Research in Industrial Projects for Students 2016

institute for pure & applied mathematics

Flow and Dispersion Patterns by OpenFOAM and FLUENT

Statement of Work

Sponsored by
AECOM

Student Members
Evan Dientsman
Emily Ho (Project Manager)
Justin Jiaze Sun
Alex Si Ao Wang

Academic Mentor
Dr. Chi-Wai Yu

Industrial Mentor
Karl An

June 22, 2016

1 Introduction

1.1 Company Background

AECOM is a globally recognized engineering firm with major offices in North America, Europe, Asia, and Australia. AECOM offers design, consulting, construction, and management services to over 150 countries around the world, employing around 95,000 individuals. In 2015, Engineering News-Record ranked AECOM first on its Top Global Design Firm and Top Green Building Design Firms, and the 2016 Fortune 500 list ranked AECOM number 156.

AECOM offers technical and management solutions to clients from all over the globe — the Americas, Europe, Asia, Oceania, Africa and Middle East, providing services in Architecture & Design, Asset Management, Construction, Engineering, Environmental Services, Planning & Consulting, Risk Management & Resilience, Technical Services, etc.

1.2 Problem Background

For decades, air pollution has been considered a serious issue in Hong Kong. In light of the fact that millions of people live and work in the densely packed urban area, it is essential to study the wind flow and pollutant dispersion pattern in these districts so that solutions can be proposed accordingly. Thanks to the advancement in computational technology, such problems are usually tackled by running simulations using Computational Fluid Dynamics (CFD) models, and two widely implemented commercial packages include ANSYS ICEM and ANSYS Fluent.

2 Problem Statement

2.1 ANSYS Fluent & OpenFOAM

ANSYS Fluent is a popular commercial CFD software package which includes a wide range of physical modelling capabilities, such as turbulence modelling, acoustics, heat transfer and radiation, etc. However, one disadvantage is its inflexibility to change its source code, which presents difficulties in certain situations.

As an alternative, OpenFOAM, developed by OpenCFD Ltd, is a free, open-source CFD software used extensively by both academic and commercial organisations in most areas of engineering and science. It also incorporates a wide range of modelling capabilities, and is able to solve complex fluid flows involving turbulence, heat transfer, acoustics, chemical reactions, solid mechanics and electromagnetics.

2.2 Goal

The goal of the project is to compare the robustness of the wind field and pollutant field simulation results obtained by OpenFOAM and ANSYS Fluent, in order to explore the possibility of using OpenFOAM to replace the more expensive Fluent.

3 Mathematical Background

3.1 Navier-Stokes (NS) Equations

The NS equations are the foundation of the study of fluid dynamics [3]. We will assume that the fluids we deal with are incompressible; that is, the fluid density ρ is constant. In Einstein notation, which will be used from here on, the incompressible NS equations are:

$$\frac{\partial u_i}{\partial x_i} = 0,$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_j} + \nu \frac{\partial^2 u_i}{\partial x_j^2},$$

where the flow velocity $\mathbf{u}(\mathbf{x}, t) = u_1(\mathbf{x}, t)\mathbf{i} + u_2(\mathbf{x}, t)\mathbf{j} + u_3(\mathbf{x}, t)\mathbf{k}$ is a function of space and time, p is the pressure, μ is the dynamic viscosity, and $\nu = \frac{\mu}{\rho}$ is the kinematic viscosity.

3.2 Reynolds Averaged Navier-Stokes Equations (RANS)

Using the Reynolds decomposition allows us to glean information about the overall flow, which is otherwise difficult to do because of the transient nature of turbulence. This technique separates any scalar variable $\phi(\mathbf{x}, t)$ of space (denoted by $\mathbf{x}$) and time (denoted by t) into a time-averaged component $\Phi(\mathbf{x})$, and a fluctuating quantity $\phi'(\mathbf{x}, t)$:

$$\phi(\mathbf{x}, t) = \Phi(\mathbf{x}) + \phi'(\mathbf{x}, t),$$

where Reynolds time-averaging is defined as

$$\Phi(\mathbf{x}) = \bar{\phi}(\mathbf{x}, t) =: \lim_{\tau \to \infty} \frac{1}{\tau} \int_t^{t+\tau} \phi(\mathbf{x}, t)\, \mathrm{d}t.$$

The RANS equations are shown below, following the convention $u(\mathbf{x}, t) = U(\mathbf{x}) + u'(\mathbf{x}, t)$, where $U(\mathbf{x}) = \bar{u}(\mathbf{x}, t)$:

$$\begin{aligned} &\text{(Continuity)} \quad \frac{\partial u_i'}{\partial x_i} = 0, \\ &\text{(Momentum)} \quad \frac{\partial}{\partial t}(\rho U_i) + \frac{\partial}{\partial x_j}(\rho U_i U_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}(\mu \overline{S_{ji}}) - \frac{\partial}{\partial x_j}(\overline{\rho u_i' u_j'}), \end{aligned}$$

where $S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$ is the mean strain rate tensor. Since we are assuming that the inflow velocity profile will be constant with respect to time, the first term of the momentum equation vanishes in all of the situations we will examine.

3.3 Turbulence Models

Turbulence models are transport equations that provide closure so that the Navier-Stokes equations can be solved. The two most popular are the $k-\omega$ and $k-\epsilon$ models, where k is the turbulent kinetic energy, ϵ is the turbulent dissipation rate, and ω is the specific turbulent dissipation rate. Despite the prevelance of these models, they both have major deficiencies; the $k-\omega$ model is overly dependent on the free stream boundary conditions, while the $k-\epsilon$ model is not sensitive to adverse pressure gradients (when pressure increases in the direction of flow) [4]. To overcome these difficulties, more advanced models have been proposed based on the original $k-\omega$ and $k-\epsilon$ models.

3.3.1 The Shear Stress Transport (SST) Model

SST combines the $k-\omega$ and $k-\epsilon$ models, blending $k-\omega$ in the inner part of the boundary layer with $k-\epsilon$ closer to the edge. Consequently, we have two equations given by:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j k - (\mu + \sigma_k \mu_t)\frac{\partial k}{\partial x_j}\right) = \tau_{ij} S_{ij} - \beta^* \rho \omega k,$$

$$\frac{\partial \rho \omega}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_j \omega - (\mu + \sigma_\omega \mu_t)\frac{\partial \omega}{\partial x_j}\right) = P_\omega - \beta \rho \omega^2 + 2(1-F_1)\frac{\rho \sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j},$$

where μ_t, β^*, τ_{ij}, F_1, P_ω, σ_k, σ_ω, and $\sigma_{\omega 2}$ are quantities as Bardina et al. (1997) defined in a NASA technical memorandum [1].

3.3.2 The Realisable $k-\epsilon$ (RKE) Model

The RKE model [5, 6] retains the turbulent kinetic energy equation of the standard model and revises the dissipation equation, assuming that $\epsilon \approx \nu \overline{\omega_i \omega_i}$ at high Reynolds numbers (the Reynolds number is defined as the ratio of inertial to viscous forces, indicating the fluid's tendency toward turbulent flow), where ω_i is the vorticity, or curl of the velocity vector. This model simplifies the standard $k-\epsilon$ model to the following two equations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}\right) + \tau_{ij}\frac{\partial U_i}{\partial x_j} - \rho\epsilon,$$

$$\frac{\partial(\rho \epsilon)}{\partial t} + \frac{\partial(\rho \epsilon U_j)}{\partial x_j} = (\mu + \frac{\mu_t}{\sigma_\epsilon})\nabla^2 \epsilon + C_1 S \rho \epsilon - C_2 \frac{\rho \epsilon^2}{k + \sqrt{\nu \epsilon}},$$

where

$$C_1 = \max[0.43, \eta/(\eta+5)],\ C_2 = 1.0,\ \sigma_k = 1.0, \text{ and } \sigma_\epsilon = 1.2.$$

4 Approach

4.1 Initial Objectives

To compare the results from OpenFOAM and ANSYS Fluent with the wind-tunnel experiment data, the project will begin with focusing on the flow and dispersion pattern around a simple rectangular cuboid with continuous release of tracer gas at the leeward side (See Fig. 1 for the experimental setup). The CFD simulations will be carried out on OpenFOAM and ANSYS Fluent respectively with the geometric model and boundary conditions obtained from Concentration Data of Street Canyons (CODASC), a database containing concentration data of street canyons. The programs will solve along the inner surfaces of the blocks, as seen in Fig. 2.

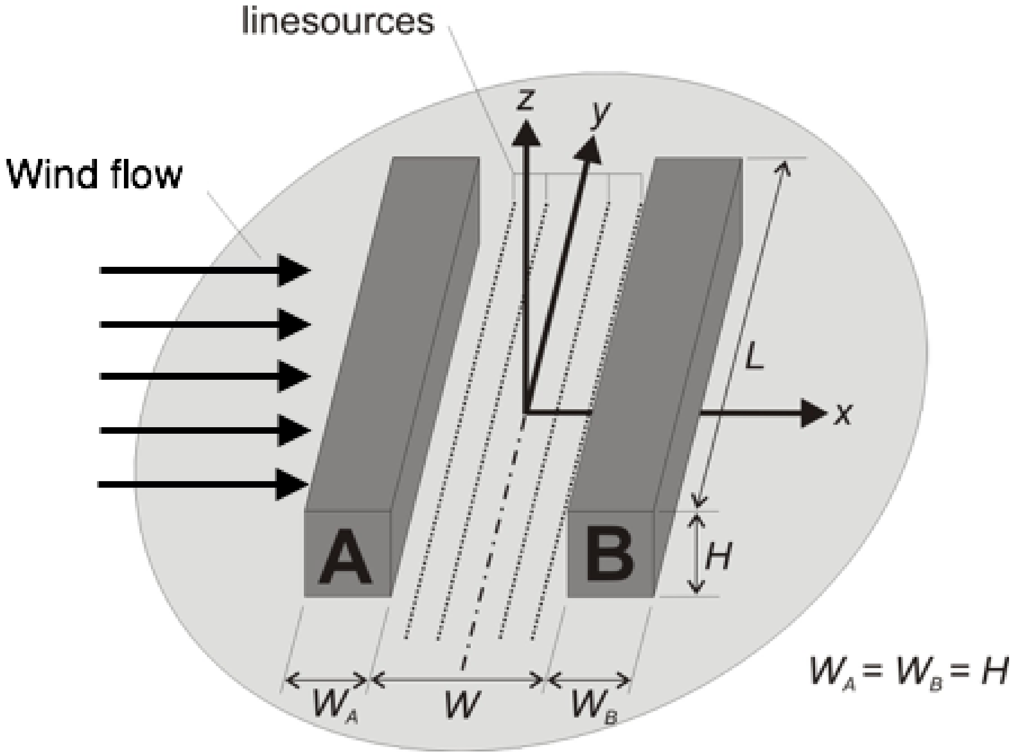

Figure 1: The street canyon models used for collecting the experimental data, which will be simulated in two different numerical platforms.

The SST and RKE models will be applied to the case and 700 grids are assessed for the comparison of the results. To obtain more comparable results, on both platforms we will resort to SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) Solver based on the same mesh grids generated from ANSYS Fluent [2]. Subsequently, the

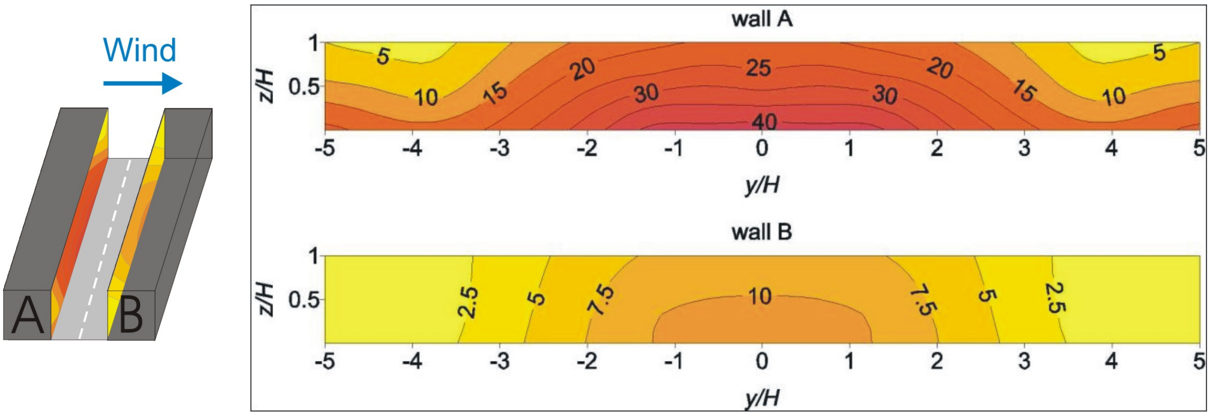

Figure 2: Assessment grids are evenly distributed at the facing surfaces of the two blocks, with 7 sensors along the z-axis and 100 sensors along the y-axis.

simulation results will be compared to a real-world wind tunnel experiment data on CODASC statistically.

4.2 Further Work

The first direction is to run the simulation in a more complex, real urban area under different wind directions using both OpenFOAM and ANSYS Fluent according to the existing Air Ventilation Assessment guidelines, and the subsequent results will be compared and contrasted.

The second direction is to compare the pollutant dispersion predictions in a real urban area obtained by using CFD and AERMOD respectively. AERMOD is an integrated atmospheric dispersion modelling system which includes three modules: a steady-state dispersion model, AERMET and AERMAP. The steady-state dispersion model is designed for the dispersion of stationary air pollutant emissions of up to 50 kilometers. AERMET is a meteorological data preprocessor which calculates parameters for the dispersion model based on surface meteorological data, upper air soundings and data from on-site instrument towers. AERMAP is a terrain preprocessor that provides a physical relationship between complex terrain features and the behaviour of air pollutant plumes. The main difference between AERMOD and CDF is that the former simulates the pollutant dispersion phenomenon using a Gaussian plume based analytical solution for concentration, whilst the latter achieves this by numerically solving nonlinear partial differential equations.

The third possibility is to improve the model currently used in OpenFOAM, by making modifications to the governing PDEs, or implementing additional environmental factors such as humidity and thermal effects in the air dynamic model, in order to obtain more realistic simulations.

5 Schedule

Week 1:

- Install OpenFOAM and FLUENT.
- Gain familiarity with the software, running basic models.
- Draft work statement.

Week 2:

- Discuss, edit, and submit work statement.
- Compare basic models OpenFOAM and FLUENT with wind tunnel data, using simpler methods of comparison.

Weeks 3-4:

- Begin using the software to simulate flow around a cuboid.
- Determine and develop a more sophisticated method for comparing OpenFOAM and FLUENT with the data.

Week 5:

- Organize all work up to this point and prepare midterm presentation.

Week 6-7:

- Continue comparison of the different packages and experimental data. If time permits, attempt to extend the testing to simulate urban environments.

Week 8:

- Draft and revise final report and presentation.

Week 9:

- Projects Day and submission of final report.

6 Deliverables

- Site visits
- Midterm presentation
- Midterm written report
- Final presentation
- Final report

References

[1] Bardina, J.E., & Huang, P.G., & Coakley, T.J. *Turbulence modeling validation, testing, and development, NASA Technical Memorandom 110446*, http://www.ewp.rpi.edu/hartford/~ferraj7/ET/Other/References/nasa_techmemo_110446.pdf, 1997

[2] Buccolieri, R., & Salim, S. M., & Leo, L. S., & Di Sabatino, S., & Chan, A., & Ielpo, P., & de Gennaro, G., & Gromke, C. *Analysis of local scale tree-atmosphere interaction on pollutant concentration in idealized street canyons and application to a real urban junction, Atmospheric Environment, Vol. 45, pp. 1702-1713*, http://dx.doi.org/10.1016/j.atmosenv.2010.12.058, 2011

[3] Celik, I.B. *Introductory turbulence modeling, West Virginia University Class Notes, Western University Press*, http://www.fem.unicamp.br/~im450/palestras%26artigos/ASME_Tubulence/cds13workbook.pdf, 1999

[4] Menter, F. R. *Improved two-equation k-ω turbulence models for aerodynamic flows, NASA Technical Memorandum 103975*, http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19930013620.pdf, 1992

[5] Rohdin, P., & Moshfegh, B. *Numerical predictions of indoor climate in large industrial premises. A comparison between different $k-\epsilon$ models supported by field measurements, Building and Environment Vol. 42, pp. 3872-3882*, http://www.sciencedirect.com.ezproxy.princeton.edu/science/article/pii/S0360132306004100, 2007

[6] Shih, T.H., & Liou, W.W., & Shabbir, A., & Yang, Z.G., & Zhu, J. *A new $k-\epsilon$ eddy viscosity model for high reynolds number turbulent flows, Computers and Fluids, Vol 24, No.3, pp. 227-238*, http://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19950005029.pdf, 1995