ABSTRACT

A numerical simulation is used to compare the particle dispersion of flows around a circular cylinder. The dispersion depending on various characteristics of particles, such as diameter and density, are investigated here. To explain particle properties, the dimensionless number, the Stokes number, is used. The flow Reynolds number is 200 and the diameter of particle change satisfying the Stokes number, 0.02, 0.1 and 0.5. It is the ratio of the characteristic time of particle to the characteristic time. In this study, an open source computational simulation tool, OpenFOAM is used to calculate flow structure and the particle position over time. The computation domain is meshed by rectangular grid and the vegetation is of straight and rigid cylinders. After getting the velocity vector of flow, this work applies an accurate technique, Lagrangian Coherent Structure (LCS), in order to understand the particle dispersion in the flow. LCS is constructed with the finite-time Lyapunov exponent (FTLE), which was obtained by post-processing velocity fields. The FTLE field is the changed distance of neighboring particle trajectories over a given time interval, so it integrates these position forward and backward in time. After getting LCS lines, the location of particles is marked to verify distribution regarding to the diameter. In addition, flow pattern and the stagnation point are investigated.

Keywords: LCS, OpenFOAM, Particle, the Stokes number

1 INTRODUCTION

The flow change around vegetation has been studied in various ways and considered being important in that the vegetation significantly alters turbulent flow near the vegetation in both of horizontal and vertical direction. Petryk and Bosmajian(1975) first introduced a force-equilibrium approach to quantify the effects of vegetation in terms of flow resistance. As a further research, there have been many experiments to identify and improve this approach by Pasche and Rouve(1985), Wu et al.(1999), Bennett et al.(2002), Stone and Shen(2002), Wilson et al.(2003), Lee et al.(2004) and Takemura and Tanaka(2007). In recent, Tanino and Nepf(2008) conducted the experiment to evaluate the drag force and drag coefficient in terms of vegetation density with changing the cylinder the Reynolds numbers in relatively low values(30-700 of the Reynolds numbers). There have also been a number of experiments to verify the flow field and turbulent structure in more detail within a plant canopy. Pasche and Rouve(1985), Tsujimoto and Kitamura(1990), Shimizu and Tsujimoto(1994), Dunn et al.(1996) Nepf(1999) have identified that the time and spatial averaged velocity profile do not follow the logarithmic law when the rigid cylinders are arranged in a row. Jacobs et al. (2004) studied the flow structure around square cylinder and it considers only vortex based on velocity field. However, such studies merely with the velocity field have limitation in understanding the particles movements such as sediment, nutrients, etc. Therefore, an accurate and appropriate technique which can take into account the particle movements is necessary in order to understand the material transport through flow.

In the present work, LCS lines can be considered as the flow particle trajectories so that we can verify particles follow LCS lines. In addition, the moment when the particles’ inertial force begins to be dominant is researched.

2 NUMERICAL MODEL & METHODOLOGY

We study numerically the particle transport around a cylinder in the viscous incompressible flow. In this study, OpenFOAM software, which solves RANS(Reynolds Averaged Navier-Stokes equation), is used. The governing equations include continuity equation and momentum equation as following:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$  \hspace{1cm} [1]

where $\rho$ is density of fluid, $t$ is time, $\mathbf{x}$ is position component and $\mathbf{u}$ is velocity component.
\[
\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right) + \rho g_i + F_i \tag{2}
\]

where \(p\) is the pressure, \(\mu\) is the dynamic viscosity, \(\rho g_i\) is the gravitational force component and \(F_i\) is the external body force component. The \(k-\omega\) turbulence model is applied here to simulate the flow. It uses equations for turbulent kinematic energy, \(k\), and specific dissipation rate, \(\omega\). This \(k-\omega\) model performs well near-wall flows, so it is capable of observing the flow separation accurately.

The computational fluid dynamics (CFD) uses two methods for modelling that are the Eulerian and the Lagrangian method. The Eulerian method treats the particle as a continuum, whereas the Lagrangian method tracks each particle individually. Therefore, in this work, RANS equation and particle tracking are solved by Eulerian method and Lagrangian method respectively. This approach is called Lagrangian Particle Tracking (LPT) method. It can quickly provide flow information and particle information such as the instantaneous full-field information of the flow velocity field and the particle dispersion. In addition, it is computationally efficient rather than using Eulerian method only. After getting the flow velocity field based on Eulerian method, the information of flow particle movement can be acquired by dynamic technique using Matlab codes which is developed to calculate Lagrangian Coherent Structure here.

LCS is constructed with the Finite Time Lyapunov Exponent (FTLE), which is obtained by post-processing velocity fields. The FTLE field is calculated by the changed distance of neighboring particle trajectories over a given time interval, so it integrates these position forward and backward in time. If the particle is traced forward, it results in the forward FTLE. Similarly, if the particle is traced backward, it leads to the backward FTLE. The maximum value of FTLE is LCS, so forward FTLE and backward FTLE brings on forward LCS and backward LCS respectively. LCS lines are the role of the fluid particle trajectory here. It is used as the criteria that how particles keep up with the fluid and it confirms that LCS is a tool that can predict particle motion. In this paper, we confirm the LCS lines can be a predictor of particle movement, and find the Stokes number when LCS lines can be hard to predict particle behaviors.

Figure 1 shows the geometry of one phase flow with a circular cylinder. This computational domain is symmetry and chooses the dimensions to be \(D = 0.04\) m and \(H = 0.8\) m. For area near a cylinder, mesh needs to be dense since turbulence is caused in this area. While for other areas, sparse mesh condition is acceptable. Difference in mesh is made through meshing tool, blockMesh in OpenFOAM, and final mesh utilized to analyze flow with weir is suggested as Figure 2. The number of grid cells is 179,904. The particles are injected at 0.04m upstream of the cylinder after the flow reaches stable state.

The Reynolds number is set to 200 for this study, and it is defined as:
where $U_0$ and $D$ are the velocity of free stream and the characteristic length, respectively.

The particle diameter and particle density influences in dispersion and it can be expressed as the Stokes number. The Stokes number is a dimensionless number and it represents the behavior of particle in the flow. It is the ratio of the characteristic time of particle to the characteristic time of flow and defined as:

$$
St = \frac{t_p}{t_f}
$$

where $t_p$ is the particle characteristic time and it is obtained by $\frac{\rho_p d_p^2}{18 \mu}$. $\rho_p$, $d_p$ and $\mu$ represent particle density, particle diameter and dynamic viscosity of flow respectively. $t_f$ is the flow characteristic time and it can be acquired through $D/U_0$. In this study, three types of the Stokes number, 0.02, 0.1 and 0.5 are considered.

3 RESULTS AND ANALYSIS

This work compares the particle dispersion with the different Stokes numbers. To study the effect of the Stokes number on the dispersion, the particle diameters are selected for three cases: 0.001265m, 0.002828m and 0.006325m. The Reynolds number in the flow is set at 200 to compare with the previous study, Vu, H.C et al. (2002), about flow structure. To compare the results of flow with previous research, the location of stagnation point, flow pattern and Strouhal number are studied. And then, this research study the flow of particles under Strouhal numbers comparing with the LCS lines.

3.1 THE LOCATION OF STAGNATION POINT AND FLOW PATTERN

Figure 3 shows averaged velocity $\bar{u}$ along the centerline of the domain. Both axes are non-dimensional. X-axis is dividing by the total length of x and y-axis is dividing by the maximum velocity. The location of stagnation point is at $x = 0.38$ m since the velocity disappears and it is a predictable result.

![Figure 3. Time Averaged Velocity versus Distance x Along the Centerline.](image)

Depending on the flow condition such as characteristic length and velocity, the flow patterns behind the cylinder change. This section confirms the Strouhal number and the flow pattern to verify that flow is fully developed. Figure 4 shows the flow patterns and figure5 shows detail information for the frequency of vortex shedding.
To find shedding frequency, this work uses the Strouhal number. Figure 5 shows that the power of the lift force coefficient. There is one dominant peak and it is related to the Strouhal number.

Both results have come out very close that of previous research, Vu, H.C et al. (2002). It means the fluid, which the Reynolds number 200, works well as the background.

3.2 PARTICLE DISPERSION

The particle diameter and particle density influences in dispersion and it can be expressed as the Stokes number. The Stokes number is a dimensionless number and it represents the behavior of particle in the flow. It is the ratio of the characteristic time of particle to the characteristic time of flow and defined as:

$$St = \frac{\tau_p}{\tau_f}$$

where $\tau_p$ is the particle characteristic time and it is obtained by $\frac{\rho_p d_p^2}{18 \mu}$, $\rho_p$, $d_p$ and $\mu$ represent particle density, particle diameter and dynamic viscosity of flow respectively. $\tau_f$ is the flow characteristic time and it can be acquired through $D/U_0$. $D$ is the characteristic length and $U_0$ is the velocity of free stream. In the present work, three types of the Stokes number, 0.02, 0.1 and 0.5 are considered.

Figure 4 shows LCS and dispersion of the particles in the wake of the circular cylinder. This figure shows instantaneous images. In all case, 30 particles are injected every 0.5 seconds and the total of 600

![Figure 4. Flow Pattern Shown by Vorticity.](image-url)

![Figure 5 Power of the lift Coefficient for Re = 200.](image-url)
In this section, the density of particles is a fixed value, $1800 \text{kg/m}^3$ and the diameters of particles are 0.001265m, 0.002828m and 0.006325m satisfying the Stokes numbers.

![Figure 5. Particle Dispersion for St = 0.02, 0.1 and 0.5 in the Flow Around Circular Cylinder.](image)

In this study, the flow around a circular cylinder was investigated. The flow over the cylinder is simulated using pimpleFoam in OpenFoam which solves the incompressible Navier-Stokes equations. The flow is set at the Reynolds number 200 and the particles are set at the various Stokes numbers; 0.02, 0.1 and 0.5. To see if the results of fluid are good, we compare the results of fluid with the previous study, Vu, H.C et al. (2002). To analysis the fluid transport a new accurate technique, Lagrangian Coherent Structure (LCS), is used here. Three kinds of particles satisfying the different Stokes numbers are observed by using LCS. When $St \leq 0.1$, particles behave like the fluid and the LCS is a good tool to predict the particle dispersion. Otherwise, at $St = 0.5$ the particles are concentrated in the boundary of the structures.

ACKNOWLEDGEMENTS

This research is supported by Korea Ministry of Environment (MOE) as “Chemical Accident Response R&D Program (ARQ201901179001)”, and the National Research Foundation of Korea (NRF) Grant (No. 2017R1A2B4007977) funded by MSIP of Korea. This research work was conducted at the Institute of Engineering Research and Institute of Construction and Environmental Engineering in Seoul National University, Seoul, Korea.
REFERENCES