Ecohydrology & Hydrobiology xxx (2016) xxx–xxx

Contents lists available at ScienceDirect

Ecohydrology & Hydrobiology

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

ELSEVIER

Original Research Article

Applications of computational fluid dynamics in fish and habitat studies

Fariborz Daneshvar, A. Pouyan Nejadhashemi *, Sean A. Woznicki, Matthew R. Herman

Department of Biosystems and Agricultural Engineering, Michigan State University, 215 Farrall Hall, East Lansing, MI 48824, USA

A R T I C L E I N F O

Article history:
Received 25 March 2016
Accepted 22 December 2016
Available online xxx

Keywords:
Flow simulation
Pressure distribution
Sediment transport
Hyporheic region
Fish habitat
Temperature

A B S T R A C T

Computational fluid dynamics (CFD) is defined as a branch of fluid mechanics that solves fluid flow problems using numerical algorithms and methods. Using CFD to model water and contaminants transport in rivers, lakes, and coastal areas can significantly improve our understanding to manage these hydrodynamic systems. In this paper, 111 studies have been reviewed on different aspects of CFD application in ecohydrology that include: flow simulation and pressure distribution, sediment transport, hyporheic zone, fish habitat and temperature distribution. In general, three-dimensional models are reported to be more capable than two-dimensional models to predict flow features and have been used more recently due to their ability to capture secondary flow. Two-dimensional models with secondary flow correction terms are acceptable and are being used as well. Discretization techniques and turbulence closures are also reported with comparisons. In general, majority of the studies were focused on flow and sediment transport; however, our knowledge about temperature distributions in near-bed regions and pool-riffle structures is limited and can be the subject of future studies.

1. Introduction

The hydrological cycle, which describes the transfer of water through the atmosphere, across the Earth's surface, and through the lithosphere, is a fundamental component of ecohydrology. Ecohydrology is an interdisciplinary field that focuses on the effects of hydrological processes on ecosystems and the effects of biotic processes on the hydrological cycle (Zalewski et al., 1997; Hannah et al., 2004). Furthermore, both physical (e.g. flow velocity) and structural (e.g. riverbed) variations within waterbodies impact aquatic ecosystems (Leclerc, 2005). Therefore, understanding how these variations change throughout a region is necessary for ecohydrological evaluation of river systems. In particular, streamflow is considered as the Master Variable (Karr, 1991; Poff et al., 1997) for riverine systems; therefore, both experimental and numerical approaches have been used to describe the relevance of streamflow to aquatic ecosystems (Sawyer et al., 2012; Plymesser and Cahoon, 2013). However, in general, numerical modeling is preferred over experimental approaches for several reasons, including lower development cost, faster execution time, and transferability to different regions (Papanicolaou et al., 2008; Abouali and Castillo, 2013).

Computational fluid dynamic (CFD) methods are numerical based approaches that solve flow equations to obtain velocity, pressure, and temperature distributions (Lane et al., 1999). This makes CFD an ideal tool for ecohydrologists, allowing them to better understand how

* Corresponding author. Fax: +1 517 432 2892.
E-mail address: pouyan@msu.edu (A.P. Nejadhashemi).

http://dx.doi.org/10.1016/j.ecohyd.2016.12.005

Please cite this article in press as: Daneshvar, F., et al., Applications of computational fluid dynamics in fish and habitat studies. Ecohydrol. Hydrobiol. (2017), http://dx.doi.org/10.1016/j.ecohyd.2016.12.005

flow patterns within environments interact with aquatic ecosystems. This study aims to synthesize the existing body of knowledge concerning the application of CFD in fish and habitat studies. The paper was organized into three sections: (1) introduction of CFD and its governing equations, (2) presentation of recent applications of CFD in ecohydrology, and (3) identification of knowledge gaps and possible future works.

1.1. Governing equations

In a system, which is characterized and controlled by moving fluid, pressure and velocity distributions can be estimated using CFD methods. In particular, CFD is used to solve the conservation of mass and conservation of momentum. Conservation of mass states that the rate of mass change in a given system must be in balance with the rate of mass entering and leaving that system. For incompressible flows, the conservation of mass can be presented as follows:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

where u, and x represent flow velocity and direction, respectively.

Conservation of momentum states that the rate of linear momentum change of a given system is in balance with external forces. The conservation of momentum equation for a viscous fluid is called the Navier–Stokes equation and for Newtonian incompressible flows (such as water) is as follows:

$$\frac{\partial u_i}{\partial t} + u_j\frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \nu\frac{\partial}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j}\right) + f_i \tag{2}$$

where ρ, ν, and P are fluid density, kinematic viscosity, and pressure respectively. The body forces (e.g. gravity) are described by the term f. Depending on the dimensions of study, i can range from 1 to 3 (Versteeg and Malalasekera, 2007).

1.2. Turbulent flow

The governing equations (1) and (2) are valid to determine the pressure and velocity distributions in laminar flows; however, for turbulent flows, additional terms are needed to capture fluctuations as follows:

$$u_i = \overline{u}_i + u_i' \tag{3}$$

$$P = \overline{P} + P' \tag{4}$$

where $\overline{u}_i$ and $\overline{P}$ are the average flow velocity and pressure and u_i' and P' are the flow velocity and pressure fluctuation terms, respectively.

In order to solve turbulent flow terms, two general approaches have been used: (1) turbulent flow simulation and (2) turbulent flow modeling (Pope, 2000).

1.2.1. Turbulent flow simulation

Turbulent flow simulation approaches solve the coupled equations (conservation of mass and momentum) for time-dependent velocities (with turbulent effects) for a given time period (Pope, 2000). One such approach is the Direct Numerical Simulation (DNS), in which all spatial and temporal ranges of turbulence flow need to be resolved (Orszag, 1970). However, the computation cost of DNS is proportional to the third power of the Reynolds number ($\propto Re^3$), which makes it difficult to use DNS for many applications (Pope, 2000). As a result, this approach is limited in its applications and is often only used for basic fundamental research (Moin and Mahesh, 1998).

Large Eddy Simulation (LES) is a more recent approach that divides turbulent motion into large and small scale eddies using a space filter function (Smagorinsky, 1963). In fully developed turbulent flow, the energy transfers from larger to smaller eddies through the concept called energy cascade. Large scale turbulent eddy motions, that are affected by boundary conditions, can be directly simulated. While for small scale in which eddies are relatively independent from boundary conditions, the turbulence flow should be modeled. Filtering large and small scale eddies in LES are accomplished by solving Eq. (5):

$$\overline{\varphi}_i(\vec{x}) = \int G(\vec{x}-\vec{\delta})\varphi(\vec{\delta})d\vec{\delta} \tag{5}$$

where G is the filter convolution kernel that differentiates between small scale variations (fluctuations) and the large scale flow movements based on a defined threshold. As a result, φ (e.g. velocity or pressure) will be decomposed into large scale ($\overline{\varphi}_i$) and sub-grid scale (φ_i') parts as follows (Pope, 2000; Sagaut, 2006):

$$\varphi_i = \overline{\varphi}_i + \varphi_i' \tag{6}$$

1.2.2. Turbulent flow modeling

For turbulent flow modeling, governing equations are averaged to get the mean pressure and velocity distributions. By combining Eqs. (1) through (4), the Reynolds Averaged Navier–Stokes (RANS) equations can be derived for incompressible Newtonian fluid flows as follows (Hinze, 1975):

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{7}$$

$$\rho\frac{\partial \overline{u}_i}{\partial t} + \rho\overline{u}_j\frac{\partial \overline{u}_i}{\partial x_j} = -\frac{\partial \overline{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right)\right] + \frac{\partial \tau_{ij}}{\partial x_j} + f_i \tag{8}$$

where τ is the Reynolds stress tensor representing the turbulence effect on fluid flow and is defined as follows (Bates et al., 2005):

$$\tau_{ij} = -\rho\overline{u_i' u_j'} \tag{9}$$

Turbulence models also known as closure models have been developed to estimate the Reynolds stress term (τ_{ij}) that is used to solve Eqs. (7) and (8).

Turbulent models can be divided into two main groups (Pope, 2000): (1) turbulent viscosity models; and (2) Reynolds stress models (RSMs).

Turbulent viscosity models obtain Reynolds stress values from turbulent viscosity by solving algebraic or