Computers and Electronics in Agriculture 93 (2013) 168–183

Contents lists available at SciVerse ScienceDirect

Computers and Electronics in Agriculture

journal homepage: www.elsevier.com/locate/compag

# The past, present and future of CFD for agro-environmental applications

In-Bok Lee [a,*], Jessie Pascual P. Bitog [a,b], Se-Woon Hong [a], Il-Hwan Seo [a], Kyeong-Seok Kwon [a], Thomas Bartzanas [c], Murat Kacira [d]

[a] Aero-Environmental and Energy Engineering Laboratory, Department of Rural Systems Engineering and Research Institute for Agriculture and Life Sciences, College of Agriculture and Life Sciences, Seoul National University, 599, Gwanakno, Gwanakgu, 151-921 Seoul, Republic of Korea
[b] Department of Agricultural Engineering, Nueva Vizcaya State University, 3700 Bayombong, Nueva Vizcaya, Philippines
[c] Laboratory of Agriculture Engineering and Environment, Institute of Technology and Management of Agricultural Ecosystems, Center for Research and Technology, Thessaly, Greece
[d] Department of Agricultural and Biosystems Engineering, College of Agriculture and Life Sciences, University of Arizona, Tucson, USA

ARTICLE INFO

*Article history:*
Received 30 November 2011
Received in revised form 1 August 2012
Accepted 7 September 2012

*Keywords:*
Agriculture
Air
CFD
Environment
Soil
Water

ABSTRACT

Computational fluid dynamics (CFD) is a proven simulation tool which caters to almost any field of study. The CFD technique is utilized to simulate, analyze, and optimize various engineering designs. In this review, the discussion is focused on the application of CFD in the external atmospheric processes as well as modeling in land and water management. With respect to its application in environmental investigations, numerous CFD studies have been done in the atmospheric processes where generally only the fluid flow characteristics are investigated. The application of CFD to soil and water management is still limited. However, with the present demand for conservation and sustainable management of our soil and water resources, CFD application in this field is fast emerging especially in structure designs of dams and reservoirs where CFD offers fast reliable results with less labor and cost. Every CFD model should be validated in order to be considered accurate and reliable. However, a benchmark or standard procedures in validating CFD models is not yet available. This probably answers why the success of the CFD models is still mostly attributed to the user's skills and experience.

At present, the degree of application of CFD to the agro-environmental field is limited by the computing power and software used, however, the fast ever computing power of PCs continually expands the potential of CFD and can be generally more flexible at accounting for the unique aspects of every CFD project. This allows easy access to conduct simulation studies from simple to complex models. In this paper, after a state of art analysis of the past and present application of CFD in the agro-environmental applications, its future directions were discussed, in order to potentially serve as a guide for researchers and engineers on what project or investigations can be conducted.



## 1. Introduction

Computational fluid dynamics (CFD) is a very powerful simulation tool which uses computers and applied mathematics to model fluid flow situations, heat, mass and momentum transfer and optimal design in agro-industrial processes (Xia and Sun, 2002). The yardstick of success is how well the results of numerical simulation agree with experiment in cases where careful laboratory experiments can be established, and how well the simulations can predict highly complex phenomena that cannot be isolated in the laboratory (Sethian, 1993). As a developing science, CFD has received extensive attention throughout the international community since the advent of the digital computer. Since the late 1960s, there has been considerable growth in the development and application of CFD to all aspects of fluid dynamics (Parviz and John, 1997). As a result, CFD has become an integral part of the engineering design and analysis environment of many companies because of its ability to predict the performance of new designs or processes before they are even manufactured or implemented (Schaldach et al., 2000). CFD has grown from a mathematical curiosity to become an essential tool in almost every branch of fluid dynamics (Xia and Sun, 2002). It allows for a deep analysis of the fluid mechanics and local effects in various fields of agro-environment. Most of the CFD results offers an improved performance, better reliability, more confident scale-up, improved product consistency, and higher plant productivity (Bakker et al., 2001) which can provide detail information that would assist the designer in arriving at good decision making and project planning. Some design engineers actually use CFD to analyze new systems before deciding which and how many validation tests need to be performed reducing sustainably in this way the production cost.

CFD has become very popular in engineering fields including agricultural engineering. In the past, the application was limited

* Corresponding author. Tel.: +82 2 880 4586; fax: +82 2 873 2087.
*E-mail address:* iblee@snu.ac.kr (I.-B. Lee).

0168-1699/$ - see front matter © 2012 Elsevier B.V. All rights reserved.
http://dx.doi.org/10.1016/j.compag.2012.09.006

only to agricultural structures such solving environmental problems of greenhouses, animal production facilities and storages. However, over the years, the versatility, accuracy and user-friendliness offered by CFD has led to its increased take-up by the agricultural engineering community (Norton et al., 2007). At present, CFD application has widely spread to various fields in the agro-environment industry such as atmospheric processes, land and water management, safety and disaster management, optimization of production systems for renewable energy and fertilizer applications. As a result, CFD has become a very important simulation tool in the investigation and analysis of various phenomena in the agricultural industry.

In this review, the discussion is focused on the application of CFD in the external atmospheric processes as well as modeling in land and water management. The applications of CFD in the internal environment especially in agricultural structures such as greenhouses, livestock and poultry houses has been enormous, thus, during the preparation of this paper, a separate review that focused on these particular subjects is also being prepared by equally well known and experienced CFD experts who has been utilizing CFD in these particular field. The review paper is now in the process for publication.

In atmospheric modeling, simulation on odor dispersion and control, air pollution, climate calculation, etc. have been investigated. Such simulations are complex when considering detailed modeling of terrain and topography of the study area. Air pollution aerodynamics concerns the interaction of noxious aerosols, gases and particles emitted into the atmosphere with surrounding structures, terrain and vegetation. This interaction can deflect materials toward sensitive areas; concentrate species above acceptable levels, or on the other hand, even mitigate concentration levels and enhance diffusion and dispersion. Several approaches have been proposed by Hong et al. (2011a) which attempted to simplify the creation of complex topography for CFD analysis. To solve arising problems in field experiment for dispersion modeling, heat dispersion models can be an alternative since heat and gas dispersion have theoretically the same transport equations during passive dispersion (Hong et al., 2011a). This approach has already been applied by Hong et al. (2011a) in wind tunnel experiments and the results were also confirmed with their simulation results.

The applications of CFD in land and water management have been recognized, however still limited. Studies on this field focused on determining how pollution is transported and dispersed through water and soil and how this influences the quality of soil, ground water, and surface water. Geo-hydrodynamic aspects are taken into account as well. The knowledge generated by this research is used to support public administrations and industries in finding cost effective solutions for pollution control when faced with soil and water pollution problems. Much of the research in land and water management focuses on the use of sensors for measuring water quality at high spatial and temporal resolutions and obtains data which can be used to parameterize numerical models of pollutant transport processes in soil and water. These data are important to predict the fate of pollutants and the exchange processes that take place in soil and water. However, such simulations are faced with considerable challenges, such as the coupling of contaminant transport models at different spatial and temporal scales throughout the soil strata, e.g. transport of contamination from surface water to ground water. For instance, the paper published by Al-Baghdadi et al. (2009) investigated the movement of chemicals through soil to the groundwater which is a major cause of the degradation of water resources. Rouholahnejad and Sadrnejad (2009) have also attempted to investigate the leachate transport into the groundwater at landfill sites using numerical simulation. The authors discussed the development of a two-dimensional numerical model established from finite difference-finite volume solution of two-dimensional advection–diffusion–linear sorption with first order decay equation. The model can be used to quantify groundwater inputs and associated contaminant discharge from a landfill facility with capacity of 2000 ton day$^{-1}$ into the nearest aquifer. In addition, the model can be used for the simulation of contaminant transport in aquifers in any scale.

In the hydrological area, the design of reliable hydraulic structures (Hong et al., 2011c) and process equipment requires an understanding of the internal flow behavior (Kim et al., 2010). Design guidelines can be defined and used for simple structures with standard flow conditions. When the application is unique, engineers need to revert to modeling to observe and understand its hydraulic behavior. At present, CFD has been widely used to understand and mitigate phenomena such as water flows that caused structural damage such as dam break and erosion (Biscarini et al., 2009; Lee and Woo, 2004). In river engineering, CFD has been utilized investigate river flows. The flow in rivers is very complicated, because it is not only turbulent and highly three-dimensional, but also has irregular boundaries of a complex geometry, a rough bed and a free surface. The ability to accurately predict the three-dimensional flow in open channels and rivers is of obvious importance for the design and construction of hydraulic systems in rivers.

Recently, CFD has been used in agricultural safety analysis such as fire prevention in the farm field, grass land and forest (Wagenbrenner et al., 2010; Koo et al., 2009). Developing a system to predict fire map danger zone through simulations requires vast information on atmospheric data, population density, and rural data related to fire occurrence, etc. However, although it is very complex to execute simulation of huge areas, the availability of information with the aid of various computer aided programs make the study feasible.

The application of CFD in the agri-environment in the future looks very bright and promising. In the next decade, it is expected that the primary role of CFD in fluid modeling particularly in air pollution aerodynamics will focus on establishing engineering design of specific facilities and agricultural structures. Furthermore, real time forecasting of air pollution levels is expected in the very near future. Fluid modeling is often not fast or flexible enough to perform the sensitivity studies commonly required by the relative industry, making engineering decisions very complex systems (Meroney, 2004). Meroney (2004) has emphasized the following directions that fluid modeling should take which can serve as guide in future CFD researches. Some were modified to highlight CFD application in the agricultural field. (1) To explore atmospheric dispersion interactions which are not yet fully understood; (2) To come up with the appropriate turbulence models incorporated into CFD models suited especially in agricultural structures; (3) To complement (or replace) numerical measurements when the reality of CFD modeling is constrained by computational capacity, understanding, or economics; (4) To develop new analytic models suitable for inclusion in larger numerical systems; (5) To validate computational modules as they are incorporated into computer design codes and establish a benchmark on validating any CFD models, and (6) To assist in the "education" of a new generation in fluid dynamics and establish more collaborative CFD projects and explore CFD application in other agricultural field of discipline.

## 2. CFD modeling in atmospheric processes

Modeling in atmospheric processes requires strong background in wind engineering. The application can be focused on air pollution which involves low and moderate winds which are relevant in dispersion of contaminants. The application of CFD in wind engineering is generally known as computational wind engineering which has significantly increased in the previous decades. There are relevant and very informative papers which discussed in detail the use of CFD in wind engineering particularly Franke et al. (2007, 2004), Tominaga et al. (2008), Tamura et al. (2008). These papers also provide some CFD guidelines in defining the physical model, the computational domain, the computational grid, the numerical approximation, and the numerical solution. CFD has been widely embraced as a very effective tool to examine different aspects of the mass, energy and circulation systems in the atmosphere. CFD can provide detail analysis of specific phenomena such as but not limited to atmospheric weather disturbance, energy transfers, pressure and winds at the primary or global scale and thermal differences.

The modeling of air quality at local, urban and regional scales is discussed in this section. Research using such models is often focused on development of techniques that can refine and improve the result of air quality forecasts. The transboundary nature of air pollution necessitates the availability of models that can support air quality policies be it in local or regional scales which can be used in national and international projects for public administrations and industries. To improve the accuracy of air pollution models the data derived from satellite imagery and ground based monitoring stations have often provided boundary conditions that could be integrated into such models (Hong et al., 2011a,b; Seo et al., 2010). The high resolution boundary conditions provided by these technologies enables a more accurate refinement of models to ensure scientifically sound policy support on actual air quality problems related to particulate matter or fine dust. For instance, Seo et al. (2010) estimated the distance area of damage caused by fugitive dust generated from the exposed surface of the Saemangeum reclaimed land located in Korea with an area of 40,100 ha. In other cases which focused on a very specific and smaller location such as in rural and urban areas, modeling of air quality within the area has been conducted. Currently, research trend is generally focused on air pollutants that will become important in the near future, such as heavy metals and ultra find particles (UFP) (Gousseau et al., 2011; Seo et al., 2010).

### 2.1. Air pollution/odor dispersion

There are typically two CFD approaches considering the modeling of air environment such as internal and external. For internally air environment, the CFD technologies have been actively used to study natural and mechanical ventilation of livestock houses, greenhouses, storage systems, etc. Meanwhile, on the external aspect, CFD is increasingly used to study various natural and artificial phenomena in the atmospheric boundary layer including pollutant dispersion (Hanna et al., 2009), odor dispersion (Maizi et al., 2010; Hong et al., 2011b; Li and Guo, 2006), wind-driven erosion (Hussein and El-Shishiny, 2009), airborne dust dispersion (Seo et al., 2010), snowdrift (Beyers and Waechter, 2008), etc.

The ammonia emissions from an aqueous solution were validated via CFD by Rong et al. (2011). The goal was to investigate the accuracy of three models for Henry's law constant (HLC) as well as functions derived from experimental vapor–liquid equilibrium (VLE) properties of ammonia water to determine the concentration on the liquid ammonia solutions surface in order to be used as boundary condition for CFD prediction of ammonia emission. Their study investigated and discussed the effects of some selected parameters on ammonia emissions such as geometry model, inlet turbulent parameters and three turbulence models (low-Reynolds number $k$–$\varepsilon$ model, renormalization group $k$–$\varepsilon$ model and Shear Stress Transport $k$–$w$ model). Then the concentration boundary condition determined by different HLC models and the VLE model is validated by ammonia emissions and concentration profiles measured in the boundary layer. Their simulation results have shown that the current HLC models generally over-predict the ammonia emissions from aqueous solution in this study whereas VLE gives better agreement between simulated and measured results. A linear relation was also observed between ammonia mass transfer coefficient obtained from the VLE relation and those from HLC models.

Focusing on odor dispersion, the extent and rate can vary significantly, depending on the odor release location, odor concentration, atmospheric stability, etc. Topographical features as well as unpredictable and unstable wind conditions, such as fluctuating weather speeds and changeable wind directions, also hinder the analysis of quantitative odor dispersion. It is therefore beneficial to utilize simulations for studying odor dispersion, verified through field experiments. To date, some field experiments and simulations have been conducted for dispersion predictions (Hong et al., 2011b; Li et al., 2006; Holmes and Morawska, 2006). Field experiments provide the most realistic results even with a very limited number of observations, and even under changing meteorological conditions. Much research on the use of CFD for the study of atmospheric dispersion has been conducted. Most have focused on modeling the dispersion phenomenon in flat or to nearby areas (Riddle et al., 2004; Li and Guo, 2006; Lin et al., 2007; Diego et al., 2009). However, topographical consideration is very important especially in mountainous areas. Most troublesome problems occur in a low atmospheric environment with low wind speeds or dips (valleys) in undulating topography which is significantly affected by the shape of the terrain. Therefore, detailed terrain modeling is crucial for the simulation of atmospheric dispersion. To a large extent, CFD meshing restricts the adaption of CFD by design engineers due to the fact that it is the most labor-intensive task in the CFD process especially in large scale complex simulations. Thus, building the geometry and meshing the grid, as a pre-processing procedure for CFD or other scientific applications, have been generally achieved by user-developed tools or commercial softwares. Several studies developed their own methods or codes to create meshes on terrain (Chin et al., 2004; Khan et al., 2005; Jung and Kwon, 2006; Yoon, 2007; Lee and Kim, 2007). The analysts used their own tools to convert raw geographic information into the desired form; commercial softwares require much time and computational cost to convert the raw information into vertices, edges, faces, and sometimes geometrical volumes (Chin et al., 2004). However, the user-developed tools mostly adopted a structured grid, which can deteriorate the mesh quality near irregular or distorted boundaries and steep mountains. The user-developed tools also need to be modified when applied to other related research. On the other hand, commercial tools have been preferred for creating various unstructured grids and analyzing various cases despite the inefficiency of regular routines demanded by the software developer. Hussein and El-Shishiny (2009) simulated wind environments around historical heritage sites of the Giza Plateau in Egypt. They used the GAMBIT commercial tool and non-conformal meshes to create unstructured meshes over such sites with complex geometry. GAMBIT is a state-of-the-art preprocessor for engineering analysis that uses advanced geometry and meshing tools in a powerful, flexible, tightly-integrated, and easy-to use interface. GAMBIT can dramatically reduce preprocessing times for many applications from simple to complex models. Models can be built directly within GAMBIT's solid geometry modeler, or imported from any major CAD/CAE system (Fluent manual, 2006). Hussein and El-Shishiny (2009) addresses the influence of wind flow structure, as an important denudation factor, on the site and its famous monuments: the Pyramids and the Great Sphinx. Their work provided more insight to the effect of wind around the Giza plateau which can be utilized in developing a global plan for conservation and protection of the site. Hanna et al. (2009) also simulated the dispersion of a pollutant in industrial sites and cities few kilometers in size. Meanwhile, appropriate grid size is becoming an important requirement for a wide range of terrain to satisfy both the accuracy and the economic efficiency of the results. Prospathopoulos and Voutsinas (2006) studied the effects of several grid refinements on 3D wind flow simulation through field measurements when a Reynolds averaged Navier–Stokes (RANS) solver was performed. However, suggested grid conditions had limitations on applications to other terrains or research, especially when the Large Eddy Simulation (LES) model, which was differentiated from the RANS model, was used.

Quinn et al. (2001) introduced some of the computational modeling methods available to predict near field concentrations of a pollutant emitted from a livestock building opening. The modeling approaches are restricted in application to effectively weightless particles, such as a low concentration pollutant gas. The flow patterns were predicted using CFD and linked to this computed flow field were two dispersion models, a Eulerian diffusion model and a Lagrangian particle tracking technique, both used to predict ensemble mean gas concentration.

Konig and Mokhtarzadeh-Dehghan (2002) and Brown and Fletcher (2003) used CFD to develop a plume model which incorporate source terms from condensation, evaporation and associated heat transfer. The CFD model was used to examine issues that cannot be assessed well with standard atmospheric dispersion models, such as the impact of condensation on plume rise and ground-level odor and the impact of ambient air addition on plume visibility. Brown and Fletcher (2003) discussed the relationship of develop models and the traditional atmospheric dispersion model. Accordingly, the different techniques can be used in a complimentary fashion to develop engineering solutions to reduce the impact of emissions from an industrial plant.

Li and Guo (2006) developed three dimensional CFD dispersion model to simulate odor dispersion from a sow farrowing farm. Atmospheric stability, wind and temperature vertical profiles in atmosphere were configured in the CFD calculation and their effects on odor dispersion were evaluated. The CFD computed results were compared with the results of CALPUFF model. CALPUFF is an advanced non-steady-state meteorological and air quality modeling system which has been adopted by the U.S. Environmental Protection Agency (U.S. EPA) in its Guideline on Air Quality Models as the preferred model for assessing long range transport of pollutants and their impacts on Federal Class I areas and on a case-by-case basis for certain near-field applications involving complex meteorological conditions (http://www.src.com/calpuff/calpuff1.htm). The results of both models showed that odor traveled farther under stable than unstable condition with the same wind speed. Under the same atmospheric stability category, odor concentrations at lower wind speed were higher than that at greater wind speed. Stronger odor was favored under stable atmospheric condition at lower wind speed. Odor concentration results predicted by the CFD model were higher than that by CALPUFF model in short distance (<300 m). CFD predictions were higher than CALPUFF predictions at the longer distance. Accordingly, the gaps of odor concentration predictions at the longer distance remained stable and were influenced by atmospheric stability category and wind speed.

Odor source does not originate only from livestock facilities. Waste water treatment plants (WWTPs) are also potential sources of offensive odors that can lead to nuisance within nearby communities (Maizi et al., 2010). Thus a CFD model was designed by Maizi et al. (2010) to examine the behavior of contaminants concentration plume released to the atmosphere, and to quantify the potential impact caused by the WWTPs on the neighborhood. In their study, the following assumptions were taken to solve odorous compounds dispersion: (1) The flow is considered to be three-dimensional, turbulent and stationary; (2) The wind speed is considered to be constant and its direction is parallel to the center line; (3) Pollutants are emitted at constant concentration; and (4) Temperature gradients were assumed to be negligible. According to the authors, the evolution of the $NH_3$ plume was assumed to be similar to the $H_2S$ plume, thus their study was limited to following the distribution of $H_2S$ concentrations. Furthermore, the evolution of the $NH_3$ plume was similar to results of their study showing the significance of the presence of buildings close to the odor source, which increases the distribution of pollutant flow and consequently increased its dispersion. Results of their several simulations have concluded that the simulated concentrations of odorous gases vary with the aerodynamic field; the most important variable is wind speed. The higher the wind speed the more the dispersion of odorous is ensured, the contaminant is almost totally carried away by wind when the wind speed is high. Low wind speeds can cause high concentrations close to the source and around nearby buildings.

Contrary to expectations, the air flow at low wind speeds was not sufficient to allow dilution of the pollutant, which generated intensive odors in these regions. Nevertheless, low wind speeds appear to ensure better dilution of the contaminant in regions away from the WWTP. Furthermore, the presence of buildings close to the plant increases the distribution of the pollutant flow and consequently increased its dispersion. The buildings created a stagnation zone, a reverse flow and a separation zone: regions of high turbulence, which could give intensive odors in the front and on the roofs of buildings, where pollutant concentrations are maximized, and also, a zone of increased turbulence downstream the buildings. This is presented in Fig. 1 which displayed the velocity vector fields from four main distinguishable zones interspersed

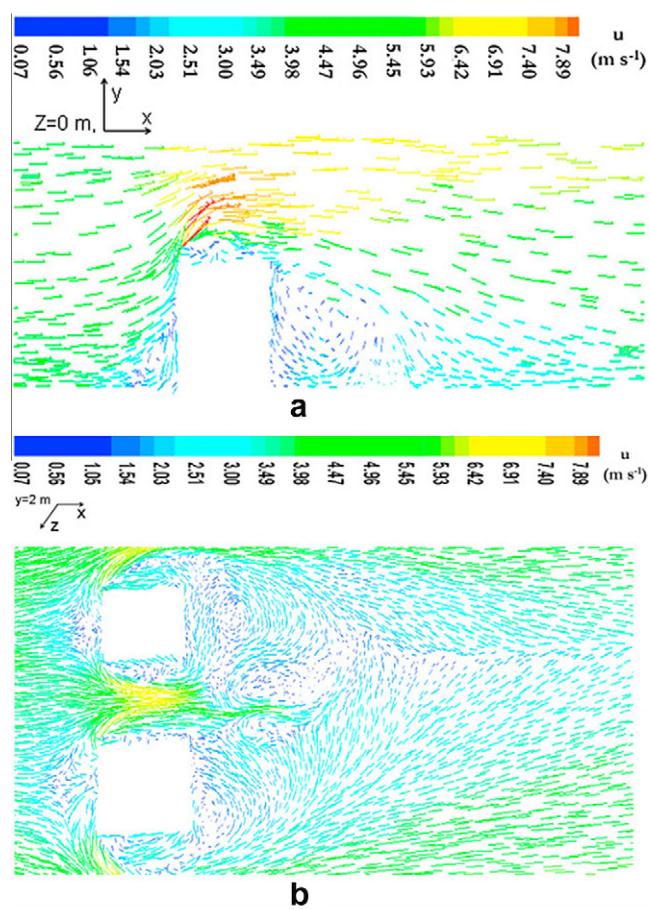


**Fig. 1.** Map of velocity vector field of the study area (reprinted with kind permission from Maizi et al., 2010. Copyright Biosystems Engineering).

between the flow and the two obstacles: frontal zone, the roof of the buildings, the recirculation zone behind buildings and the passage between buildings. Fig. 1 shows the first zone created by the fluid reverse flow at the flow impacts with the buildings. This is shown by the negative mean velocity values indicating stagnation zone. Second zone was situated on the roof of buildings where a reverse flow and a separation zone are formed. The third zone was the passage between both blocks where a mass of fluid moved vertically some height. Finally, the fourth zone was the region behind buildings. This zone was characterized by negative velocities immediately downstream behind the obstacles.

The use of pesticides in greenhouse operations in order to control pests and diseases increases the potential risk exposure of workers and the pollution of environment since the application of pesticides is usually followed by natural ventilation. Kittas et al. (2010) numerically simulated the emission and dispersion of the fungicide Pyrimethanil in the indoor air of the greenhouse after its application with a low volume sprayer. Numerical results show that the concentration of the pesticide decreased first in the windward part of the greenhouse and afterwards in the rest of the greenhouse volume. This distribution is due to the air movement inside the greenhouse. The wind direction has a major role in the dispersion of pesticide in the ambient environment. As can be seen in Fig. 2, a North–South (N–S) direction transfers the pesticide outside of the greenhouse and disperse it in the nearby greenhouses and buildings, whereas the pesticide was dispersed to the exactly opposite direction with an East–West (E–W) wind direction. The distance in which the pesticide can be found in high concentrations depends on the initial concentration of the pesticide inside the greenhouse and on the wind velocity.

Understanding the process of emission and dispersion of pesticides from greenhouses will be a useful tool for responsible authorities in order to specify the frame of pesticide legislation, integrating all necessary precautions to protect workers, bystanders, surrounding communities and the environment.

### 2.2. Large-scale modeling

In spite of the existing models, CFD is more appropriate for applications that involve flow and dispersion in complex geometrical situations (Riddle et al., 2004). In industrial complexes and agricultural lands, however, a typical area of interest has horizontal dimensions of at most a few kilometers and a vertical dimension of 0.5–1.0 km. Generation of a computational grid for such geometry is very complicated due to the complex terrain features. It is also not possible to maintain high resolution within the overall area of interest. Such issues were previously discussed in Bergeles et al. (1996), but the best compromise between reliable calculations and computational cost is still one of the ongoing important issues.

The size of mesh is the most basic and effective design factor to reach the best compromise. Table 1 shows some cases which determined the size of the mesh according to the size of the whole computational domain. In most cases, the mesh resolution was designed to be dense at the center of the domain where the study is focused on and to be gradually coarser further away. However as reported by Lee et al. (2007), it is very critical not to make a big and sudden change of mesh size between adjacent meshes because a rapid change of mesh size between adjacent meshes could make the truncation error larger. To provide a better numerical solution, some suggestions on designing mesh size were presented, as regards growth rate (Franke et al., 2004), grid density (Prospathopoulos and Voutsinas, 2006), height of the first mesh adjacent to the ground (Riddle et al., 2004; Blocken et al., 2007; Pontiggia et al., 2009), $y+$ distance (Mohammadi and Pironneau, 1994; Fluent Inc., 2006; Hussein and El-Shishiny, 2009;), etc. Regarding the shape of the meshes, hexahedra are generally to be preferred over tetrahedral because hexahedra are known to produce smaller truncation errors and lead to better iterative convergence (Franke et al., 2004). However the tetrahedral cells can be created much faster in complex geometries while they may increase the levels of numerical diffusion (Hefny and Ooka, 2009).

Elaborate terrain features also become an important issue as the area of interest is getting bigger and more complicated. Accurate digital map based on a Geographical Information System (GIS) as well as various graphics programs are being introduced to make a sophisticated geometrical model. Hussein and El-Shishiny (2009) used Autodesk MAYA modeling software to generate the terrain and Seo et al. (2010) and Hong et al. (2011a) used Rhinoceros, commercial NURBS-based 3D modeling tool, to make a large and complicated topographical geometry. In most studies dealing with a large scale atmospheric simulation, CAD software were basically used to clean the surface meshes, e.g. removing very small faces and edges, and sharp angles.

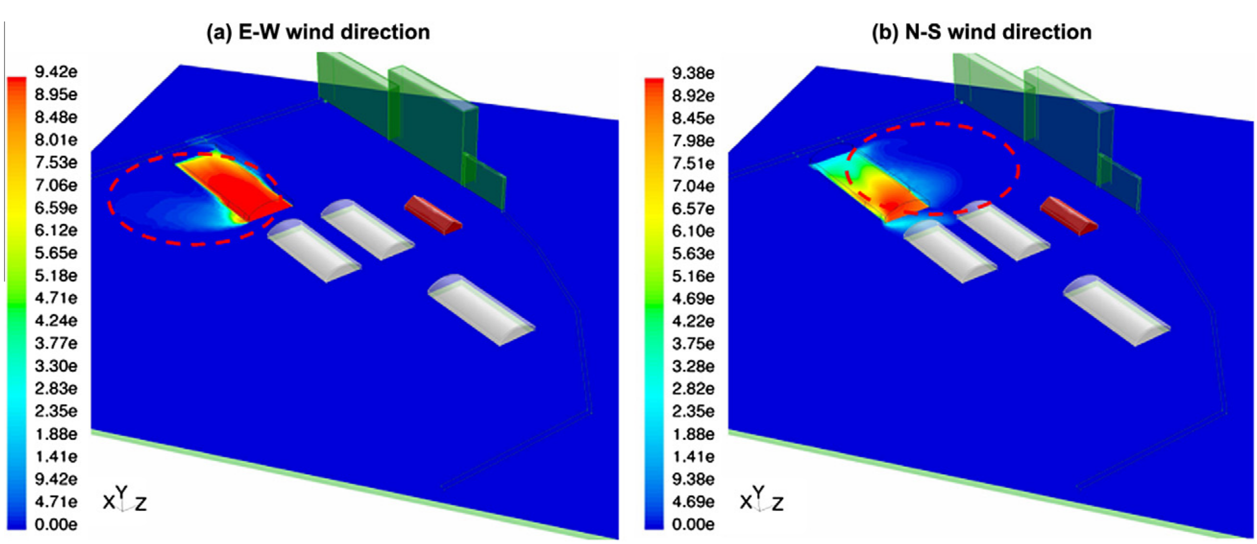


**Fig. 2.** Simulated contours of pyrimethanil concentration inside the experimental greenhouse and at the ambient environment. Effect of wind direction on the dispersion of pesticide (a) E–W wind direction; and (b) N–S wind direction.

**Table 1**
References of large-scale CFD studies and their mesh sizes.

| Reference | Mesh size | Computational domain size | Remarks |
|---|---|---|---|
| Brown and Fletcher (2003) | 0.8–100 m (model 1)<br>0.5–40 m (model 2) | 200 m (upstream)<br>2000 m (downstream)<br>1200 m (width) | CFX-4, CFX-5<br>Plumes dispersion from the stack |
| Riddle et al. (2004) | 0.1–20 m | 1000 × 500 m (horizontal)<br>150 m (vertical) | FLUENT<br>Atmospheric dispersion |
| Scargiali et al. (2005) | 1–50 m (200 cells: geometric ratio of 1.02) (model 1)<br>50 × 50 × 25 cells: geometric ratio of 1.2 (model 2) | 2500 m (1-D)<br>30 × 30 × 2.5 $km^3$ (3-D) | CFX<br>Heavy gas dispersion |
| Pullen et al. (2005) | 6 m (horizontal and vertical) | 860 × 580 × 40 $m^3$<br>360 × 360 × 55 $m^3$ | FAST3D-CT urban CFD model |
| Li and Guo (2006) | Total 200,000 meshes | 5000 m diameter 200 m height | FLUENTodor dispersion |
| Prospathopoulos and Voutsinas (2006) | About 73 m (horizontal)<br>0.5, 1, 2, 3 m (vertical, 4 cases) | 3600 × 3600 m | Wind prediction for installing wind turbine |
| Hussein and El-Shishiny (2009) | Minimum 0.05 m with 1.24 growth rate (sub-domain 1)<br>Minimum 0.225 m with 1.26 growth rate (sub-domain 2)<br>Minimum 1.2 m with 1.3 growth rate (sub-domain 3) | 20.64 × 16.26 × 3.80 $km^3$ | OpenFOAM CFD toolkit ver 1.4.1<br>Wind environment over the historical heritage sites |
| Hanna et al. (2009) | 0.2 × 0.2 × 0.2 $m^3$ 1 × 1 × 0.2 $m^3$ (model 1)<br>6 × 6 × 2 $m^3$–18 × 18 × 2 $m^3$ (model 2) | Not mentioned<br>1300 × 2400 m | FLACS CFD<br>Chlorine dispersion |
| Seo et al. (2010) | 5 × 5 $m^2$–30 × 30 $m^2$ (horizontal) | 27 × 24 $km^2$ | FLUENT<br>Dust dispersion from reclaimed land |
| Gousseau et al. (2011) | Few centimeters | 1000 × 425 × 330 $m^3$<br>1150 × 460 × 330 $m^3$ | FLUENT<br>Near field pollutant dispersion on a high-resolution grid |
| Hong et al. (2011b) | 5–10 m | 2.4 km diameter<br>2500 m height | FLUENT<br>Odor dispersion |

Some additional techniques accompanied with the above methods can also improve the quality of computational meshes as well as economically reduce the number of meshes. In atmospheric odor dispersion modeling by Li and Guo (2006) and Hong et al. (2011a), the cylindrical domain was divided into 16 parts like a pizza in accord with the meteorological definition of wind directions, which helped wind direction configuration in boundary conditions very convenient. Hussein and El-Shishiny (2009) and Hong et al. (2011a) also split the computational domain into multiple sub-domains considering geometrical complexity and the degree of importance. Different sizes of meshes were applied to each sub-domain. They also adopted non-conformal interface between sub-domains. The non-conformal interface allows relaxing the requirement for point–point matching at the interface between two adjacent boundaries.

Fossum et al. (2012) have utilized large-eddy CFD simulation model to investigate aerosol dispersion in an area surrounding an existing biological treatment facility. In their study, the aerosol sources consist of two large aeration ponds that slowly diffuse aerosols into the atmosphere. These sources are modeled as dilute concentrations of a non-buoyant non-reacting pollutant diffusing from two horizontal surfaces. The time frame of the aerosol release is restricted to the order of minutes, justifying a statistically steady inlet boundary condition. To predict particle deposition, the authors simulated discrete particle transport and try to find out if both the large (18 μm) and small (2 μm) particles are deposited at approximately the same rate, or if only the smaller aerosols are transported over longer distances. To this date, no experimental reference data are available for particle deposition, however, in light of the author's previous simulation results, it is likely that the simulation provides a good indication as to how particles emitted from the aeration ponds are deposited in the domain. However, the obtained results are only tentative, as a more complete analysis requires knowledge of second-order effects (evaporation, agglomeration, etc.), a much higher number of particles, and a longer release time, in order to approach statistical convergence. Presented in Fig. 3 shows the deposition on the ground and building roofs after all (>99%) released particles have left the domain or been deposited. As the authors discussed, the deposition is chaotic, indicating the presence of turbulence. Within the near-field domain simulated, no significant difference in the deposition patterns for large and small particles was observed. Their simulation results have shown that 23% of the released particles are deposited, of which 58% are large particles. Hence, there is no significant difference between the deposition of large and small particles within the first few hundred meters downstream of the ponds.

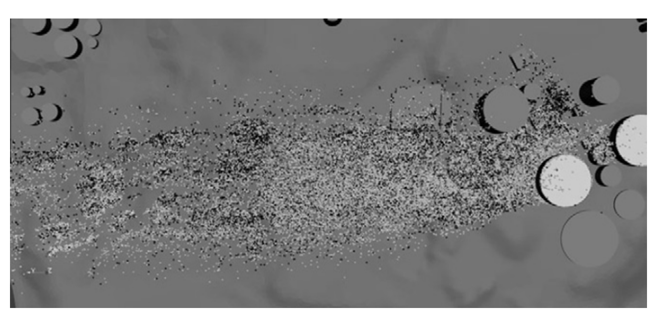

**Fig. 3.** Accumulated deposition at the end of the discrete particle simulation. *Dark color* heavy particles; *light color* light particles (reprinted with kind permission from Fossum et al., 2012. Copyright Boundary-Layer Meteorology).

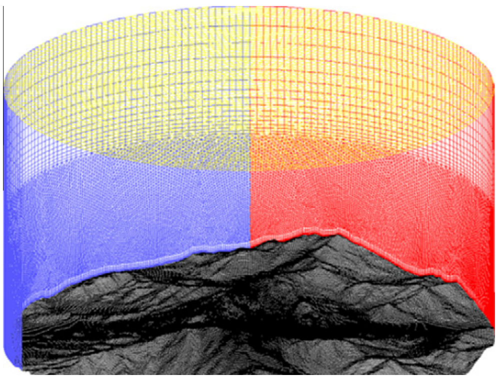

**Fig. 4.** 3-Dimensional computational domain with 3.6 km (D) and 2.5 km (H) designed in TGrid (reprinted with kind permission from Hong et al., 2011a. Copyright Biosystems Engineering).

One of the recent models on dispersion modeling was published by our team (Hong et al., 2011a, 2011b) with the ultimate goal of developing an aerodynamic model to qualitatively and quantitatively predict odor dispersion originating from livestock facilities. In the first paper (Hong et al., 2011a), as an initial stage of the research, the methodology for designing a complicated topography is suggested, and a three-dimensional grid model is presented with respect to the study area (Fig. 4). Grid construction method, selection of fundamental design criteria and topographical modeling were discussed. The mesh model of complex topography, with a 3.6 km diameter and 2.5 km height, was developed with a fine resolution. Well known commercially available computational tools presented in Fig. 4 were used for the topographical modeling (Hong et al., 2011a). An earlier wind tunnel experiment contributed to the selection of the grid size (to ensure grid independence), and the selection of time step and turbulence model for CFD simulation. In the related subsequent paper (Hong et al., 2011b), methodologies for modeling of the dispersion phenomenon including related User Define Function (UDF) and module designs were presented considering time-dependently changed wind speed and direction. Modules for modeling physical atmospheric phenomena as shown in Fig. 5 is developed and linked to a main computational process to predict the dispersion of livestock odor under various atmospheric conditions. The developed model was used to ameliorate odor conflicts as well as predict odor dispersion according to various meteorological and geographical conditions. For instance, in the study area in Cheongyang, Chungcheongnam province, Korea, westerly winds were the most influential in creating potential odor problems; north-westerly winds were the second most influential, with the longest distance being 71% that of the westerly wind. In Yesan area which is also located in the same province, the most influential wind direction was from the northwest; the second most was from the southwest. The most critical conditions were a westerly wind with neutral or stable atmospheric stability for the Cheongyang area, and a north-westerly wind and stable atmospheric stability for the Yesan area. On average, the simulation results found that the Cheongyang area had a 30% greater dispersion distance than the Yesan area under identical wind environments. Therefore, the Cheongyang area could be expected to require more remediation to address odor problems (Hong et al., 2011b).

### 2.3. Windbreaks

In the present decade, the application of CFD in studying wind flow characteristics around windbreaks becomes very popular. The most recent studies such as Steffens et al. (2012), Rosenfeld et al. (2010), Bitog et al. (2009), Gromke and Ruck (2008) and Santiago et al. (2007) have exploited the power of CFD technique to investigate and analyze wind flows over an area as affected by windbreaks. The availability of more powerful computers with higher memory has now allowed more reliable simulations of 3-dimensional models. In Steffens et al. (2012), CFD investigation was conducted in exploring the effects of a vegetation barrier on particle size distributions in a near road environment. It has been believed that roadside vegetation barrier can be a potential mitigation strategy for near-road air pollution. Thus, the authors conducted simulation studies to gain proper understanding on how road side barriers affect pollutant transport and transformation on and near roadways especially under different meteorological conditions and barrier properties. In the simulation procedures, the representations of particle aerodynamics and deposition mechanisms were incorporated into a Comprehensive Turbulent Aerosol Dynamics and Gas Chemistry (CTAG) model, and explored the effects of vegetation barriers on near-road particulate air pollution by comparing the simulation results against field measurements. Accordingly, CTAG is an environmental turbulent reacting flow model, designed to simulate transport and transformation of multiple air pollutants in complex environments, e.g., from emission sources to ambient background. The model shows generally adequate agreement with concentrations of particles larger than 50 nm, but tends to over-predict concentrations of particles less than 50 nm behind a vegetation barrier. It was found that an increase in leaf area density (LAD) further reduces particle concentration, but the responses were non-linear. Increases in wind speed were shown to enhance particle impaction, but reduce particle diffusion, which result in reduction in concentration for particles larger than 50 nm but have a minimal effect on particles smaller than 50 nm. Further improvements in representing particle deposition and aerodynamics in near-road environments are needed to fully capture the complex effects of roadside vegetation barriers.

Bitog et al. (2009) examined the quantitative effect of windbreak fences on wind velocity in the reclaimed land in Korea using CFD simulation, and its validity was also examined by conducting wind tunnel experiment. Simulation results revealed a maximum decrease of up to 93% of velocity as affected by the fences. A study

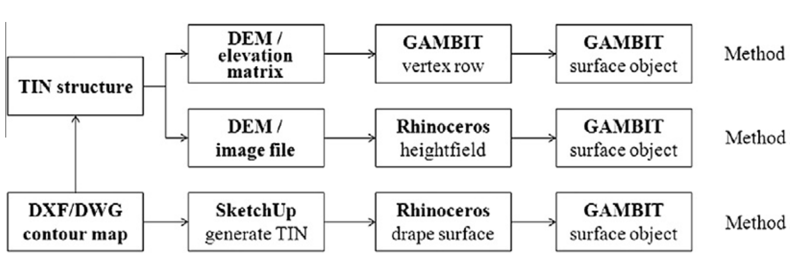


**Fig. 5.** Three methods for topographical modeling based on the DXF/DWG contour map (reprinted with kind permission from Hong et al., 2011a. Copyright Biosystems Engineering).

by Rosenfeld et al. (2010) established the significance and extent the 3-dimensional flow patterns across tree windbreak comprising of individual cypress trees. The cypress tree is modeled as a solid cylindrical stem and a conic porous canopy. Three dimensional flow was found in the vicinity of the windbreak up to a leeward distance of 1–2 tree-heights, depending on the density of the canopy, and is manifest as significant lateral variations and reduced vertical flow. Their simulation study was validated by comparing the results with experimental data which showed better agreement.

Lin et al. (2007) used also the technology to simulate odor dispersion downwind from natural windbreaks and to test the effect of tree characteristics such as tree porosity, type and height, and windbreak distance from the odor source. The airflow internal resistance of windbreaks was defined as proportional to the square of the tree diameter. Results showed that a less porous or denser windbreak (aerodynamic porosity of 0.2 versus 0.4 and 0.66) produced a shorter and wider odor plume, but with a higher odor level immediately downward from its position (Fig. 6). Bourdin and Wilson (2008) investigated the suitability of CFD with regard to windbreak aerodynamics. They used CFD to predict 2- and 3-dimensional turbulent flows through porous barriers. Mostly, the wind speed and pressure distributions of windward and leeward on the ground were analyzed to investigate the porous windbreaks. The measured and CFD computed horizontal wind speeds upwind and downwind from the Ellerslie windbreak (Wilson, 2004) were compared for validating the model.

Bitog et al. (2011a,b) attempted to find the effectiveness of trees as windbreak on soil erosion in agricultural field. Initially, they conducted wind tunnel tests (Bitog et al., 2011a) to find its drag coefficient and then proceed with 3-dimensional CFD simulation (Bitog et al., 2011b). The drag values of Black pine trees, one of the most typical tree windbreak in Korea, was found through wind tunnel tests, and then it was used as input value in the simulation. The effect of gap distance between trees, rows of trees and trees arrangement, in reducing wind velocity at various heights, was thoroughly and quantitatively analyzed.

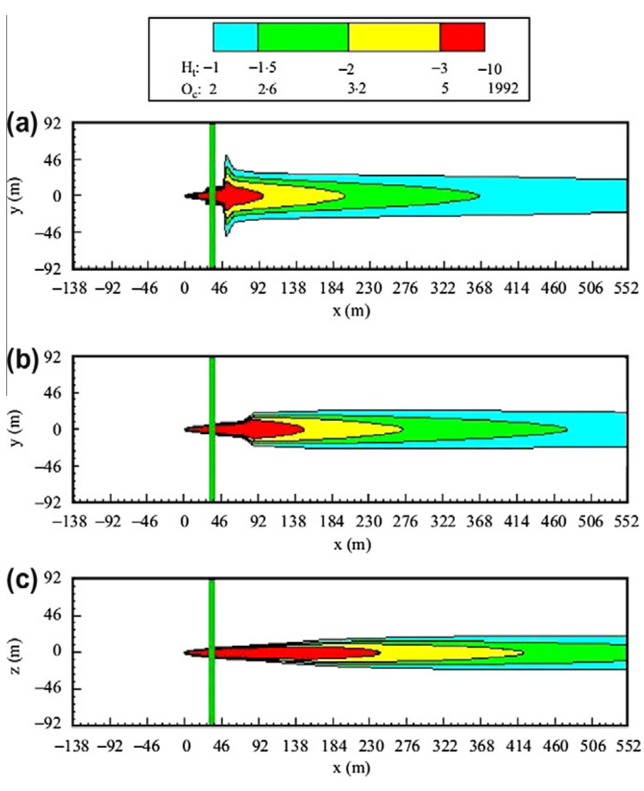


**Fig. 6.** Effect of windbreak aerodynamic porosity; contours of the odor plume ($z$ = 1.5 m) for an aerodynamic porosity of (a) 0.2 (simulation 1), (b) 0.4 (simulation 2) and (c) 0.66 (simulation 3), respectively; the green bar is the windbreak and the unit of the odor concentration is OU $m^{-3}$ (reprinted with kind permission from Lin et al., 2007. Copyright Biosystems Engineering).

A more detailed modeling on tree canopy was conducted by Endalew et al. (2006), who used separate models for the leaves and the branches of the canopy in their 3-dimensional CFD modeling of airflow within model plant canopies (Fig. 7). As discussed by Endalew et al. (2006), the determination of the typical patterns of physical quantities within vegetation canopies is difficult because of the complex airflow dynamics determined by the spatial variability of the canopy elements. However the general notion is that there is an overall reduction of air velocity through the canopy due to flow resistance by the canopy elements. They modeled a 3-dimensional structure of the canopy using relatively simple mathematical growth and architectural models and introduced it into the CFD main model to resolve the real effects of the plant and its branches on airflow. Results of their study shows a reduction of the average longitudinal air velocity where the extent of reduction depends on canopy density. It was clearly shown in their visualization analysis that the reduction is higher at about half the height of the tree where the density is relatively high and it decreases on the upper and lower parts.

### 2.4. Dust dispersion/sand erosion/sandstorm

A 3-D aerodynamic modeling on dust emission coupled with field monitoring at a reclaimed land was earlier attempted by our research team (Hwang et al., 2006). The aim of the study was to exploit CFD as an effective tool to build prediction and alarm system on dust diffusion. The simulation model was continuously upgraded based on the monitored data in the field. Results provided an estimate of dust concentration at every location in the computational domain. It also revealed the characteristics of dust dispersion based on the topography and weather condition of the area. These data were necessary to obtain accurate simulation results for predicting the dust concentration as confirmed from field measured data. A follow-up study was conducted by Seo et al. (2010) utilizing other computer aided design tools such as Geographic Information Systems (GISs), Triangular Irregular Network (TIN) and Digital Elevation Models (DEMs) (Fig. 8). Details

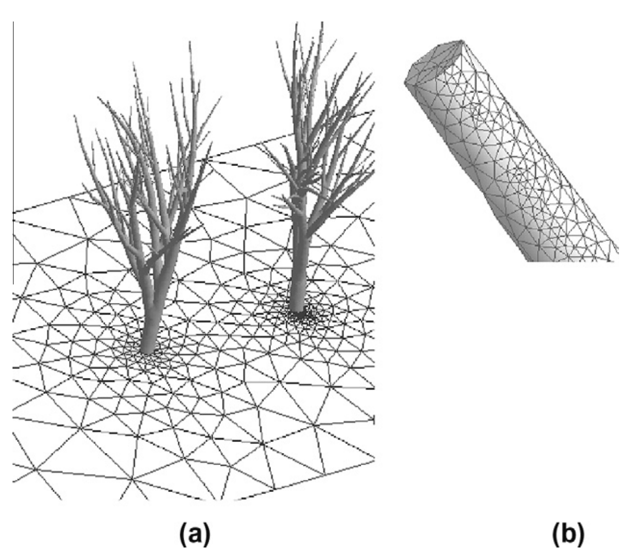


**Fig. 7.** The trees on the ground surface showing mesh refinement near the tree on the ground (a) and mesh on the surface of the smallest branch (b) reprinted with kind permission from Endalew et al., 2006 (Copyright Aspects of Applied Biology).

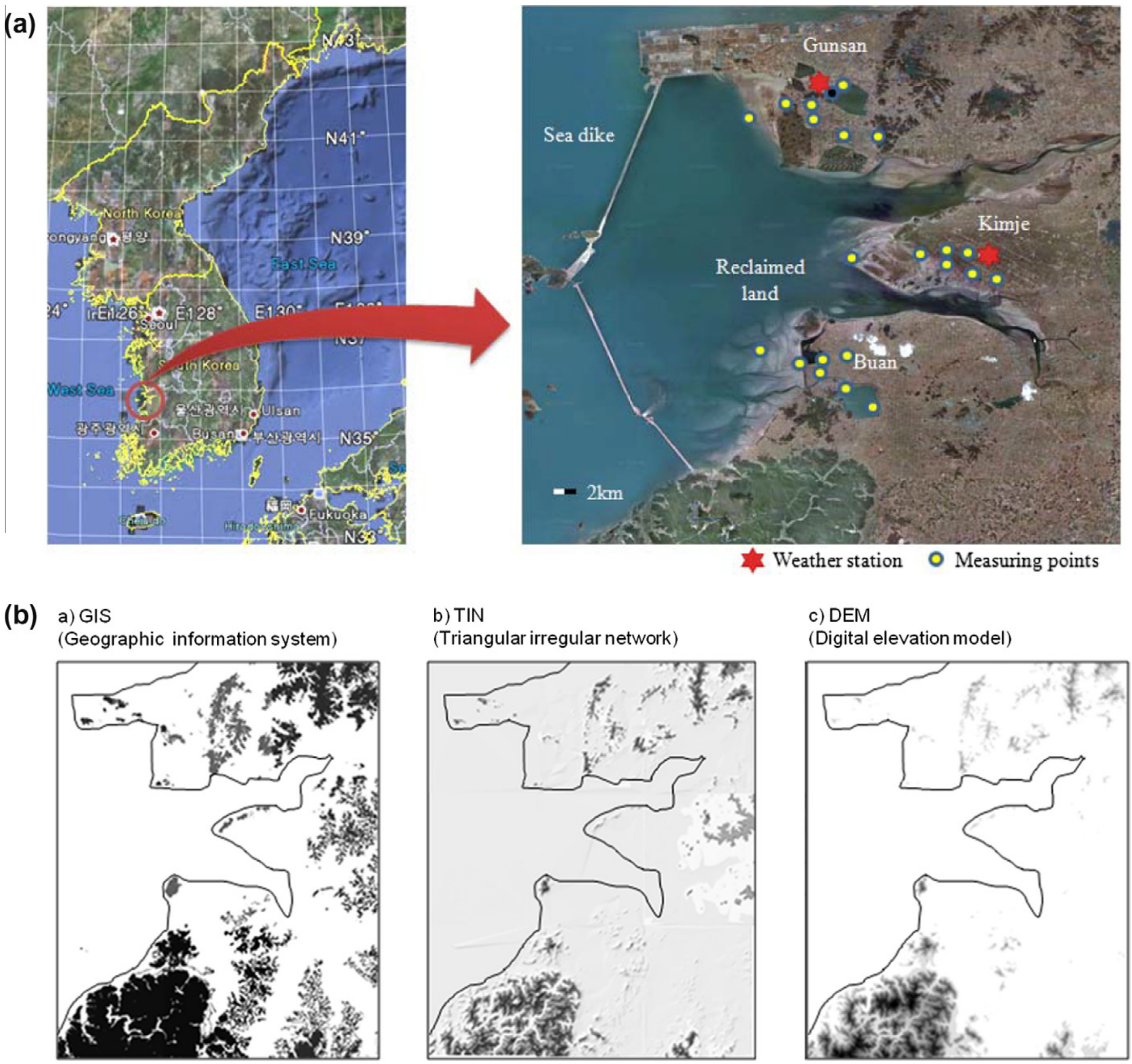


**Fig. 8.** (a) Satellite picture of Saemangeum reclaimed land at Jeonbuk Province (35°49′N, 126°36′W); (b) process of topographical modeling using GIS information of the Saemangeum area including reclaimed land and nearby areas at a 36 km width and a 38.5 km length (reprinted with kind permission from Seo et al., 2010. Copyright Transactions of the ASABE).

of using these tools in the CFD model were also discussed in the paper. The reliability of using these tools in the CFD models was earlier validated from experimental data and an average error of −6.8% was obtained which is within the acceptable range. Accordingly, the CFD model can be still improved using the time dependent weather conditions as well as the realistic dust distributions generated from the reclaimed land. The technique applied in the study can be utilized further to investigate the effect of artificial or natural barriers in order to minimize dust dispersion.

Sand storm is a serious environmental threat to humans as well as animals. In order to prevent and predict sand storms, the causes and the manners of particle motions resulting in saltation and suspension causing soil erosion in one place and deposition in another must be studied in detail. A CFD model was used for the gas phase simulation and the discrete element method (DEM) is used to predict the movements of particles using an in-house procedure. Then the data were summarized in an Eulerian–Eulerian regime after simulation to get the statistical particle Reynolds stress and particle collision stress (Zhang et al., 2010; Kang et al., 2008). Qiu et al. (2003) simulated the distribution of wind velocities in different straw checkerboard sizes on fixing sand. In this study, the importance of roughness length was emphasized in the simulation model since it is an important parameter in reflecting the resistance of the ground to the wind. Accordingly, a large value of the roughness length indicates a larger resistance to flow (Qiu et al., 2003). Results from the field experiment and simulation show that 10–20 cm height for the straw checkerboard has a substantial effect on dune fixation.

### 2.5. Forest fire

Some forest fire studies are available in literature, however limited. The earliest CFD study was done by Morvan and Dupuy (2001) where they predicted fire propagation in Mediterranean shrub land by representing the vegetation as a collection of solid fuel particles distributed with appropriate size, moisture content, density, etc. Separate layers were created to represent ground cover, crown canopy regions, thinning, and fire breaks. The model captures the degradation processes (drying, pyrolysis, char combustion) and ignition. Calculations were performed over a domain 5 m tall and 20 m long. The authors considered different cell sizes (5, 10 and 20 cm) and compared rate of spread, mass fluxes, contributions of radiation and convection. The model predicted the temperature and velocity field which is detrimental to fire when the maximum wind speed is 1 and 5 m $s^{-1}$. Wagenbrenner et al. (2010) presented a CFD modeling approach for predicting ash and dust emissions in forest-fire environments with complex terrain as well as calibration and validation methods for model evaluation. The study investigated the erosion mechanism that governs the emissions from burned soil and ash and attempted to predict the local terrain effects on winds in the mountainous regions where wildfires often occur. The authors proposed linkage of an existing computational fluid dynamics (CFD) code with an existing dust algorithm to generate gridded $PM_{10}$ vertical fluxes from burned landscapes. They also outlined the development of the linked model and described the ongoing CFD validation effort, which is a critical first step in model development.

In Korea, a tragic accident happened when a supposed traditional event of burning dried grasses went out of control and the fires spread over Mt. Hwawang on February 9, 2009 due the strong wind. This accident prompted Koo et al. (2009) to analyze the fatal wind based on wind flow simulations over a digitized complex terrain of the mountain with a localized heating area using a three-dimensional CFD model. Three levels of the fire intensity were simulated: no fire, 300 °C, and 600 °C of surface temperature at the site on fire. They insisted that the model can be utilized in turbulence forecasting over a small area due to surface fire in conjunction with a mesoscale weather model to help fire prevention at the field.

## 3. CFD modeling in land and water management

CFD application in soil and water management is fast emerging. The discussion presented here divides the topic into two: Soil management and water pollution and design of hydraulic structures.

### 3.1. Soil management and water pollution

Generally, water and soil pollution is now becoming one of the global problems which should be given immediate attention. Thus, simulation studies on small scale pollution of water and soil pollution has already been done. These studies are hereby discussed.

A paper by Jia et al. (2010) presents a 3-D numerical model to simulate morphological changes in alluvial channels due to bank erosion. They have established a method to simulate bank erosion and incorporated into a 3-D mathematical model for turbulent flow and non-uniform, non-equilibrium sediment transport. Accordingly, the bank erosion module that was developed also includes other factor affecting the rate of bank erosion, such as longitudinal length of failed bank, the thickness of each layer in the double layer structure, and the season resisting effect of cohesive material from the top layer of failed bank. They have also proposed a locally-adaptive grid system to efficiently simulate the lateral migration of alluvial channel due to bank erosion.

The production, processing, and storage of petroleum and its products are one of the main contributors to soil and groundwater contamination was the focus of the study by Al-Baghdadi et al. (2009). They employed CFD to model the contaminant transport in soils including the effect of chemical reactions. The movement of chemicals through the soil to the groundwater, or their discharged to surface waters, represents a degradation of these resources. Al-Baghdadi et al. (2009) developed a CFD model which they applied to study contaminant transport through a column of sandy soil including the combined effect of advection and dispersion. Their investigation results showed that the contaminant transport model is capable in simulating various phenomena governing miscible contaminant transport in soils including advection, dispersion, diffusion, adsorption and chemical reaction effects. Their model performed well in predicting transport of contaminants through the soil. Comparison with experimental results shows that the CFD model is capable of predicting the effects of chemical reactions with very high accuracy.

Earlier numerical studies on soil forces and stresses was conducted by Chi and Kushwaha (1989) where they used a non-linear 3-dimensional finite element modeling to find that soil forces on a tillage tool edge were larger than the force at the center of the tool. The soil deformation pattern around a tillage tool was studied considering the soil as a visco-plastic material using CFD (Karmakar and Kushwaha, 2005). In a follow-up study, Karmakar et al. (2007) investigated the pressure distribution over the surface of a flat tillage tool and the soil stress pattern due to forward motion of the tool for high-speed tillage. CFD was also employed in analyzing soil stresses due to the tool motion which are very important in identifying the soil mechanical behavior. Furthermore, soil pressure on tillage tools and its distribution over the tool surface are also considered as one important factor for tool design with respect to tool wear. The soil–tool interaction was analyzed from fluid flow perspective. A narrow, rigid and vertical blade was considered as stationary tool in the middle of the viscoplastic soil flow domain. The soil was characterized for its rheological behavior as a Bingham material, and then three-dimensional analyzes were carried out by the control volume method with structured mesh with a single-phase laminar flow. Though the rectangular tillage tool was not a streamlined body, it was compared to the hydrodynamic nature of streamlined body with respect to the contribution of pressure and viscous drag on the tool.

In a study by Wu and Crapper (2009, Fig. 9) on polluted soils, they insisted that biopiles are a common worldwide treatment for the ex situ remediation of the contaminated soil. They conducted CFD simulations to model a biopile under the influence of wind pressure, with and without forced aeration including in the simulation the temperature and the bioreactions. Gas flow within the pile has been modeled with qualitative accuracy and bioremediation including microbial degradation and temperature variation has been represented. Preliminary results indicate that the cooling effect of ambient wind on the surface of the pile is significant and that considerable contaminant is lost via diffusive flow to the atmosphere, or flow via aeration pipes, before being degraded. The results indicated that a very high proportion of contaminant loss from the pile is due to venting to the atmosphere, rather than to microbial degradation. Unfortunately, validation of the study was not presented due to lack of available data in the literature related to the topic. This study therefore requires further work which will include experimental validation of the model and the representation of more complex reaction models.

### 3.2. Design of hydraulic structures

Hydraulic structure design requires knowledge of the drag forces and pressures that the flow will impart on the structure. Physical model studies can be performed in certain situations to quantify the forces on the structure and its response. In other cases CFD studies need to be performed to obtain this information. CFD is used extensively by engineers to model and analyze complex

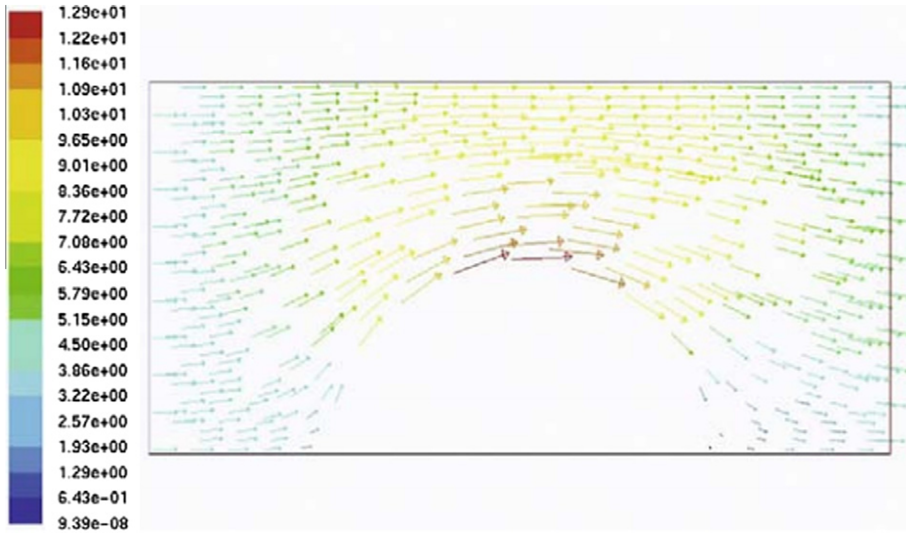


**Fig. 9.** Gas flow field around biopile at 350 h, 5 m s$^{-1}$ wind (reprinted with kind permission from Wu and Crapper, 2009. Copyright Desalination).

issues related to hydraulic design, planning studies for future generating stations, civil maintenance, supply efficiency, and dam safety. The integrity of computed values from CFD models is of considerable economic importance in the design, upgrading and maintenance of hydroelectric generating stations (Chanel and Doering, 2007).

Computer simulations used for the optimum design of hydraulic structures have been performed to study the stress and surface pressure analysis of dam gate (Lee and Woo, 2004). CFD studies have also been conducted to find the amount of water discharge, cavitation and fluid analysis at spillways, and flow distribution around lock canals (Kim et al., 2003; Jean and Mazen, 2004; Tabbara et al., 2005; Bhajantri et al., 2007; Lee et al., 2007; Chanel and Doering, 2008; Gaston et al., 2009). Hong et al. (2011c) attempted to combine the flow analysis and the gate movement. In this CFD study, the dynamic mesh method was used to perform the time-dependent dynamic movements of the overturnable gate affected by water pressure and level as well as characteristics of water flow before and after gate opening. During the overturning and restoration of the gate, its hydrodynamic characteristics were analyzed quantitatively as well as qualitatively, and those time-dependently computed data were used for the optimum design of the gate.

Biscarini et al. (2009), using the Volume of Fluid (VOF) method designed a multi-phase model to study dam break flow. Experimental and numerical literature data were used for the CFD validation. They compared the experimental data with the modeling results deriving from shallow water and detailed Navier–Stokes numerical models. The former is based on two-dimensional hydrodynamics and sediment transport model for unsteady open channel flow and the latter on Reynolds-averaged Navier–Stokes algorithm. In the latter, the water–air interface was captured with the VOF method. In this paper, the RNG k–ε model was used in both the shallow water approximation and the detailed three-dimensional simulation. They insisted that LES model is in fact unacceptable for this study because of large-scale problems. The VOF approach was also utilized by Wang and Yan (2007) when they simulated the effect of bed discordance on flow dynamics at Y-shaped open channel confluences. In their study, they focused on determining the different characteristics between the asymmetrical river confluences and symmetric confluences such as the Y-shape confluence. Their study revealed a lot of quantitative flow differences between the confluences and the discordant bed height play a very important role at the Y-shaped junction.

CFD models of the forebay of dams have been reported by Meselhe and Odgaard (1998), Meselhe et al. (2000), Muste et al. (2001), Lai et al. (2003) and Khan et al. (2008). These studies have focused on dams on the Columbia and Snake Rivers, where sluiceways, combined with partial screening of flows, are used for juvenile fish passages. The CFD was used to investigate juvenile fish passage systems. Khan et al. (2008) also simulated three-dimensional CFD model of the forebay of the dam to investigate forebay hydrodynamics generated by the floating surface collector (FSC) and operation of the powerhouse. It was validated against field data.

Nguyen and Nestmann (2010) presented various applications and developments of CFD technology in hydraulics and river engineering as well as navigation. The flow in rivers is very complicated, because it is not only turbulent and highly three-dimensional, but also has irregular boundaries of a complex geometry, a rough bed and a free surface. The ability to accurately predict the 3-dimensional flow in open channels and rivers is of obvious importance for the design and construction of hydraulic systems in rivers. Its accuracy was examined using the experimental data of water free surface. They used two methods for designing free surface such as free surface tracking and VOF models. The advantage of the free surface tracking method was to obtain a sharp shape of a free surface, but they found that the numerical implementation becomes very difficult when the free surface is strongly enfolded (e.g. flow over spillways, weirs, sluices, etc.). On the other hand, the VOF model can overcome this limitation, but it has also some inherent disadvantages of larger CPU time and storage space due to the extension of the solution domain. One recent study on river channels was done by Huang (2009) where they considered curved channels to represent the rivers. They have considered the helical flow structure which has a very important bearing on sediment transport, riverbed evolution, and pollutant transport study. Furthermore, they compared the different turbulent models with different pressure solution techniques by comparing the vertical-averaged velocities with the experimental data and found out satisfactory results. Discussion on the turbulent discrepancies with respect to surface elevations, super elevations and secondary flow patterns were presented.

### 3.3. Aquaculture

Mohammadi (2008) and Nagata et al. (2005) used CFD for typical hydraulic engineering cases such as the effect of flow over weirs through bridge piers and dam breaks. Fragmentation and other effects of dams have been linked to the loss of populations and species of fish. Successful guidance and passage designs reduce effects of dams or other barriers that obstruct the dispersal and migration of organisms. The passage of downstream out-migrating juvenile fishes around hydropower dams has historically been difficult to manage and not entirely successful. Emigrants navigate in a dense, relatively incompressible fluid (water) that distorts as it flows over or around static features of the channel and in response to solid objects moving within the fluid. The distortion of the flow field creates spatial gradients in velocity that may or may not lead to the formation of vortices and turbulence, a flow field attribute known to be important to fish because it impacts swimming efficiency and sensory acuity. That kind of turbulence was computed from variables commonly output by steady-state Reynolds-averaged Navier–Stokes (RANS) and could also link characteristics of the flow field to known capabilities of the fish mechanosensory system (Nestler et al., 2008). The Unsteady, Unstructured RANS (U2RANS) CFD model was used to capture the 3-dimensional steady-state attributes of the flow field for all evaluation cases. Attributes of CFD models important for biological application are described in Weber et al. (2006). In their paper, the discussion was focused on three parts:(1) an agent-based model, that simulates the movement decisions made by individual fish, (2) an Eulerian CFD model that solves the 3D Reynolds-averaged Navier–Stokes (RANS) equations with a standard k–ε turbulence model with wall functions using a multi-block structured mesh, and (3) a Lagrangian particle-tracker used to interpolate information from the Eulerian mesh to point locations needed by the agent model and to track the trajectory of each virtual fish in three dimensions.

CFD modeling has been used also in simulating water flow velocities patterns and sediment conditions in aquaculture ponds (Peternson et al., 2000, 2001). Their methodology is capable of simulating any combination of paddlewheels and propeller-aspirators in a single pond. Pond bathymetry is modeled with a smooth bottom and a piece-wise series of inclined banks, to generally represent any convoluted shoreline. Huggins et al. (2004) utilized CFD to analyze sediment transport modeling for aquaculture raceways. The results of their simulation were used to evaluate the efficiency of solids settling in the quiescent zone of an existing trout raceway. They have developed a methodology for analyzing the raceway sediment transport in terms of its percentage of solids removed based on CFD simulations which can also be used to examine raceway design alternatives for improving the particle removal efficiency. Huggins et al. therefore made a follow-up study on this field when they tested a number of potential raceway design modifications using CFD model of a "standard" aquaculture raceway (Huggins et al., 2005). They attempted to design aquaculture raceways to evaluate the impact of potential raceway design modifications on the in-raceway settling of solids (Fig. 10). Their simulation results show quantitatively the effect of settling velocity on sedimentation effectiveness, with small differences in the particle settling velocity causing large changes in percent solid removal values.

CFD simulations of aquaculture systems were used to describe water flow and solids removal in circular tanks (Montas et al., 2000; Veerapen et al., 2002). Validation of the tank CFD model was carried out in a qualitative manner based on experimental observations (Montas et al., 2000). Montas et al. (2000) and Veerapen et al. (2002) agreed with some of the well-known advantages of using CFD modeling over laboratory physical models and found CFD models to be more flexible, faster to develop, and less expensive than physical models.

## 4. Validation of CFD models

CFD is very important technology not only to complement field limitations but also to get numerous quantitative and qualitative data for complex flow problem. Appropriate validation progress of CFD model for each research purpose is extremely important no matter how complex or simple are the models. Validation is defined as the process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model (AAIA, 1998). Validation should depend on direct comparison between CFD computed result and measured experimental result. A number of validation studies have been performed in the past; however, Table 2 only summarizes the most recent literatures on validation of CFD model with various methods. Generally it is very difficult to obtain accurate and reasonable result of fluid dynamic factors from field experiment because the experimental situation is changeable and unstable in spite of time and labor consumption. Therefore many research groups considered wind tunnel test, PIV test, scale-model and so onto optimize the experimental situation. Many research groups also used previous data conducted by earlier researchers in order to validate the CFD model with similar research purpose (Bourdin et al., 2008; Ruhaak et al., 2008; Wang et al., 2009; Biscarini, 2010; Stoesser et al., 2009; Bartzanas et al., 2010; Rosenfeld et al., 2010).

In atmospheric field, some validation tests were conducted in real field by means of measurement of odor, gas, and dust concentrations (Hong et al., 2011; Seo et al., 2010). Wind tunnel test which uses real air with scaled model for certain phenomenon on their research purposes are also actively used to get reliable data for CFD validation (Gromke and Ruck, 2008; Bitog et al., 2009; Dimitrova et al., 2009; Endalew et al., 2009; Mohaned et al., 2009). The research groups have conducted wind tunnel test to get mainly air velocity magnitude, velocity distribution, heat distribution, tracer gas concentration and so on. In water field, scale model in the laboratory have been commonly used to get wave

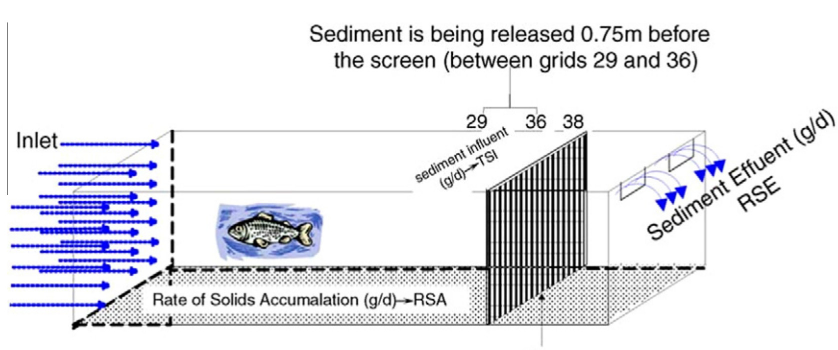


**Fig. 10.** Schematic diagram of the simulated standard raceway (SSR). The sediments are being released on the surface of the raceway 0.75 m before the screen (reprinted with kind permission from Huggins et al., 2005. Copyright Aquacultural Engineering).

**Table 2**
References of the most recent CFD validation results according to their research fields and targets.

| Field | Target | Tool | Domain size (m) and cell number | Method | Accuracy | Reference |
|---|---|---|---|---|---|---|
| Atmosphere | Odor dispersion in complex terrain | FLUENT | 4800 m dia. 4,223,735 | Field experiment | $R^2$ = 0.863 | Hong et al. (2011) |
| | Wind break | FLUENT | 450 × 633 × 1080 5,090,000 | Compared to the literature | Acceptable | Rosenfeld et al. (2010) |
| | Non-uniform hydrogen mixture explosion | FLUENT | 1.5 m dia. 5.7 m high 1 millions | Lab experiment | 9.1–46.8% | Makarov et al. (2010) |
| | Fugitive dust dispersion | FLUENT | 6000 × 6000 × 1000 1.2 millions | Field experiment | 6.80% | Seo et al. (2010) |
| | Pollutant dispersion from vehicle exhaust | FLUENT | 3 × 1.5 × 1.5 461,070 | Wind tunnel | Acceptable | Yassin et al. (2009) |
| | Airflow with plant canopies | CFX | 0.5 × 0.4 × 2 6,577,243 | Wind tunnel | 16.6%, 13.3% | Endalew et al. (2009) |
| | Wind break | FLUENT | 23 × 0.5 × 6 636,250 | Wind tunnel | 7.20% | Bitog et al. (2009) |
| | Thermal effects on wind field | CHENSI | 2.1 × 1.6 × 1.1 0.0095 cell size | Wind tunnel | Acceptable | Dimitrova et al. (2009) |
| | Street canyons with wind planting | FLUENT | 180 × 18 × 18 300,000 | Wind tunnel | $R^2$ = 0.97 NMSE = 0.09 | Gromke and Ruck (2008) |
| | Wind break | FLUENT | 120 × 120 × 50 | Compared to the literature | Acceptable | Bourdin et al. (2008) |
| Water | Turbidity currents | FLUENT | Overall 18.6 $m^3$ 440,058 | Lab experiment | Acceptable | Georgoulas et al. (2010) |
| | Landslide generated waves | FLUENT | 0.45 m channel With 45° angle 40,000 | Compared to the literature | Acceptable | Biscarini (2010) |
| | Meandering channel | Hydro3D | 0.25 × 0.2 × 10 36,236 | Compared to the literature | Acceptable | Stoesser et al. (2009) |
| | Gross pollutant trap | FLUENT | 0.6 × 0.6 × 19 36,236 | Scale model | Acceptable | Madhani et al. (2009) |
| | Air entrainment at spillway aerators | FLUENT | 1.65 × 1.5 × 8 266,934 | Scale model | 17% | Aydin and Oztuk (2009) |
| | Spilling breaking wave | FVM solver | 0.4 × 22 (2D) 28,800 | Compared to the literature | Acceptable | Wang et al. (2009) |
| | Nutrient transport | FLUENT | 20 × 100 × 75 360,000 | Not clear | – | Williamson et al. (2009) |
| | Cryogenic spill of LNG in complex domain | FLUENT | 500 × 500 × 50 Not clear | Field experiment | Acceptable | Gavelli et al. (2008) |
| Soil | Water quantity in cut grass | FLUENT | 10 × 250 × 200 324,000 | Compared to the literature | Acceptable | Bartzanas et al. (2010) |
| | Rock mass hydraulic behavior | FLUENT | 0.02 × 0.3 × 0.0005 890,000 | Field experiment | 3–17% | Javadi et al. (2010) |
| | Groundwater and heat transport | CHEMAT | 200 × 100 (2D) Not clear | Compared to the literature | Acceptable | Ruhaak et al. (2008) |

height, wave shape, velocity distribution, gas concentration and so on because experimental sites were generally huge with unstable environmental weather condition (Aydin and Oztuk, 2009; Madhani et al., 2009; Georgoulas, et al. 2010). As the experiments using water and other liquids are more controllable than atmospheric experiment, already published works could be used for the validation of CFD model. The CFD analysis in ground field is rare, and some research groups have used scale models to get pressure drop for comparison between CFD models and already proven analytical models (Ruhaak et al., 2008; Bartzanas et al., 2010).

## 5. Conclusions

The application of CFD in the agro-environment has been remarkable. CFD is already a well-proven tool and economically feasible since the advances in computing make it possible to conduct simulation studies in desktop PC. Many CFD validation studies have also shown quite comparable results to real world or wind tunnel studies. The values obtained in CFD calculations may not be sufficiently exact; however, for engineering purposes, the degree of error is within reasonable bounds. No doubt, CFD analysis is able to increase quality and reduce cost for research and engineering development in the agro-environmental field involving fluid flows, dispersions, heat transfer, mass transfer and reactions.

Currently, the size of the most CFD projects models are limited by the computing power and software used, however, the fast ever computing power of PCs continually expands the potential of CFD and can be generally more flexible at accounting for the unique aspects of every CFD project.

The direction of CFD in agro-environment is gearing towards more 3D flow dynamics modeling of simple to very complex computational domains. The high utilization of several computer aided designs (CAD) and imported for CFD simulation has also been attempted by modelers to exactly create their geometry almost similar or the same with the real world. Because computational mesh has great influence on the final result, powerful tools and methods for complex mesh design should be developed and introduced to the CFD simulation. Meshless or meshfree CFD codes are also possible solutions in this connection.

In terms of reliability and accuracy, CFD models are often validated from field experimental data. However, until now, a standard or a benchmark on validating any CFD model is not yet available.

The problem that the researcher solves is getting more complicated and needs collaboration with various fields of researches. Past purpose of the CFD simulation was focused on the flow analysis and its application. However recent agro-researches request more complicated analyzes, such as operation of machinery, growth of animal and crop and energy efficient building. The recent problems need not only basic flow simulation but also combination of various theories and mechanisms in physics, chemistry, or biology. The theories are solved in the CFD simulation coupled with the basic flow-related governing equations. For instance, many physical phenomena should be linked to CFD to investigate other factors such as modeling actual particle so that the general assumption where particles follow fluid flow can be avoided.

In the near future, CFD can be utilized for accurate forecasting of air pollution levels for odor, dust, aerosols, etc. in a short period of time. CFD back data from various weather conditions can be utilized to developed forecasting system which can be used for various purposes.

## References


AIAA, 1998. Guide for Verification and Validation of Computational Fluid Dynamics Simulation. AIAA-G-77. American Institute of Aeronautics Astronautics.

Al-Baghdadi, A.R.S., Abdolhamid, H.R., Elgarini, M.M., 2009. CFD modeling of contaminant transport in soils including the effect of chemical reactions. Ovidius University Annals of Chemistry 20 (1), 40–43.

Aydin, M.C., Oztuk, M., 2009. Verification and validation of a computational fluid dynamics (CFD) model for air entrainment at spillway aerators. Canadian Journal of Civil Engineering 36 (5), 826–836.

Bakker, A., Ahmad, H.H., Lanre, M.O., 2001. Realize greater benefits from CFD. Fluid/Solids Handling March, 45–53.

Bartzanas, T., Bochtis, D.D., Sorensen, C.G., Sapounas, A.A., Green, O., 2010. A numerical modeling approach for biomass field drying. Biosystems Engineering 106, 458–469.

Bergeles, G., Glekas, I., Prospathopoulos, I., Voutsinas, S.G., 1996. Statistical and physical modelling of wind resources in complex terrain: assessment of the applicability of a 3D Navier Stokes code. In: Proceedings of EUWEC'96, Goteborg, Sweden.

Beyers, M., Waechter, B., 2008. Modeling transient snowdrift development around complex three-dimensional structures. Journal of Wind Engineering and Industrial Aerodynamics 96, 1603–1615.

Bhajantri, M.R., Eldho, T.I., Deolalikar, P.B., 2007. Numerical modelling of turbulent flow through spillway with gated operation. International Journal for Numerical Methods in Engineering 2007 (72), 221–243.

Bitog, J.P., Lee, I.B., Shin, M.H., Hong, S.W., Hwang, H.S., Seo, I.H., Yoo, J.I., Kwon, K.S., Kim, Y.H., Han, J.W., 2009. Numerical simulation of an array of fences in Saemangeum reclaimed land. Atmospheric Environment 43, 4612–4621.

Bitog, J.P., Lee, I.B., Shin, M.H., Hong, S.W., Hwang, H.S., Seo, I.H., Yoo, J.I., Kwon, K.S., Kim, Y.H., Han, J.W., 2011a. A wind tunnel study on aerodynamic porosity and windbreak drag. Forest Science and Technology 7 (1), 8–16.

Bitog, J.P., Lee, I.-B., Hwang, H.-S., Shin, M.-H., Hong, S.-W., Seo, I.-H., Kwon, K.-S., Mostafa, E., Pang, Z., 2011b. A wind tunnel and numerical simulation study of a tree windbreak. Biosystems Engineering, http://dx.doi.org/1016/j.biosystemseng.2011.10.06.

Biscarini, C., 2010. Computational fluid dynamics modelling of landslide generated water waves. Landslides 7, 117–124.

Biscarini, C., Francesco, S.D., Manciola, P., 2009. CFD modeling approach for dam break flow studies. Hydrology and Earth System Sciences Discussions 6, 6759–6793.

Blocken, B., Stathopoulos, T., Carmeliet, J., 2007. CFD simulation of the atmospheric boundary layer: wall function problems. Atmospheric Environment 41, 238–252.

Bourdin, P., Wilson, J.D., 2008. Windbreak aerodynamics: is CFD reliable? Boundary-Layer Meteorology 126, 181–208.

Brown, G.J., Fletcher, D.F., 2003. CFD prediction of odour dispersion and plume visibility for alumina refinery calciner stacks. In: Third International Conference on CFD in the Minerals and Process Industries, Melbourne, Australia, pp. 111–120.

Chanel, P.G., Doering, J.C., 2007. An evaluation of computational fluid dynamics for spillway modelling. In: Proceedings of the 16th Australasian Fluid Mechanics Conference, 2-7 December 2007, Gold Coast, Australia.

Chanel, P.G., Doering, J.C., 2008. Assessment of spillway modeling using computational fluid dynamics. Canadian Journal of Civil Engineering 35 (12), 1481–1485.

Chi, L., Kushwaha, R.L., 1989. Finite element analysis of forces on a plane soil blade. Canadian Agricultural Engineering 31, 135–140.

Chin, S.M., Won, C.S., Hur, N., 2004. The development of a mesh generation program using contour line data. Korean Society of Computational Fluids Engineering 9 (4), 7–12.

Diego, I., Pelegry, A., Torno, S., Torano, J., Menendez, M., 2009. Simultaneous CFD evaluation of wind flow and dust emission in open storage piles. Applied Mathematical Modelling 33 (7), 3197–3207.

Dimitrova, R., Sini, J.-F., Richards, K., Schatzmann, M., Weeks, M., Garcia, E.P., Borrego, C., 2009. Influence of thermal effects on the wind field within the urban environment. Boundary-Layer Meteorology 131, 223–243.

Endalew, A.M., Hertog, M., Gebrhiwot, M.G., Baelmans, M., Ramon, H., Nicolai, B.M., 2006. Modelling airflow through 3D canopy structure of orchards. Aspects of Applied Biology 77, 465–472.

Endalew, A.M., Hertog, M., Gebrhiwot, M.G., Baelmans, M., Ramon, H., Nicolai, B.M., Verboven, P., 2009. Modelling airflow within model plant canopies using an integrated approach. Computers and electronics in Agriculture 66, 9–24.

Fluent Inc., 2006. Fluent Manual. New Hampshire, USA.

Fossum, H.E., Pettersson Reif, B.A., Tutkun, M., Gjesdal, T., 2012. On the use of computational fluid dynamics to investigate aerosol dispersion in an industrial environment: a case study. Boundary-Layer Meteorology. http://dx.doi.org/10.1007/s10546-012-9711-z.

Franke, J., Hirsch, C., Jensen, A.G., Krüs, H.W., Schatzmann, M., Westbury, P.S., Miles, S.D., Wisse, J.A., Wright, N.G., 2004. Recommendations on the use of CFD in wind engineering. In: van Beeck, J.P.A.J. (Ed.), Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics. COST Action C14, Impact of Wind and Storm on City Life Built Environment, von Karman Institute, Sint-Genesius-Rode, Belgium 5e7, May 2004.

Franke, J., Hellsten, A., Schlünzen, H., Carissimo, B., 2007. Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment. COST 732: Quality Assurance and Improvement of Microscale Meteorological Models.

Gaston, E., Ferrari, G.E., Politano, M., Weber, L., 2009. Numerical simulation of free surface flows on a fish bypass. Computers & Fluids 38, 997–1002.

Gavelli, F., Bullister, E., Kytomaa, H., 2008. Application of CFD (Fluent) to LNG spills into geometrically complex environments. Journal of Hazard Materials 159 (1), 158–168.

Georgoulas, A.N., Angelidis, P.B., Panagiotidis, T.G., Kotsovinos, N.K., 2010. 3D numerical modelling of turbidity currents. Environmental Fluid Mechanics 10 (6), 603–635.

Gousseau, P., Blocken, B., Stathopoulos, T., van Heijst, G.J.F., 2011. CFD simulation of near-field pollutant dispersion on a high-resolution grid: a case study by LES and RANS for a building group in downtown Montreal. Atmospheric Environment 45 (2), 428–438.

Gromke, C., Ruck, B., 2008. Aerodynamic modelling of trees for small-scale wind tunnel studies. Forestry. http://dx.doi.org/10.1093/forestry/cpn027.

Hanna, S.R., Hansen, O.R., Ichard, M., Strimaitis, D., 2009. CFD model simulation of dispersion from chlorine railcar releases in industrial and urban areas. Atmospheric Environment 43, 262–270.

Hefny, M.M., Ooka, R., 2009. CFD analysis of pollutant dispersion around buildings: effect of cell geometry. Building and Environment 44, 1699–1706.

Holmes, N.S., Morawska, L., 2006. A review of dispersion modelling and its application to the dispersion of particles: an overview of different dispersion models available. Atmospheric Environment 40, 5902–5928.

Hong, S., Lee, I., Hwang, H., Seo, I., Bitog, J., Kwon, K., Song, J., Moon, O., Kim, K., Ko, H., Chung, S., 2011a. CFD modeling of livestock odour dispersion over complex terrain, Part I: Topographical modeling. Biosystems Engineering 108, 253–264.

Hong, S., Lee, I., Hwang, H., Seo, I., Bitog, J., Kown, K., Song, J., Moon, O., Kim, K., Ko, H., Chung, S., 2011b. CFD modeling of livestock odour dispersion over complex terrain, Part II: Dispersion modeling. Biosystems Engineering 108, 265–279.

Hong, S.W., Lee, I.B., Lee, K.Y., Kim, H.D., Seo, I.H., Hwang, H.S., Bitog, J.P., Kwon, S.H., 2011c. CFD analysis on the performance of an overturnable water gate for reservoir spillway. Applied Engineering in Agriculture 27 (5), 719–728.

Huang, J., 2009. Three-dimensional numerical modeling of secondary flows in a wide curved channel. Journal of Hydrodynamics 21 (6), 758–766.

Huggins, D., Piedrahita, R.H., Rumsey, T., 2005. Use of computational fluid dynamics (CFD) for aquaculture raceway design to increase settling effectiveness. Aquacultural Engineering 33, 167–180.

Huggins, D., Piedrahita, R.H., Rumsey, T., 2004. Analysis of sediment transport modeling using computational fluid dynamics (CFD) for aquaculture raceways. Aquacultural Engineering 31, 277–293.

Hussein, A.S., El-Shishiny, H., 2009. Wind flow modeling and simulation over the Giza Plateau. ACM Journal on Computing and Cultural Heritage 2 (2). Article 6.

Hwang, H.Y., Lee, I.B., Chang, P.W., Hong, S.W., Seo, I.-H., Lee, S.Y., 2006. Monitoring of Dust Emission at a Reclaimed Land and Its 3-D Aerodynamic Modeling. ASABE Paper No. 062055. St. Joseph, Mich.: ASABE.

Javadi, M., Sharifzadeh, M., Shahriar, K., 2010. A new geometrical model for non-linear fluid flow through rough fractures. Journal of Hydrology 389 (1–2), 18–30.
Jean, C., Mazen, T., 2004. Computational modeling of flow over an ogee spillway. Computers and Structures 82, 1805–1812.
Jia, D., Shao, X., Wang, H., Zhou, G., 2010. Three-dimensional modeling of bank erosion and morphological changes in the Shishou bend of the middle Yangtze River. Advances in Water Resources 33, 348–360.
Jung, M.S., Kwon, O.J., 2006. Development of an unstructured overset mesh method for 2-D unsteady viscous flow simulation with relative motion. Korean Society of Computational Fluids Engineering 11 (2), 1–7.
Kang, L., Guo, L., Liu, D., 2008. Reconstructing the vertical distribution of the Aeolian saltation mass flux based on the probability distribution of lift-off velocity. Geomorphology 96, 1–15.
Karmakar, S., Kushwaha, R.L., Lague, C., 2007. Numerical modeling of soil stress and pressure distribution on a flat tillage tool using computational fluid dynamics. Biosystems Engineering 97, 407–414.
Karmakar, S., Kushwaha, R.L., 2005. Simulation of soil deformation around a tillage tool using computational fluid dynamics. Transactions of ASABE 48 (3), 923–932.
Khan, M.N., Odman, M.T., Karimi, H.A., 2005. Evaluation of algorithms developed for adaptive grid air quality modeling using surface elevation data. Computers, Environment and Urban Systems 29, 718–734.
Khan, L.A., Wicklein, E.A., Rashid, M., Ebner, L.L., Richards, N.A., 2008. A case study on the application of a CFD model to the forebay of The Dalles Dam, Oregon. Journal of Hydraulic Engineering 135 (4), 509–519.
Kim, Y.H., Oh, J.S., Seo, I.H., 2003. Analysis of hydraulic characteristics upstream of dam and in spillway using numerical models. Journal of Korea Water Resources Association 36 (5), 761–776.
Kim, S.D., Lee, H.J., An, S.D., 2010. Improvement of hydraulic stability for spillway using CFD model. International Journal of the Physical Sciences 5 (6), 774–780.
Kittas, N. Katsoulas, Bartzanas, T., Tsiropoulos, N., Boulard, T., 2010. Measurements and modelling of pesticides residues in the greenhouse air after low volume spray application. In: International Conference of Agricultural Engineering towards Environmental Technologies – AgEng 2010, Clermont Ferrand 6–8 September, France.
Konig, C.S., Mokhtarzadeh-Dehghan, M.R., 2002. Numerical study of buoyant plumes from a multi-flue chimney released into an atmospheric boundary layer. Atmospheric Environment 36, 3951–3962.
Koo, H.-J., Choi, Y.-J., Kim, K.-R., Byon, J.-Y., 2009. Simulation of detailed wind flow over a locally heated mountain area using a computational fluid dynamics model, CFD-NIMR-SNU. Korean Journal of Agricultural and Forest Meteorology 11 (4), 192–205.
Lai, Y.G., Weber, L.J., Patel, V.C., 2003. Nonhydrostatic three-dimensional method for hydraulic flow simulation II: Validation and application. Journal of Hydraulic Engineering 129, 206–214.
Lee, B.J., Kim, B.S., 2007. Automatic generation of an unstructured surface grid system using CAD surface data. Korean Society of Computational Fluids Engineering 12 (4), 68–73.
Lee, S.-H., Woo, S.-I., 2004. Model tests study on flow-induced vibration of Tainter Gate in Estuary sluices – flow from the gate inside to the gate outside. Transactions of the KSAE 46 (2), 41–47.
Lee, H.J., An, S.D., Jun, K.W., Son, Y.K., 2007. Analysis of flow in the spillway of flood control reservoir using CFD model. Journal of The Korean Society of Hazard Mitigation 7 (4), 51–57.
Li, Y.L., Guo, H., 2006. Comparison of Odor Dispersion Predictions Between CFD Model and CALPUFF Model. 2006 ASABE Meeting Presentation. Paper Number: 064102.
Lin, X.J., Barrington, S., Choiniere, D., Prasher, S., 2007. Simulation of the effect of windbreaks on odor dispersion. Biosystems Engineering 98, 347–364.
Madhani, J.T., Kelson, N.A., Brown, R.J., 2009. An experimental and theoretical investigation of flow in a gross pollutant trap. Water Science Technology 59 (6), 1117–1127.
Maizi, A., Dhaouadi, H., Bournot, P., Mhiri, H., 2010. CFD prediction of odourous compound dispersion: case study examining a full scale waste water treatment plant. Biosystems Engineering 106, 68–78.
Makarov, D., Verbecke, F., Molkov, V., Kotchourko, A., Lelyakin, A., Yanesz, J., Baraldi, D., Heitsch, M., Efimenko, A., Gavrikov, A., 2010. An intercomparison of CFD models to predict lean and non-uniform hydrogen mixture explosions. International Journal of Hydrogen Energy 35 (11), 5754–5762.
Meroney, R.N., 2004. Wind tunnel and numerical simulation of pollution dispersion: a hybrid approach. Working paper, Croucher Advanced Study Institute on Wind Tunnel Modeling, Hong Kong University of Science and Technology 6 (10), 60.
Meselhe, E.A., Odgaard, A.J., 1998. 3D Numerical flow model for fish diversion studies at Wanapum Dam. Journal of Hydraulic Engineering, 1203–1214.
Mohammadi, M., 2008. Boundary shear stress around bridge piers. American Journal of Applied Sciences 5 (11), 1546–1550.
Mohammadi, B., Pironneau, O., 1994. Analysis of the k-ε Turbulence Model, John Wiley and Sons, New York.
Montas, J.H., Prabhakar, G.V.S., Wheaton, F., 2000. CFD analysis of flow in aquaculture ranks. In: Proceedings of the 93rd Annual International Meeting of ASAE. Biological Resources Engineering Department. University of Maryland at College Park, ASAE No 003111, pp. 1–24.
Morvan, D., Dupuy, J.L., 2001. Improvement of the Wildland fire Behavior Model, Fire Star: A Decision Support System for Fuel Management and Fire Hazard Reduction in Mediterranean Wildland – Urban Interfaces, Contract EVG1-CT-2001-00041, p. 18.
Muste, M., Meselhe, E.A., Weber, L.J., Bradley, A.A., 2001. Coupled physical numerical analysis of flows natural waterways. Journal of Hydraulic Research 39, 51–60.
Nagata, N., Hosoda, T., Nakato, T., Muramoto, Y., 2005. Three-dimensional numerical model for flow and bed deformation around river hydraulic. Journal of Hydraulic Engineering 131, 1074–1087.
Nestler, J.M., Goodwin, R.A., Smith, D.L., Anderson, J.J., Li, S., 2008. Optimum fish passage and guidance designs are based in the hydrogeomorphology of natural rivers. River Research and Applications 24, 148–168.
Nguyen, V.T., Nestmann, F., 2010. Application of CFD in hydraulics and river engineering. International Journal of Computational Fluid Dynamics 18 (2), 165–174.
Norton, T., Sun, D.W., Grant, J., Fallon, R., Dodd, V., 2007. Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: a review. Bioresource Technology 98, 2386–2414.
Parviz, M., John, K., 1997. Tackling turbulence with supercomputers. Scientific American 1, 276.
Peternson, E.L., Harris, J.A., Wadhwa, L.C., 2000. CFD modeling pond dynamic processes. Aquaculture Engineering 23, 61–93.
Peternson, E.L., Wadhwa, L.C., Harris, J.A., 2001. Arrangement of aerators in an intensive shrimp growout pond having a rectangular shape. Aquaculture Engineering 25, 51–66.
Pontiggia, M., Derudi M., Busini, V., Rota, R., 2009. Hazardous gas dispersion: a CFD model accounting for atmospheric stability classes. Journal of Hazardous Materials 171, 739–747.
Prospathopoulos, J., Voutsinas, S.G., 2006. Implementation issues in 3D wind flow predictions over complex terrain. Journal of Solar Energy Engineering 128, 539–553.
Pullen, J., Boris, J.P. Young, T., Patnaik, G., Iselin, J., 2005. A comparison of contaminant plume statistics from a Gaussian puff and urban CFD model for two large cities. Atmospheric Environment 39, 1049–1068.
Qiu, G.Y., Lee, I.B., Shimizu, H., Gao, Y., Ding, G., 2003. Principles of sand dune fixation with straw checkerboard technology and its effect on the environment. Journal of Arid Environments 56, 449–464.
Quinn, A.D., Wilson, M., Reynolds, A.M., Couling, S.B., Hoxey, R.P., 2001. Modelling the dispersion of aerial pollutants from agricultural buildings – an evaluation of computational fluid dynamics (CFD). Computers and Electronics in Agriculture 30, 219–235.
Riddle, A., Carruthers, D., Sharpe, A., McHugh, C., Stocker, J., 2004. Comparisons between FLUENT and ADMS for atmospheric dispersion modeling. Atmospheric Environment 38, 1029–1038.
Rong, L., Elhadibi, B., Khalifa, H.E., Nielsen, P.V., Zhang, G., 2011. Validation of CFD simulation for ammonia emissions from an aqueous solution. Computers and Electronics in Agriculture 75, 261–271.
Rosenfeld, M., Marom, G., Bitan, A., 2010. Numerical simulation of the airflow across trees in a windbreak. Boundary-Layer Meteorology 135, 89–107.
Rouholahnejad, E., Sadrnejad, S.A., 2009. Numerical simulation of leachate transport into the groundwater at landfill sites. In: 18th World IMACS/MODSIM Congress, Cairns, Australia, 13–17.
Ruhaak, W., Rath, V., Wolf, A., Clauser, C., 2008. 3D finite volume groundwater and heat transport modeling with non-orthogonal grids, using a coordinate transformation method. Advances in Water Resources 31 (3), 513–524.
Santiago, J.L., Martin, F., Cuerva, A., Bezdenejnykh, N., Sanz-Andres, A., 2007. Experimental and numerical study of wind flow behind windbreaks. Atmospheric Environment 41, 6406–6420.
Scargiali, F., Rienzo, E.D., Ciofalo, M., Grisafi, F., Brucato, A., 2005. Transactions of IchemE, Part B – Process Safety and Environmental Protectio 83 (B5), 1–16.
Schaldach, G., Berger, L., Razilov, I., Berndt, H., 2000. Computer simulation for fundamental studiesand optimisation of ICP spray chambers. ISAS (Institute of Spectrochemistry and Applied Spec-troscopy) Current Research Reports, Berlin, Germany.
Seo, I.H., Lee, I.B., Shin, M.H., Lee, G.Y., Hwang, H.-S., Hong, S.W., Bitog, J.P., Yoo, J.I., Kwon, K.S., Kim, Y.H., Bartzanas, T., 2010. Numerical prediction of fugitive dust dispersion on reclaimed land in Korea. Transactions of the ASABE 53 (3), 891–901.
Sethian, J.A., 1993. Computational fluid dynamics. In: From Desktop to Teraflop: Exploiting the USLead in High Performance Computing, NSF Publications, National Science Foundation, Washing-ton, DC, USA.
Steffens, J.T., Wang, Y.J., Zhang, K.M., 2012. Exploration of effects of a vegetation barrier on particle size distributions in a near-road environment. Atmospheric Environment 50, 120–128.
Stoesser, T., Ruether, N., Olse, N.R.B., 2009. Calculation of primary and secondary flow and boundary shear stresses in meandering channel. Advance, s in Water Resources 33 (2), 158-170.
Tabbara, M., Chatila, J., Awwad, R., 2005. Computational simulation of flow over stepped spillways. Computers and Structures 83, 2215–2224.
Tamura, T., Nozawa, K., Kondo, K., 2008. AIJ guide for numerical prediction of wind loads on buildings. Journal of Wind Engineering and Industrial Aerodynamics 96 (10–11), 1974–1984.
Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., Shirasawa, T., 2008. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. Journal of Wind Engineering and Industrial Aerodynamics 96, 1749–1761.
Veerapen, J.P., Brooks, M.J., Lowry, B.J., Counturier, M.F., 2002. Solids removal modeling in recirculating aquaculture systems. In: Proceedings of the Fourth International Conference on Recirculating Aquaculture. Virginia Tech, pp. 491–498.

Wang, X., Yan, Z., 2007. Three-dimensional simulations for effects of bed discordance on flow dynamics at Y-shaped open channel confluences. Journal of Hydrodynamics 19 (5), 587–593.

Wang, Z., Zou, Q., Reeve, D., 2009. Simulation of spilling breaking waves using a two phase flow CFD model. Computers and Fluids 38 (10), 1995–2005.

Wagenbrenner, N.S., Lamb, B.K., Foltz, R.B., Robichaud, P.R., 2010. Modeling Post-fire Ash and Dust Emissions in Complex Terrain. ASABE Meeting, Paper Number 1008348.

Weber, L.J., Goodwin, R.A., Li, S., Nestler, J.M., Anderson, J.J., 2006. Application of an Eulerian–Lagrangian-Agent method (ELAM) to rank alternative designs of a juvenial fish passage facility. Journal of Hydroinformatics 8 (4), 271–293.

Williamson, N., Komiya, A., Maruyama, S., Benhia, M., Armfield, S.W., 2009. Nutrient transport from an artificial upwelling of deep sea water. Journal of Oceanography 65, 349–359.

Wilson, J.D., 2004. Oblique, stratified winds about a shelter fence I: Measurements. Journal of Applied Meteorology 43, 1149–1167.

Wu, T., Crapper, M., 2009. A computational fluid dynamics based model of the ex-situ remediation of hydrocarbon contaminated soils. Desalination 248, 262–270.

Xia, B., Sun, D.-W., 2002. Applications of computational fluid dynamics (CFD) in the food industry: a review. Computers and Electronics in Agriculture 34, 5–24.

Yassin, M.F., Kellnerova, R., Jañour, Z., 2009. Numerical simulation on pollutant dispersion from vehicle exhaust in street configurations. Environmental Monitoring and Assessment 156 (1–4), 257–273.

Zhang, Y., Yang, J., Liu, D., Wei, X., Yu, L., 2010. Sand storms: CFD analysis of Reynolds stress and collision stress of particles near sand bed. Particuology 8, 325–331.