Numerical analysis of air pollutant dispersion in steam power plant area using the finite volume method

A Fatahillah1,a, M A Masyhudi1 and T B Setiawan1

1 University of Jember, Jember - Indonesia

E-mail: a.arif.fkip@unej.ac.id

Abstract. In this research a mathematical model will be developed for the distribution of air pollutants. The model is built on the basis of momentum equations and mass continuity equations solved using the finite volume method. Based on the model obtained, the effect of wind speed and initial concentration on the distribution of pollutants in the air will be analyzed. An air pollutant with a certain concentration released through the chimney into the atmosphere will undergo a process of transportation and transformation. The characteristics of emission sources such as concentration when the emission and wind speed around the chimney really determine the spreading of the concentration. The case study in this research is located in the Paiton-Probolinggo Steam Power Plant area. The analysis of the distribution of pollutants was solved numerically using the Gauss Seidel iteration method assisted by MATLAB software and simulated using a CFD approach assisted by FLUENT software. The results obtained in this research indicated that the greater the wind speed around the chimney, the lower the concentration of SO2 pollutants in the air, while the effect of the initial concentration showed that the greater the initial concentration, the greater the concentration of pollutants scattered in the air.

1. Introductions

The Steam Power Plant is one industry which serves to meet electricity needs. The electrical energy is a vital energy needed by humans to support their needs in various aspects of life. But in the process of producing electricity, Steam Power Plant activities also have a negative impact on the environment due to the pollutants it produces. As we know now, air pollution is a big with a strong impact about the environment [1]. The use of fuel in the form of coal is one of the causes. Coal combustion in addition to producing heat will also produce sources of emissions or air pollutants in the form of sulfur dioxide (SO2) gases released through the factory chimney.

In Indonesia, air pollutants or exhaust emissions from industry contribute around 10-15% of air pollution [2]. Air pollutant with certain concentrations released through the chimney into the air will undergo transportation and transformation processes in the atmosphere [3]. The characteristics of emission sources such as concentration and discharge determine the spreading of these pollutants. The influential factor is the wind. The wind is air that moves due to the difference in air pressure with the direction of wind flow from a place that has high pressure to a place that is low pressure or from an area that has low temperature to a high temperature area. The wind has direction and speed. The direction of the wind plays an important role in bringing it to the direction where the pollutant is dispersed, while the wind speed influences the concentration of the pollutant when dispersed.
Xiaofei in his research entitled "Modeling of pollutant dispersion with atmospheric instability in an industrial park" has simulated pollutant pollution in the industry Park with the effect of the Combined Large Eddy Simulation (LES) with scale models Smagorinsky Subgrid is dynamic. In conclusion, the direction of pollutant dispersion profile is different when passing through the plant canopy under unstable and neutral weather conditions, this proves that the plant can protect the downstream area in unstable atmospheric conditions [4].

In numerical processes both meshing and iteration, the equations used are fluid regulating partial differential equations, which originate from the basic laws of physics [5]. In this research, the governing equation for this model are the momentum equation and the mass continuity equation.

\[
\frac{\partial \rho C_0}{\partial t} + \nabla \rho u_i u_j = - \frac{\nu P}{\rho} + \nu_d \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{\partial \tau_{ij}}{\partial x_j} + f_d \quad (1)
\]

\[
\frac{\partial \rho C_0}{\partial t} + \nabla u_i u_j = D \frac{\partial^2 C_0}{\partial x_j \partial x_j} + \frac{\partial J_A}{\partial x_j} + q_c \quad (2)
\]

Equation (1) is a momentum equation and Equation (2) is a mass continuity equation. Where \( C_0 \) is the initial concentration, \( \rho \) is density, \( P \) is pressure, \( \nu_d \) is dynamic viscosity, \( u \) is the wind speed, \( v \) is the jet velocity, \( D \) is the molecular mass, \( f_d \) is the drag force, \( J_A \) is the amount of mass flux and \( q_c \) is the intensity of contamination.

A mathematical model was built using the finite volume method. The finite volume method is based on an integral form of conservation law. The finite volume method is chosen because it can be used in irregular objects so that it becomes more easily discretized to determine the values to be sought in the discretization process. So that in completing mathematical equations it will be easier and the solutions obtained approach true values. The discretization technique used is QUICK discretization. The model obtained is then solved numerically using the Gauss Seidel iteration method assisted by MATLAB software. In addition, the SO\(_2\) pollutant dispersion pattern is also simulated using Computational Fluid Dynamics (CFD) gas flow that moves into the atmosphere through the chimney based on the influence of differences in wind speed, initial concentration. Computational Fluid Dynamics (CFD) is a very powerful tool extensively used nowadays for dealing with the micro-scale simulation of pollutants dispersion in environments [6], [7], [8]. Computational fluid dynamics (CFD) is one of the branches of fluid mechanics predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process [9]. GAMBIT and FLUENT are some CFD based software because of their ease of use and fluid flow analysis with good results [10].

2. Research method

Mathematical modelling is a branch of mathematical field, can represent and explain the physical systems or the problem in the real life into mathematical expression. Representations of real systems that are described in the form of symbols and mathematical statements are mathematical models. The mathematical model in this research was built based on the momentum equation and the mass continuity equation which were then solved using the finite volume method. The finite volume method is locally conservative because it is based on a "balance" approach: a local balance is written on each discretization cell which is often called "control volume"; by the divergence formula, an integral formulation of the fluxes over the boundary of the control volume is then obtained. The fluxes on the boundary are discretized with respect to the discrete unknowns [11].

The method employed in this research is simulation. A research requires a procedure that contains a series of events until the results are obtained or data analyzed to reach conclusions related to the research objectives. The procedure applied in this research is to doing a literature review of the factors that influence the spread of air pollutants. Then build a mathematical model of air pollutants dispersion from the equation of momentum and equation of mass continuity. After the mathematical model is obtained, the mathematical model will be solved using the finite volume method the QUICK discretization approach. Then made a computer program with MATLAB and FLUENT to simulate the process of spreading air pollutants into the environment. The next step in this research is to analyze
the relationship between the influence of wind direction and speed on the spread of pollutants into the environment and the effect of the initial concentration of pollutants on the spread of pollutants into the environment. The final step in this research is to evaluate the test results and then provide conclusions and results.

3. Results and discussion
This research aims to develop mathematical models of pollutant dispersions. The study of mathematical models of dispersion pollutants is built on the basis of momentum equations and mass continuity equations solved using the finite volume method. Then, the equation will be solved by the QUICK discretization and simulated by MATLAB and FLUENT software. After collecting data from various literature, data is obtained in table 1.

| Table 1. Research Data |
|------------------------|
| Explanation | value |
| density ($\rho$) | 2.63 kg/m$^3$ |
| pressure ($P$) | 40 Pascal |
| dynamic viscosity ($\nu_d$) | $2.1 \times 10^{-5}$ kg/ms |
| drag force ($f_d$) | 0.00023 N |
| intensity of contamination ($q_c$) | 0.0073 kg/m$^3$s |

The data which has been obtained then entered into the mathematical model variables found in MATLAB.

The first case was analyzed by the influence of wind speed on the distribution of pollutant concentrations. The MATLAB simulation results with the influence of several different wind speeds are presented in table 2.

| Table 2. The maximum concentration of SO$_2$ pollutants is influence by different wind speeds |
|---|
| Case | wind speeds (m/s) | concentrations of SO$_2$ pollutant (mg/Nm$^3$) |
| 1 | 7.0 | 15,968 |
| 2 | 6.8 | 17,153 |
| 3 | 6.6 | 18,256 |
| 4 | 6.5 | 19,279 |
| 5 | 6.3 | 20,188 |
| 6 | 6.1 | 20,952 |
| 7 | 6.0 | 21,557 |

The second case was analyzed by the effect of the initial concentration on the distribution of SO$_2$ pollutant concentrations. The MATLAB simulation results with the influence of several different initial concentrations are presented in table 3.

| Table 3. The maximum concentration of SO$_2$ pollutants is influence by different initial concentrations |
|---|
| Case | initial concentrations (mg/Nm$^3$) | concentrations of SO$_2$ pollutant (mg/Nm$^3$) |
| 1 | 4.00 | 11,336 |
| 2 | 4.05 | 14,383 |
| 3 | 4.11 | 16,368 |
| 4 | 4.16 | 17,754 |
| 5 | 4.22 | 18,756 |
| 6 | 4.27 | 19,494 |
| 7 | 4.33 | 22,291 |

The simulation on FLUENT is presented in the geometric form of the simulation results of SO$_2$ pollutant spread from the chimney of the Paiton Power Steam Plant. The simulations shown include the spread of concentration from SO$_2$. The simulation images generated are based on object formation,
definition, and mesh with GAMBIT software, then simulated using FLUENT software. The geometry design in GAMBIT is presented in Figure 3 below.

After constructing the computational geometry design, the mesh generation is then performed and determine boundary types on each face. The last stage is simulating the FLUENT program.

1. The simulation of the Effect of Wind Speed on SO₂ Pollutant Concentration Distribution

   (a)  
   (b)  
   (c)  

   Figure 3. Contour Distribution of Sulfur Dioxide Pollutants Based on Wind Speed (a) 5.5 m/s, (b) 6.25 m/s, (c) 7 m/s.

2. The simulation of the Effect of Initial Concentration on SO₂ Pollutant Concentration Distribution

   (a)  
   (b)  
   (c)  

   Figure 4. Contour Distribution of Sulfur Dioxide Pollutants Based on Initial Concentration (a), 4 mg/Nm³, (b) 4.25 mg/Nm³, (c) 4.5 mg/Nm³

   Based on MATLAB and FLUENT simulations, the effect of wind speed on the distribution of SO₂ pollutant concentrations shows that the greater the wind speed around the chimney, the lower the concentration of SO₂ pollutants. This is in accordance with Tasic's statement which states that high wind speeds reduce concentration due to the dilution effect [13]. While the MATLAB and FLUENT simulation results with the effect of the initial concentration on the distribution of SO₂ pollutant concentrations showed that the greater the initial concentration, the greater the concentration of SO₂ pollutants spread in the air.

3. Display of Mapping of SO₂ Pollutant Spreads Around the Paiton-Probolinggo Power Steam Plant

   The output of the simulation of SO₂ pollutant dispersion assisted by FLUENT software is combined with the mapping display of the area around the Paiton-Probolinggo Power Steam Plant using Google Earth. The first case is simulated the distribution of SO₂ pollutants based on the
influence of local winds, namely land wind and sea breeze. The simulation results are obtained in Figure 5.

![Figure 5](image1.png)

**Figure 5.** Simulation of the Contour Distribution of Sulfur Dioxide Pollutants by Local Wind Direction: (a) Land Winds and (b) Sea Winds

Based on the influence of land winds such as Figure 5 (a), the direction of the distribution of SO$_2$ pollutants is carried from the land to the sea which generally occurs at night ranging from 08.00 pm to 06.00 am. Whereas sea winds as in Figure 5 (b) carry SO$_2$ pollutants from the sea to the land which generally occur during the day ranging from 09.00 am to 04.00 pm.

The second case simulated the distribution of SO$_2$ pollutants based on the influence of monsoons, namely the west monsoon and the east monsoon. The simulation results are obtained in Figure 6.

![Figure 6](image2.png)

**Figure 6.** Simulation of the Contour Distribution of Sulfur Dioxide Pollutants by Wind Direction: (a) West Musoosons and (b) East Musoosons

Based on the influence of the western monsoon like Figure 6 (a), the direction of the distribution of SO$_2$ pollutants is carried from the northwest to the southeast. The west monsoon winds that occur in October-March with an average wind speed of 4.5 m/s. Whereas the east monsoon as shown in Figure 6 (b) carries SO$_2$ pollutants from the southeast to the northwest. East monsoon winds occur in April-September with an average wind speed of 5 m/s.

4. **Conclusion and suggestion**

Based on the above discussion on the influence of wind speed toward the distribution of SO$_2$ pollutant concentrations showed that the greater the wind speed around the chimney, the lower the concentration of SO$_2$ pollutants in the air. While the effect of the initial concentration on the distribution of SO$_2$ pollutant concentrations showed that the greater the initial concentration is, the greater the concentration of SO$_2$ pollutants scattered in the air.

Based on the results of research which has been done, the further research is needed with more complex pollutant objects and simulation of air pollutants from steam power plant can be done using the other MATLAB and FLUENT simulation software.
Acknowledgement
We would like to thank to Jember University, Indonesia, which offers grants and opportunities to disseminate intellectual results from our research.

References
[1] Clark P U, Shakun J D, Marcott S A, Mix A C, Eby M, Kulp S, Levermann A., Milne G A, Pfister P L, Santer B D, Schrag D P, Solomon S, Stocker T F, Strauss B H, Weaver A J, Winkelmann R, Archer D, Bard E, Goldner A, Lambeck K, Pierreumbert R T and Plattner G K 2016 Consequences of twenty-first-century policy for multi-millennial climate and sea-level change Nat. Clim. Change 6 pp 360–369
[2] Fatahillah A 2018 Numerical analysis of fluidal flow in heat exchangers using finite element method to reduce exhaust emission level in air J. Phys. Conf. Ser 1108 (2018) 012089 pp 1-7
[3] Pierce A M, Loría-salazar S M, Holmes H A and Gustin M S 2019 Investigating horizontal and vertical pollution gradients in the atmosphere associated with an urban location in complex terrain , Reno , Nevada , USA Atmos. Environ pp 103–117
[4] Ma X, Zhong W, Feng W, and Li G 2017 Modelling of Pollutant Dispersion with Atmospheric Instabilities in an Industrial Park Powder Technol pp 577–588
[5] Ghaustsun A, Boro K, and Sabdo A 2009 Simulasi Dispersi Gas Polutan dari Cerobong ke Lingkungan dengan Pendekatan Computational Fluid Dynamics (CFD) Pros. Semin. Nas. Himpun. Inform. Pertan. Indones pp. 978–979
[6] Toja-silva F, Pregel-hoderlein C and Chen J 2018 On the urban geometry generalization for CFD simulation of gas dispersion from chimneys: Comparison with Gaussian plume model J. Wind Eng. Ind. Aerodyn pp 1–18
[7] Lateb M, Meroney R N, Yataghene M, Fellouah H, Saleh F and Boufade M C 2016 On the use of numerical modelling for near-field pollutant dispersion in urban environments - a review Environmental Pollution 208 pp 271–283
[8] Tominaga Y and Stathopoulos T 2016 Ten questions concerning modeling of near-field pollutant dispersion in the built environment Building and Environment 105 pp 390–402
[9] Jeong W and Seong J 2014 Comparison of effects on technical variances of computational fluid dynamics (CFD) software based on finite element and finite volume methods Int. J. Mech. Sci pp 19–26
[10] Srinivas G and Potti S R 2014 Numerical Simulaton of Axial Flow Fan Using Gambit and Fluent,” Int. J. Res. Eng. Technol 3(3) pp 586–590
[11] P.G. Ciarlet and J.L. Lions 2000 Handbook of Numerical Analysis vol 7, ed Eymard R, Gallouet T, and Herbin R (Elsevier) p 717
[12] Wang X 2006 Numerical simulation of wind-induced dispersion of emissions from rooftop stacks (Canada:Concordia University) p 73
[13] Tasic V, Kovacevic R, and Milosevic N 2013 Investigating the Impacts of Winds on SO2 Concentrations in Bor, Serbia J. Sustain. Dev. Energy, Water Environ. Syst 1(2) pp 141–151