

Applications of CFD Simulations in Environmental Science

Guanni Qu*

College of Letters and Science, University of California, Davis, CA, USA

*Corresponding author: jgqu@ucdavis.edu

Abstract. Computational Fluid Dynamics (CFD) Simulations are an essential tool in environmental engineering. Since its development, it has been widely utilized in different subject fields to compute numerical solutions to solve complicated fluid dynamics equations. To better understand the strengths and limitations of the application of Computational Fluid Dynamics Simulations, research papers using CFD are analyzed, and their particular methods are discussed. The simplification of the modeling process and the generation of mesh are likely to induce errors. The geometry of the model in the porous media shows that the choice of the model makes a huge difference on the simulation results. Thus, qualifying the discrepancies between the real on-site measurements and the simulation results is of great importance by verifying both the CFD modeling and other modeling methods used. The meshing should also be verified by comparing the velocity of the fluid in the medium mesh and that in the fine mesh to ensure the quality of the mesh.

Keywords: Computational Fluid Dynamics, Porous Media, Air Pollutant Tracking.

1. Introduction

Computational Fluid Dynamics (CFD) Simulations are developed in the late 20 centuries by engineers and mathematicians to solve complicated equations in fields of engineering, such as mechanical engineering [1,2]. It is a major way to compute numerical solutions to solve complicated fluid dynamics equations. Velocity, density, energy and mass are calculated numerically. Since fluid dynamics problems usually involve the transfer of mass, heat, and energy, CFD is built upon a series of conservation equations, including the conservations of mass equation, momentum equation, energy equation, and the thermodynamic equation of state [3]. Since the conservation of momentum equation is three dimensional, there exist three equations of motion under the principle of conservation of momentum [4]. Since fluid dynamics problems are too complex to solve empirically, CFD relies on the computational power of modern technology [5-7]. To apply CFD, researchers first develop a model to represent the structure of interest and set the initial conditions, boundary conditions and the fluid flow inlet and outlet. The initial conditions include the viscosity of the fluid and the inlet and outlet velocity. Then, researchers generate a mesh of the model to divide the space of fluid fields surrounding the model into small increments to perform analysis using finite element methods, and solve for the differential equations for each mesh. It should be noted that most models are simplified models with simple geometric shapes representing the real structure to reduce the complexity of modeling the realistic structure. The simplification process is usually carefully designed so the numerical solutions are close to the wind tunnels test results in order to achieve accurate simulation [8]. It should also be noted that the density of mesh is not uniformly distributed such that more mesh is generated in fluid fields with more complex fluid flows, such as those with vortices, usually on the boundaries of the model. In application, CFD is widely applied in different fields in engineering. Environmental engineering involves many scenarios with fluid flows, and the simulation of air flow and water flow is usually necessary in environmental science [9,10]. This review discusses the advantages and disadvantages of CFD in applications of fluid dynamics in environmental science. It is a summary about the application of CFD in a few selected topics in environmental science. In particular, it discusses the application of CFD in air pollutant tracking and porous media. It discusses the methodology of each application, the design of geometry model, mathematical models, and different conclusions reached using the CFD simulation.

2. Analysis of different application of CFD

2.1. Air pollutant tracking near road using weather research and forecasting (WRF) and CFD

Lee et al. used WRF-CFD modeling based off of data in WRF to simulate air pollution near roads [10]. NO_x, black carbon, and ultra-fine particles are simulated. The site of study is based in a residential area in Chuncheon in South Korea, around the Toegye Intersection. For WRF-CFD modeling, data is first collected from the mesoscale meteorological fields simulated by the WRF model. The CFD model is a Reynolds-averaged Navier-Stokes (RANS) model, which is according to the idea of time-averaged motion of fluid flows. It is good for modeling turbulent flows. The digital topographic and building maps are used for boundary conditions. The max grid size is 13 m, and the number of grids is 2856280 (202 × 202 × 70). The results are illustrated in Figure1. The WRF modeling is compared to the data observed, and the similarity of the values between the data observed and the data forecasted demonstrate that the WRF model is suitable. The CFD model is compared with the data observed alongside the location. The similar air temperature with a mean absolute error of ~1 °C and the similar wind speed agree with the data that is measured and collected. This shows that the CFD model for the near-road air pollutant tracking is appropriate.

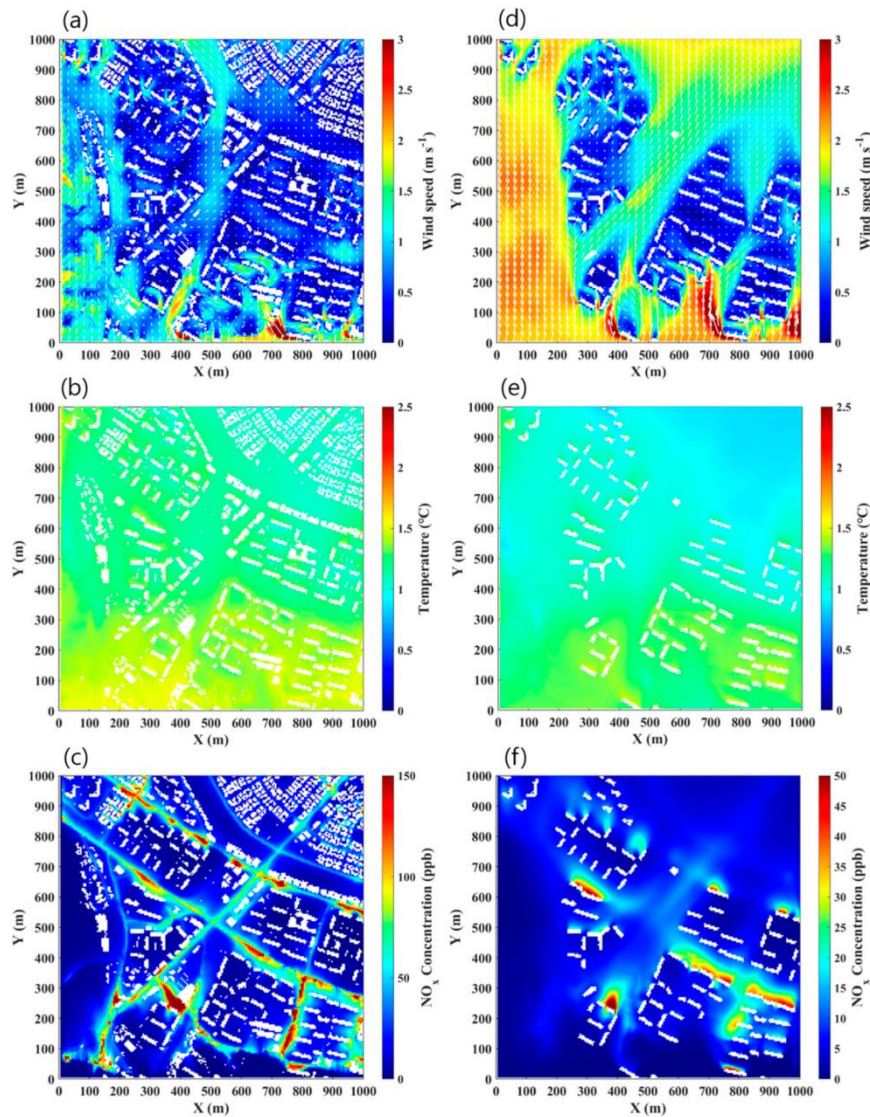


Figure 1. Simulation results.

Data for air pollutant emissions are simulated by emission rate according to the Clean Air Policy Support System. Different vehicles are classified into six different types. The emission rate of

nitrogen oxides and black carbon are calculated based on the traffic recording of different types of vehicles, with different emission rates.

The study reaches the conclusion that the highest concentration of air pollutant emission appeared between apartment buildings due to the recirculating flows. Based on both simulation results and measured data, this study shows that the air pollutants are highest within 120 m horizontally of the intersection, and 50 m vertically of the intersection.

2.2. Turbulent airflow from coal cutting and coal dust pollution

Zhang et al. investigated the turbulent flow's impacts on coal dust using CFD [8]. The system ventilation airflow and the turbulent airflow from coal cutting are simulated in the modeling. The mathematical model uses the Euler-Lagrange approach, treating the fluid phase as a continuum by the Reynolds time averaged equation, and calculating the dispersed phase by tracking particles through the flow field. The CFD modeling uses a 3D geometry that realistically mimics the physical Fucun Coal Mine, with front and rear drum and hydraulic supports. The mesh is generated using the Ansys Bench software, and the Skewness values are inspected to ensure the mesh quality, with all of them being less than 0.95. The boundary conditions including velocity, turbulence, and hydraulic diameter were adapted from the actual data. To verify the suitability of this model, measurements from the operational sites were considered with comparison of the numerically simulated results. The relative error is shown to be insignificant, which verifies the numerical simulation.

Based on analysis of the airflows and the coal dust tracking, this study shows that the turbulent airflow from coal cutting has a non-trivial effect on the behavior of coal dust. Turbulent airflow on coal dust intensifies the intensity of turbulence and creates the lateral shifting of airflow.

2.3. Diesel particulate matter (DPM) pollution

Chang et al. investigated a model for minimizing diesel particulate matter (DPM) pollution in underground mines, using CFD to optimize the ventilation system [9]. Because miners are under greater risk of DPM which is dangerous and lethal to humans, it is important to investigate and come up with a solution to solve the DPM pollution problems.

The computational modeling of a mine in Australia is built for computational fluid dynamics simulations. The authors use OpenFOAM to solve the CFD simulations. The mathematical model adapts the Eulerian-Lagrangian approach and the Navier-Stokes equation. In meshing processes, tetrahedral and hexahedral meshes are generated for complex geometry regions. To verify the mesh, air velocity profiles are monitored and compared at different locations. The air velocities of the medium mesh are in accordance with the air velocities of the fine mesh, achieving the meshing independence. To test the accuracy, the measurement onsite has been compared with the simulation result of the airflow field and the DPM simulation. Nine cross sectional points were measured, and errors of the velocity at each point are compared to the measurement data. The average error is less than 15% and the results of the simulation are taken as significant.

The distributions of DPM concentrations were contrasted with the AIOH DPM standards (0.1 mg/m^3). The findings showed that the current ventilation system was unable to significantly lower the DPM content, due to the simulation results that CPM content is beyond the limit of the CPM concentration. The study also shows that the ventilation system with a duct length 10.2 m is a more suitable system for DPM dilution compared with the two other scenarios.

2.4. Indoor environments of livestock houses

The previous study examines the effect of different CFD models on the simulation of indoor environments of livestock houses. Three different simplified geometries are adapted to simulate the geometrical model of the pigs, as porous media. The ellipsoid model (EM), half ellipsoid model (HEM) and half cylinder model (HCM) are coupled with pigs under different body masses (30-105 kg).

The porous media model is validated by comparing simplified pig model (SPM) to the porous media model. The mean relative errors of velocity magnitudes of the simulated and measured results are shown to be 0.46%, 0.12%, 0.05% respectively in the x, y and z axes, indicating that the simulation results are acceptable. The study shows that HEM is the most appropriate for the SPM, taking into account the different masses of the pigs.

3. Conclusion

The computational fluid dynamics simulation approach to fluid dynamics problems are proven to be effective in solving complex fluid dynamics problems. Its flexibility in adapting to different scenarios with different initial conditions, its application for a wide array of structures, and computational power make it a great option to track air pollutants in the field of environmental science. Coupled with other models to simulate data such as the weather research and forecast model, CFD can be extremely useful. However, the simplification of the modeling process and the generation of mesh are likely to induce errors. The geometry of the model in the porous media of the pigs shows that the choice of the model makes a huge difference on the simulated results. Thus, it is of importance to qualify the discrepancies between the real on-site measurements and the simulation results by verifying both the CFD modeling and other modeling methods used. The meshing should also be verified by comparing the fluid velocity in the medium mesh and the fine mesh to guarantee the quality of the mesh. Despite the errors and the limitation, it is highly hopeful that CFD simulations will be improved and more widely implemented in the future.

References

- [1] Montavon, C.: 1998, Simulation of atmospheric flows over complex terrain for wind power potential assessment, PhD thesis, Lausanne, EPFL.
- [2] Murena, F., d'Allesandro, S. & Gioia, F.: 1997, CO dispersion in urban street canyons, PHOENICS Journal, 10, 4, 407-415.
- [3] Schatzmann, M., Rafailidias, S., Britter, R. & Arend, M. (Editors): 1997: Database, monitoring and modelling of urban air pollution. Inventory of models and data sets. EC COST 615 Action. 109p.
- [4] Sini, J.-F., Anquetin, S. & Mestayer, P.G.: 1996, Pollutant dispersion and thermal effects in urban street canyons. Atmos. Envir., 30:15, pp. 2659-2677.
- [5] Delaunay, D., Flori, J.P. & Sacré, C.: 1996, Numerical modelling of gas dispersion from road tunnels in urban environments: Comparison with field experiment data, Ostende, Belgique, 361-368.
- [6] Baklanov, A., Burman, J. & Näslund, E.: 1996, Numerical modelling of three-dimensional flow and pollution transport over complex terrain during stable stratification, Scientific report, FOA, Umea, Sweden (FOA-R-96-00376-4.5-SE), 40 p.
- [7] Baklanov, Alexander. (2000). Application of CFD Methods for Modelling in Air Pollution Problems: Possibilities and Gaps. Environmental Monitoring and Assessment. 65. 181-189.
- [8] Gonzalez Olivardia, Franchesca G., Qi Zhang, et al. 2019. "Analysis of Pollutant Dispersion in a Realistic Urban Street Canyon Using Coupled CFD and Chemical Reaction Modeling" Atmosphere 10, no. 9: 479.
- [9] Ping Chang, Guang Xu, Fubao Zhou, et al. Minimizing DPM pollution in an underground mine by optimizing auxiliary ventilation systems using CFD, Tunnelling and Underground Space Technology, 87, 2019, 112-121.
- [10] Lee SH, Kwak KH. Assessing 3-D Spatial Extent of Near-Road Air Pollution around a Signalized Intersection Using Drone Monitoring and WRF-CFD Modeling. Int J Environ Res Public Health. 2020 Sep 22;17(18):6915.