, so the flow at the outlet will contain high PM. Compared to Stairmand's method type in Figure 3 (b), the velocity at high area has lower value, the turbulence is lower for this cyclone type. Then it will make more PM will fall to lower body, so the flow at outlet area cleaner.

Figure 3 (c) & 3 (d) shows the particle pathline in cyclone type with modification of Perry's method and Stairmand's method, respectively. It shows that for these two cases, high turbulence occur at the high body area, even has very high turbulence with not only move centrifugally but also move up and down to the middle area for the cyclone of modification Stairmand's method, Figure 3 (d). From these two turbulence phenomenon, PM will have difficulty to move to lower body, then again the gas flow will contain high PM concentration at the outlet.

The comparison of the four cases shows that Stairmand's method of cyclone has the lowest turbulence at the high area, so that PM will move to lower body easily. Then the gas flow at the outlet more clean Than the other cases.

4. Conclusion

The simulation of PM behavior in different type of cyclone separator has been done by using CFD. From the results, each type has different particle movement due to the shape of cyclone. However, Stairmand's method has the lowest turbulence and it effects the PM movement, the swirling flow will lead PM to lower body, so only gas flow with few PM will move out to outlet.

Acknowledgments

The authors wish to recognize the outstanding contributions of their technical staff at the internal combustion engine laboratory, Department of Marine Engineering, Sepuluh Nopember Institute of Technology (ITS) Indonesia.

References

Buekers, A., (2012) 'Mechanical and cyclonic separator', Department of Chemical Engineering, Brussel University, Belgium.

Funk, P., Holt, G., and Whitelock, D. (2014) 'Novel cyclone empirical pressure drop and emissions with heterogeneous particle'. *Journal of Aerosol Science*, Vol.74: 26–35.

Geng, P., Cao, E., Tan, Q., and Wei, L. (2017) 'Effects of alternative fuels on the combustion characteristics and emission products from diesel engines: A review', *Renewable and Sustainable Energy Reviews*, Vol.71: 523–534.

Holbrow, P., (2013) 'Dust explosion venting of small vessel and flamless venting', *Proses Safety and Environmental Protection*, Vol.91: 183–190.

Hsiao, T., Chen, D., and Son, Y., (2009) 'Development of mini-cyclones as the size-selective inlet of miniature particle detectors', *Journal of Aerosol Science*, Vol.40: 481-491.

Luo, Y., Zhu, L., Fang, J., Zhuang, Z., Guan, C., Xia, C., Xie, X., and Huang, Z. (2015) 'Size distribution, chemical composition, and oxidation reactivity of particulate matter from gasoline direct injection (GDI) engine fueled with ethanol-gasoline fuel', *Applied Thermal Engineering*, Vol.7: 647–655.

Perry, H., (1999) 'Chemical Engineers Handbook 7th Edition, McGraw-Hill, Kogakusha, Ltd, Tokyo.

Savic, N., Rahman, M., Miljevic, B., Saathoff, H., Naumann, H., Leisner, T., Richies, J., Gupta, B., Motta, N., and Goyal, R., (2014) 'Influence of biodiesel fuel composition on the morphology and microstructure of particles emitted from diesel engine', *Carbon*, Vol.104: 179–189.

Sun, X., Zhang, Z., and Chen, R., (2017) 'Numerical modeling of miniature cyclone', *Powder Technology*, Vol.380: 325–339.

Sobolev, A., Melnikov, A., and Tiutiunyyk, O., (2011) 'The movement of particles in the air stream', *Mechanical Engineering*, Vol.17: 82–86.

Stairmand, C., (1949) 'Pressure drop in cyclone separators', *Engineering*, 168, 409.